

EspressDashboard

Version *7.1*

EspressDashboard 7.1 User's Guide

EspressDashboard User's guide	vii
Q. QuickStart	1
Q.1. Overview	1
Q.1.1. Setup	1
Q.2. Getting Started	1
Q.2.1. Create a User	2
Q.2.2. Start the Organizer	3
Q.3. Setup Data Sources	8
Q.3.1. Create a Data Registry	8
Q.4. Chart Designer	21
Q.4.1. Chart Mapping	21
Q.4.2. Basic Chart Formatting	28
Q.4.3. Advanced Charting Features	33
Q.5. QuickDesigner Reports	36
Q.5.1. Create a Query	36
Q.5.2. Create a Report	41
Q.6. QuickDesigner Charts	46
Q.6.1. Create a Chart	47
Q.7. Online Maps	51
Q.7.1. Create Coordinates	51
Q.7.2. Create Online Map	55
Q.8. SVG Maps	62
Q.8.1. Create a Map	62
Q.8.2. Set SVG Map Thresholds	64
Q.8.3. Set SVG Map Drilldowns	67
Q.9. Publishing	68
Q.9.1. Dashboards	68
Q.9.2. URLs	80
Q.9.3. The Menu Page	81
Q.10. Alerts	82
Q.10.1. Alerts in Charts	82
Q.10.2. Alerts in Maps	84
Q.10.3. Alerts in Dashboards	85
Q.10.4. Alert Monitoring	86
1. Administration	91
1.1. Overview & Architecture	91
1.1.1. EDAB Components	91
1.1.2. EDAB Architecture	92
1.2. New Features List V7.0	92
1.2.1. New EspressDashboard Features	92
1.3. Installation and Configuration	94
1.3.1. Installing EDAB	94
1.3.2. Configuration	103
1.3.3. Backward compatibility patches	106
1.3.4. Run Applets in WebStart with JNLP file	108
1.4. Administration	109
1.4.1. The Console Interface	109
1.4.2. User Settings	137
1.A. EDAB Details Log	138
1.B. Server Commands	143
2. Organizer	147
2.1. Using the Organizer	147
2.1.1. Starting the Organizer	147
2.1.2. The Organizer Interface	148
2.1.3. Projects and Folders	151
2.1.4. Files	152
2.1.5. URL Mapping	155
2.1.6. Repairing Broken Links	155

2.1.7. Update Directory	156
2.1.8. Searching in Organizer	159
2.1.9. Limiting Browse Directories	160
2.2. Security and Security Administration	160
2.2.1. Security Concepts	160
2.2.2. Setting User Privileges	162
2.2.3. Security Levels	165
3. Data Sources	167
3.1. Data in Organizer	167
3.1.1. Managing Data Registries	167
3.1.2. The Data Source Manager	169
3.1.3. Data from a Database	172
3.1.4. Data from XML and XBRL Files	205
3.1.5. Data from Text Files	209
3.1.6. Data from Class Files	210
3.1.7. Data from EJBs	211
3.1.8. Data from SOAP with WSDL support	213
3.1.9. Data from Salesforce	215
3.1.10. Data from Excel files	218
3.1.11. Using Data for Charts	220
3.1.12. Data Source Updating	223
3.1.13. CData JDBC drivers	225
3.2. Data in QuickDesigners and Maps	230
3.2.1. Select a Data Source	230
3.2.2. Queries	231
4. Designing Reports & Charts	244
4.1. Chart Designer	244
4.1.1. Introduction to Chart Designer	244
4.1.2. Charting Basics	245
4.1.3. Chart Types and Data Mapping	250
4.1.4. The Chart Designer Interface	282
4.1.5. Chart Drill-Down	349
4.1.6. Saving & Exporting Charts	354
4.2. QuickDesigner Reports	357
4.2.1. Start	357
4.2.2. Select a Data Source	358
4.2.3. Toolbar	359
4.2.4. Format Report Elements	360
4.2.5. Parameter Setting	366
4.2.6. Aggregation/Group	367
4.2.7. Data Formatting	369
4.2.8. Sorting	372
4.2.9. Data Filtering	372
4.2.10. Save the Report	373
4.2.11. Export the Report	373
4.2.12. Open the Saved Report	374
4.2.13. Exit	374
4.3. QuickDesigner Charts	374
4.3.1. Start	375
4.3.2. Select a Data Source	375
4.3.3. Data Mapping and Ordering	376
4.3.4. Toolbar	378
4.3.5. Data Formatting	379
4.3.6. Save the Chart	399
4.3.7. Export the Chart	400
4.3.8. Open the Saved Chart	401
4.3.9. Exit	402
5. Designing Maps	403

5.1. Introduction to EDAB Maps	403
5.2. Online Maps	403
5.2.1. Generating Google Maps API Key	403
5.2.2. Data Sources	404
5.2.3. Start Online Maps	405
5.2.4. Coordinates Editor	406
5.2.5. Coordinates Mapping	415
5.2.6. Create Online Maps	416
5.2.7. Save Map	428
5.2.8. Open Saved Map	429
5.2.9. Exit	430
5.3. SVG Maps	430
5.3.1. Introduction to SVG Maps	430
5.3.2. Set Area IDs Using Inkscape Editor	430
5.3.3. Area ID Mapping	431
5.3.4. SVG Maps Designing	431
5.3.5. Dynamic SVG Maps	442
5.3.6. Save Map	446
5.3.7. Open Saved Map	447
5.3.8. Exit	447
5.4. Map Viewer	447
5.4.1. Writing Map Viewer URL	448
5.5. Migrating Maps	449
5.A. List of SVG Map Images	449
6. Designing Dashboards	451
6.1. Introduction to Dashboards	451
6.2. Create Dashboard	452
6.2.1. Toolbar	453
6.2.2. Responsive Dashboard	454
6.2.3. Add Charts, Reports and Maps	455
6.2.4. Shared Parameters	458
6.2.5. Insert Labels	473
6.2.6. Add Background	475
6.2.7. Additional Options	476
6.2.8. Drilldown	478
6.2.9. Folders	479
6.2.10. Template Linkage	483
6.2.11. Dashboard Preview	484
6.3. Dashboard Migration	484
6.3.1. Migrating from previous EDAB versions	484
6.4. Save Dashboard	486
6.5. Open Saved Dashboard	487
6.6. Exit	488
7. Publishing (Menu & URLs)	489
7.1. The Menu Page	489
7.1.1. Launching the Menu Page	489
7.1.2. Using the Menu Page	489
7.1.3. Mobile MenuPage	495
7.2. Dashboard URLs	499
7.2.1. Generating URLs in Organizer	499
7.2.2. Writing Dashboard URLs	501
7.3. Menu Page Listener	502
7.3.1. EDAB Listener Manager	502
7.3.2. Using the Menu Page Listener	503
7.4. Dashboard Viewer	505
7.4.1. Preview Toolbar	505
7.4.2. Preview Options	506
8. Programming	510

8.1. Managing Users and Groups	510
8.1.1. Introduction	510
8.1.2. Users and Groups	510
8.1.3. Single Sign-On	511
8.1.4. Login Listener	512
8.1.5. Javadoc	514
8.1.6. Summary	514
8.2. EDAB Menu API Overview	514
8.2.1. Introduction and Setup	514
8.2.2. Using the API	514
8.2.3. Javadoc	517
8.2.4. Summary	517
8.3. Organizer	517
8.3.1. Introduction	517
8.3.2. Look and Feel	518
8.3.3. Inserting and Removing Items	518
8.3.4. Javadoc	519
8.3.5. Summary	519
8.4. Menu Page Listener	519
8.4.1. EDAB Listener Manager	519
8.4.2. Using the Menu Page Listener	520
8.5. Class File Data Source	521
8.5.1. Non Parameterized	521
8.5.2. Parameterized	522
8.5.3. Custom Selection Choices	523
9. Configuration	525
9.1. Using Other Databases	525
9.1.1. Create Table Scripts	525
9.1.2. Specifying the Database Connection	526
9.1.3. Migrating the EDAB Database	527
9.2. Integrating Existing Users/Groups	528
9.2.1. Using a Database	528
9.2.2. Implementing UserSecurityProvider	530
9.3. Using Other Application Servers	535
9.3.1. Tomcat 4.1/5.x/6.0.x/7.0.x	537
9.3.2. Resin	537
9.3.3. WebLogic Server	538
9.3.4. WebSphere	544
9.3.5. JBoss 3.2.3 (with Tomcat 5.0) / JBoss 4.0.5/6.0.0	551
9.3.6. Oracle Containers for J2EE (OC4J) 10g (9.0.4.0/10.1.3.5)	552
9.3.7. Oracle Application Server 10g R3 (10.1.3)	553
9.3.8. GlassFish Server 3.0.1	554
9.4. Clustering/Load Balancing	555
9.4.1. Tomcat 5.0	556
9.4.2. JBoss 3.2.5 (with Tomcat 5.0)	562
9.4.3. JRun 4 (with Apache Web server)	564
9.4.4. WebSphere 6.0	566
10. Internationalization	568
10.1. Internationalizing EDAB	568
10.1.1. Specifying Locales	568
10.1.2. Language and Encoding	568
11. Alerts	574
11.1. What is an Alert	574
11.2. Specifying alerts	574
11.2.1. Online Maps	574
11.2.2. SVG Maps	574
11.2.3. Charts	574
11.3. Alert types	575

11.3.1. Dashboard alerts	575
11.3.2. Visual alert display	575
11.3.3. Alert messages	575
11.3.4. Sound alert	576
11.3.5. User interface	576
11.4. Monitoring	576
11.4.1. Alert notifications	577
11.4.2. Failed emails	577
11.4.3. User interface	577
11.4.4. Monitoring log	590
11.5. Handling special situations	595
11.5.1. Change in script/control area/range	595
11.5.2. Missed monitoring jobs	596

EspressDashboard 7.1

EspressDashboard User's guide

Welcome to the EspressDashboard Online User's Guide.

Here you will find information about all the functions and features in EspressDashboard. For additional API resources, please refer to the API JavaDoc for both the Report API and EDAB API.

EspressDashboard Users guide consists of several elements:

QuickStart	This is a good starting point for EspressDashboard first time users. It explains some of the most commonly used features, and provides examples of administration, report and chart design, and publishing.
Administration	This section provides an overview of EDAB along with installation instructions, and information for administering the reporting environment.
Organizer	This section explains how to use the Organizer interface, security features, and provides information about using and connecting to different data sources.
Designing Reports & Charts	This section explains how to use all the design tools including the Chart Designer, thin-client QuickDesigner Reports and QuickDesigner Charts.
Designing Maps	This section explains how to use Map Builder interface for creating Google and Bit maps.
Designing Dashboards	This section explains how to use Dashboard Builder interface to create and deploy dashboards as well as migrating Dashboards from one EDAB Server to another.
Publishing (Menus & URLs)	This section covers the built-in publishing features available with EDAB.
Programming	This section explains how to use the EDAB API to manage users and groups, customize the menu page, call the Organizer, and create a menu page listener.
Configuration	This section explains how to configure EspressDashboard to run with different application servers and databases.
Internationalization	Explains how to set EDAB to create dashboards in virtually any language and locale-specific formatting.
Alerts	Explains how to monitor your KPI (Key Performance Indicator) alerts.

If you upgraded from an older EDAB version, you might be interested in seeing the list of V7.0 new features.



Tip

If you're reading the PDF version, enable *Bookmarks* toolbar in your PDF viewer.

Please select an option from the menu on the left to begin.

QuickStart

Q.1. Overview

Welcome to the EspressoDashboard Quick Start Guide. This guide is the recommended starting point for users who are new to EDAB and would like to get up to speed fairly quickly. This guide can also help users to quickly evaluate the product capabilities.

This guide provides step-by-step tutorials for basic administration features, report and chart creation, and publishing features. Only a small portion of the features are described in this guide, so for more information and detail, please refer to the other sections in the User's Guide.

Q.1.1. Setup

The QuickStart assumes that you are working with EDAB in default configuration. This means a clean installation using the Tomcat server that comes with EDAB and running connected to the default HSQL database. For more information about installation options, please see Section 1.3.1 - Installing EDAB. The information and exercises in this guide can be used in other configurations, however, references to certain options and file paths may not be applicable.

For more information about EDAB components and architecture, see Section 1.1.1 - EDAB Components.

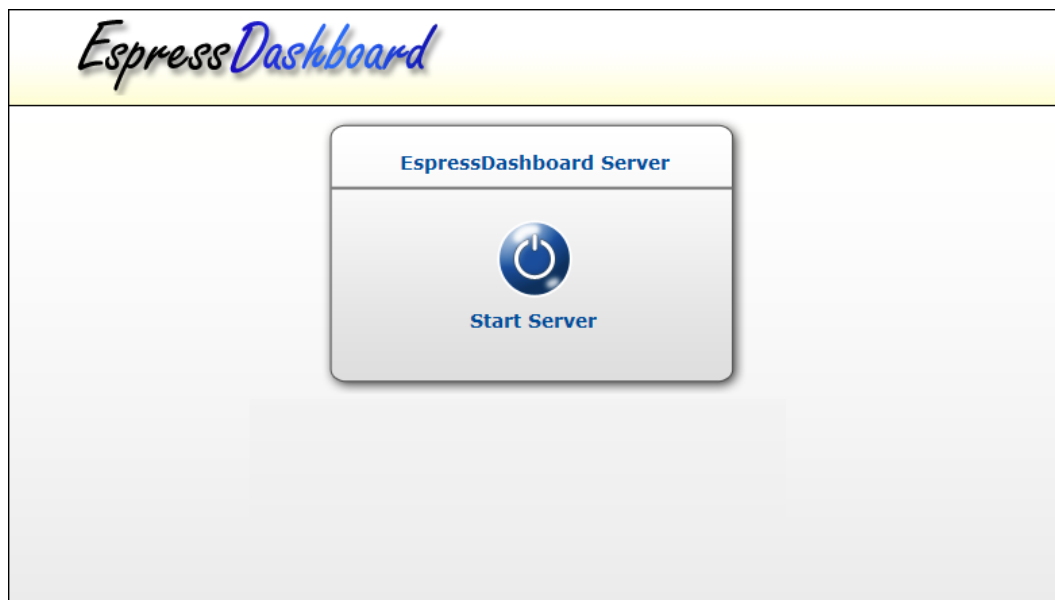
Q.2. Getting Started

The first step to run EDAB is to start the application server, if it didn't auto-start (installation default). In this case, it means starting the Tomcat server that comes with the EDAB installation. To start the Tomcat, execute `startup.bat/.sh` in the `/bin/` directory of your Tomcat installation. If you installed EDAB on Windows, you can also start Tomcat from a shortcut in the Start Menu.

After the Tomcat is successfully started, you can load the Start page for EDAB from the following location:

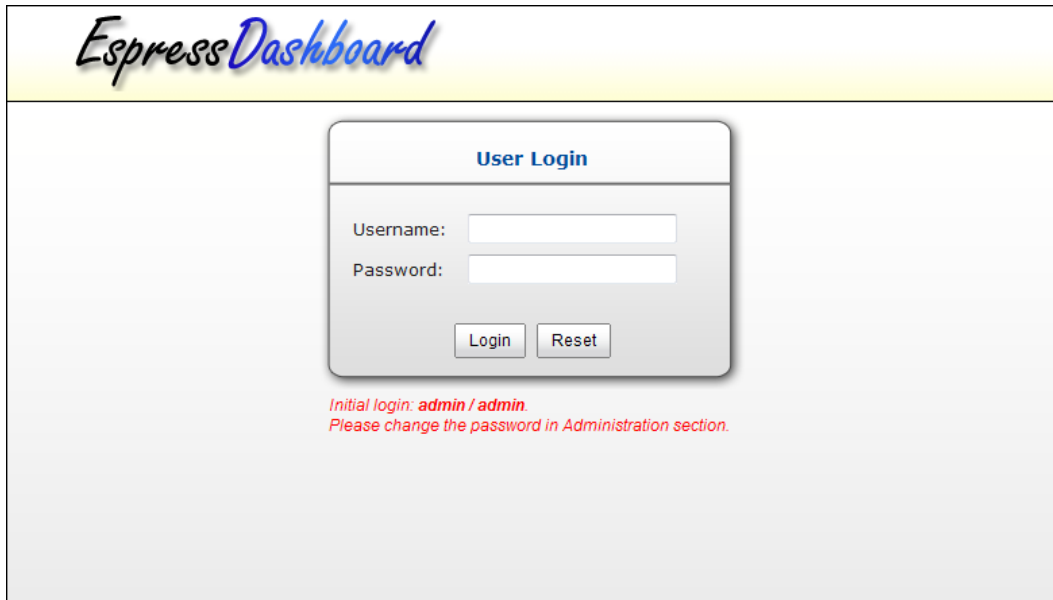
`http://<machinename>:<port>/EDAB/index.jsp`

Replace `<machinename>` with your server hostname or IP address, and replace port with the port you specified for Tomcat during installation. The default port is 8080. If you installed EDAB on Windows, you can launch the Start page from the Start Menu.



EDAB Start Page - EDAB Server is OFF

Before we start working with EDAB, the EDAB Server needs to be running. If the Autostart feature has been enabled during installation process (see Section 1.3.1 - Installing EDAB for more details), the EDAB Server should be now running. If the server is off, turn it on by clicking on the *Start Server* button.

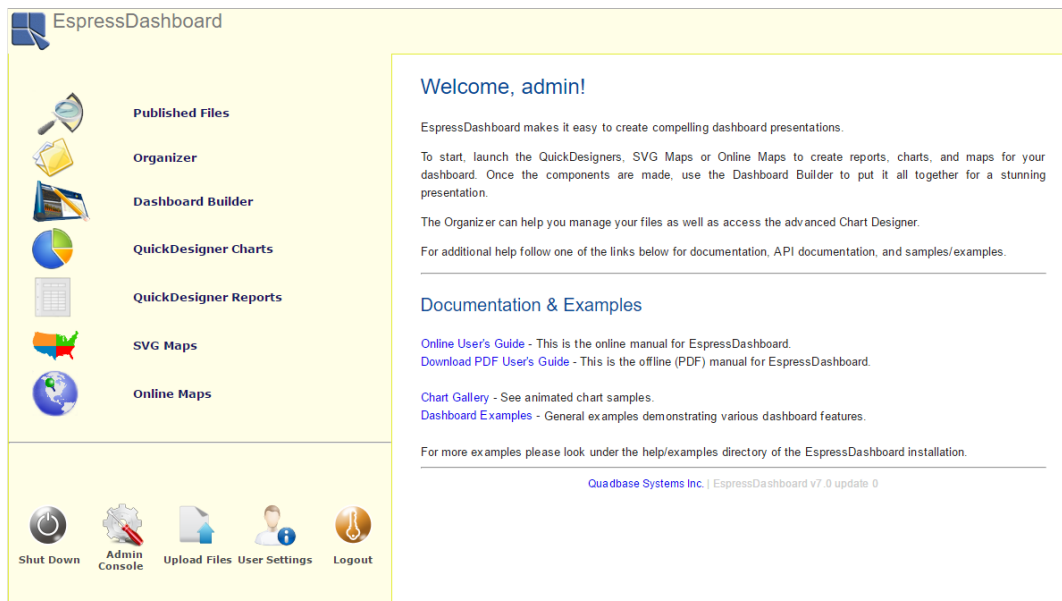


EDAB Server Started

Now you need to first login before continuing.

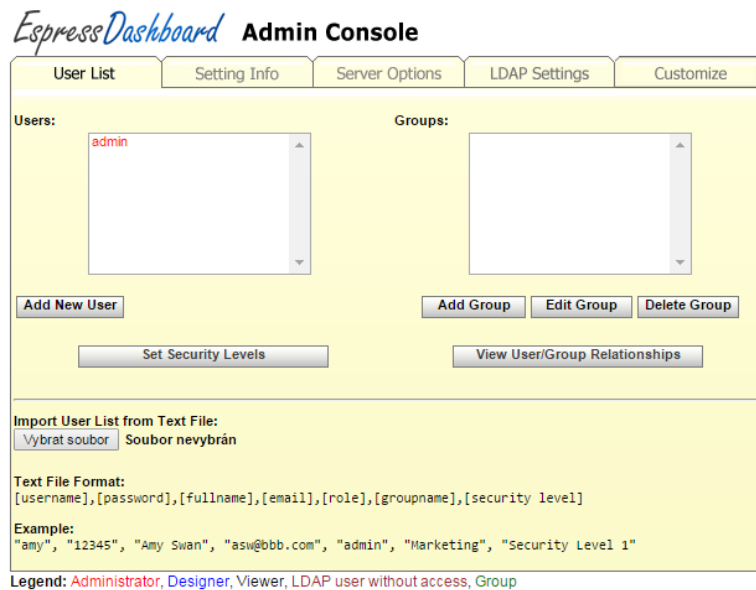
Q.2.1. Create a User

With the server running, login as the system administrator. The default login for the administrator is username: **admin**, and password: **admin**. Once you have logged in, the Start page will be displayed.



Admin Logged into Start Page

To launch the Admin Console, click the *Admin Console* button in the lower left corner of the Start page.



EspressoDashboard Admin Console

Users: admin

Groups:

Add New User Add Group Edit Group Delete Group

Set Security Levels View User/Group Relationships

Import User List from Text File:
 Vybrat soubor Soubor nevybrán

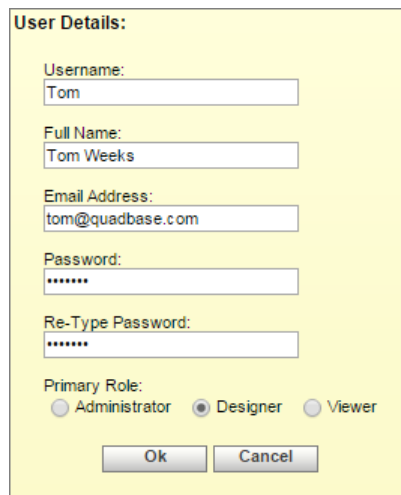
Text File Format:
 [username],[password],[fullname],[email],[role],[groupname],[security level]

Example:
 "amy", "12345", "Amy Swan", "asw@bbb.com", "admin", "Marketing", "Security Level 1"

Legend: Administrator, Designer, Viewer, LDAP user without access, Group

Admin Console

The Admin Console opens the *User List* tab. By default, the administrator is the only defined user. To add a new user to the system, click the *Add New User* button. This will open a window allowing you to add details about a new user.



User Details:

Username:

Full Name:

Email Address:

Password:

Re-Type Password:


Primary Role:
☐ Administrator ☒ Designer ☐ Viewer

Ok Cancel

Add User Dialog

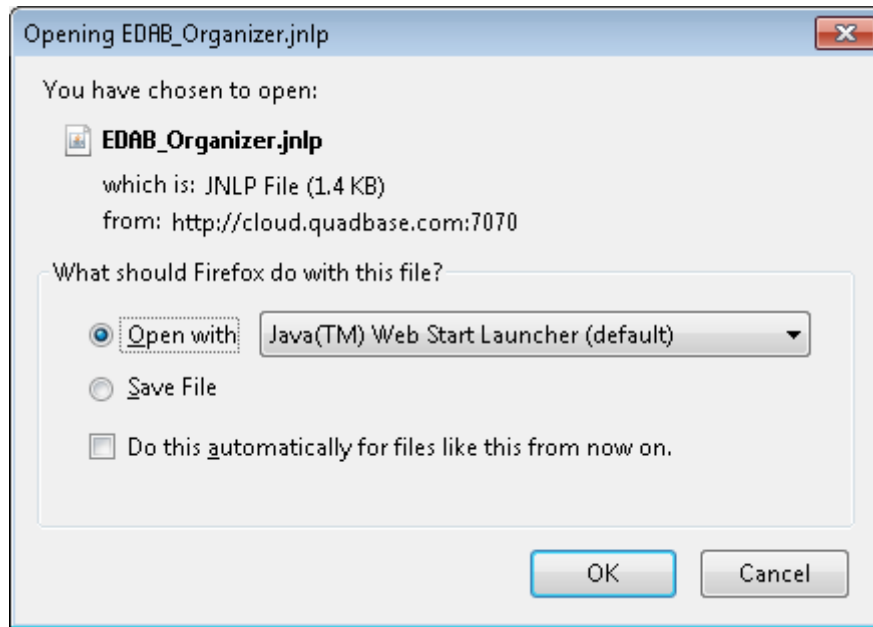
Enter details for a new user and make sure to select *Designer* as the primary role. Please note that if you select *Viewer* as the primary role, you will not have access to the core design/development tools like Chart Designer, QuickDesigner Reports, QuickDesigner Charts and Organizer. For more information about creating users, please visit Section 1.4.1.1 - User List.

When you're done, click the *Ok* button to create a new user. The dialog will close and the new user will be added to the user list in the Admin Console.

Now that you have entered a new user, return to the Start page by clicking on the  **Home** Home icon in the upper left corner of the Admin Console.

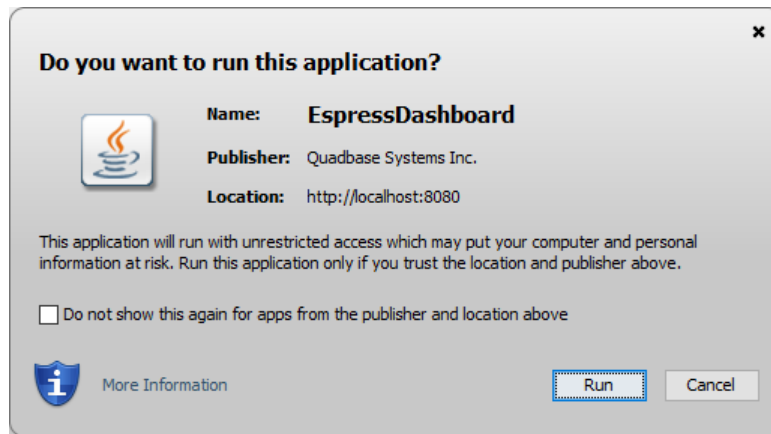
Q.2.2. Start the Organizer

While we are still logged in as admin, let's do one more thing that will facilitate some of the exercises later. Click on the *Organizer* in the left menu of the Start page. This will open a pop-up window. Confirm `EDAB_Organizer.jnlp` file open by Java Web Start Launcher and the Organizer will open.



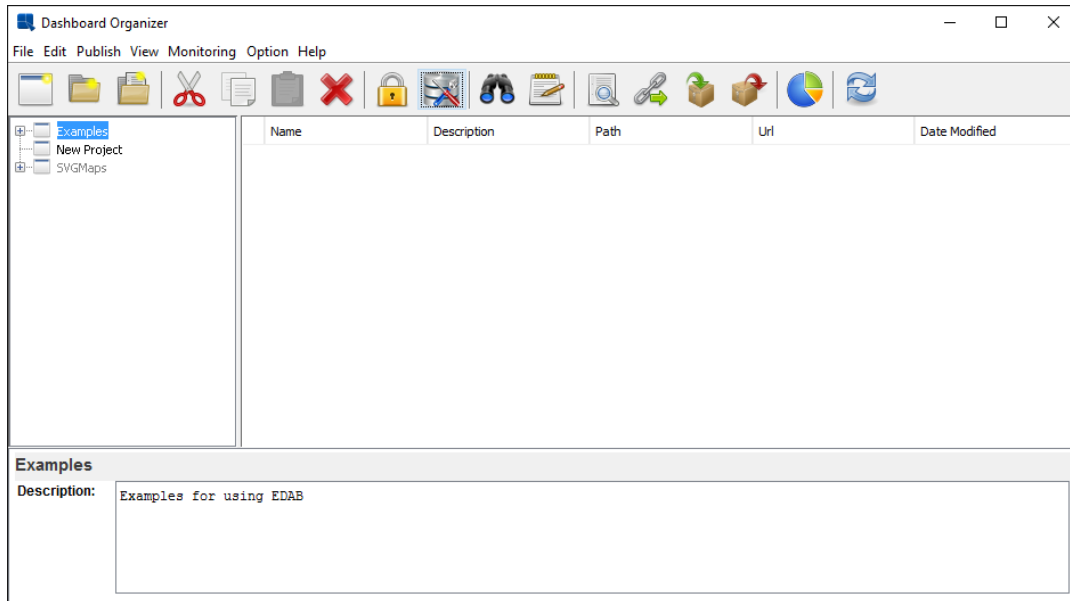
Opening Organizer

If you're using a different browser like Chrome, you have to save the `EDAB_Organizer.jnlp` file first and then open it from your download directory. You will see a window with a question: "Do you want to run this application?" Click the *Run* button and the Organizer will open.



Opening Organizer in Chrome

Once the Organizer is opened, you will see this window:

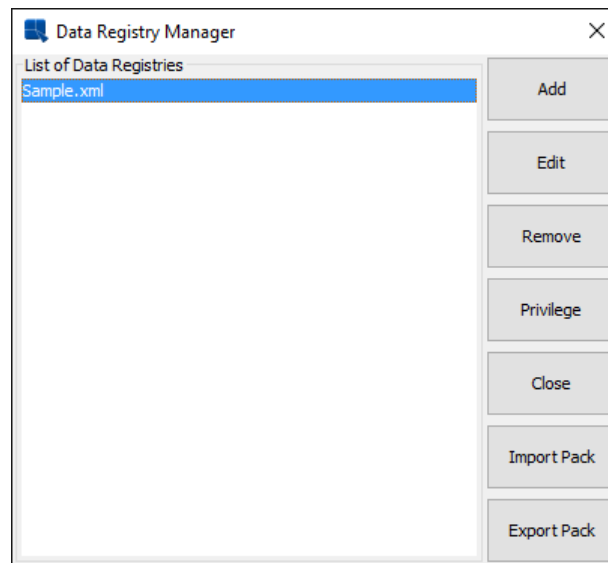


Organizer Interface

The Organizer is a virtual file management system that allows you to create, manage, publish, and schedule reports and charts. The Organizer is used in conjunction with most of the other interfaces in EDAB and will be referenced extensively throughout this guide.

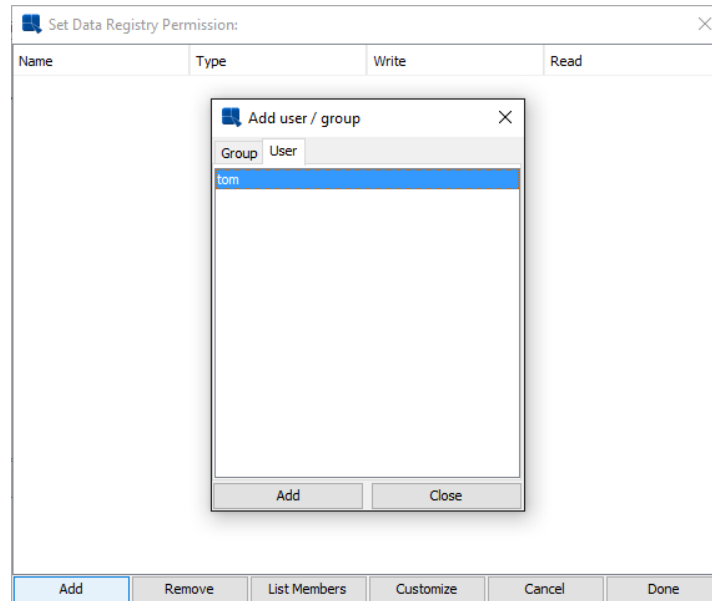


Once the Organizer is opened, click the *Manage Data Sources* button and the Data Registry Manager window will pop-up. Highlight `Sample.xml` and click the *Privilege* button in the lower right corner of the window.



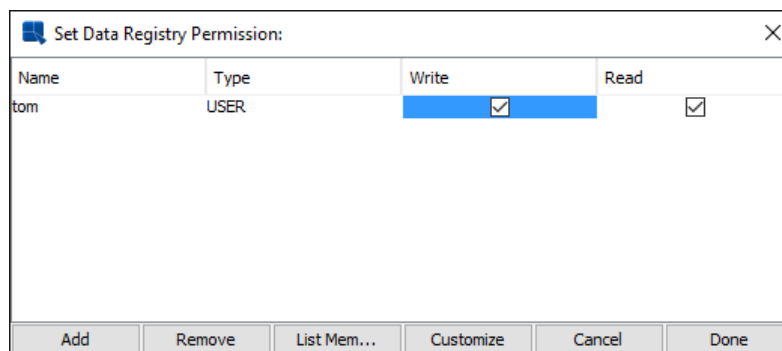
Data Registry List

A new dialog *Set Data Registry Permission* will be displayed. Click the *Add* button and switch to the *User* tab. Highlight the user you previously created via Admin Console and click the *Add* button. Close the window by clicking the *Close* button.



Add Data Registry Privilege

Now check the *Write* and *Read* checkboxes in the Set Data Registry Permission window and click the *Done* button to close it.



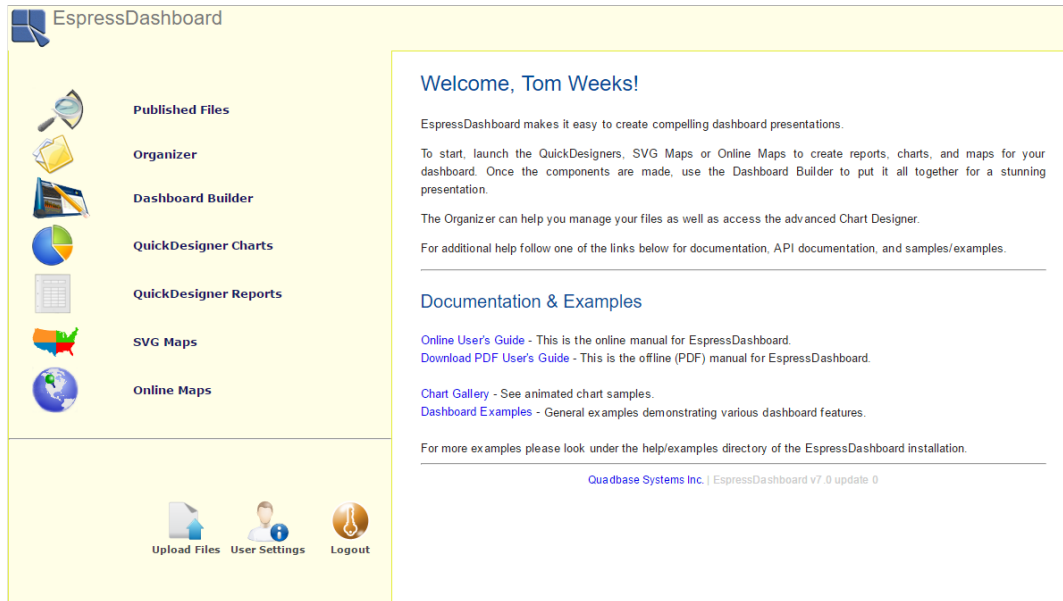
Data Registry Read Write Privilege

Close the Data Registry Manager window by clicking on the *Close* button and then close the Organizer. You should now be on the Start Page.

Q.2.2.1. Login as User

We are done with all the administration activities, so you can now login as the designer user you created previously. On the Start page, click the *Logout* button. This will logout the administrator out of the system.


Now log in again using the user that you created in Section Q.2.1 - Create a User. After you login, notice that the administration functions are not available. Click on the *Organizer* on the left to open the Organizer.



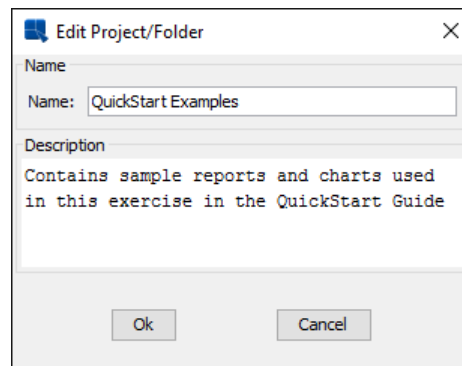
User Logged into Start Page

Q.2.2.2. Add a Project

Reports and charts in the Organizer are grouped into projects and folders. Now we are going to create a new project in which to add reports and charts that we will use later in this guide. To add a new project, click the *New Project*

button on the toolbar . A new node will be added with the name *New project* or *New project(1)*, if a node of that name already exists.

To edit the new project, right click on the new node and select *Edit* from the pop-up menu. This will bring up a dialog allowing you to re-name the project and to specify a description.




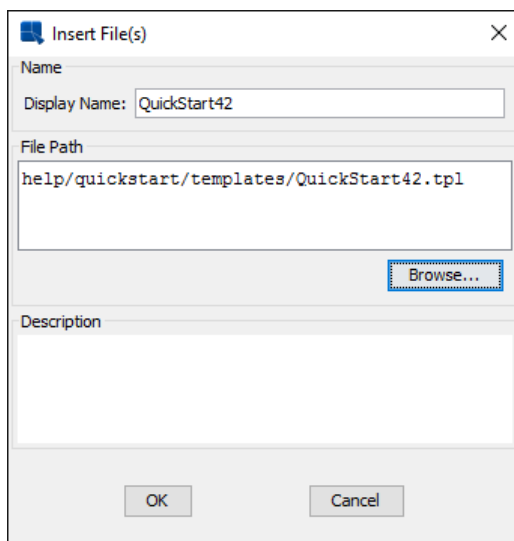
Edit Project Dialog

Specify a new name and description for the project and click *Ok*. The changes will be applied. When you select a project in the Organizer, you will see its description at the bottom of the page.

Q.2.2.3. Inserting a File

If you do not have the Organizer open, open it. Then select your project in the left side window. Next, click the

Insert File button on the toolbar.  a new dialog allowing you to select a file to insert into the Organizer.




Insert File Dialog

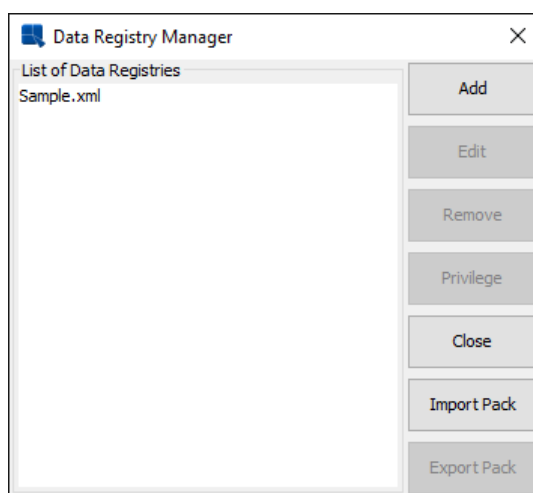
In the dialog, click the *Browse* button, navigate to `help/quickstart/templates` and select `QuickStart42.tpl`. The display name and corresponding URL should be automatically filled. Click the *Ok* button to add the chart. The dialog will close and you will see an entry in the Organizer for the newly added chart.

Q.3. Setup Data Sources

Data sources in EDAB are maintained in XML data registry files. These files are created and managed from within the Organizer. Detailed documentation about the data sources can be found in Section 3.1 - Data in Organizer.

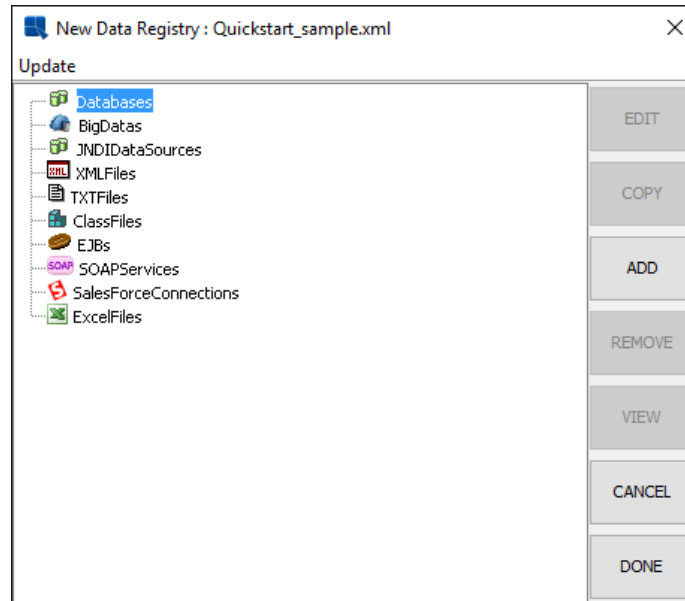
Q.3.1. Create a Data Registry

To create a new data registry, open the Organizer and click the *Manage Data Sources* button . This will pop up a dialog containing all the registries defined in EDAB.



Registry List Dialog

Click the *Add* button in this dialog to add a new data registry. A dialog will open prompting you to specify a name for the registry. Enter any name you like (e.g. **Quickstart_sample**) and click the *Ok* button. The registry will be created and the Data Source Manager window for the new registry will open.



Data Source Manager Window

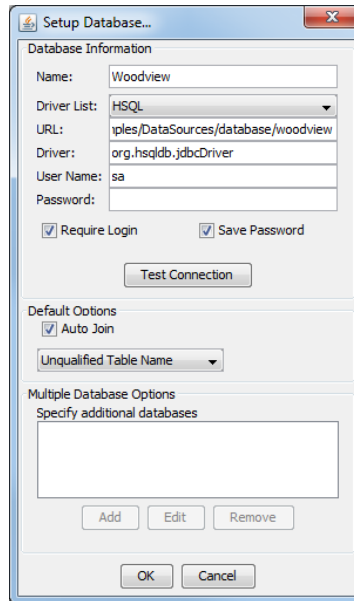
Q.3.1.1. Setup Database Connections

EDAB allows you to connect to JDBC compliant data sources. Few examples are included in your installation.

Q.3.1.1.1. Setup a JDBC Connection

In this tutorial, we will set up a JDBC connection to the Woodview HSQL database that comes with the EDAB Installation. If you are running EDAB using the configuration described in Section Q.1.1 - Setup then the HSQL JDBC driver will already be in the classpath of your Tomcat server, as HSQL is also used as the default EDAB database. If this is not the case, or if you are using a different database for EDAB, you will need to make sure that your database driver (hsqldb.jar for HSQL) is in the <EDABInstallDir>/WEB-INF/lib directory, or in Tomcat's classpath.

The classpath and Woodview sample database were already created during installation, so you can select the Woodview database in your registry. From the Data Source Manager window, click on the *Databases* node in the left panel, select Woodview and hit the *Edit* button to reach the following *Setup Database* panel, where HSQL is on the *Driver List*, **`jdbc:hsqldb:help/examples/DataSources/database/woodview`** for the *URL*, and **`org.hsqldb.jdbcDriver`** as the *Driver*. Click on both the *Require Login* and *Save Password* boxes. Next, enter **`sa`** for the *User Name* and leave the *Password* blank will connect to the built-in HSQL database.



Add Database Dialog

The procedure for connecting to other databases is very similar. First, select the type from the database list and then fill in the values. The JDBC Driver will be automatically provided so it is usually not necessary to modify this field. Here is an example connecting to a MySQL database.

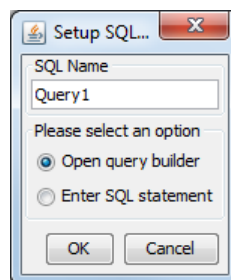
```
URL: jdbc:mysql://192.168.0.55:3306/woodview
Driver: com.mysql.jdbc.Driver
User Name: root
Password: *****
```

Leave the *Auto Join* and *Table Name* properties alone, and click the *Test Connection* button to make sure you've entered the information correctly. Click *OK* to bring up the Data Source Manager window, where there will be an existing node under *Databases* for Woodview.

Q.3.1.2. Create a Query

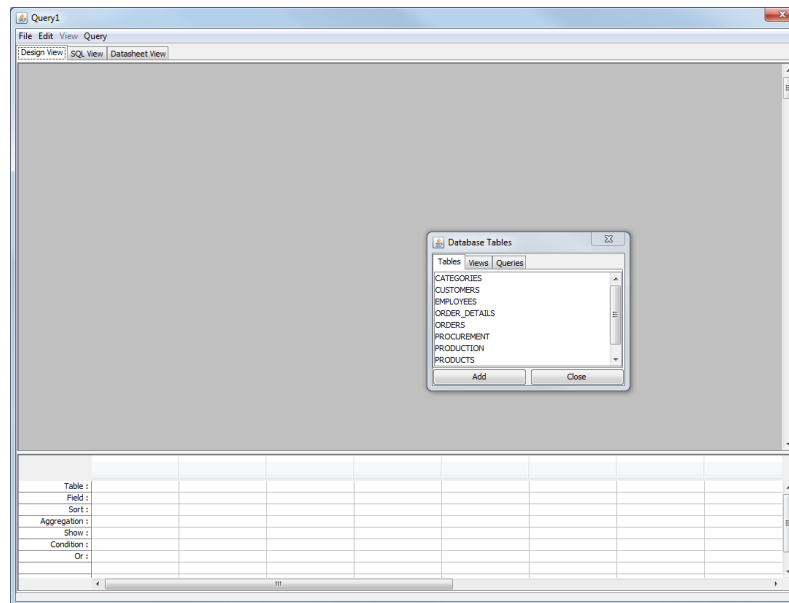
EDAB provides a number of different interfaces to query a database to retrieve the report data. You can type a SQL statement, use the Query Builder, or use data views to create a query interface that insulates the end user from the database structure. In this example, we will use the Query Builder to create a query.

To create a new query, click to expand the *Woodview* node in the left-hand frame of the Data Source Manager. Two sub-nodes will appear, one called *Queries* and one called *Data Views*. Select the *Queries* node and click *Add*. A dialog will appear prompting you to specify a name for the query and to select whether to launch the Query Builder or to enter an SQL statement.



Query Name Dialog

Enter any name you would like, select *Open query builder*, and click on *OK*. The Query Builder will launch. You will see a separate window containing all of the tables for Woodview sitting over top of the main Query Builder window.

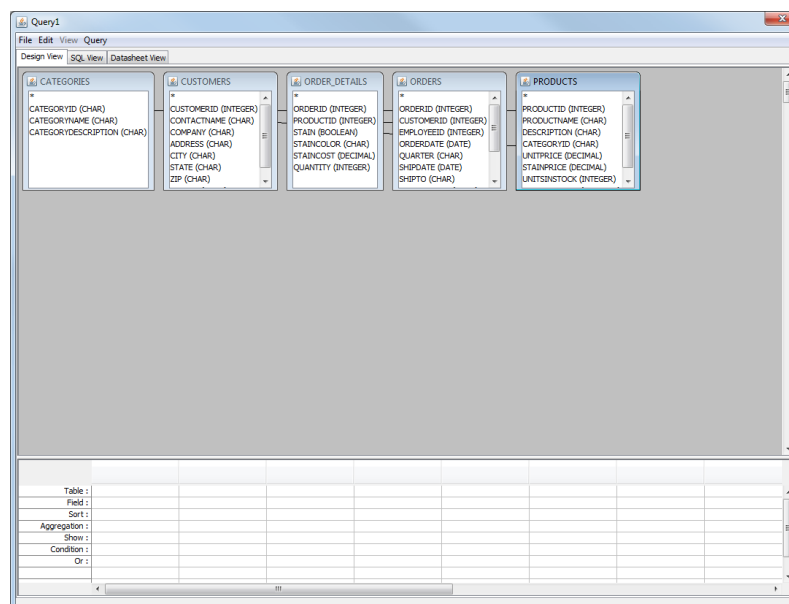


Query Builder Dialog

To add a table to the query, select a table in the Tables window and click the *Add* button. You can also double click on the table name. Using one of the two methods, add the following tables to the query:

CATEGORIES
CUSTOMERS
ORDER_DETAILS
ORDERS
PRODUCTS

The tables will appear in the top half of the Query Builder window. You will see the join lines connecting various fields in the tables.

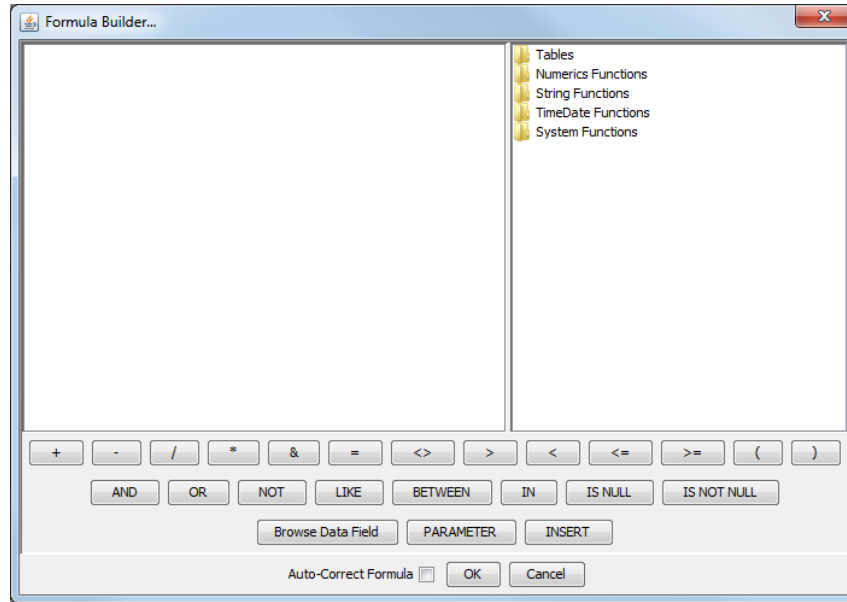


Query Builder with Tables

To add a field to the query, you can either double click on the field in the table window or double click on the *Table* and *Field* fields in the lower (QBE) portion of the Query Builder window and select the table and field from the drop-down menu. Using either method, add the following fields to the query.

ORDERID from ORDERS
COMPANY from CUSTOMERS
REGION from CUSTOMERS
CATEGORYNAME from CATEGORIES
PRODUCTNAME from PRODUCTS
UNITPRICE from PRODUCTS
QUANTITY from ORDER DETAILS

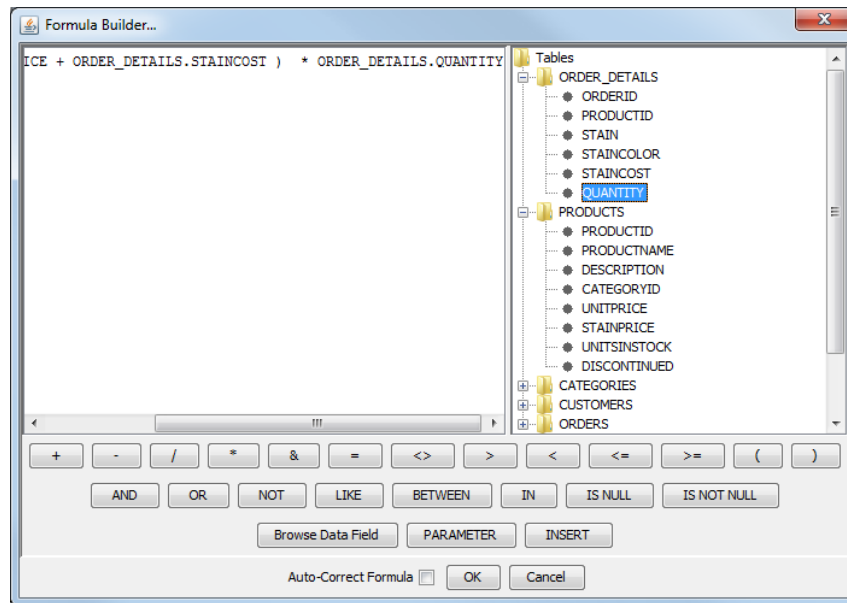
In the eighth column, which should be blank, right click in the *Field* field and select *Build* from the pop-up menu. This will open the Formula Builder interface allowing you to create a computed column.



Formula Builder Window

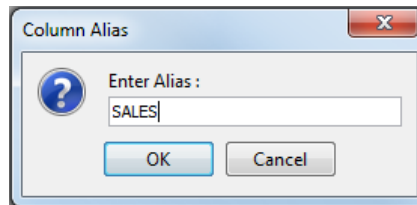
To build a column, first click the *left parenthesis* button. Then double click on the *Tables* folder. It will expand into five nodes, one for each table you selected for the query. Opening a table folder will list all of the column fields in that table. Open up the *PRODUCTS* folder, select *UNITPRICE* and click *Insert*. Then click the *add (+)* button. Next, insert *STAINCOST* from the *ORDER_DETAILS* table. Then click the *right parenthesis* button. Click the *multiply (*)* button and finally insert *QUANTITY* from the *ORDER_DETAILS* column. The finished formula should look like this:

`(PRODUCTS.UNITPRICE + ORDER_DETAILS.STAINCOST) * ORDER_DETAILS.QUANTITY`



Formula Builder Window with Formula

Click the *OK* button and the built column will be added to the query. Next, we will give the column you built an alias. Right click on the column and select *Alias* from the pop-up menu. A window will appear asking you to enter a column alias. Enter "SALES" (without quotation marks) and click the *OK* button.



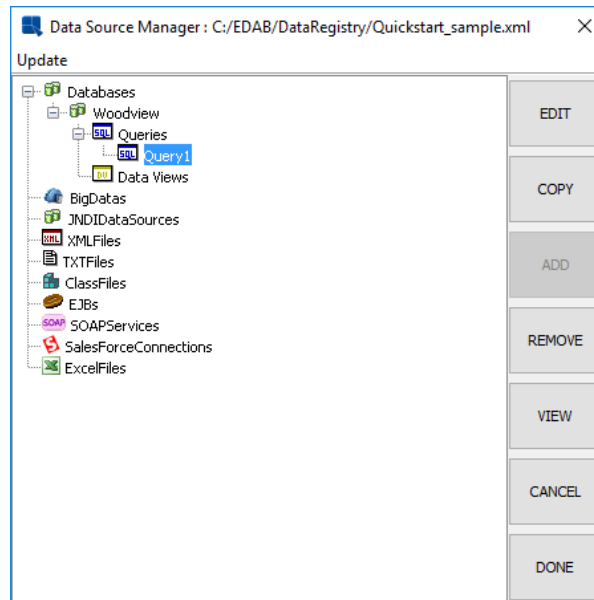
Column Alias Dialog

Click the *OK* button and you will see the column name change in the Query Builder. Now click on the *Datasheet View* tab in the Query Builder. Your query will run and you should see the first thirty records of the query results.

ORDERID (INTEGER)	COMPANY (CHAR)	REGION (CHAR)	CATEGORY (CHAR)	PRODUCTNAME (CHAR)	UNITPRICE (DECIMAL)	QUANTITY (INTEGER)	SALES (NUMERIC)
10001	P & S Furniture East		Side Chairs	Enfil Chair	450.00	12	5724.00
10001	P & S Furniture East		Single Dress...	Ria Dresser	1745.00	12	20940.00
10001	P & S Furniture East		Arm Chairs	Shimaliya Chair	424.00	14	6440.00
10002	P & S Furniture East		Double Dres...	Set Dresser	1645.00	16	26320.00
10002	P & S Furniture East		Arm Chairs	Nisaba Chair	414.00	21	8694.00
10003	Alfano Furni...	East	Round Tables	Arubis Table	1687.00	13	25363.00
10003	Alfano Furni...	East	Side Chairs	Nergal Chair	335.00	4	1340.00
10003	Alfano Furni...	East	Arm Chairs	Shouenume...	1445.00	41	18245.00
10004	P & S Furniture East		Round Tables	Apep Table	1587.00	15	23805.00
10004	P & S Furniture East		Oval Tables	Neith Table	1798.00	17	35870.00
10004	P & S Furniture East		Arm Chairs	Nusku Chair	425.00	22	9350.00
10005	Ebert Furnit...	Midwest	Single Dress...	Salt Dresser	1977.00	15	35655.00
10005	Ebert Furnit...	Midwest	Round Tables	Ningoda Table	1499.00	15	22485.00
10006	Room & Boa...	Midwest	Rectangular...	Bes Table	1141.00	17	23069.00
10006	Room & Boa...	Midwest	Round Tables	Arubis Table	1687.00	22	42922.00
10007	Alfano Furni...	East	Side Chairs	Venhunag C...	369.00	14	5166.00
10007	Alfano Furni...	East	Arm Chairs	Cula Chair	468.00	16	7488.00
10007	Alfano Furni...	East	Oval Tables	Ma'at Table	1875.00	2	4506.00
10008	Design With...	South	Round Tables	Amen Table	1327.00	21	27867.00
10008	Design With...	South	Oval Tables	Isis Table	1789.00	12	21468.00
10009	Lou Rippner...	South	Single Dress...	Ria Dresser	1745.00	18	31410.00
10009	Lou Rippner...	South	Single Dress...	Salt Dresser	1977.00	12	28692.00
10009	Lou Rippner...	South	Arm Chairs	Nusku Chair	425.00	11	4675.00
10009	Lou Rippner...	South	Side Chairs	Zabada Chair	312.00	19	5928.00
10010	Munire Furni...	East	Arm Chairs	Ningrau Chair	478.00	4	2052.00
10010	Munire Furni...	East	Double Dres...	Serket Dresser	1761.00	14	30450.00
10011	Room & Boa...	Midwest	Side Chairs	Tahar Chair	319.00	18	6102.00
10011	Room & Boa...	Midwest	Side Chairs	Enki Chair	425.00	18	7650.00
10011	Room & Boa...	Midwest	Side Chairs	Enfil Chair	450.00	18	8100.00

Query Builder Datasheet View

Now that you have finished designing the query, select *Done* from the File menu to save the changes. This will close the Query Builder window and return you to the Data Registry Manager window. There will now be a node under *Queries* for the query you have just designed.



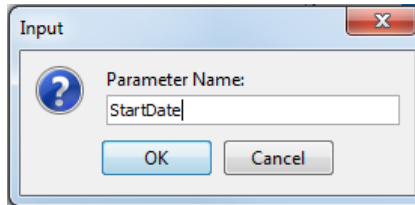
Data Source Manager With Query

Q.3.1.2.1. Add Query Parameters

EDAB allows you to easily parameterize queries, allowing report and chart data to be dynamically filtered at run-time. In this tutorial we will add parameters to the query created in Section Q.3.1.2 - Create a Query.

To open the query that you created, select it, click the *Edit* button in the Data Source Manager, and click the *OK* button. Your query will re-open in the Query Builder. The Tables window will open on top of the Query Builder. Click *Close* to close the Tables window and scroll down in the lower (QBE) portion of the Query Builder window until you see the *Condition* field. Right click in the *Condition* field under the *ORDERID* column and select *Build* from the pop-up menu. This will bring up the Formula Builder allowing you to construct a condition for the query.

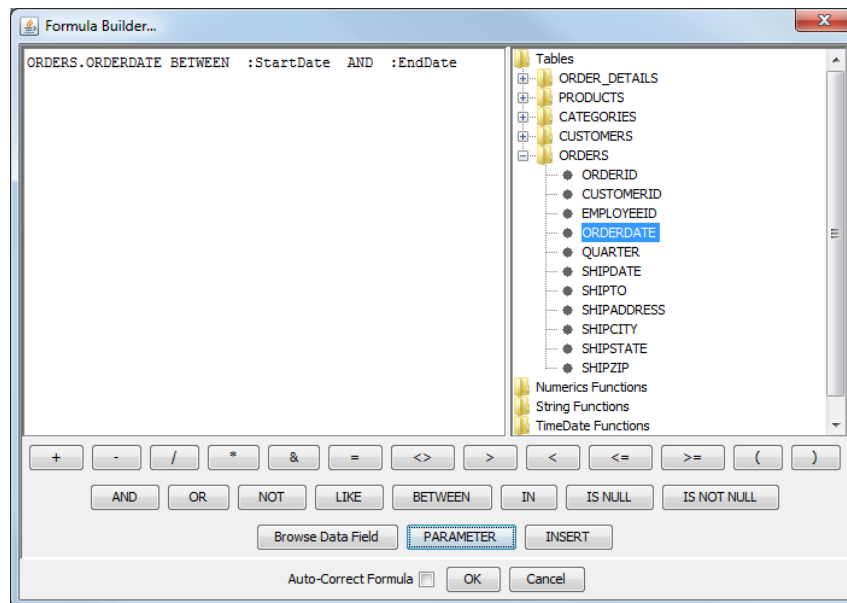
Double click on the *Tables* folder within the Formula Builder to expand it. Then expand the *ORDERS* node and double click on the *ORDERDATE* field. Next click the *Between* button and then click the *Parameter* button. This will bring up a dialog prompting you specify a name for the query parameter.



Parameter Name Dialog

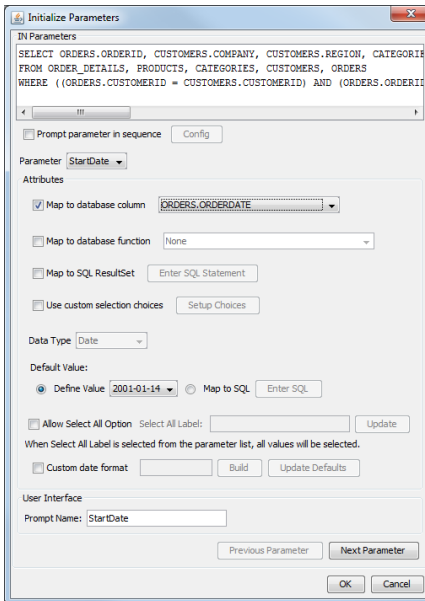
Enter **StartDate** as the parameter name and click the *OK* button. The parameter will be added to the query. Then click the *And* button. Click the *Parameter* button again. Enter **EndDate** as the second parameter name. The finished condition should look like this

`Orders.OrderDate BETWEEN :StartDate AND :EndDate.`



Formula Builder with Conditions

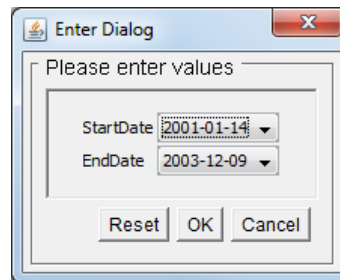
Click *OK* to close the Formula Builder and return to the Query Builder window. Now click on the *Datasheet View* tab. Because you have just added two parameters to the query, an initialization dialog will appear, asking you to specify a few properties for the query parameters.



Parameter Initialization Dialog

From this window, click on *Map to a database column* and select *ORDERS.ORDERDATE* from the drop-down menu. This will automatically fill the *Default Value* and *Data Type* options. Next, enter **Start Date** into the *Prompt Name*, then click the *Next Parameter* button and map the *EndDate* parameter to the same column. Click on the *Define Value* drop-down menu to select an end date. Select a date far enough from the start date that by default you will have more than a couple records to work with (this makes report design easier). Change its *Prompt Name* to **End Date**.

Click *OK* to close the initialization window once you have specified all the options. A new dialog will appear prompting you to select a date range by which to filter the result set.



Parameter Selection Dialog

Select the Start and End date that you would like and click *OK*. You will now see the filtered result in the datasheet window. Now, click *Done* from the File menu to save the changes you have made to the query.

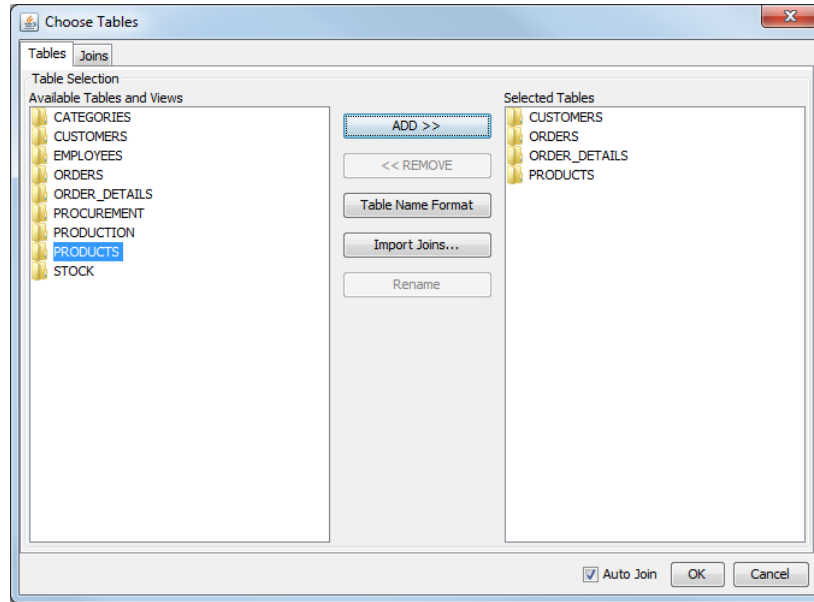
Q.3.1.3. Create a Data View

A unique feature in EDAB is the ability to create data views. Data views are local schemas/views that allow an administrator to pre-configure a group of tables and fields, so that the end users only need to select fields and define simple conditions to create a query. Data views are also used with the ad hoc QuickDesigner. To create a data view, select the *Data Views* node under *Woodview* and click *Add*.

This will open a new dialog asking you to select database tables you would like to use. Select the following tables and add them to the *Selected Tables* panel by clicking on the *ADD >>* button:

CUSTOMERS
ORDERS
ORDER_DETAILS

PRODUCTS

*Data View Tables Dialog*

Next, click on the *Joins* tab. You will see a representation of the tables like in the Query Builder. You can see the auto-join lines between the tables. This window can be used to join the tables or modify the auto-joins if necessary (for more about joins please see Section 3.1.3.2.1.2 - Joins). Click *Ok* to finalize the table selection. The next window allows you to select and group fields for the view. At the top of the window you can specify a name for the view. Name it **Invoicing**.

Next, double click on the *CUSTOMERS* folder to reveal the fields for that table. Add the following fields by double clicking them or selecting them and clicking on the *Add >>* button:

COMPANY
CONTACTNAME
ADDRESS
CITY
STATE
ZIP

Now add fields from the other tables as follows:

ORDERS :

ORDERDATE
SHIPDATE
SHIPTO
SHIPADDRESS
SHIPCITY
SHIPSTATE
SHIPZIP

ORDER_DETAILS :

ORDERID
STAIN
STAINCOLOR
QUANTITY

PRODUCTS :

PRODUCTNAME
UNITPRICE
STAINPRICE

Now click the *Add Heading* button. At the prompt, specify the name **Customer Info**. Add two more headings in the same way, one called **Shipping Info** and one called **Order Info**. Once they are created, select the following fields (Using **CTRL+Click** or **SHIFT+Click** for multiple selection):

COMPANY
CONTACTNAME
ADDRESS
CITY
STATE
ZIP

Once the fields are selected, click the *Group Fields* button and select *Customer Info* from the drop-down list. The fields will be moved under that heading. Next, select the following fields in the same way:

SHIPTO
SHIPADDRESS
SHIPCITY
SHIPSTATE
SHIPZIP
SHIPDATE

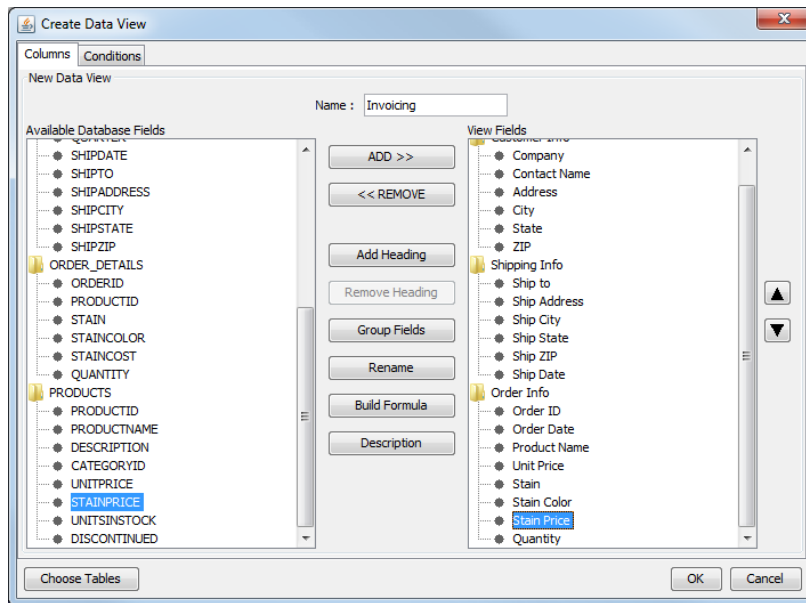
Add these fields to the *Shipping Info* group the same way as before. Next, select the following fields:

ORDERDATE
ORDERID
STAIN
STAINCOLOR
QUANTITY
PRODUCTNAME
UNITPRICE
STAINPRICE

Add these fields to the *Order Info* group. Next, select the *CONTACTNAME* field on the right side and click the *Rename* button. In the dialog, specify the name **Contact Name**. Repeat this for every field in order to give it proper names.

Next, select the *Order ID* field on the right side and click on the up arrow button to move the field to the top of the *Order Info* heading. Use the arrows to arrange the items in the *Order Info* heading in the following order:

Order ID
Order Date
Product Name
Unit Price
Stain
Stain Color
Stain Price
Quantity



Data View Fields Window

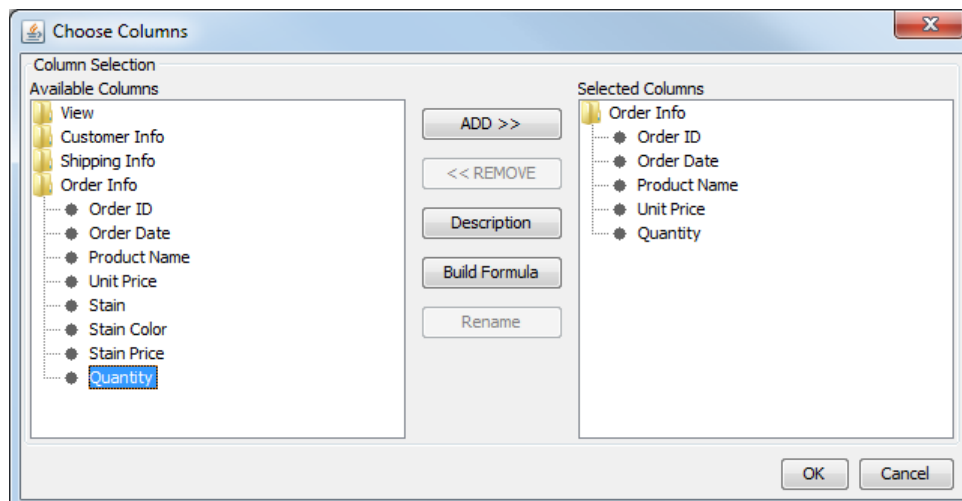
Now that the data view has been created, click the *OK* button in the fields window to save the view. It will be saved as a new node under *Data Views* in the Data Source Manager.

Q.3.1.3.1. Query a Data View

Now that the data view has been created, you can write queries against the view. This allows users to develop queries without knowing the underlying structure of the database. It also allows administrators to limit which database elements the user has access to. In this tutorial we will create a query for the data view you created in. Section Q.3.1.3 - Create a Data View.

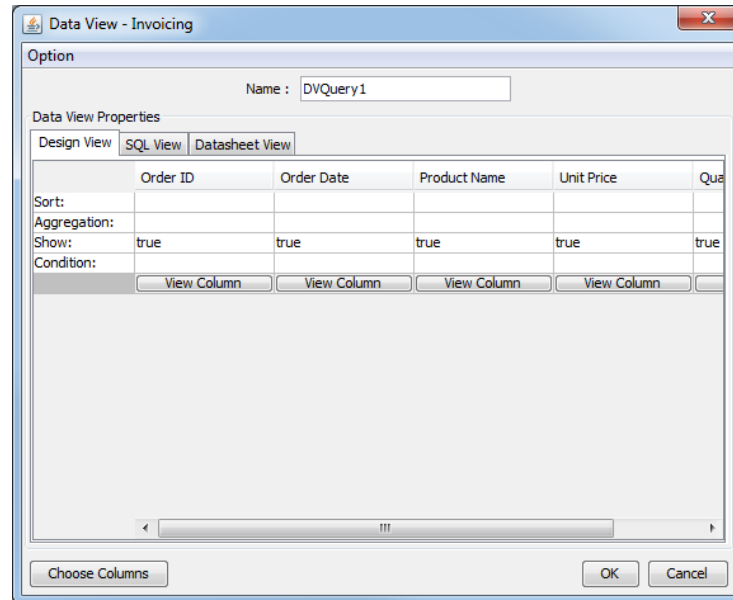
In the Data Source Manager, select the *Invoicing* data view. Then click the *View* button. This will open a dialog, prompting you to select fields from the view. To select fields, first double click on a heading to expand it. Add the following fields to the query by double clicking or selecting them and clicking on the *ADD* button:

Order ID
Order Date
Product Name
Unit Price
Quantity



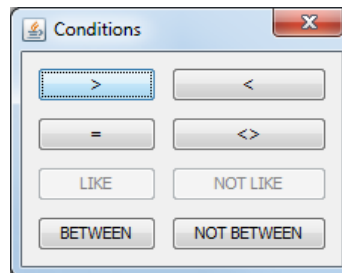
Data View Query Field Selection Dialog

Once you have finished adding the fields, click *OK*. This will bring up a new window allowing you to set conditions, grouping, and ordering for the query. Like the Query Builder this window also allows you to preview the query result with the *Datasheet View* tab.



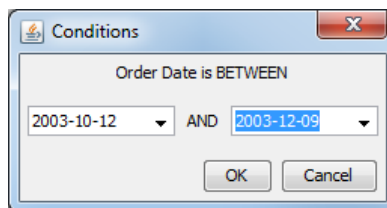
Data View Conditions Dialog

First specify a name for the query in the space provided at the top. Then double click on the *Condition* field for the *Order Date* column. This will bring up a dialog allowing you to specify a condition for the field.



Specify Condition Dialog

Click the *Between* button. A new dialog will appear prompting you to specify a start and end date with which to filter the results. Select **2003-10-12** as the first date and **2003-12-09** as the second.



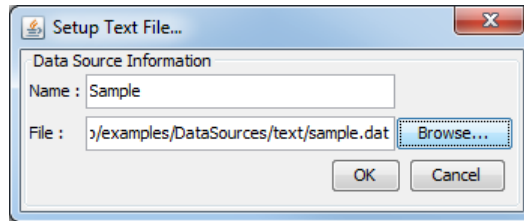
Specify Condition Fields Dialog

Click *OK* to close the dialog and add the condition. You will be taken back to the conditions window. Now you can click on the *Datasheet View* tab to preview the query. Click *OK* in the main window and the query will be saved using the name you provided. You will then be taken to a screen showing the result of the data view query. Options in this screen allow you to use the results to create a report or chart. Click *Cancel* in this dialog to close the interface and return to the Data Source Manager. There will now be a new node for your query under the *Invoicing* data view.

Q.3.1.4. Setup a Text Data Source

In addition to database data, EDAB can also draw data from flat files(XML and text). In this tutorial we will setup a text file data source in the registry. To add a new text file, select the node labeled *TXTFiles* and click the *ADD* button. This will bring up a dialog allowing you to specify a display name and the location for the text data source.

Enter any display name you would like (e.g. *Sample*). Then click the *Browse* button and browse to the *help/examples/DataSources/text* directory. Select the *sample.dat* file.



Setup Text File Dialog

After you have finished entering the information, click *OK*. You will see a new node in the Data Source Manager for the text file.

Q.4. Chart Designer


After setting up data sources and creating queries, the next step to creating charts is to take the results of the data source and map them to the chart. This section only covers basic charting from mapping the data to modifying the look and feel. For more information, please refer to Section 4.1 - Chart Designer.

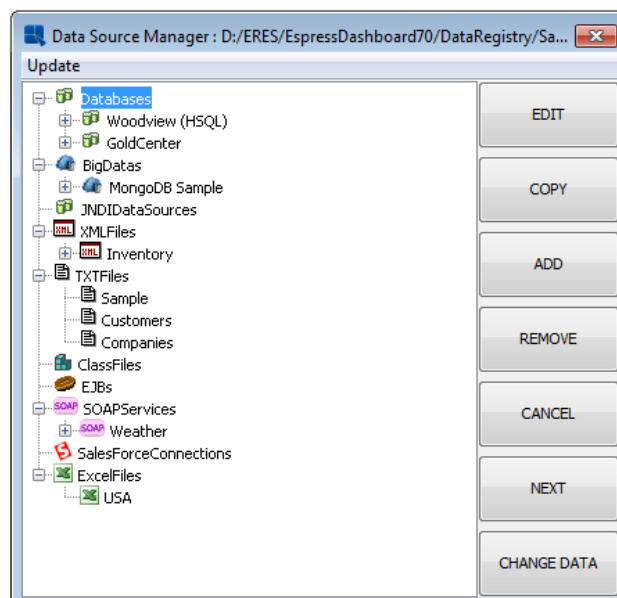
Q.4.1. Chart Mapping

This section will look at several ways that data can be mapped to charts. This section will use the text file data source that was setup in Section Q.3.1.4 - Setup a Text Data Source.

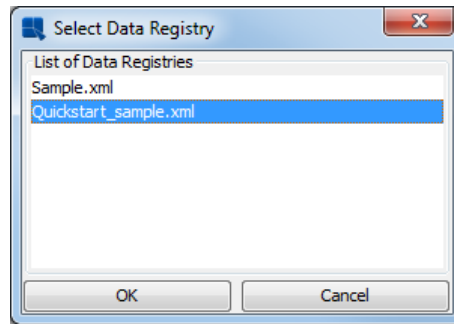
Q.4.1.1. Column Chart

Column charts are a good starting point as the mapping for column charts is very similar to that of bar, area, and line

charts. To begin creating a chart, click the *Chart Designer* button in the Organizer toolbar . This will launch the Chart Designer interface and open the *Data Source Manager*. Click *CHANGE DATA* option in the lower right corner, select *Quickstart_sample.xml* data registry and click *OK*.



Change Data Registry



Select Data Registry

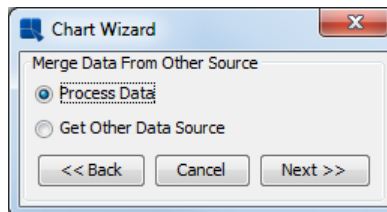
Now, select the *Sample* text data source and click the *NEXT* button. A dialog will open a table with the contents of the text file (first 20 records).

 A dialog box titled "Chart Wizard" with a close button (X) in the top right corner. It displays a "Query Result" table with 8 columns: INDEX, Day, Drink, timing, quantity, volume, high, and low. The table contains 20 rows of data. Below the table is a scroll bar and a "Show All Records" checkbox. At the bottom are "<< Back", "Cancel", and "Next >>" buttons. A status bar at the very bottom says "Showing the first 20 of 168 records".

INDEX	Day	Drink	timing	quantity	volume	high	low
TYPE	Varchar	Varchar	Varchar	Integer	Double	Double	Double
1	Mon	Coffee	day	1	32.2	30.0	21.1
2	Mon	Coffee	night	10	128.11	34.0	30.0
3	Mon	Soft Drinks	day	8	3.2	33.0	22.0
4	Mon	Soft Drinks	night	8	10.2	34.0	20.0
5	Mon	Fruit Juice	day	3	9.12	34.0	30.0
6	Mon	Fruit Juice	night	1	13.1	34.0	30.0
7	Mon	Water	day	7	23.1	40.0	32.0
8	Mon	Water	night	6	40.1	41.0	5.0
9	Tue	Coffee	day	20	23.6	12.0	8.0
10	Tue	Coffee	night	4	23.0	10.0	6.0
11	Tue	Soft Drinks	day	9	3.1	24.0	22.0
12	Tue	Soft Drinks	night	9	13.3	44.0	44.0
13	Tue	Fruit Juice	day	9	23.8	35.0	30.0
14	Tue	Fruit Juice	night	7	7.1	30.6	22.0
15	Tue	Water	day	15	23.1	34.0	30.0
16	Tue	Water	night	2	13.1	34.0	30.0
17	Wed	Coffee	day	2	18.0	34.1	22.4
18	Wed	Coffee	night	8	13.1	9.0	2.4
19	Wed	Soft Drinks	day	10	23.0	34.0	30.0
20	Wed	Soft Drinks	night	5	40.0	24.0	20.0

Contents of Text File

At the bottom of the dialog, click the *Next* button. A dialog will open asking you if you would like to select an additional data source for the chart.



Add Data Source Dialog

Select the *Process Data* option and click *Next*. You will then be taken to a dialog prompting you to select which type of chart you would like to create.

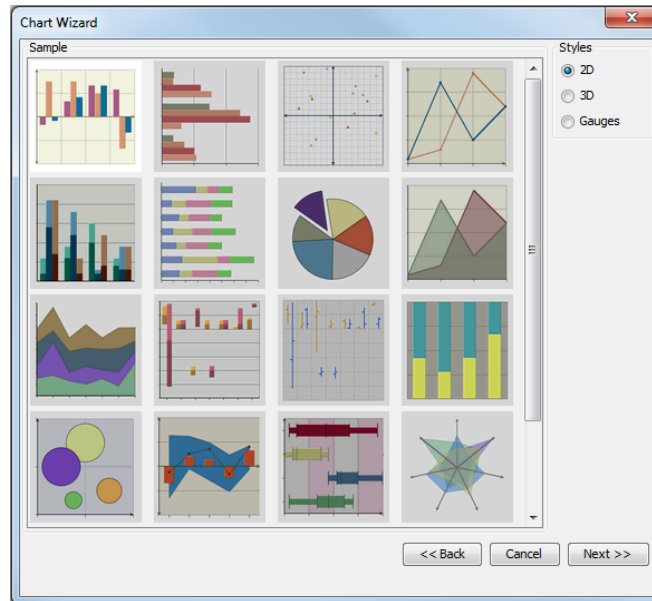


Chart Types Dialog

In this dialog, leave selected a *Column Chart* (the first image) and a *2D* option (two-dimensional chart) as the data type and click *Next*. You will then be taken to the data mapping dialog which allows you to map columns from the data source to chart elements.

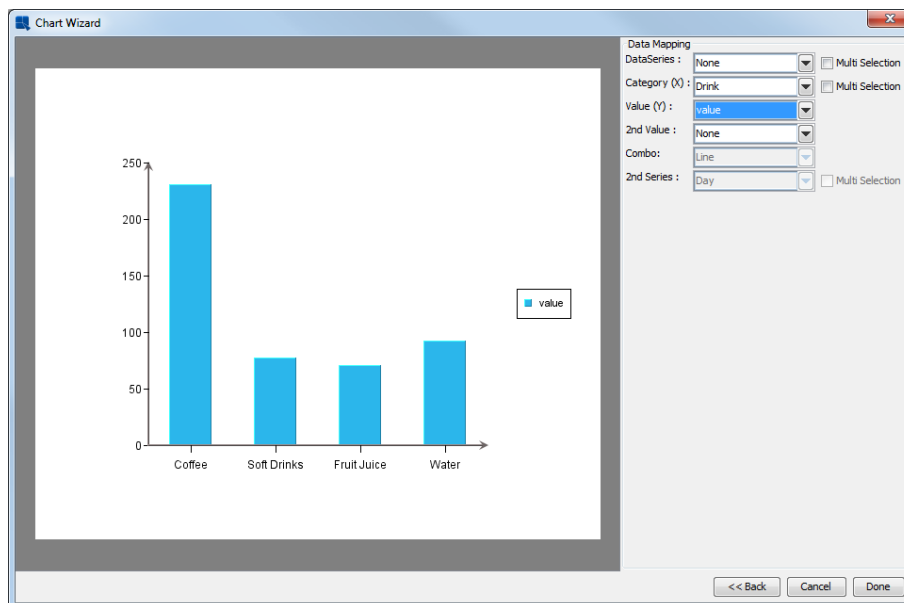


Chart Data Mapping Dialog

Set the *Data Series* to **None**, the *Category* to **Drink** and the *Value* to **value**. Then click *Done* to finish the Wizard and go to the Chart Designer interface.

If the generated chart does not fit in the viewport of the Chart Designer, select *Canvas* from the *Format* menu. This will bring up a dialog allowing you to set the size of the chart.

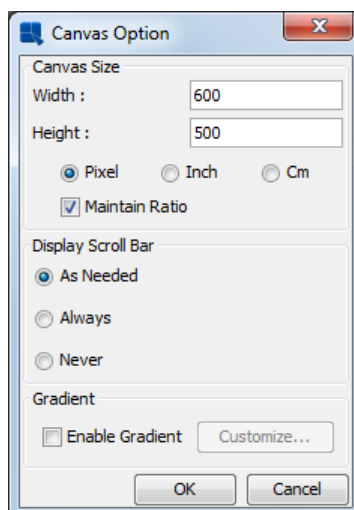


Chart Canvas Dialog

Enter a smaller size for the chart canvas and click *OK* to close the dialog. You should now see the whole chart in the viewport.

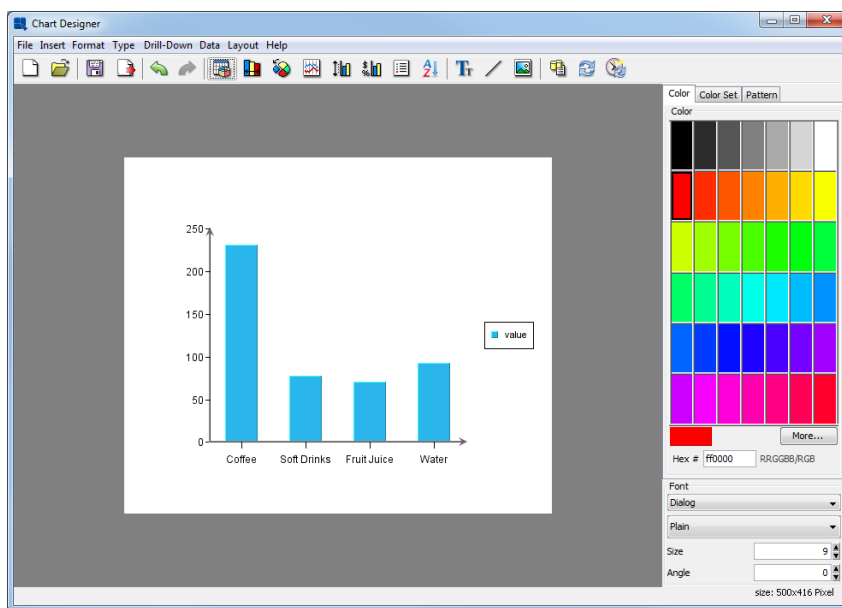



Chart Designer Showing Column Chart

Notice that the column chart contains a column for each distinct value in the Drink column and shows the corresponding value.

Q.4.1.1.1. Add a Data Series

At this point, the column chart only contains data points for the categories Drink column. However, EDAB supports adding another dimension to this data by way of a series. To add a series to the column chart, click the *Data*

Mapping button in the toolbar . This will return you to the data mapping window.

In the mapping window, change the *Data Series* option from **None** to **Day** and click *Done* to return to the Chart Designer.

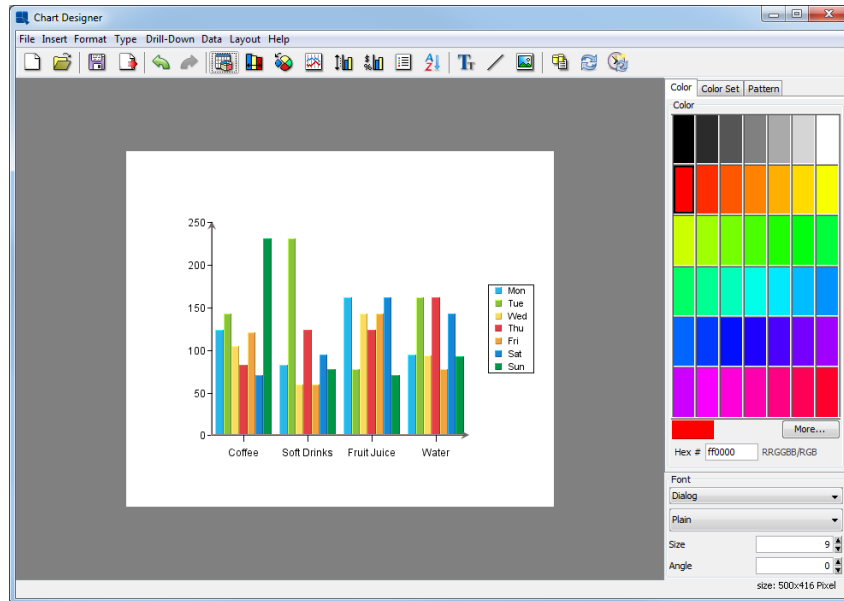


Chart Designer Showing Column Chart with Series

Notice now that instead of a single data point for each drink, the column for each drink is now comprised of seven small columns, one showing the data point for each day.

Q.4.1.1.2. 3D Column Chart

In a two-dimensional chart, the data series is represented in-line along the X axis. However, a three-dimensional representation provides another axis to work with.

In the Chart Designer select *3D Chart* from the *Type* menu to convert the column chart to a 3D representation.

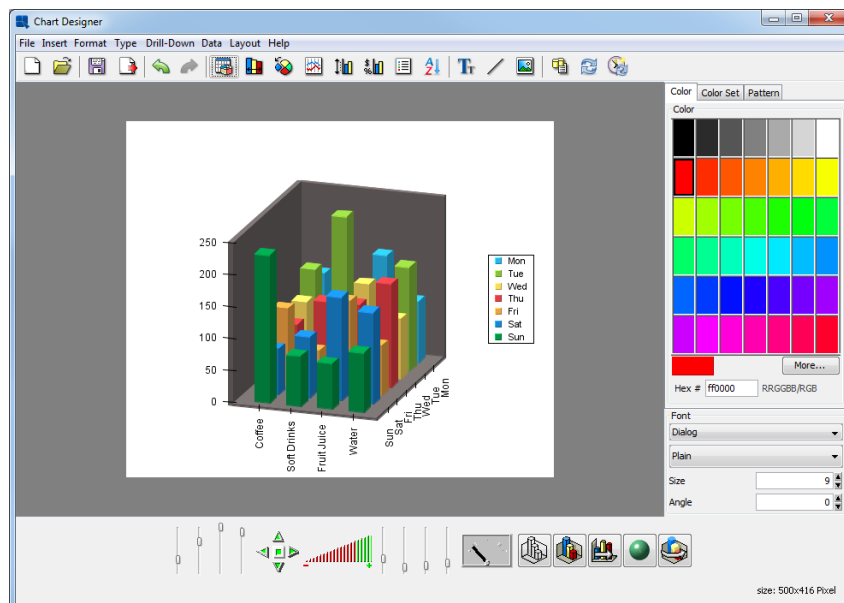


Chart Designer Showing 3D Column Chart

The 3D toolbar will automatically appear at the bottom of the Chart Designer when the chart is converted to 3D. The chart may appear squished when you convert it. You can use the sliders next to the zoom function to change the X, Y, and Z scaling for the 3D chart. You can also use the navigation buttons to position the chart in space. For more information about 3D features, see Section 4.1.4.4 - The Navigation Panel.

Notice that with the chart in 3D, the data series is now moved to the Z axis of the chart by default.

Q.4.1.2. Pie Chart

Pie charts are another commonly used chart type that shows values as a percentage of a whole. To convert your chart to a pie chart, first go to the *Type* menu and select *2D Chart* to convert your column chart back to two-dimensional form. Then select *Pie* from the *Type* menu. The chart will then be converted to a pie representation.

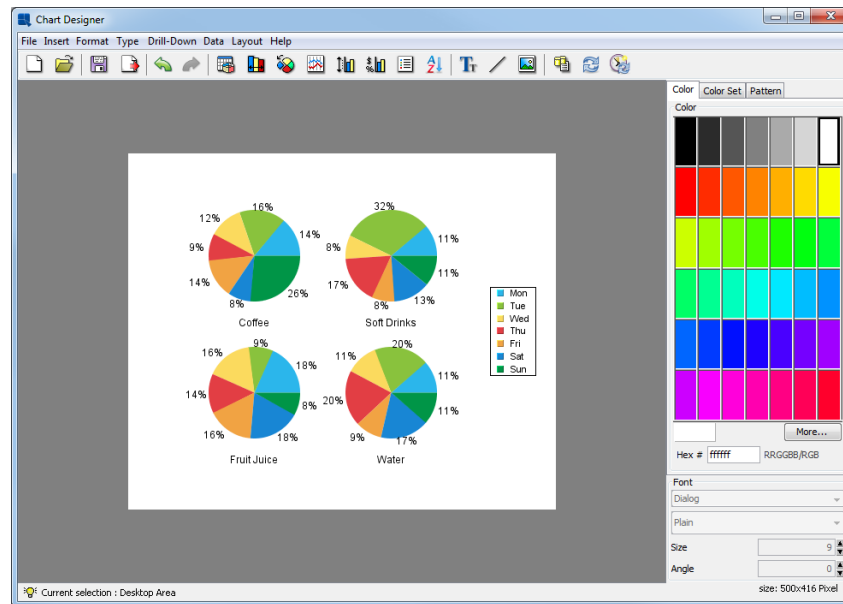

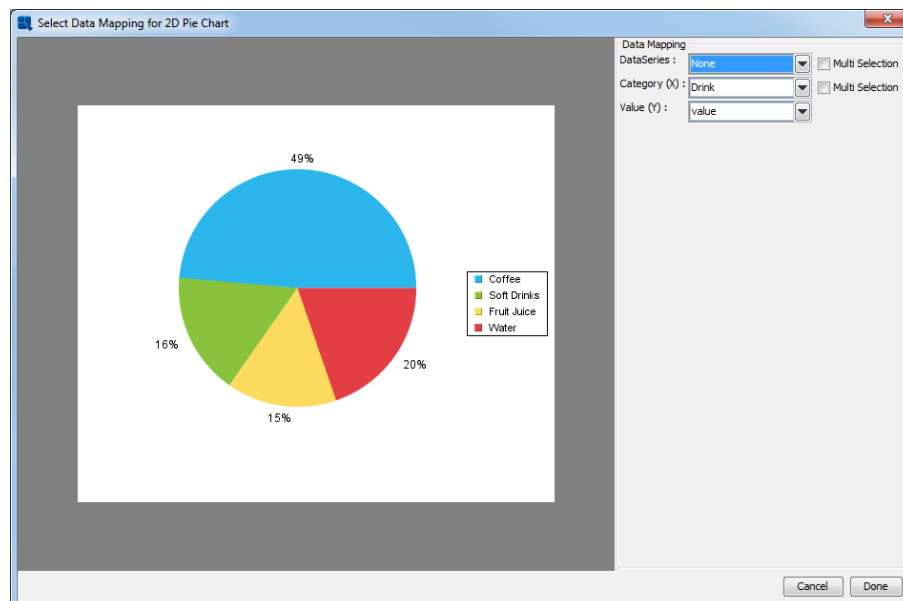


Chart Designer Showing Pie Chart with Series

Multiple pies are drawn when you have a data series; one for each category. Each category is broken down showing percentage contribution for the series elements.

In order to turn the chart into a single pie, we will remove the series. Click the *Change Data Mapping* button  on the toolbar to bring up the mapping options.



Data Mapping Options for Pie Charts

Change the *Data Series* option to **None** and click *Done*. Now in the Chart Designer you will see a single pie made up of your drink categories.

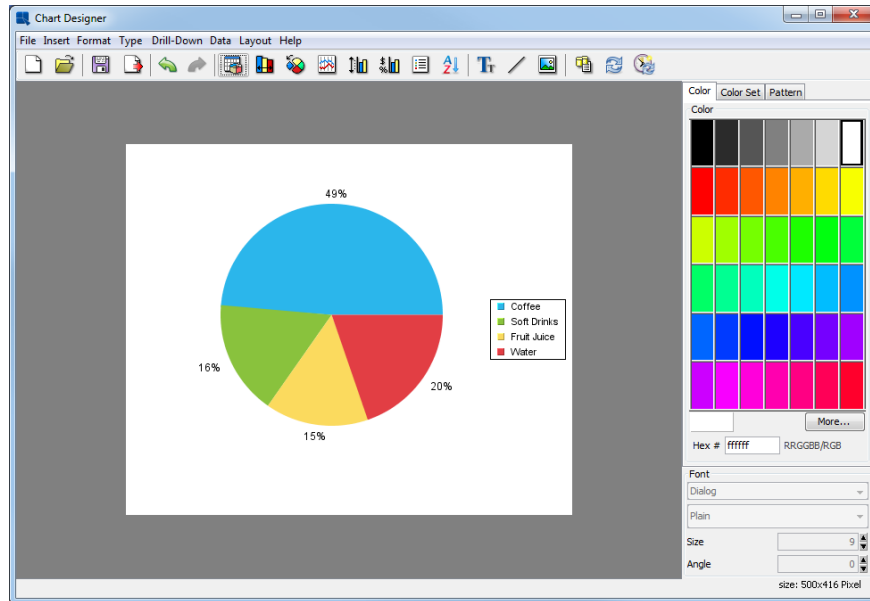


Chart Designer Showing Pie Chart

Q.4.1.3. Stack Column Chart

Another way to display multi-dimensional data is to use a stack type representation to show contributions to a total. To convert your pie chart into a stack column chart, select *Stack Column* from the *Type* menu. Chart Designer will ask whether you want to redo data mapping for the new chart type. Click *Yes* and confirm the change by clicking *Done* in the “Select Data Mapping for 2D Stack Column Chart” dialog. The chart will then be converted to a stack column layout.

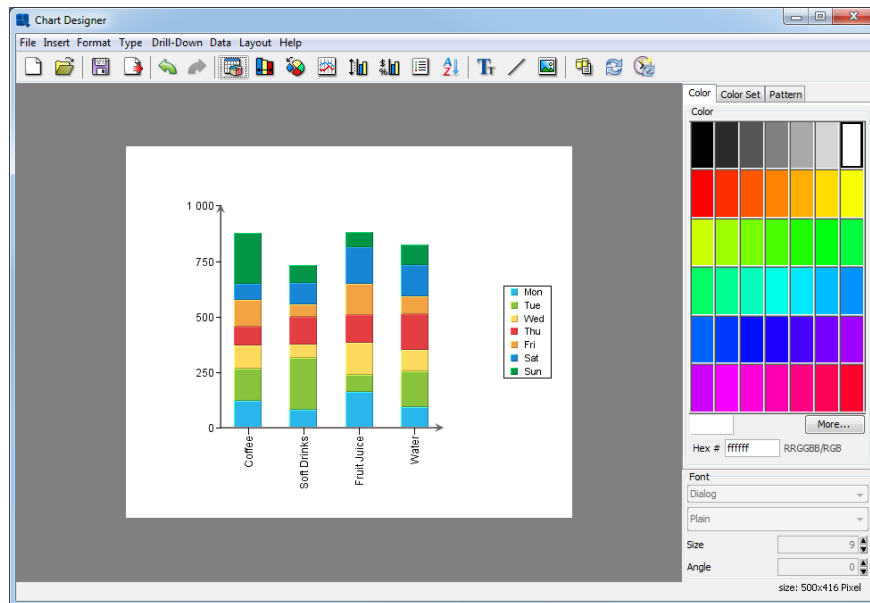

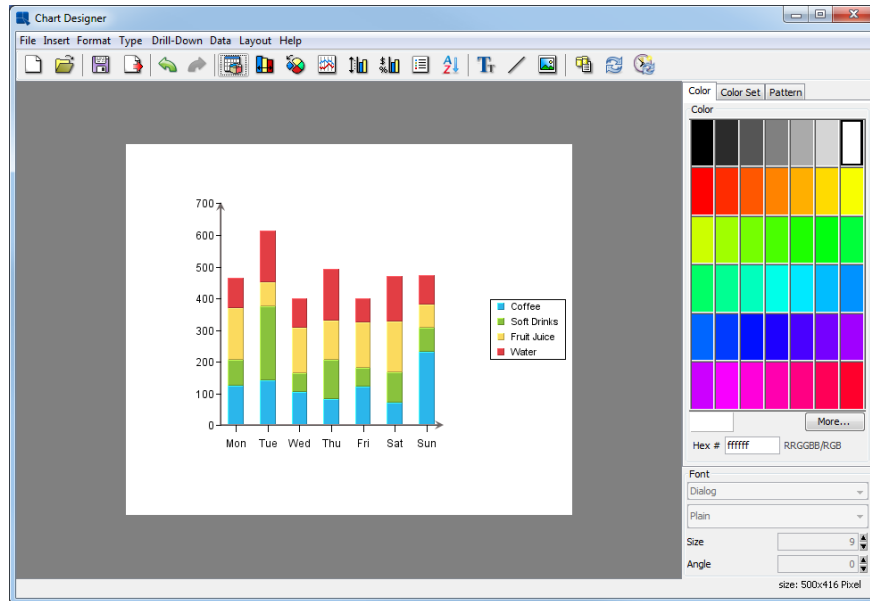



Chart Designer Showing Stack Column Chart

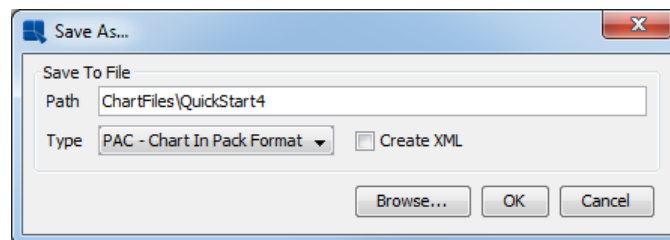
Notice that the chart now shows the columns as stacks made up of the values for each drink. Now click the *Data*

Mapping button  on the toolbar. In the data mapping window there is a new option called *Sum by*. This is set to the *Day* column. The *Sum by* column provides the individual stacks in these types of charts. Set *Sum by* to *Drink* and *Category* to *Day*. Click *Done* to return to the Chart Designer. You should see the following chart.



Stack Column Chart

Now that you have finished designing a chart, click the *Save* button on the Toolbar . A dialog will open prompting you to specify a location and file name for the chart.



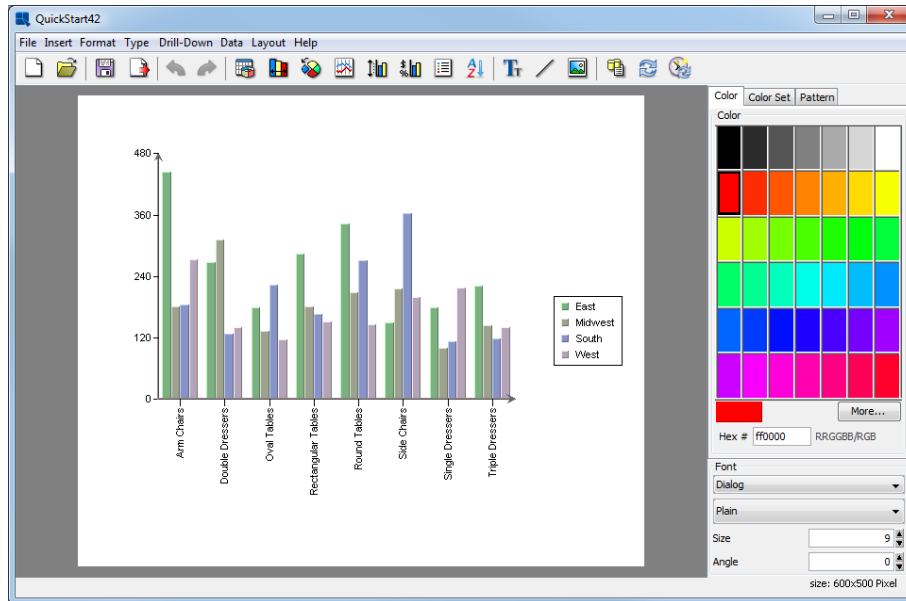
Save As Dialog

Enter a file name, and select to use either a PAC, CHT, or TPL format. For more about chart formats, see Section 4.1.6.1 - Saving Charts. The chart will be saved in the /ChartFiles/ directory under the EDAB installation by default. You will then be prompted to add the file to the Organizer. Select the *QuickStart Examples* Organizer folder and click *OK*. Close the Chart Designer.

Q.4.2. Basic Chart Formatting


In this section, we will use some of the basic formatting features in the Chart Designer to change the look and feel of an unformatted chart.

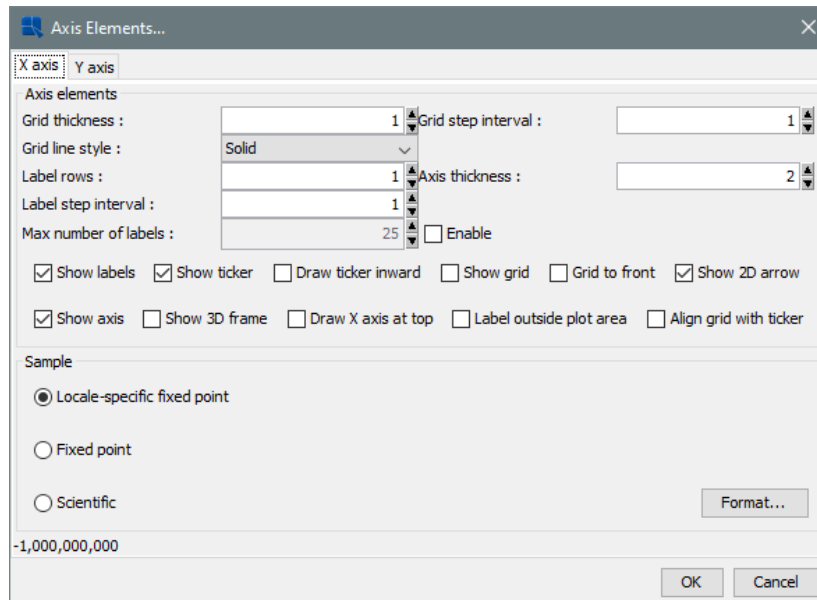
Right click on the QuickStart42.tpl file (which you added earlier in the Section Q.2.2.3 - Inserting a File chapter) and select Open File from the pop-up menu. This will open the chart in the Chart Designer. As you can see, it is relatively un-formatted.



Unformatted Chart in Designer

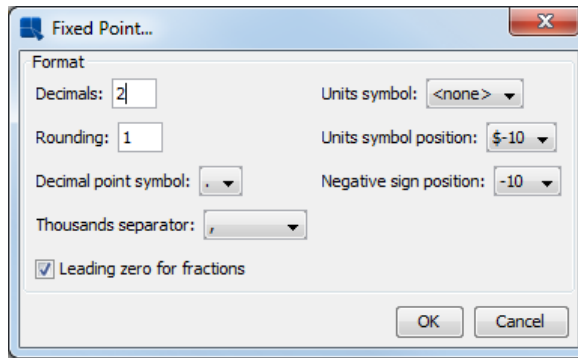
Q.4.2.1. Axis Options

Many chart appearance options are controlled using the Axis Elements dialog. To bring up this dialog, click the *Format Value Elements* button on the toolbar . This will bring up a tabbed dialog allowing you to set different options for each chart axis. Click on the *Y Axis* tab to bring up options for the value axis.



Axis Elements Dialog

Check the box marked *Show grid* to add grid lines to the Y Axis. Select *Fixed point* for the data format and click the *Format* button. This will bring up an additional dialog allowing you to set format options for the numeric data. Select the number of decimals as **2** and click *OK*.



Numeric Format Dialog

Click *OK* again to dismiss the axis elements dialog and you will see the specified changes reflected in the chart.

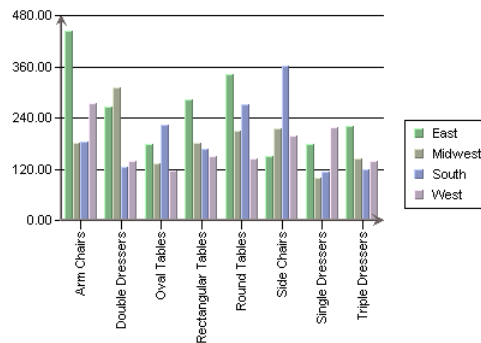
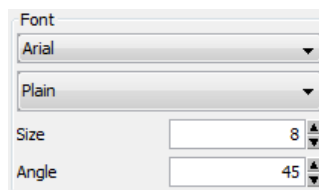


Chart with new Axis Formatting

Q.4.2.2. Modify Color and Font

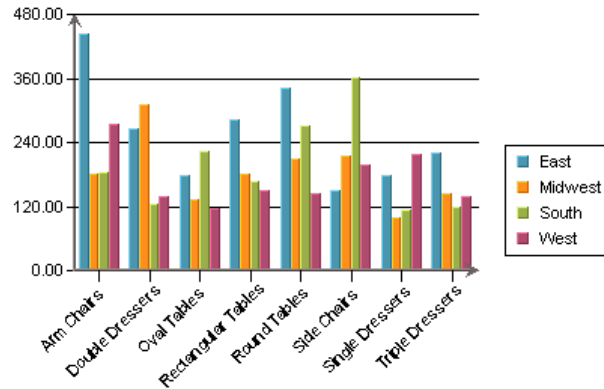
Chart colors can be selected individually or you can select predefined color set. In Chart Designer, click on the *Color set* tab on the right and select a color combination you like.

Now click on one of the X axis (category) labels. The lower left corner of the design window will reflect your selection. In the font dialog in the lower right corner, change the angle of the labels from 90 to 45 degrees and press Enter.



Font Panel

The chart labels will then rotate. You may need to adjust the position of the labels slightly. To do this, click and drag on a label. All the X axis labels will follow your cursor around the chart canvas.



Q.4.2.3. Add Titles

Next, we will add titles to the chart. To do this, select *Titles* from the *Insert* menu. This will bring up a dialog prompting you to enter a main title for the chart as well as titles for each of the axes.

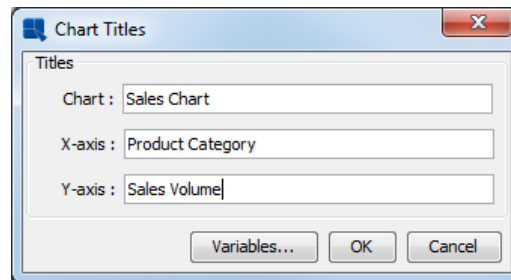


Chart Titles Dialog

Enter any titles that you would like for the various elements and click *OK*. The titles will then be added to the chart. Titles are placed automatically but you can manually adjust their positions by clicking and dragging the text on the chart canvas.

If you do not see chart titles, click and drag on the chart plot to move the chart. You should then be able to see the titles. You can adjust the font and font size of the chart titles by clicking them and changing the settings in the font panel.

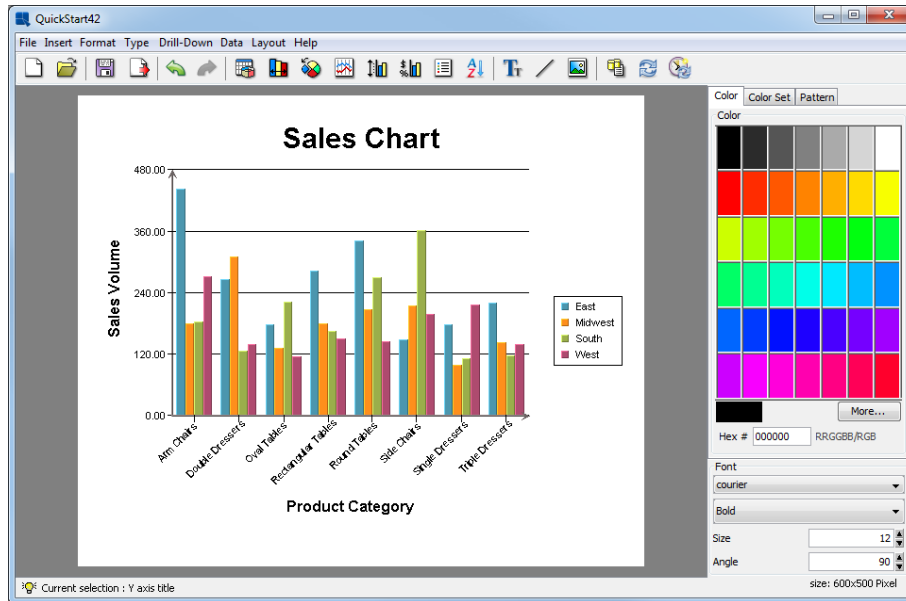
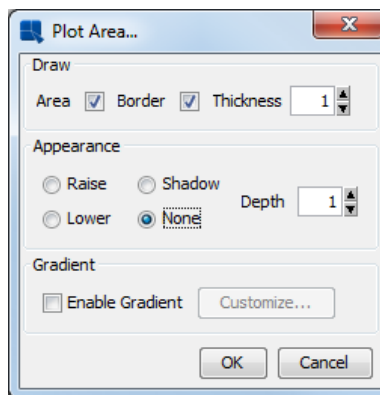


Chart With Titles

Q.4.2.4. Customize Plot Area

You can also customize the plot of two-dimensional charts, creating a separate background for the data points. To do this, select *Plot Area* from the *Format* menu. This will bring up a dialog allowing you to set display options for the chart plot.



Plot Area Dialog

Select to draw both the *Area* and the *Border* with a *Thickness* of **1**. Specify *None* for the *Appearance*. Once you have specified the options, click *OK* and the plot area for the chart will be modified. You can change the background color of the plot area by clicking to select it and then modifying the color in the color panel.

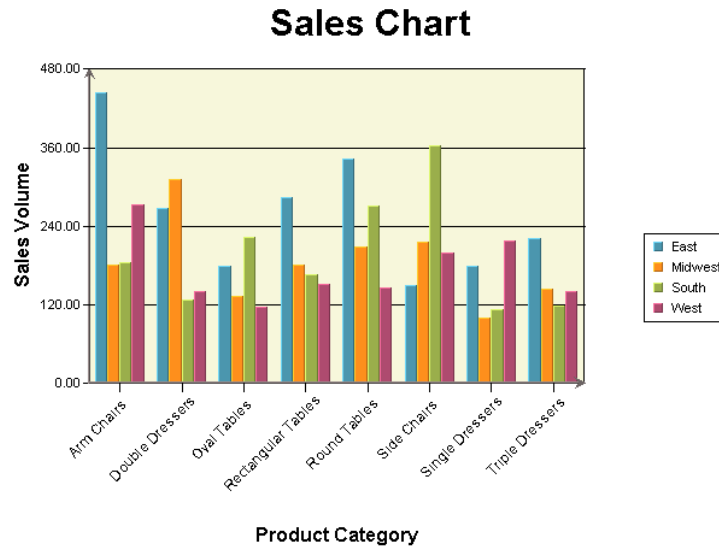


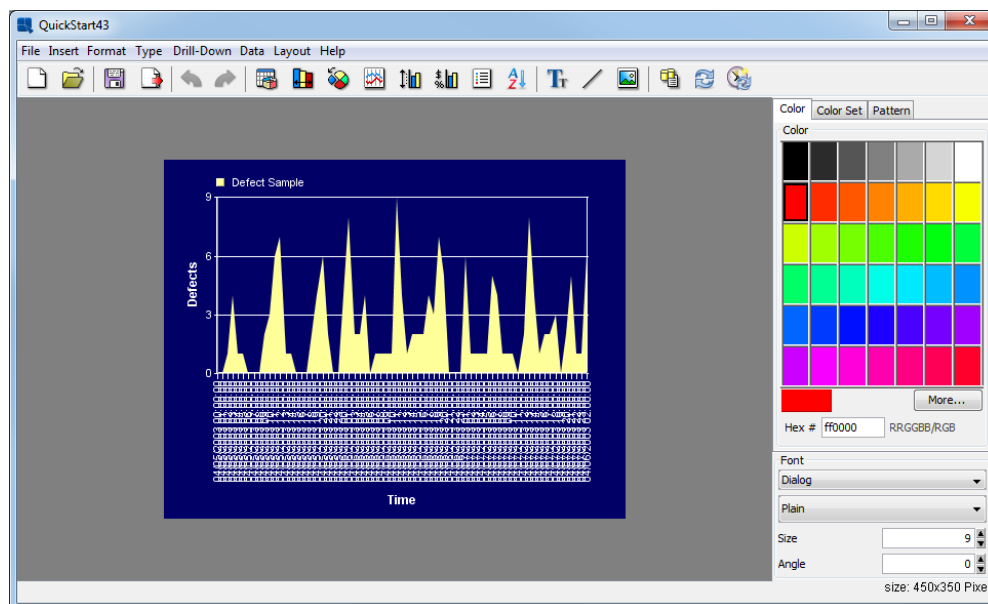
Chart With Colored Plot Area

Now save the changes you have made to the chart and exit the Chart Designer.

Q.4.3. Advanced Charting Features

In this section we will use some of the charting features in EDAB to provide a chart that quickly conveys salient information to the viewer. For detailed information about the features discussed in this section and other charting features, please see Section 4.1.4 - The Chart Designer Interface.

Following the same procedure shown earlier, add the QuickStart43.tpl file under help/quick-start/templates into your project in the Organizer. Then right-click on the entry for this file in Organizer and select Open File from the pop-up menu. This will open the chart in Chart Designer.



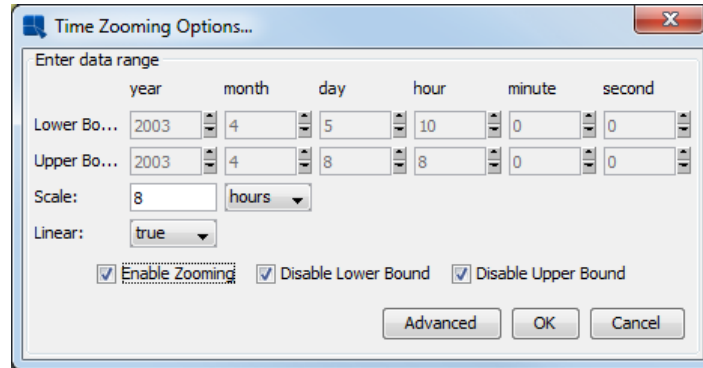
QuickStart43.tpl in Chart Designer

The chart shows hourly defect counts for a manufacturing process. Since the chart shows three days worth of data it is difficult to make out the individual points and the X axis labels are not legible because they are drawn on top of each other.

Q.4.3.1. Time-Series Zooming

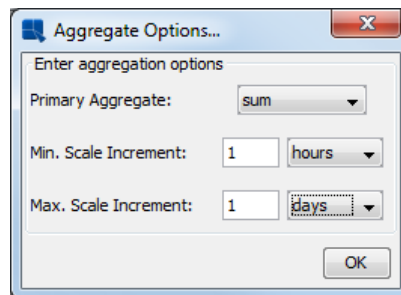
One way to improve the chart, is to use the time series zooming feature to aggregate the data into fewer data points. In this instance, let's look at the total number of defects for each eight hour shift, instead of plotting the data for each hour.

To turn on zooming, select *Time Zooming Options* from the *Format* menu. This will bring up the following dialog.



Zoom Options Dialog

In this dialog, set the *Scale* to **8 hours** and *Linear* option to **true**. Leave the *Disable Upper Bound* and *Disable Lower Bound* options checked. Check the *Enable Zooming* option. This will bring up a new dialog prompting you to select the aggregation for the zooming.



Aggregation Options Dialog

In this dialog, set the *Primary Aggregate* to be **sum**, the *Minimum Scale* to be **1 hour**, and the *Maximus Scale* to be **1 day**. Then click *OK* to return to the zoom options dialog, and *OK* again to return to the zoom setting. The chart now shows the total number of defects for each eight hour period.

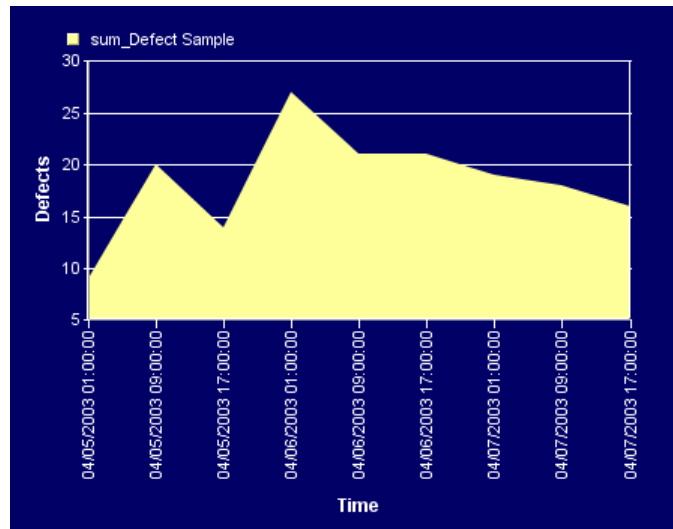
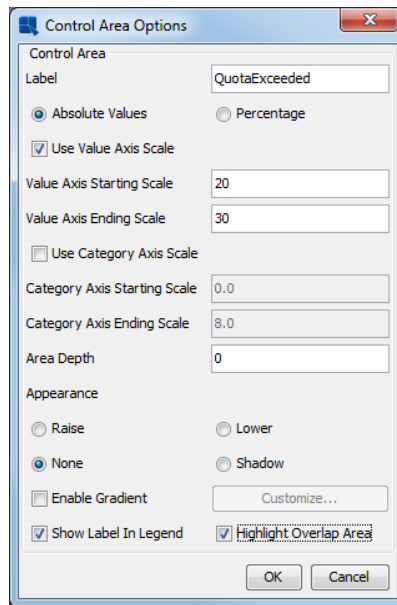


Chart With Zooming Applied

Q.4.3.2. Control Areas

Next, we will use the control area feature to highlight any shift where there were more than 20 defects.

To add a control area, select *Control Area* from the *Insert* menu and then click *Insert* in the control range list. This will bring up a dialog allowing you to select options for the new control area.



Control Area Dialog

In this dialog, enter a name for your control area. Then check the option marked *Use Value Axis Scale*, and enter **20** for the *Starting Scale*, and **30** for the *Ending Scale*. Then at the bottom of the dialog check the options marked *Show Label In Legend* and *Highlight Overlap Area*. Then click *OK* to apply the control area. Notice how only the areas where the number of defects is colored. You can set the color, by selecting the overlapped area, and selecting a new color from the palette on the right-hand side.

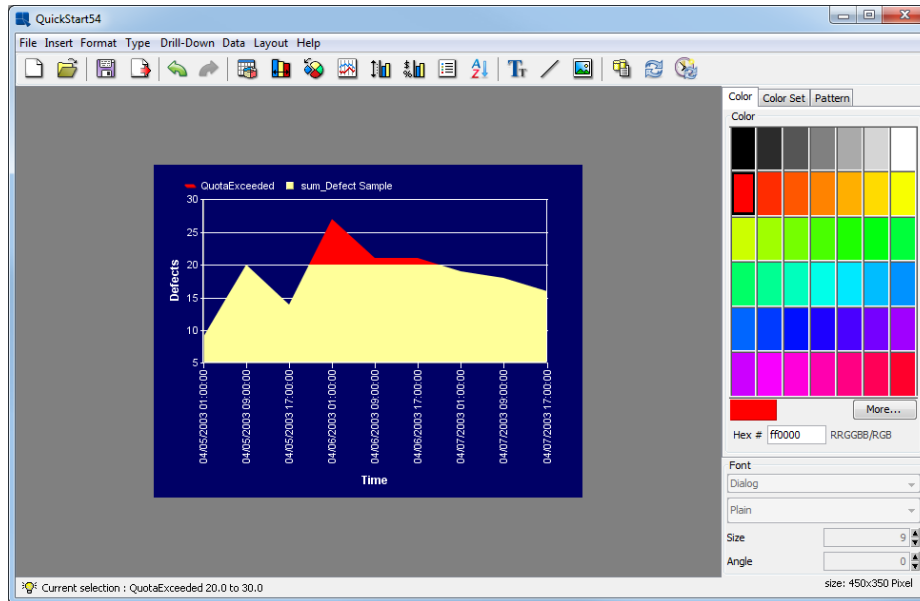


Chart With Control Areas Applied

Once you have finished modifying the chart, click the *Save* button on the toolbar to save the changes you have made and close the Chart Designer.

Q.5. QuickDesigner Reports

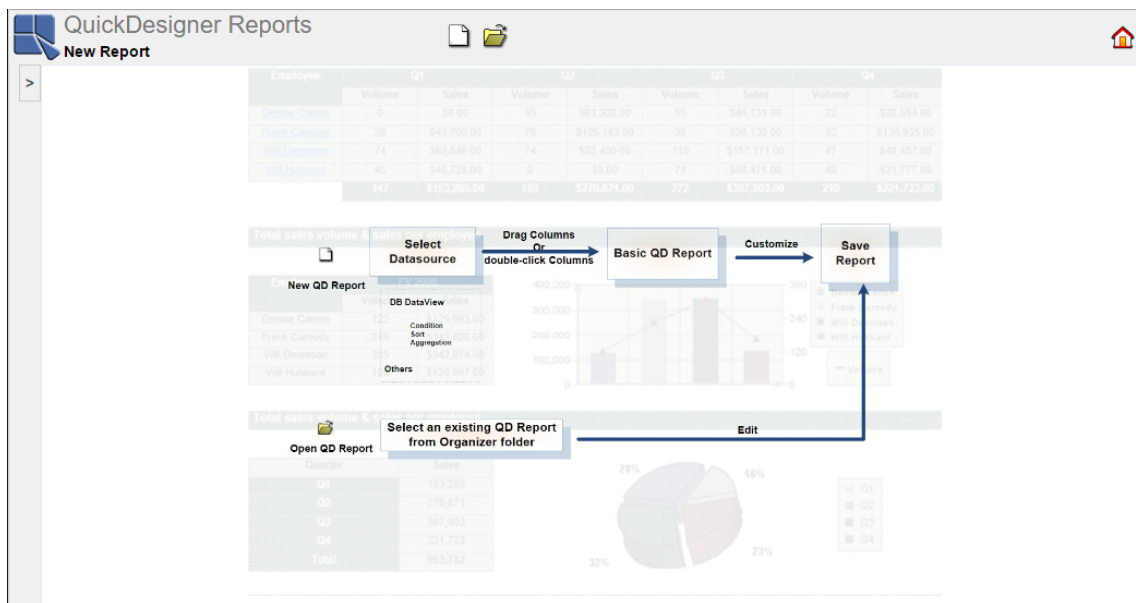
QuickDesigner Reports interface provides a thin-client interface for creating ad hoc queries and reports. In this chapter, we will create a query and a report using QuickDesigner Reports.

To start QuickDesigner Reports, go to the EDAB Start page and login using the user created in Section Q.2.1 - Create a User. When you login click the link labeled *QuickDesigner Reports* to launch QuickDesigner Reports.


Q.5.1. Create a Query

QuickDesigner Reports allows you to build a report using any data source from the Organizer. However, if you want to modify a data source, you can only query DataViews or modify existing DataView queries using QuickDesigner Reports.

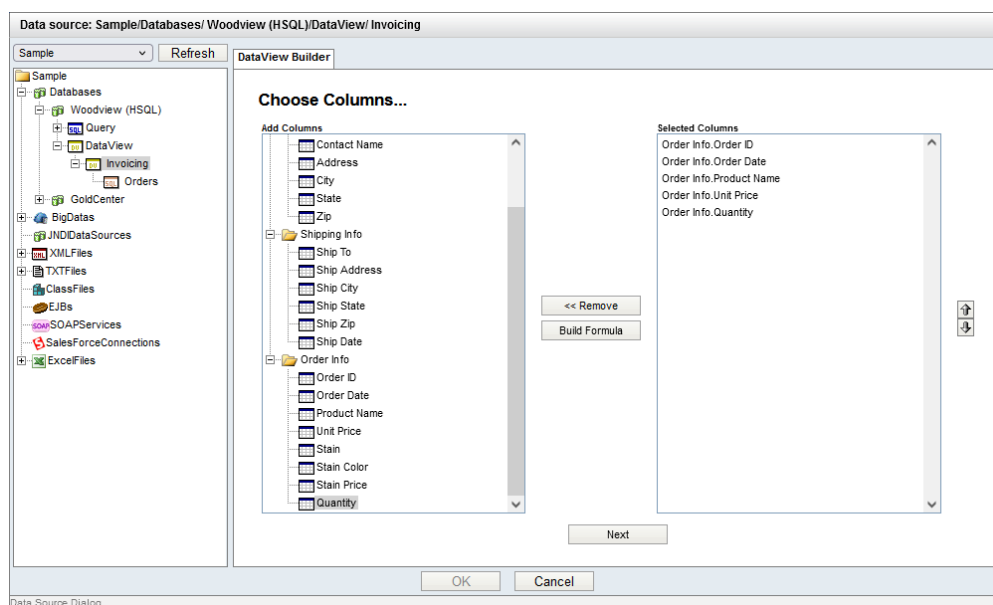
When QuickDesigner Reports opens, there are only two options accessible, to create a new report or open an existing one.



QuickDesigner Reports

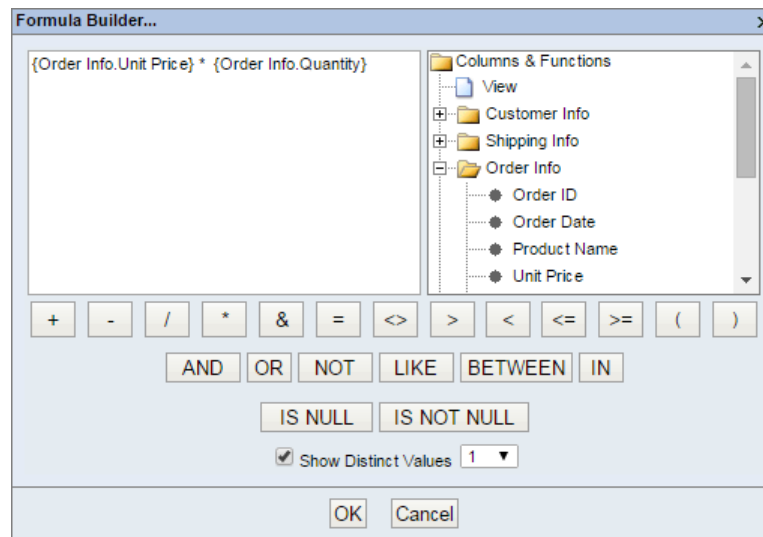
Click the  *New* button on the toolbar and the *Data Source Dialog* will open. Select the *Quickstart_sample* data registry from the left top drop-down list (the data registry you created in Section Q.3.1 - Create a Data Registry). The tab below shows all of the defined data sources in the registry. Expand the nodes and select the *DataView Invoicing* that you created in Section Q.3.1.3 - Create a Data View. You will see all headings and fields of the view in *DataView Builder* in the right pane of *Data Source Dialog*. Add the following fields from the *Order Info* heading by clicking on them in the left side panel.

Order ID
Order Date
Product Name
Unit Price
Quantity



Data Source Dialog

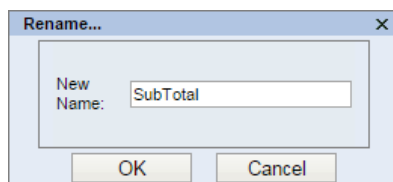
After adding the columns, click the *Build Formula* button to build a computed column. This will launch the Formula Builder in a new window.



Formula Builder

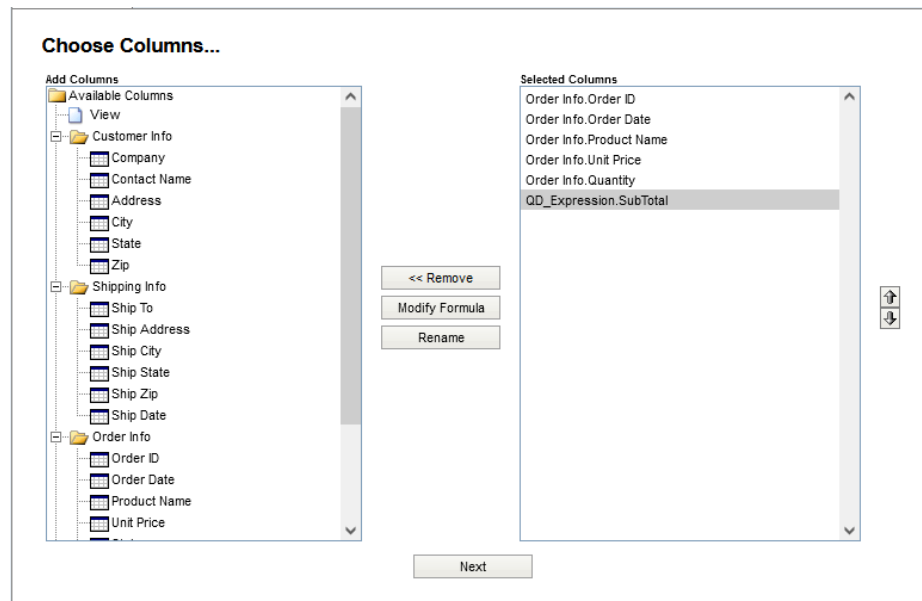
In the Formula Builder, expand the *View* node in the right-hand side. Then expand the *Order Info* node. Under this node click the *Unit Price* entry. The text will be added to the main formula window. Click the *multiply button* (*). Finally, select the *Quantity* field. The finished formula should look like this: {Order Info.Unit Price} * {Order Info.Quantity}. Once you have finished, click *OK* to add the formula. A new column will be added in the column selection dialog.

To provide an alias for the expression column, select it and click the *Rename* button. This will bring up a dialog allowing you to specify the alias.

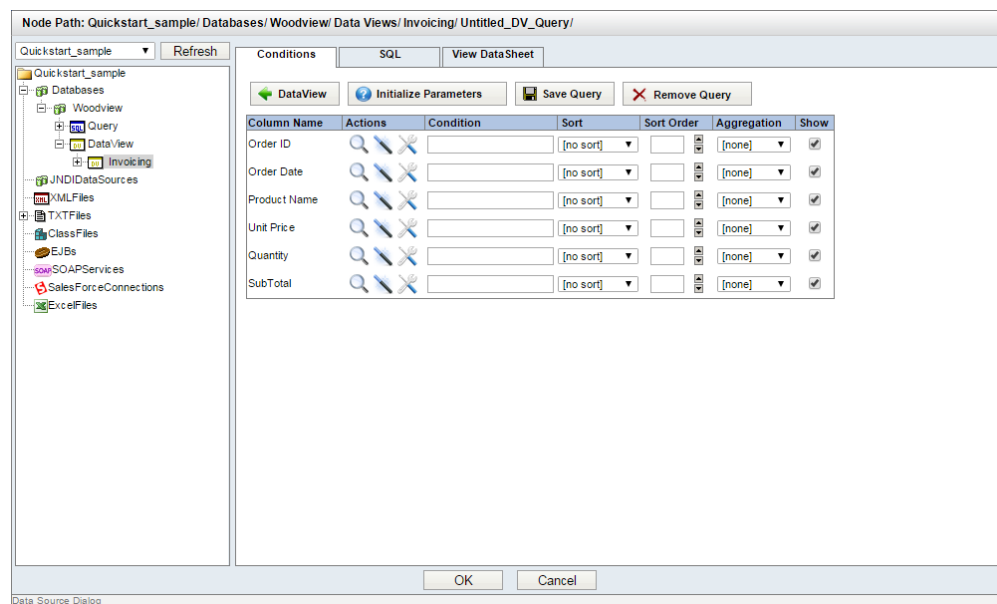



Rename Dialog

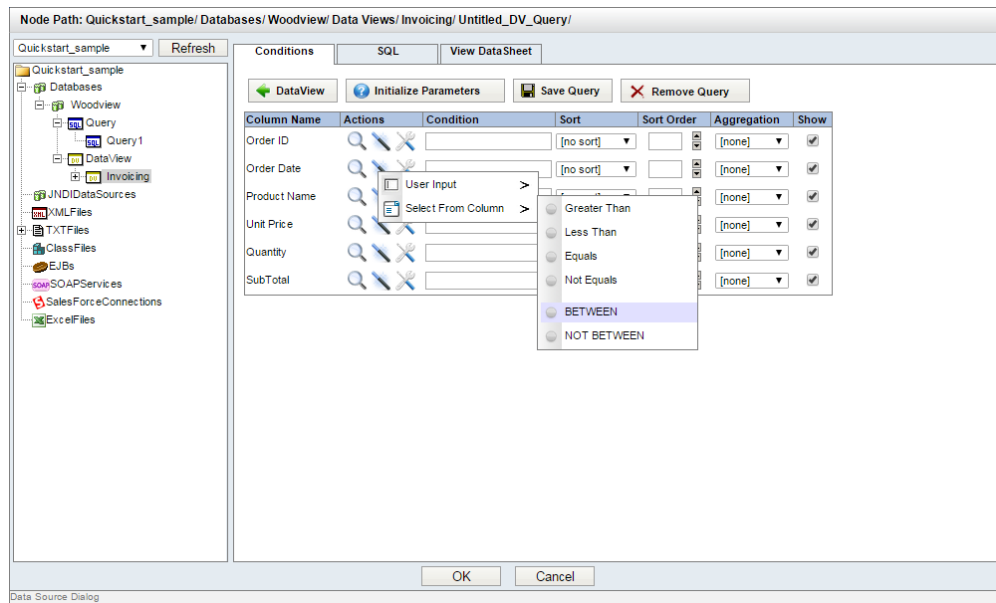
Enter **SubTotal** as an alias for the computed column and click *OK*. The computed column is renamed.

*Renamed Column*

Click the *Next* button to submit the column selection. The next dialog that opens allows you to set sorting, grouping, and conditions for the DataView query.

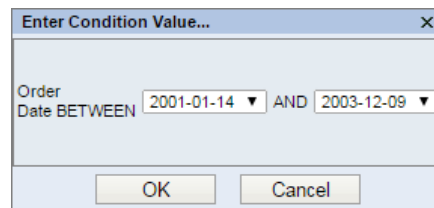
*Conditions Dialog*

Click the  *Condition Wizard* button for the Order Date field. Select the *Select From Column* option and then click the *Between* operator.



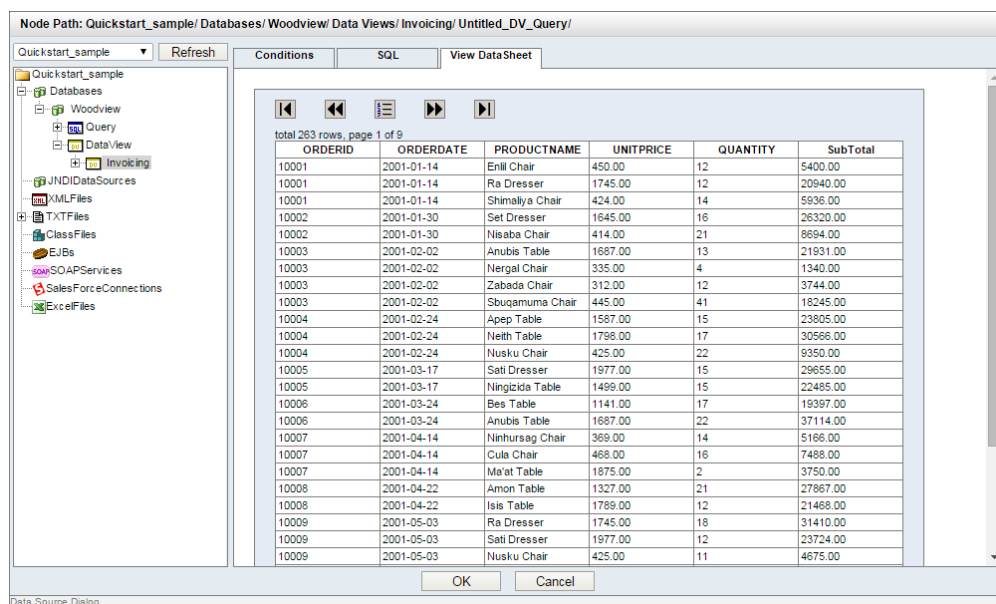
Condition Operators

This will open a dialog allowing you to select a date range by which to filter the query.



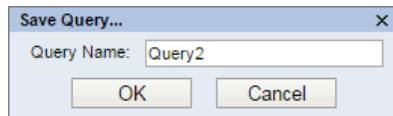
Select Condition Value Dialog

Select the date range that you would like and click *OK*. The condition will be added for the Order Date field. Now you can preview the finished query by clicking the *View DataSheet* tab.



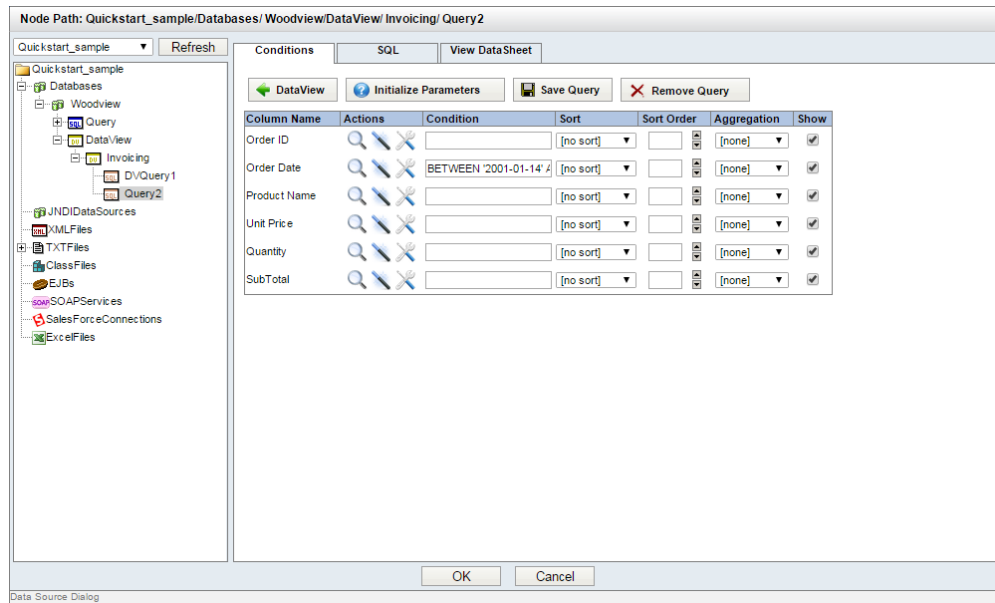
Data Preview

Next, go back to the *Conditions* tab and click the *Save Query* button to save this DataView query back to the registry. This will bring up a dialog prompting you to select a name for the query.



Save Query Dialog

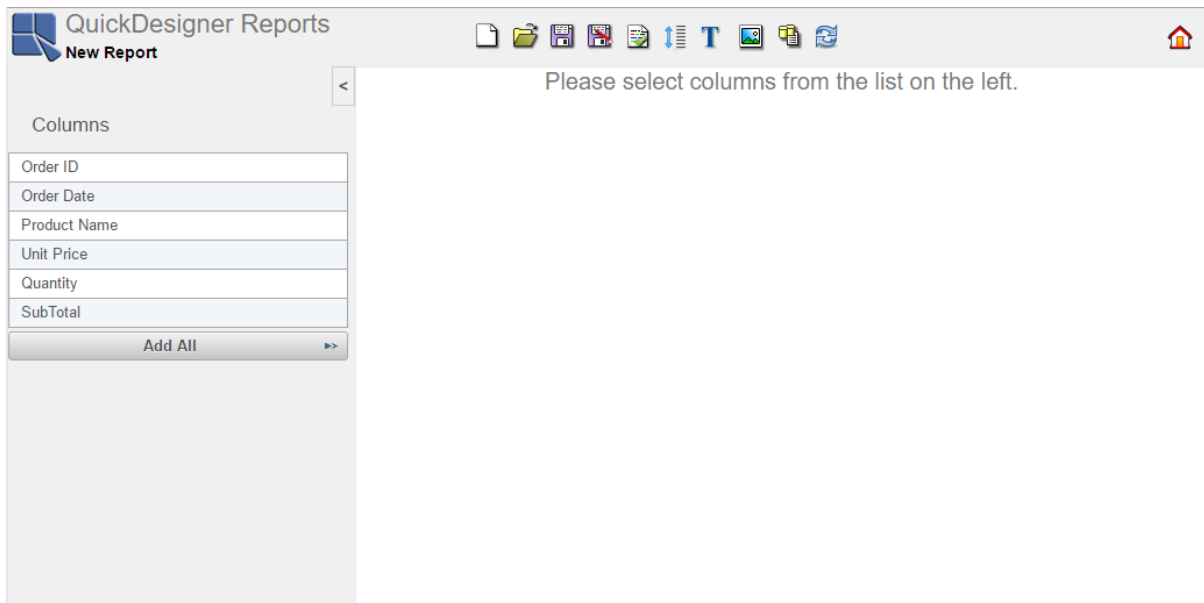
Enter the name you would like to use for the query and click *OK*. The new query appears under *DataView* node in the left pane of the *Data Source Dialog*.



New DataView Query

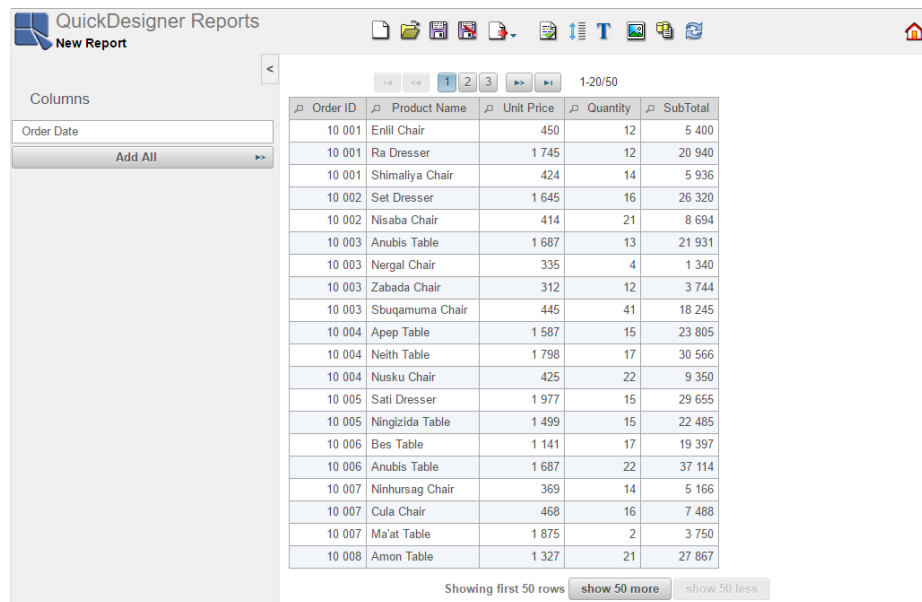
Q.5.2. Create a Report

Now that you have completed the query, click *OK* to close the *Data Source Dialog* and to open the main page of QuickDesigner Reports.




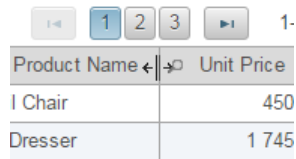
QuickDesigner Reports Main Page

You can insert any column to the report by using a double-click on the column or by dragging it to the report. So, insert columns *Order ID*, *Product Name*, *Unit Price*, *Quantity*, *SubTotal*.



Columns Selecting

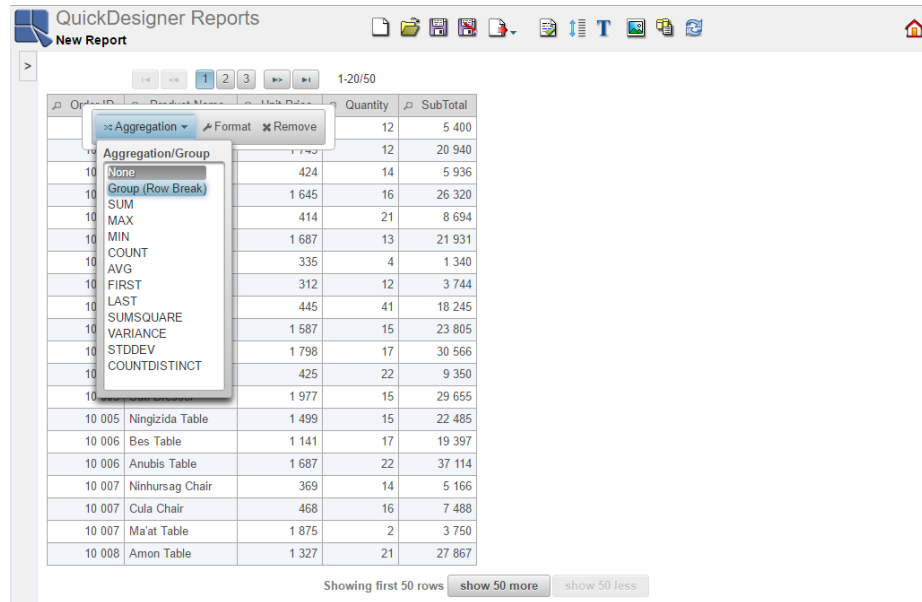
Collapse the left *Column* pane by clicking the  *Collapse* button and adjust the width of columns. To do this, move your mouse over the right side of the column header. You will see a double arrow. Then left click and drag it.



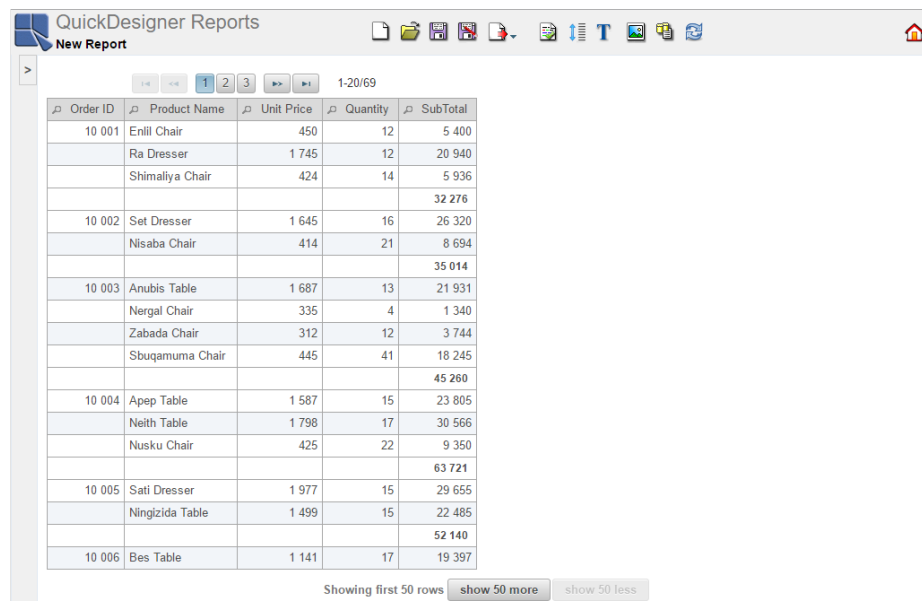
Product Name	Unit Price
I Chair	450
Dresser	1 745

Column Width Setting

Right click on the *Order ID* header, select *Aggregation* and then *Group(Row Break)* option.

*Aggregation Dialog*

Right click on the *SubTotal* header, select *Aggregation* and then *SUM* option. Your report should look like this now:



Order ID	Product Name	Unit Price	Quantity	SubTotal
10 001	Enlil Chair	450	12	5 400
10 001	Ra Dresser	1 745	12	20 940
10 001	Shimaliya Chair	424	14	5 936
10 001				32 276
10 002	Set Dresser	1 645	16	26 320
10 002	Nisaba Chair	414	21	8 694
10 002				35 014
10 003	Anubis Table	1 687	13	21 931
10 003	Nergal Chair	335	4	1 340
10 003	Zabada Chair	312	12	3 744
10 003	Sbuqamuma Chair	445	41	18 245
10 003				45 260
10 004	Apep Table	1 587	15	23 805
10 004	Neith Table	1 798	17	30 566
10 004	Nusku Chair	425	22	9 350
10 004				63 721
10 005	Sati Dresser	1 977	15	29 655
10 005	Ningizida Table	1 499	15	22 485
10 005				52 140
10 006	Bes Table	1 141	17	19 397

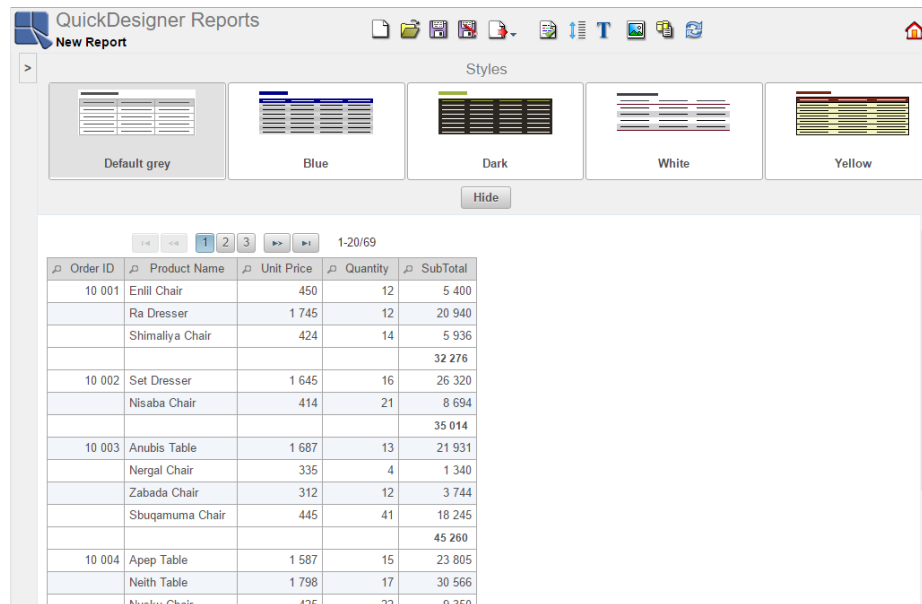
Report with Aggregation

Q.5.2.1. Format Report Elements

QuickDesigner Reports allows you to change the style of the report by using several predefined styles. Click the

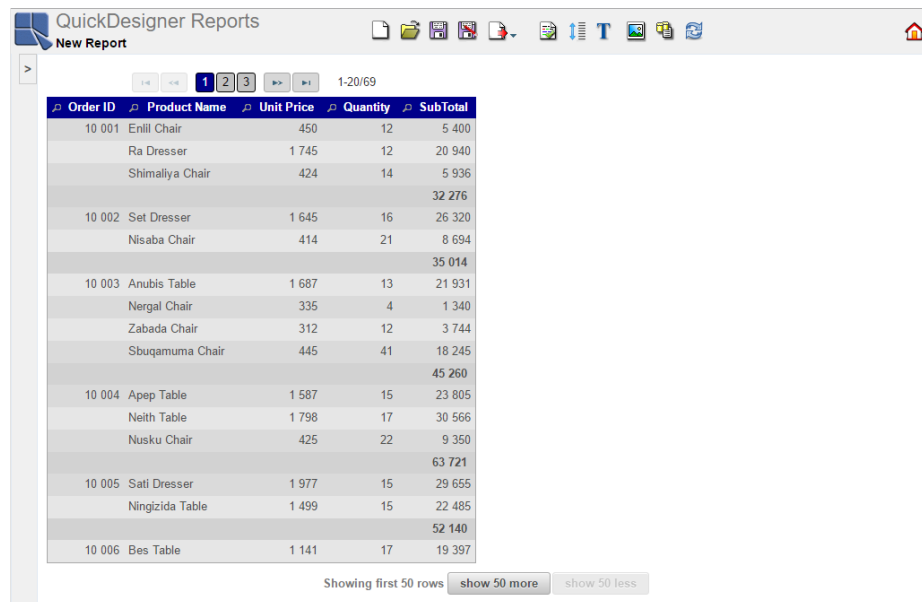


Styles button on the toolbar. The *Styles* dialog will open above the report.



Styles Dialog

Select the style you like, e.g. the *Blue* style, and close the *Styles* dialog by clicking *Hide* button.



Style Applied



Next, place a pagination under the report. Click *Pagination/Scrolling* button on the toolbar and *Pagination/Scrolling* dialog opens. Select *Bottom* option for *Paginator Position* and click *OK*.

Pagination/Scrolling Dialog

Now, add a report title. Click the **T** *Report Title* button on the toolbar and the *Report Title* dialog appears. Type **Order Information** as the title and click *Apply* button.

Report Title Dialog

The report title appears on the left side of the report by default. Move it to the center of the report. Click on the title, hold the mouse button and move the mouse slightly. Dashed rectangles appear. Move the title to the middle rectangle and release the mouse button.

Order ID	Product Name	Unit Price	Quantity	SubTotal
10 001	Enlil Chair	450	12	5 400
	Ra Dresser	1 745	12	20 940

Title Placement


Your report should look like the report below. For more detail about all the formatting option available in Quick-Designer Reports, see Section 4.2.4 - Format Report Elements.

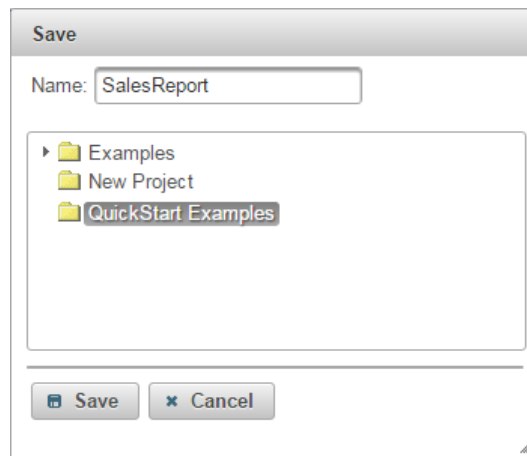
Order Information

Order ID	Product Name	Unit Price	Quantity	SubTotal
10 001	Enlil Chair	450	12	5 400
	Ra Dresser	1 745	12	20 940
	Shimaliya Chair	424	14	5 936
				32 276
10 002	Set Dresser	1 645	16	26 320
	Nisaba Chair	414	21	8 694
				35 014
10 003	Anubis Table	1 687	13	21 931
	Nergal Chair	335	4	1 340
	Zabada Chair	312	12	3 744
	Sbuqamuma Chair	445	41	18 245
				45 260
10 004	Apep Table	1 587	15	23 805
	Neith Table	1 798	17	30 566
	Nusku Chair	425	22	9 350
				63 721
10 005	Sati Dresser	1 977	15	29 655
	Ningizida Table	1 499	15	22 485
				52 140
10 006	Bes Table	1 141	17	19 397
1 20/69				

Report With Changes Applied

Q.5.2.2. Save the Report

Now that you have finished formatting, you can save the report you have created back to Organizer so that other users can view it. To save the report, click the *Save* button on the toolbar . This will bring up a dialog allowing you to specify a name for the template.



Save Report Dialog

Enter a name for the report, and select the *QuickStart Examples* project that you created in Section Q.2.2.2 - Add a Project. Click *Save* button to save the report.

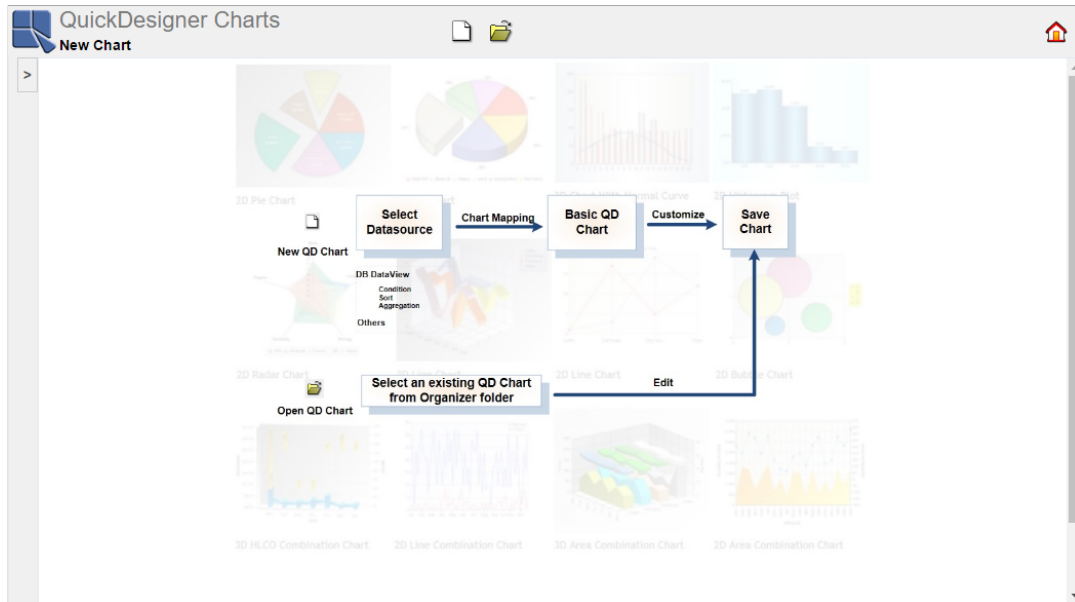
Q.6. QuickDesigner Charts

QuickDesigner Charts is a thin-client ad-hoc charting interface. It provides users a scaled-down design tool to create charts. With QuickDesigner Charts, end users can easily select, filter, and present data without mastering database structures, all with zero client download.


For more detailed information about all the chart options available in QuickDesigner Charts, see Section 4.3 - QuickDesigner Charts.

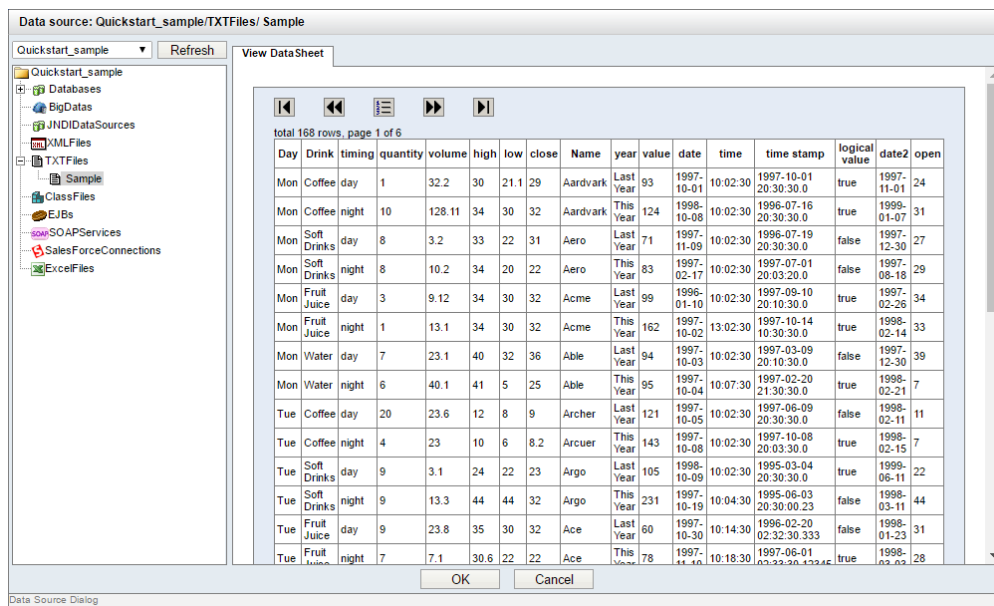
Q.6.1. Create a Chart

To start QuickDesigner Charts, go to the EDAB Start page and click the link labeled *QuickDesigner Charts*. QuickDesigner Charts should open, and you can create a new chart or open an existing one.



QuickDesigner Charts

Click the *Create New Chart* button on the toolbar . You will be taken to the *Data Source Dialog*, where you can select a data registry and a data source for a new chart. Select the *Quickstart_sample* data registry from the top left drop-down list, and then select the *Sample.dat* text file data source that you created in Section Q.3.1.4 - Setup a Text Data Source. You can see records of the text file in the *View Data Sheet* window on the right side of the *Data Source Dialog*. Click the *OK* button to continue.



Data Source Dialog

You can see the *Chart Mapping* dialog that allows you to select the chart type and mapping options. Select to create a **2D High Low** chart using the first options in the dialog.

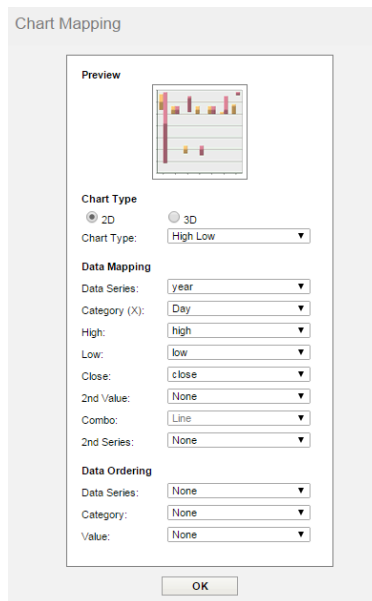
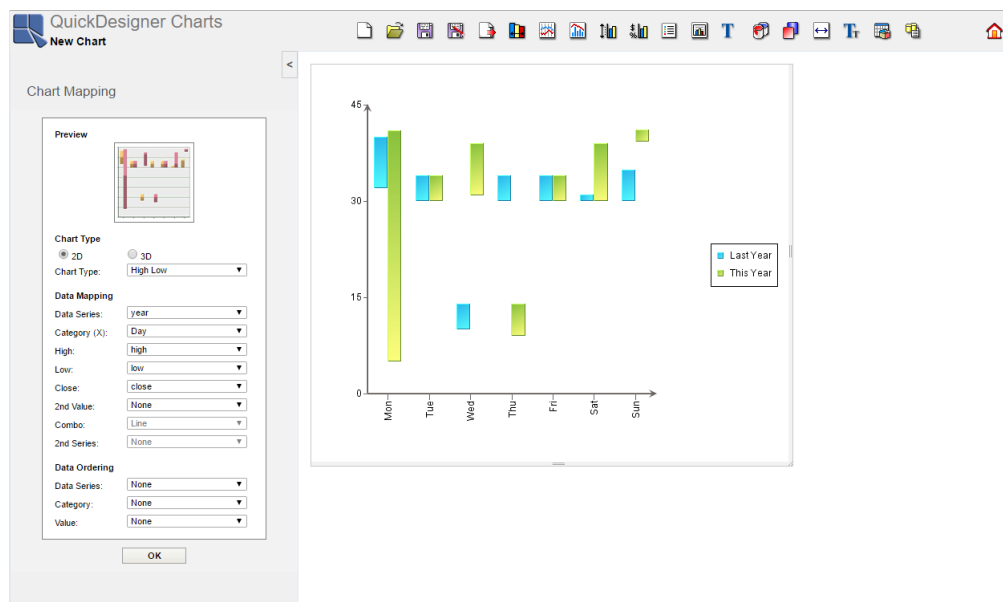



Chart Mapping Dialog

For the mapping options select **year** as the *Data Series*, **Day** as the *Category*, **high** as the *High*, **low** as the *Low*, and **close** as the *Close*. Click *OK* and the chart will open on the right side of QuickDesigner Charts.



QuickDesigner Charts Interface

Q.6.1.1. Format Chart Elements

Click the *Canvas Size* button on the toolbar . This will bring up a dialog allowing you to set the size of the chart canvas. Set the size to be **500** by **400** pixels and click *OK*.

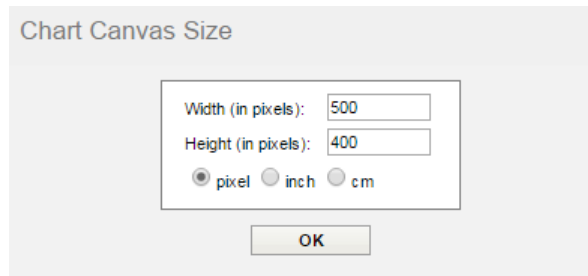



Chart Canvas Size Dialog

Next, click the *Font Settings* button on the toolbar . This will bring up a dialog allowing you to set the font options for the different text elements in the chart. From the drop-down list in the top of the dialog select **X Axis Labels**. Then set the font to be **Arial**, **8 pt**, and **Plain**.

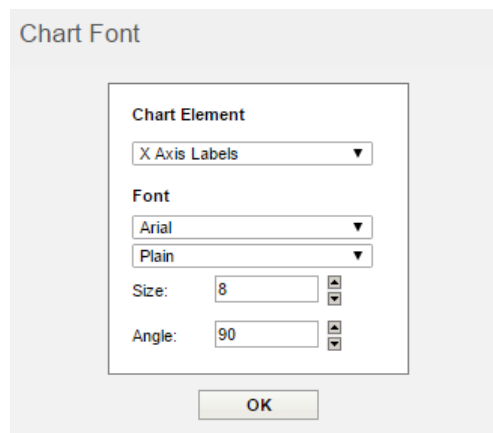

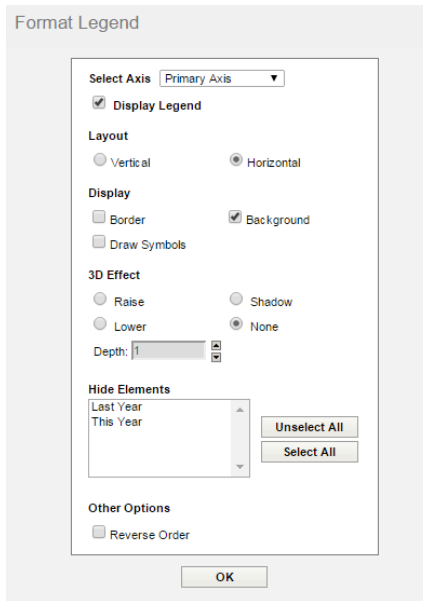


Chart Font Dialog

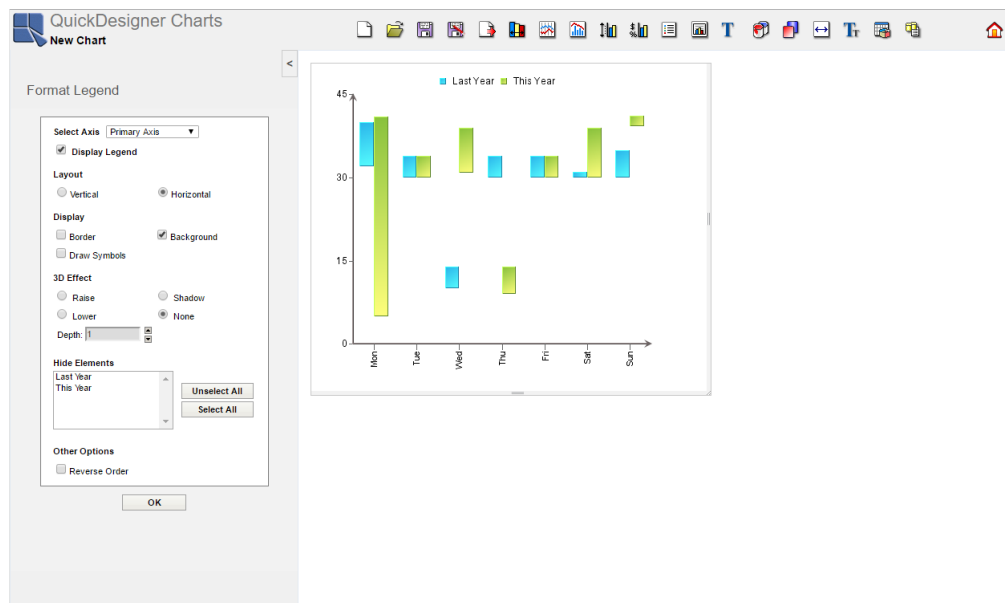
Now repeat this process for the Y axis labels and the legend text, setting them all to be **Arial**, **8pt** and **Plain**. Click *OK*.

Next, click the *Format Legend* button on the toolbar . This will bring up a dialog allowing you to set display options for the chart legend. Change the *Layout* option to **Horizontal**, uncheck the *Border* option, and change the *3D Effect* option to **None**. Click *OK* to apply the changes.



Format Legend Dialog


Now left click on the chart, hold the mouse button down and move the chart to keep the best position. Do the same with the legend. You can also set the plot area size by dragging the sides or dragging the corners of the chart. To do this, mouse over the side or the corner of the chart until you see the arrow. Then left click and drag.

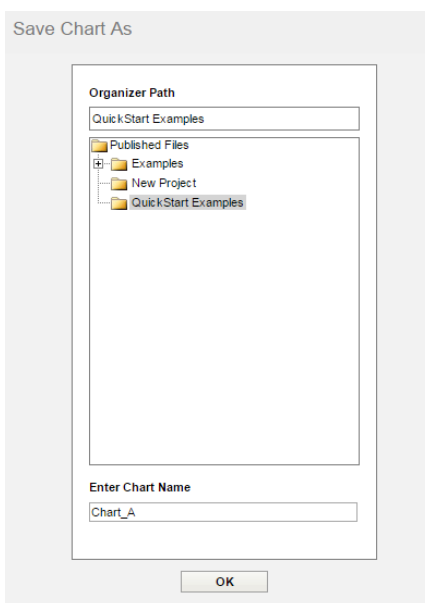


Finished Chart

For more detailed information about all the chart formatting options available in QuickDesigner Charts, see Section 4.3.5 - Data Formatting.

Q.6.1.2. Save the Chart

Now that you have finished formatting the chart, you can save it back to the Organizer. To save the chart, click the *Save* button on the toolbar . This will bring up a dialog allowing you to specify a name for the chart.



Save Chart Dialog

Select the *QuickStart Examples* project that you created in Section Q.2.2.2 - Add a Project and enter a name for the chart. Click *OK* to save the chart. The window will then give you a message to indicate that the chart was saved successfully. Click *OK* to close the dialog.

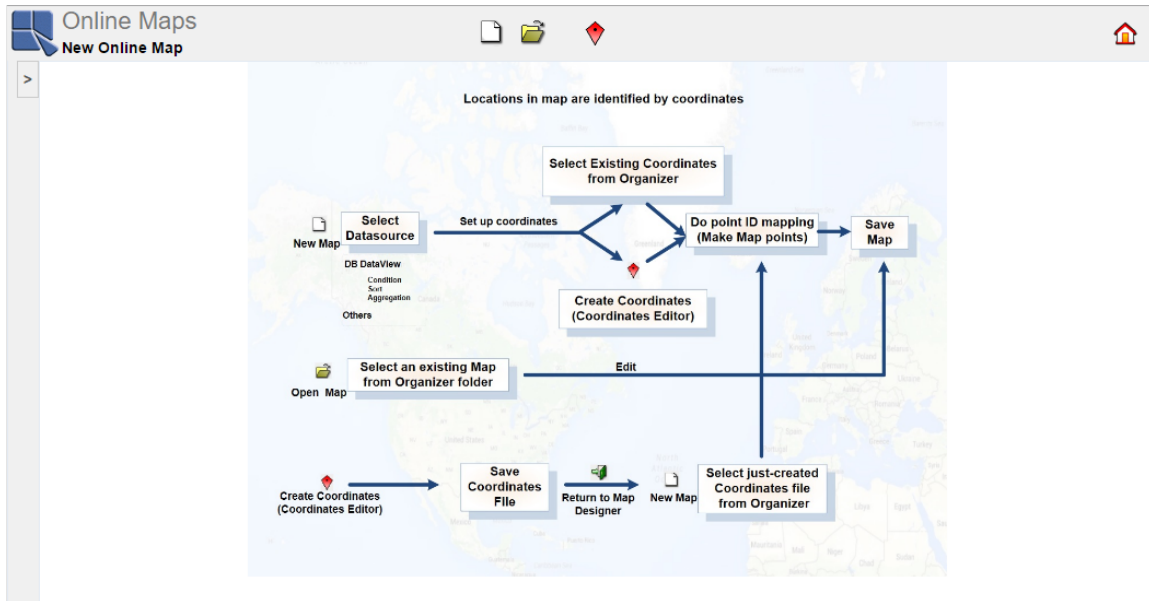
Q.7. Online Maps

An alternative way of presenting your data is by using the EDAB maps feature. EDAB maps are designed to report geographical data from data sources. They fetch geographical data from a data source and mark them on a map. There are two types of maps: Online Maps and SVG Maps. We will work with Online Maps at first and then with SVG Maps. (SVG Maps are described in Section Q.8 - SVG Maps.)


Q.7.1. Create Coordinates

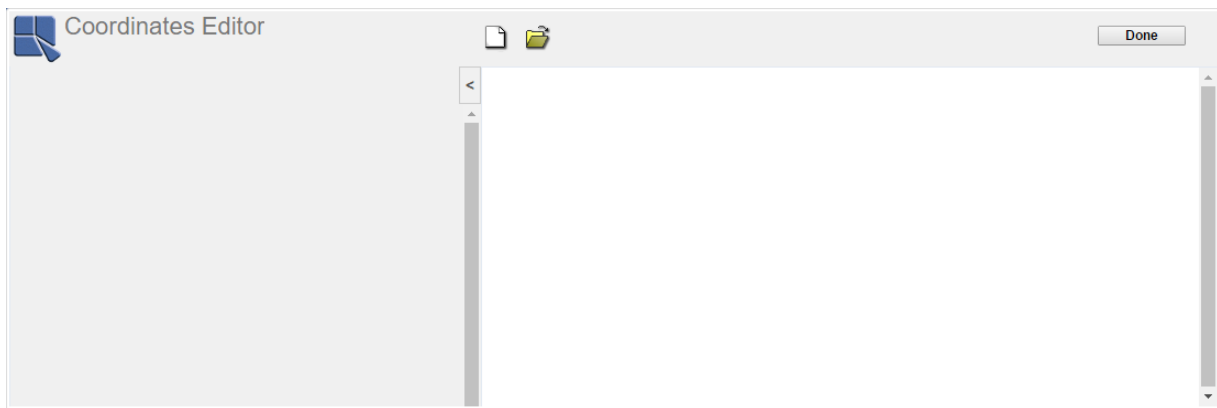
Before creating an Online Map, you will need to create a coordinates file. Coordinates files contain coordinates of places that you want to mark on the map. For more details about coordinates, see section Section 5.2.4.2 - Create Coordinates.

After launching Online Maps, you can create a new map, open some existing map, or create/edit coordinates.




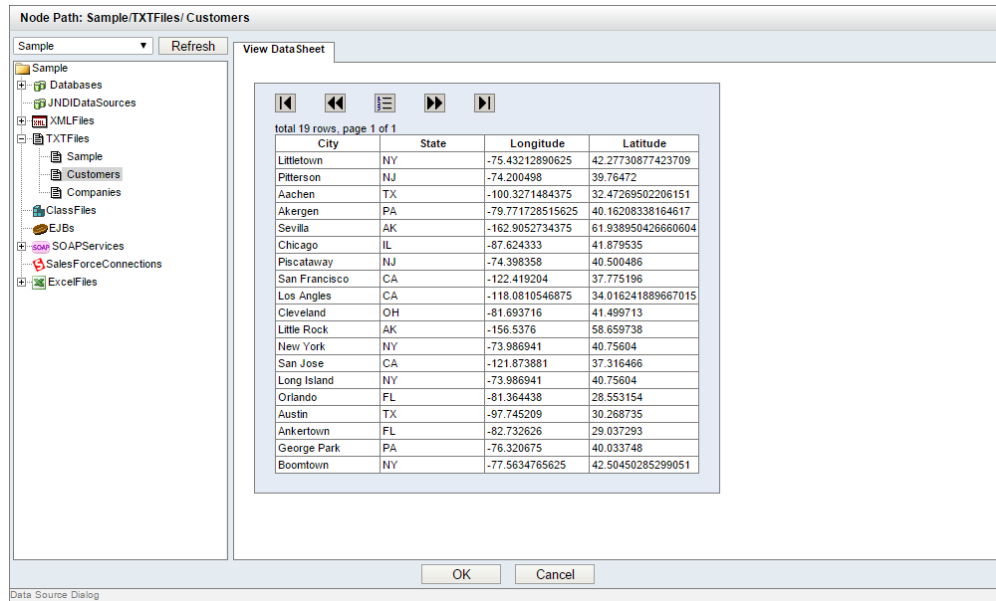
EDAB Online Maps

Click the  *Coordinates Editor/Edit Coordinates* icon on the toolbar. The Coordinates Editor will open.



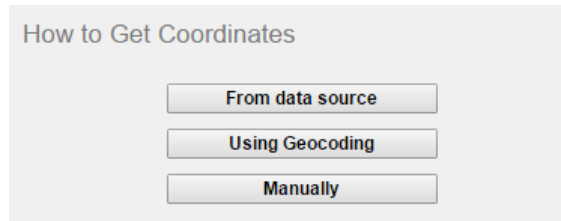
Coordinates Editor Start Options

Click the  *New Coordinates* icon. The *Data Source Dialog* will appear, allowing you to select a data source for the coordinates file. Select the *Sample* data registry and click on the *TXTFiles* node to show all TXT data sources. Select the *Customers* data source. You can see a table displaying records of the selected data source in the right pane. It allows you to check whether you chose the right data source. Click *OK* to close the *Data Source Dialog*.



Selecting Data Source

You will be prompted to choose a method of obtaining coordinates from the data source. There are three methods for this purpose (to learn more about how to choose the right method, see section Section 5.2.4.2 - Create Coordinates). The best method for our *Customers* data source is to obtain the coordinates directly from the data source, so click the *From data source* option.



How To Get Coordinates

Next step is to select data source fields that contain the coordinates.



Note

This step would be different if you chose some other method of obtaining coordinates (all possibilities described in Section 5.2.4.2 - Create Coordinates).

Each Coordinates data record consists of latitude and longitude (obtained automatically or manually) and a Point ID. The **Point ID** is an identifier of the place (e.g. city name, branch name, etc) and it is loaded from the data source you chose to create Coordinates from. The Point ID can consist of several **Point ID fields** - each data source column creates one field (except the fields, which contain longitude and latitude). Our data source has four columns *City*, *State*, *Longitude*, and *Latitude*. Therefore, the Point ID will consist of two fields - *City* and *State*. The upper option is labeled *Longitude* and the lower option is labeled as *Latitude*. Just set the upper field to data source field that contains information about longitude (in this case, it is the *Longitude* field) and the lower field to the data source field that contains latitude (*Latitude*). The following screenshot shows the correctly set dialog:

Coordinates From Data Source

☒ Lon/Lat Longitude Longitude ▾
☐ WKT WKT Latitude ▾
☐ WKT WKT City ▾

< Back Apply

Select Fields With Coordinates

Your dialog should look like the one in the previous screenshot. If it does, click the *Apply* button.

You can see a map with coordinates table in the right pane of the Coordinates Editor. When you move the mouse arrow over one of the map markers or over a Point ID Table row, it will highlight the map marker and Point ID Table row that match together. (For more information see Section 5.2.4.3 - Coordinates Editor Interface).

Coordinates Editor

Coordinates From Data Source


☒ Lon/Lat Longitude Longitude ▾
☐ WKT WKT Latitude ▾
☐ WKT WKT City ▾

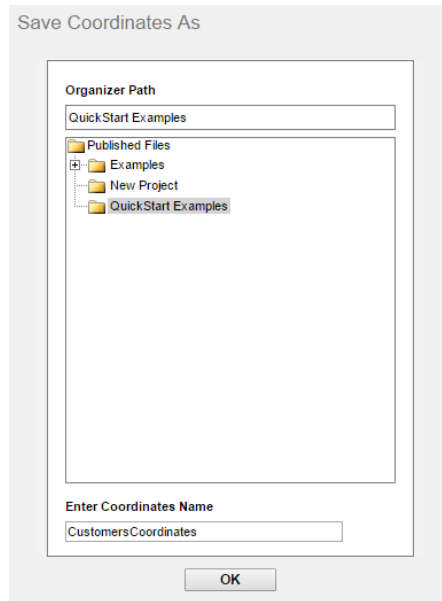
< Back OK

From DataSource

City	State	Longitude	Latitude
Littletown	NY	-75.43212890625	42.27730877423709
Pitterson	NJ	-74.200498	39.76472
Aachen	TX	-100.3271484375	32.47269502206151
Akergen	PA	-79.771728515625	40.16208338164617
Sevilla	AK	-162.9052734375	61.938950426660604
Chicago	IL	-87.624333	41.879535
Piscataway	NJ	-74.398358	40.500486
San Francisco	CA	-122.419204	37.775196
Los Angeles	CA	-118.0810546875	34.016241889667015
Cleveland	OH	-81.693716	41.499713
Little Rock	AK	-156.5376	58.659738
New York	NY	-73.986941	40.75604
San Jose	CA	-121.873881	37.316466
Long Island	NY	-73.986941	40.75604
Orlando	FL	-81.364438	28.553154
Austin	TX	-97.745209	30.268735
Ankertown	FL	-82.732626	29.037293
George Park	PA	-76.320675	40.033748
Boomtown	NY	-77.5634765625	42.50450285299051

Coordinates Editor

Now you may save the coordinates file. To save, click on the  *Save* icon on the toolbar.



Save Coordinates

Type **CustomersCoordinates** into the *Enter Coordinates Name* textfield and select the *QuickStart Examples* Organizer folder created in Section Q.2.2.2 - Add a Project. Click *OK* to save the coordinates file.


Close the Coordinates Editor and go back to the Online Maps designer by clicking the *Done* button on the right side of the toolbar.

Q.7.2. Create Online Map

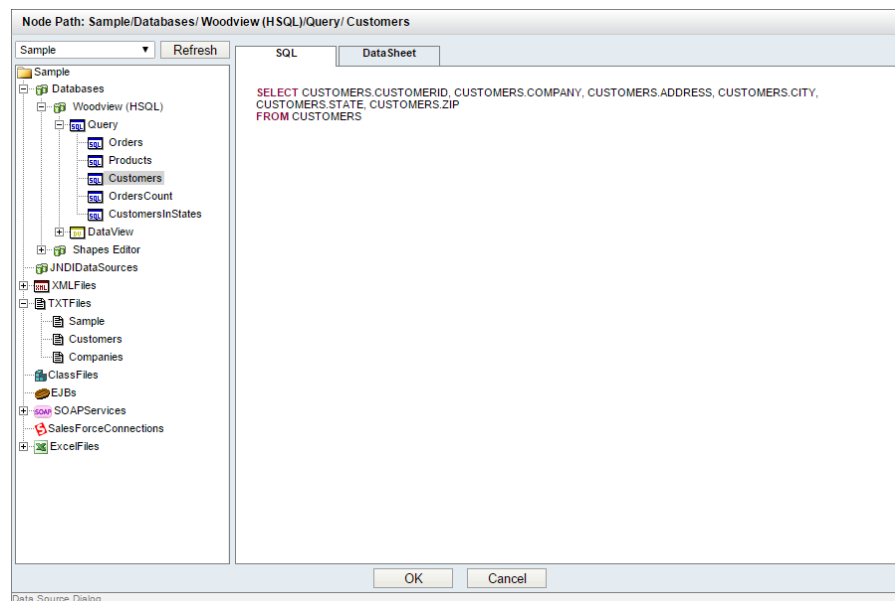


Note

If you have not practiced creating coordinates, please do so. It may be extremely difficult to create an Online Map without proper coordinates.

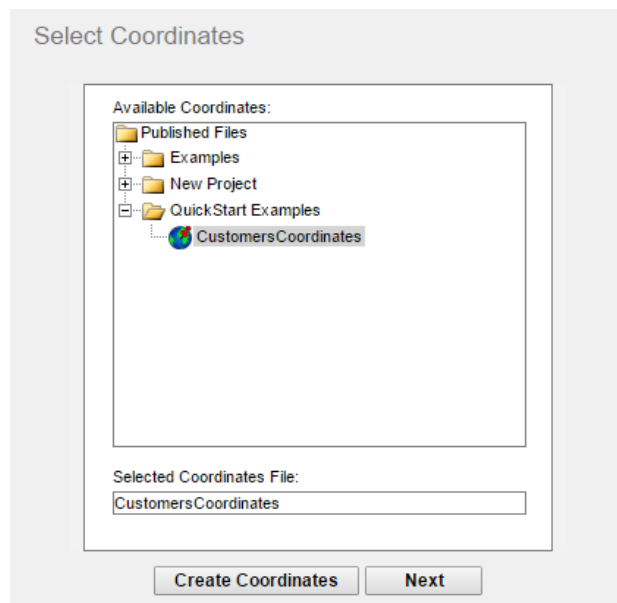
Open the *Data Source Dialog* by clicking the  *New Map* icon on the Online Maps toolbar. Select the *Sample* data registry.

We will use a different data source than the one we used for the coordinates in previous section. Open the *Databases* node, then the *Woodview (HSQL)* and the *Query* node and select the *Customers* query. Click *OK* to close the *Data Source Dialog*.



Selecting Data Source

Now, you will have to select a coordinates file from the Organizer. You will be able to see all projects, folders, and coordinates that were inserted into EDAB Organizer. To select the coordinates file we created in previous section, open the QuickStart Examples project (click on it), then select the *CustomersCoordinates* coordinates and click *Next*.




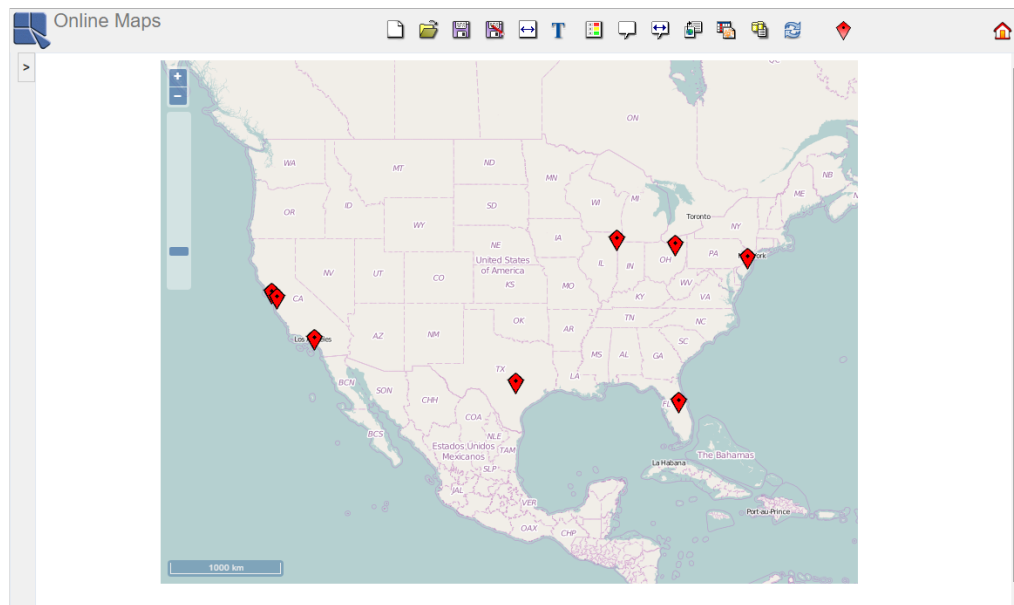
Selecting Coordinates File

Next, you will need to set point mapping. To do this, you need to map at least one field from the EDAB Online Maps data source to a Point ID field in the coordinates file data source. Both fields should contain the same type of information (for example, city name). But you can map more than one field if that is required to uniquely identify the location on the map (for example, city and state). When determining the map location of the EDAB Online Maps data source record, the mapping function automatically searches for the same value of the mapped field(s) in EDAB Online Maps data source and the mapped field(s) of the coordinates data source, and only the matched points will be visible on the map. In a nutshell, the coordinates data source contains information about location of data points on the map and EDAB Online Maps data source contains the data that you want to report on the map. You may read more about point mapping in Section 5.2.5 - Coordinates Mapping.

In this tutorial, we will map the `City` field to the `CITY` field and the `State` field to the `STATE` field. Just map the fields and click *Apply*.


Set Point Mapping

Setting point mapping was the final step of the Maps Wizard. You should be able to see the Online Maps toolbar and the Online Map with some marker on it (the left pane is collapsed by clicking the  *Collapse* button on the screenshot).



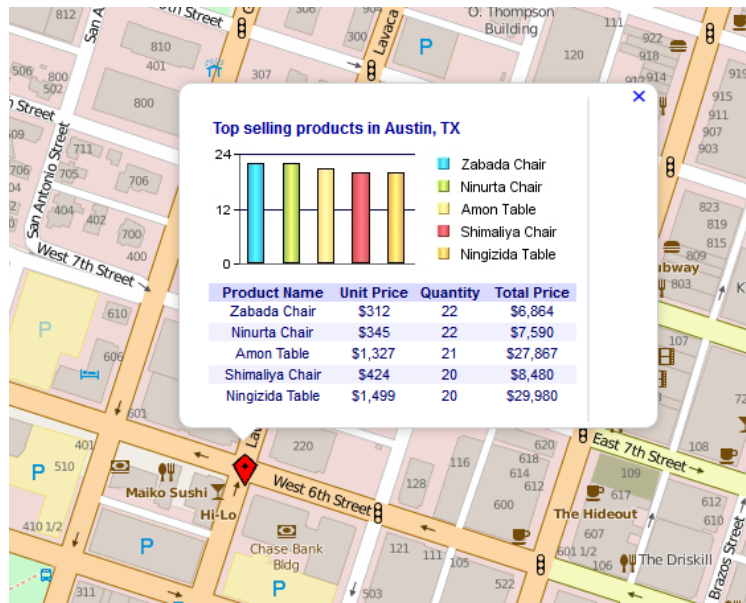
Online Maps Designer

The Online Maps toolbar allows you to configure the Online Map. You can immediately see results of all changes you make on the Online Map.

Save the map by clicking on the  *Save* icon on the toolbar. Type **CustomersOnlineMap** into the *Enter Map Name* textfield. Select the **QuickStart Examples** Organizer folder and click *OK* to save the map.

Q.7.2.1. Configure Tooltips

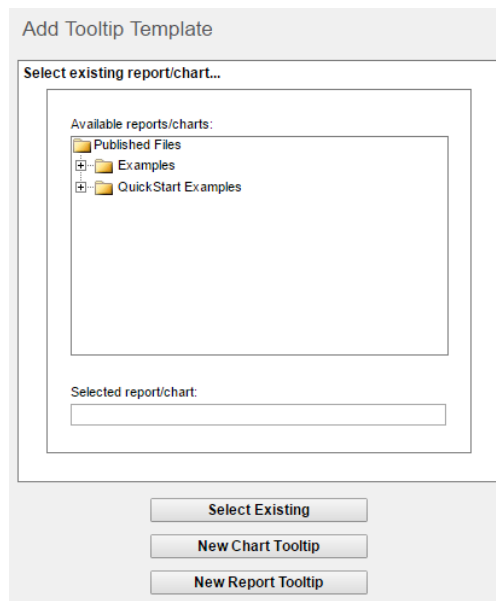
Tooltips show a brief report for a particular map marker. When the tooltips are enabled, each marker on the Online Map has its own tooltip. After setting the tooltip template (report or chart) for the Online Map, upon mouse over any map marker, the tooltip bubble will appear, displaying data relevant to the particular marker. Detailed information about the tooltips can be found in Section 5.2.6.7 - Tooltips.



Example: Google Map with Tooltip

Lets configure the tooltips for our CustomersOnlineMap that we created in the previous QuickStart section.

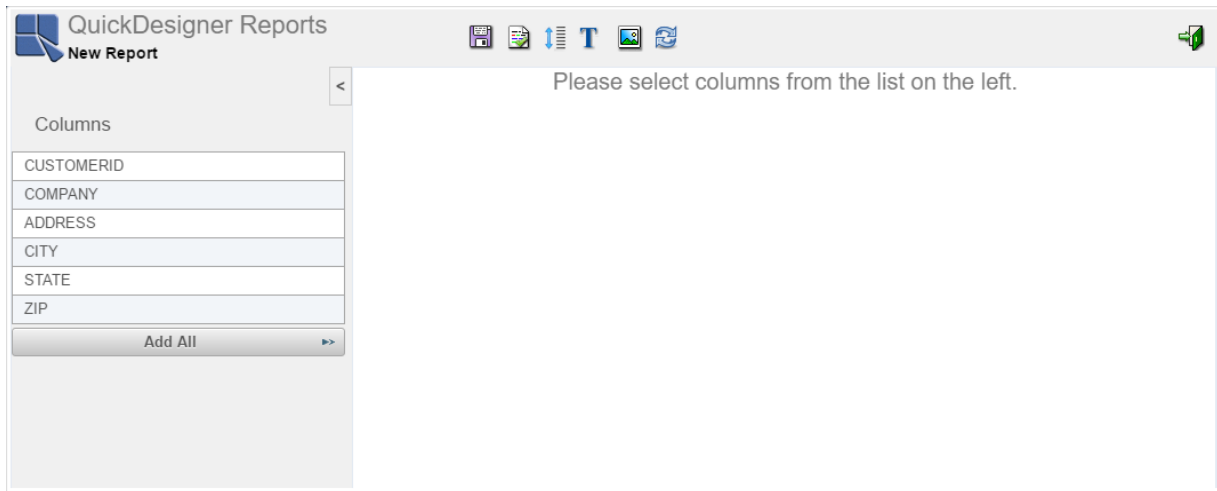
Click on the  *Tooltip template* icon on the toolbar. The *Add Tooltip Template* dialog will appear.



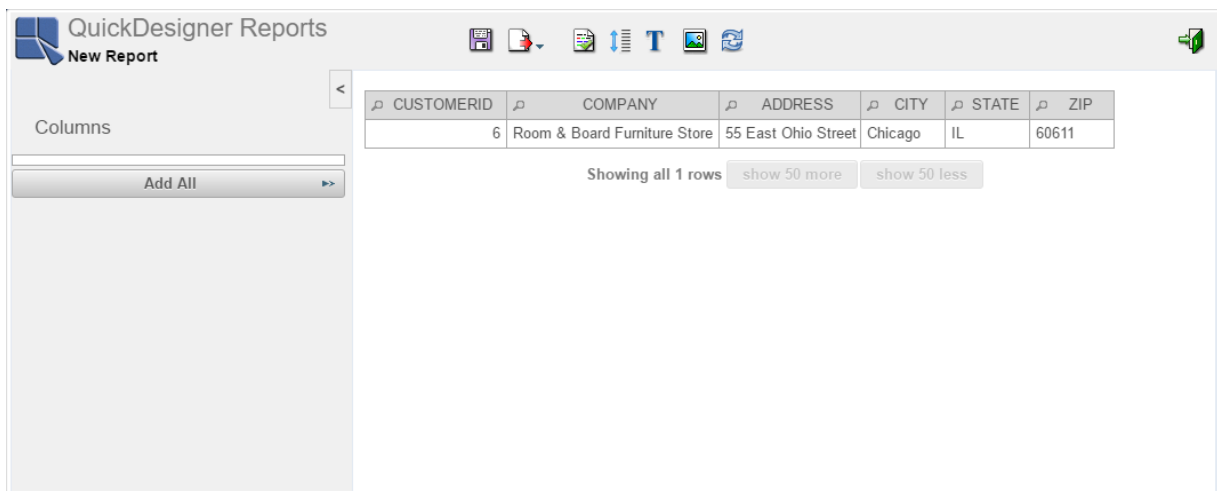
Add New Tooltip


You can select an existing tooltip template or create a new chart or report tooltip template. When you choose to select new chart template, QuickDesigner Charts will launch. When you choose to select new report template, QuickDesigner Reports will launch. The launched QuickDesigner Report or QuickDesigner Charts will use the Map Data as a datasource to create a report or chart as a tooltip. In our example, Customer Query under Woodview(HSQL) Query node is the datasource used by Quick Designer to create tooltip. For now, we will just make a very simple report in QuickDesigner Reports. Visit Section Q.5 - QuickDesigner Reports, or the Section 4.2 - QuickDesigner Reports for information about how to use QuickDesigner Reports.

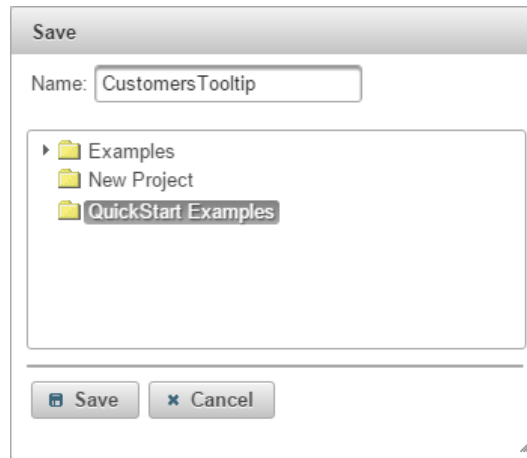
Click the *New Report Tooltip* option to open QuickDesigner Reports.

*QuickDesigner Reports*

Click *Add All* to add all columns to the report.

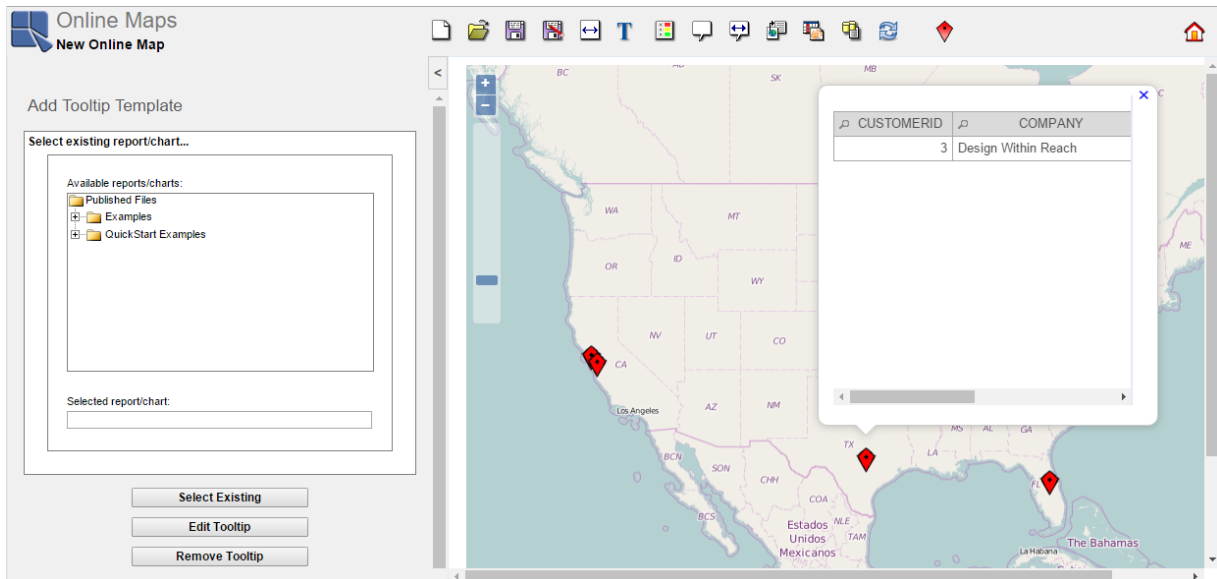
*Tooltip Report*

Click on the  Save icon on the QuickDesigner Reports toolbar. Enter **CustomersTooltip** as the *Name* and select the *QuickStart Examples Organizer* folder. Click *OK* to save the report.




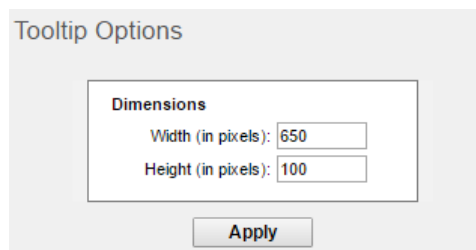
Save Report Dialog

Close QuickDesigner Reports by clicking the  *Return to Map Designer* icon on the toolbar.



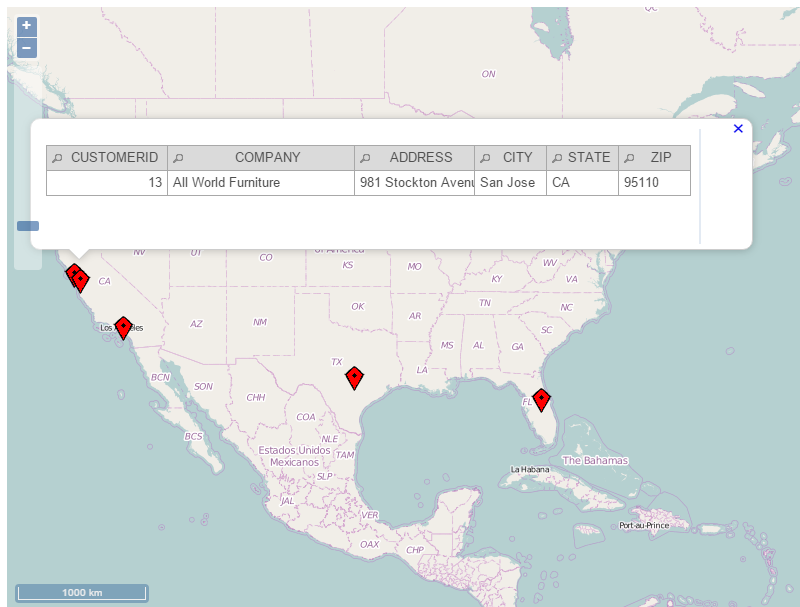
Adding Tooltip

Move the mouse arrow over some map marker and a tooltip bubble will appear. As you can see, the bubble has inconvenient dimensions for your report. To adjust the bubble dimensions, click on the  *Tooltip Options* icon on the toolbar. Set width to **650** and height to **100** and click **Apply**.




Tooltip Option Dialog

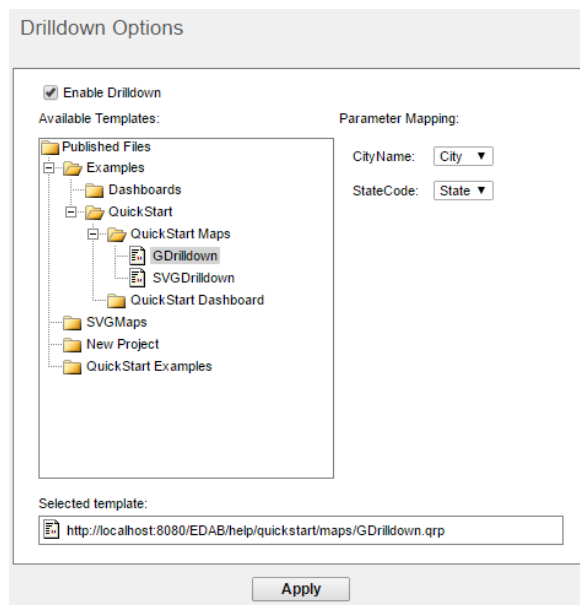
Close the currently open tooltip bubble (if any) and move mouse arrow over some marker. The tooltip should look better now.



Online Map with Tooltip

Q.7.2.2. Setting Drilldowns

Click on the  *Drilldown Options* icon on the toolbar. Select the *Enable Drilldown* option. The *Available Templates* treeview will open, showing all projects, folders and parameterized reports, charts and maps inserted in the Organizer. Open the *Examples* project, then open the *QuickStart* and *QuickStart Maps* folder. Select the *GDrilldown* report. This report has two parameters: *CityName* and *StateCode*. You have to select the *Online Map* database fields containing corresponding data. In this case, map the *CityName* parameter to the *City* field and the *StateCode* parameter to the *State* field. Click *Apply*. The drilldowns are configured now. Click on a map marker, report will open in a new window. More info in Section 5.2.6.9 - Drilldowns.



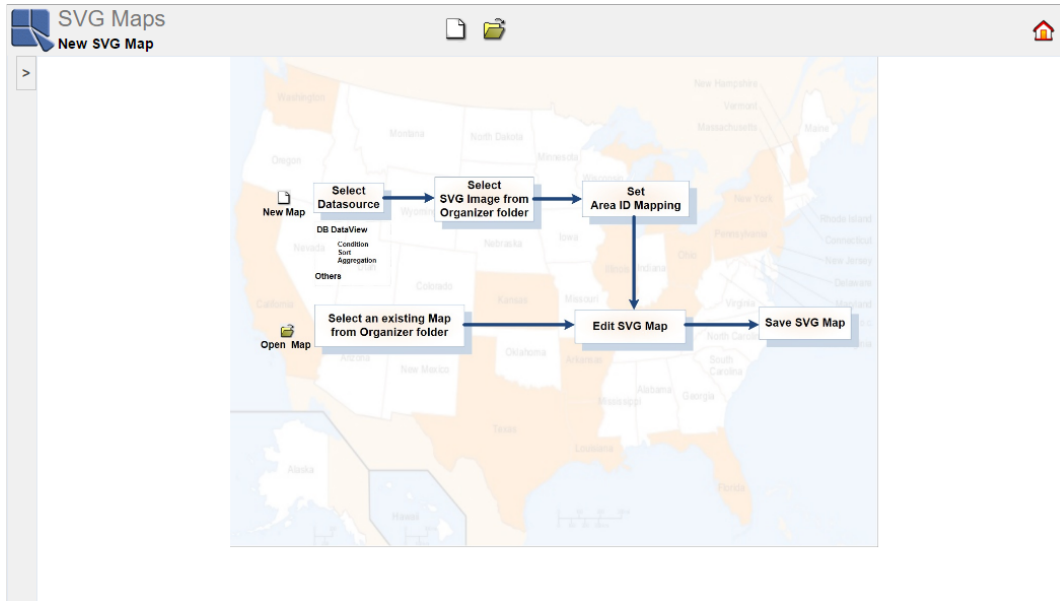
Drilldown Options

Q.8. SVG Maps


Unlike Online Maps, SVG Maps do not use map points. They use **map areas**. The map areas can be colored according to some values from the map data source. Coordinates of the map areas are defined in **SVG map image**. For more information see Section 5.3 - SVG Maps.

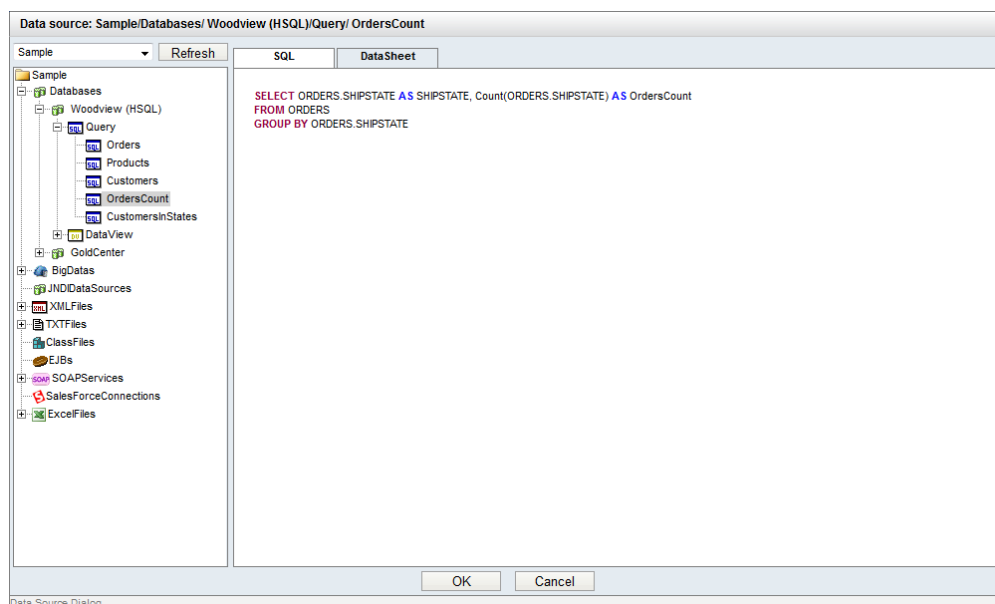
Q.8.1. Create a Map

To create a SVG Map, go to the EDAB Start page and click the link labeled *SVG Maps*. You can create a new SVG map or open some existing SVG map.



New/Open Selection

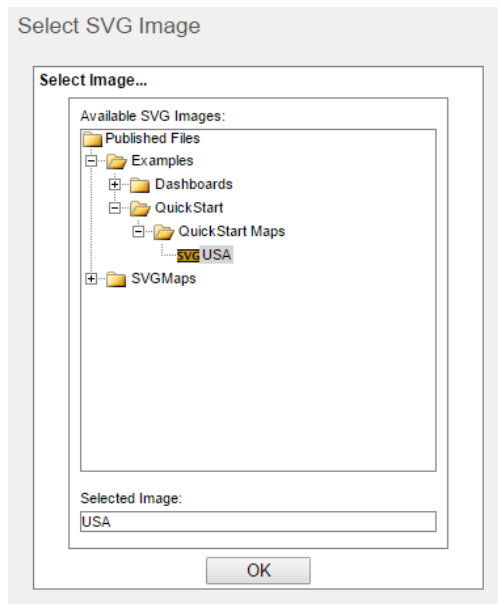
Click the  *Create New Map* button on the toolbar. You will be taken to the *Data Source Dialog*, where you can select a data registry and a data source for a new SVG Map. Select the *Sample* data registry (the left top corner of the window). Open the Databases/Woodview (HSQL) /Query node, and select the OrdersCount query.



Data Source Dialog

You can see records of the data source file in the *DataSheet* tab. Then click *OK* to close the *Data Source Dialog*.

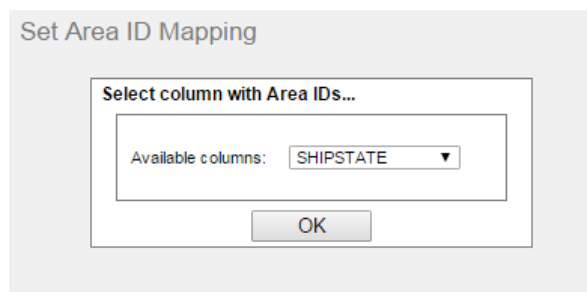
The next dialog allows you to select a Map Image. Map Images are SVG image files with data structures containing geographic data(see Section 5.3.3 - Area ID Mapping for more information about required image format), and must be inserted into the Organizer before they can be used.



Select SVG Image Dialog

The *Select SVG Image* dialog contains a tree showing all Projects, Folders and SVG images inserted in the Organizer. Open the *Examples* project, then the *QuickStart* and the *QuickStart Maps* folder. You should be able to see the SVG image named *USA* in the *QuickStart Maps* folder. Select it and click *OK*.

Now, you can set the connection between the data source and the SVG image. This is done by assigning a data source field to the SVG map Area IDs (see Section 5.3.3 - Area ID Mapping to learn more about area ID mapping). In this case, we have a map of the USA with state name abbreviations in its Area IDs. So we want to assign a field containing the same type of data, which is the *SHIPSTATE* field. Set the *Available columns* drop-down menu to *SHIPSTATE* and click *OK*.



Select Column with Area IDs

SVG map will open (*Set Area ID Mapping* dialog is collapsed on the screenshot).

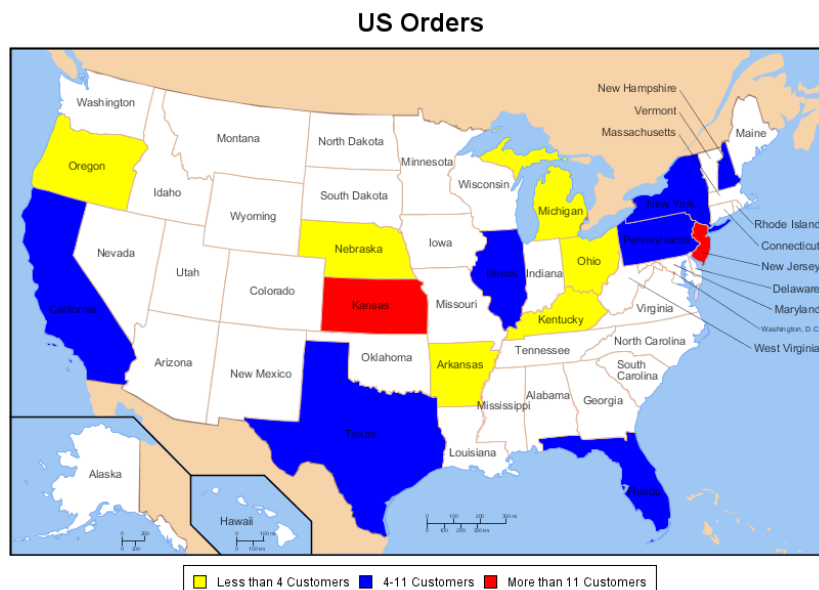


SVG Map Builder


As you can see, right now we have only the basic map with no data highlighted. There are two ways of adding data to the SVG Map: Thresholds and Drilldowns.

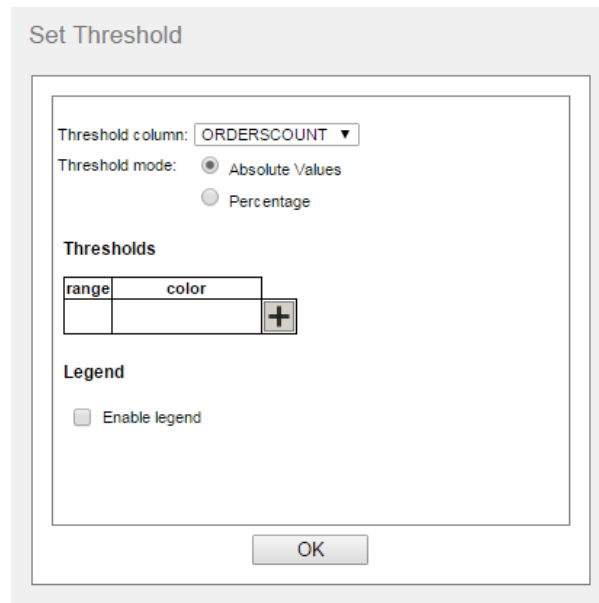
Q.8.2. Set SVG Map Thresholds

Thresholds allow you to configure rules for coloring map areas according to its value in a particular data source field (see Section 5.3.4.5 - Thresholds).



Example: SVG Map with Thresholds

To set thresholds, click on the  *Set Thresholds* icon on the toolbar. First of all, you have to select a data source column containing numerical data that you want to report. We will use a column labeled as **ORDERSCOUNT**, which holds information about number of orders made in a particular state.



Set Threshold

Threshold column: ORDERSCOUNT ▾

Threshold mode: ☒ Absolute Values ☐ Percentage

Thresholds

range	color

Legend

☐ Enable legend

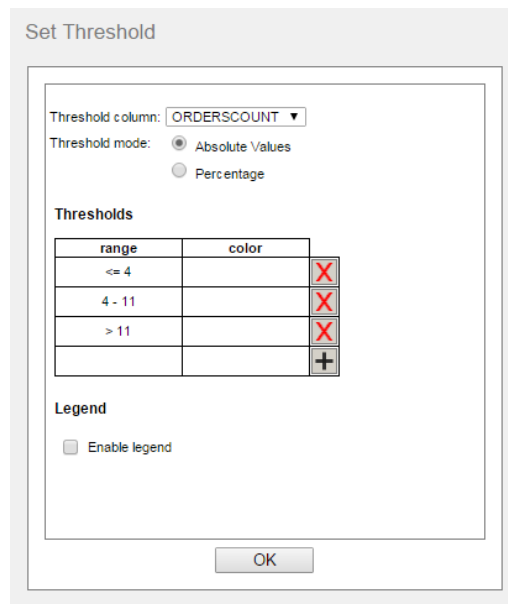
OK

Set Threshold Dialog

Next, you can add some threshold value(s) in actual values, or percentage unit. We will use actual values for our

example. Click on the **+** Plus icon next to the blank field labeled as *color*. You will be prompted to enter threshold value, which can be done by two ways – you can type it manually into the text field or you can select the value from the drop-down menu. (There are three ways to enter threshold values for percentage units. The third way is to enter a number of intervals.) Select the value **4** from the drop-down menu and confirm the dialog by clicking

OK. Click on the **+** Plus icon once again, and choose the value **11** from the drop-down menu and click OK. EDAB has automatically set the ranges between values you added.



Set Threshold

Threshold column: ORDERSCOUNT ▾

Threshold mode: ☒ Absolute Values ☐ Percentage

Thresholds

range	color
<= 4	
4 - 11	
> 11	

Legend

☐ Enable legend

OK

Setting Range

You can set color for each of the intervals. Click on the white color field next to the <=4 range. Click on some yellow field in the swatches color table and click OK. Color of the <=4 range has changed to yellow. Click on the color field next to the 4-11 range, choose a blue color and click OK. Click on the color field next to the >11 range, choose a red color and click OK. The Set Thresholds dialog should look like the one on the following screenshot:

Set Threshold

Threshold column: **ORDERSCOUNT**

Threshold mode: ☒ Absolute Values ☐ Percentage

Thresholds

range	color	
<= 4	Yellow	X
4 - 11	Blue	X
> 11	Red	X
		+

Legend

☐ Enable legend

OK

Setting Color

Confirm the dialog by the *OK* button. Areas of some states will be filled with the colors you previously set. Without a legend, it can be unclear what the colors mean. So let's create one. Select the *Enable legend* option in the Set Threshold dialog. The *Thresholds* table has changed. It has one more column now labeled as *legend*. You should be able to write into the legend field. Fill in the column according to the following screenshot.

Set Threshold

Threshold column: **ORDERSCOUNT**

Threshold mode: ☒ Absolute Values ☐ Percentage

Thresholds

range	color	legend	
<= 4	Yellow	Less than 4 Customers	X
4 - 11	Blue	4-11 Customers	X
> 11	Red	More than 11 Customers	X
			+

Legend

☒ Enable legend

Legend layout: **Horizontal**


Legend position: **Bottom**

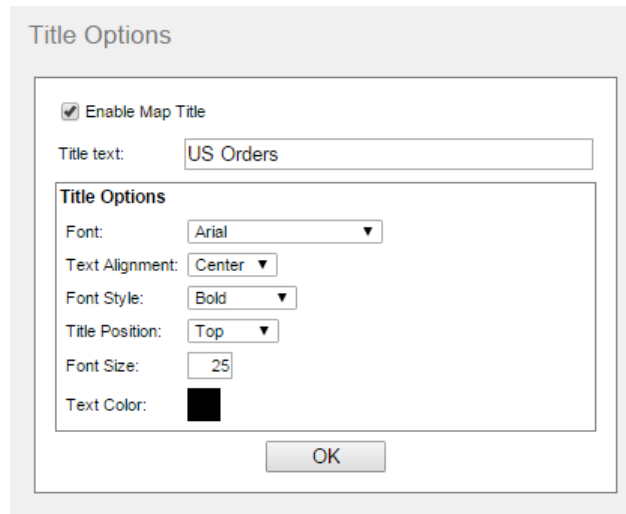
OK

Set Thresholds- Legend

Set the *Legend layout* drop-down menu to **Horizontal** and set the *Legend position* to **Bottom**. Click *OK*. Then, without map title, you have the same SVG Map image shown in the beginning of this part.

Q.8.2.1. Add a Map Title


Click on the  *Map Title* icon on the toolbar. Select the *Enable Map Title* option. Type **US Orders** into the *Title text* field, set *Font Style* to **Bold** and confirm by the *OK* button.

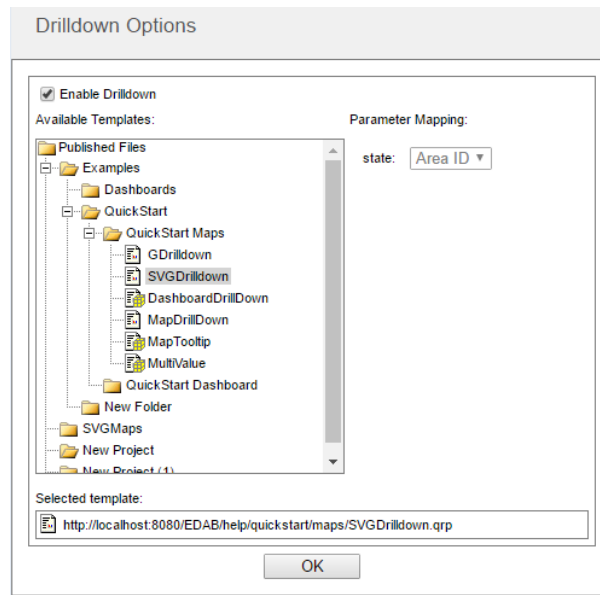


The 'Title Options' dialog box is shown. It has a title bar 'Title Options'. Inside, there is a checked checkbox 'Enable Map Title'. Below it is a text field 'Title text:' containing 'US Orders'. There is a sub-section titled 'Title Options' containing several settings: 'Font:' set to 'Arial', 'Text Alignment:' set to 'Center', 'Font Style:' set to 'Bold', 'Title Position:' set to 'Top', 'Font Size:' set to '25', and 'Text Color:' represented by a black color swatch. At the bottom right is an 'OK' button.

Map Title

Q.8.3. Set SVG Map Drilldowns

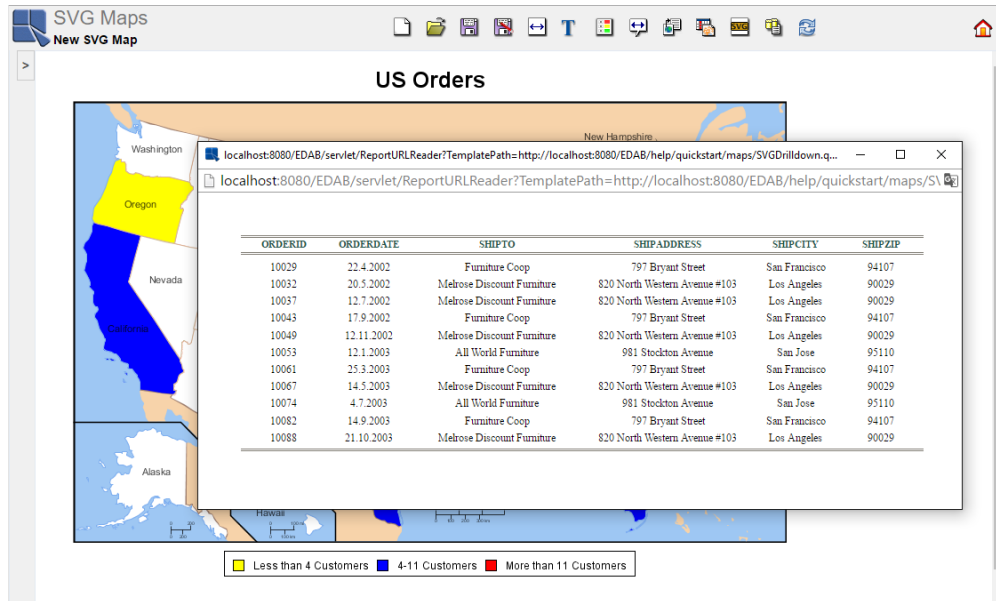
Click on the  *DrillDown Options* icon on the toolbar. Select the *Enable Drill-Down* option. Organizer structure will show up in *Available Templates* treeview. You will be able to see only folders and parameterized reports, charts and maps inserted into Organizer. Open the *Examples/QuickStart/QuickStart Maps* folder and select the *SVGDrilldown* report. Click *OK*.




The 'Drilldown Options' dialog box is shown. It has a title bar 'Drilldown Options'. Inside, there is a checked checkbox 'Enable Drilldown'. Below it is a treeview 'Available Templates:' showing a hierarchy: 'Published Files' > 'Examples' > 'Dashboards' > 'QuickStart' > 'QuickStart Maps' > 'SVGDrilldown'. To the right is a 'Parameter Mapping:' section with 'state:' set to 'Area ID'. At the bottom is a 'Selected template:' text field containing the URL 'http://localhost:8080/EDAB/help/quickstart/maps/SVGDrilldown.qrp'. An 'OK' button is at the bottom right.

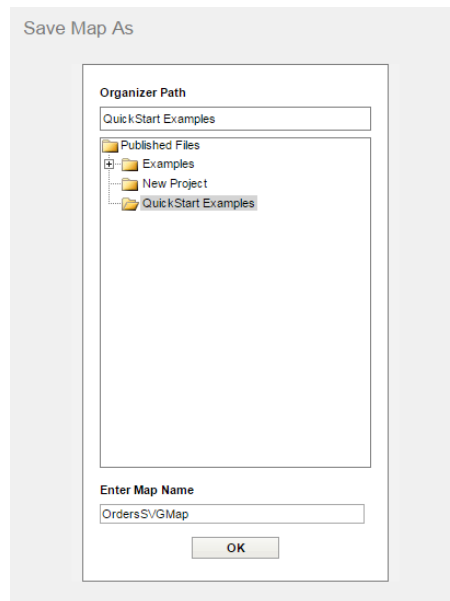
DrillDown Options

When you click on a colored state area, a drilldown report should open in a new window.



Map with Drilldown

To save the SVG Map, click on the  Save icon on the toolbar. Type **OrdersSVGMap** into the *Enter Map Name* text field and select the QuickStart Examples folder from the Organizer treeview.



Saving Map

Q.9. Publishing

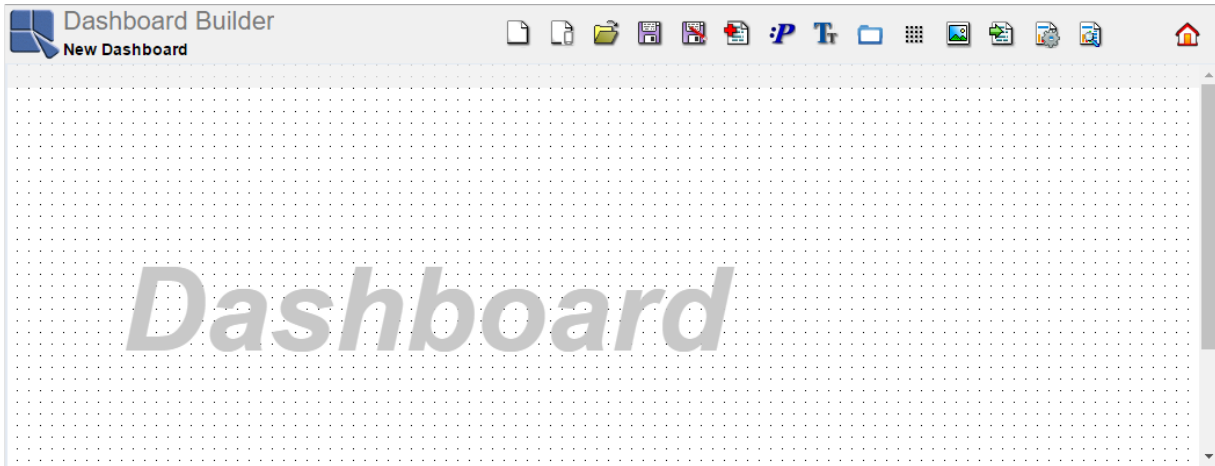
In the previous sections we looked at setting up EDAB and creating reports, charts, and maps. This chapter discusses the automated publishing features in EDAB.

Q.9.1. Dashboards

In addition to URLs and the menu page, EDAB also allows you to publish reports and charts using a dashboard interface. Dashboards can place multiple charts and report tables into a single presentation page. Users can define common filters for the dashboard items and set up drill-down for individual dashboard items.


Q.9.1.1. Create a Dashboard

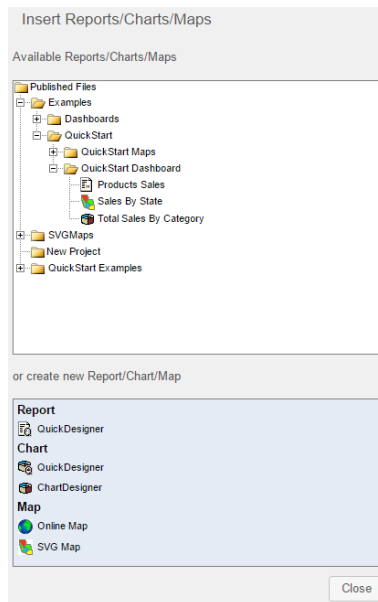
Dashboards are created in the thin-client Dashboard Builder interface. To launch the Dashboard Builder, click the *Dashboard Builder* link in the EDAB start page. The interface will allow you to build a new dashboard.



Dashboard Builder Interface

In this section we will guide you through creating a dashboard from report, chart, and map templates that come with EDAB installation. At first we need to add some templates to the dashboard. To add reports, charts, or maps to the

dashboard, click the *Add Report/Chart/Map* icon  on the Dashboard Builder toolbar. After you have clicked the button, the *Insert Reports/Charts/Maps* dialog will appear. The dialog contains a tree that mirrors the folder structure in the Organizer. Next, expand the *Examples* node and then the *QuickStart* and *QuickStart Dashboard* sub-nodes as shown on the image below.

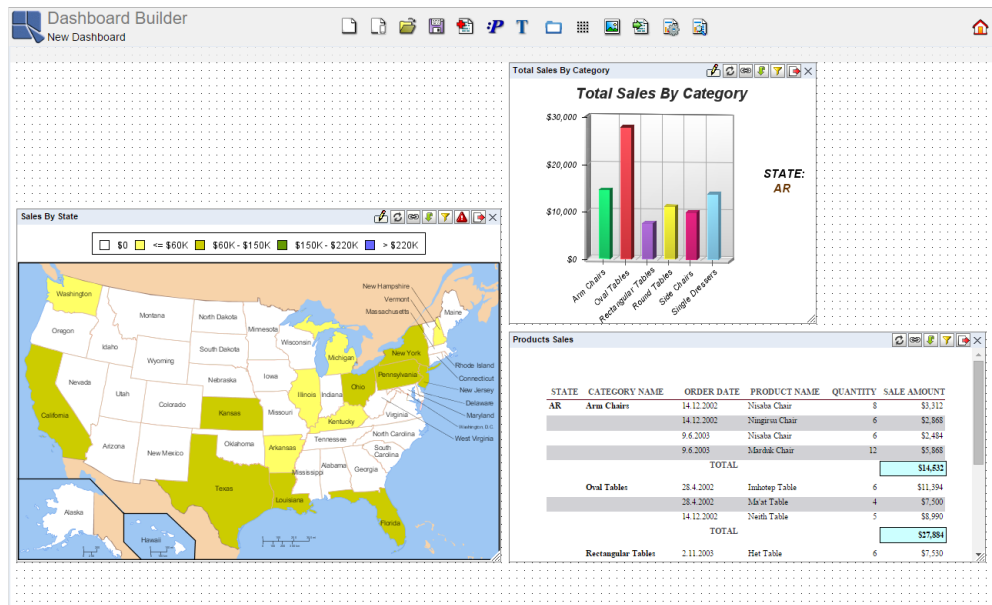


Insert Reports/Charts/Maps Dialog

From the dialog click on all the templates (*Sales By State* map, *Products Sales* report, and *Total Sales By Category* chart) in the *QuickStart Dashboard* node. This will insert the templates into the dashboard. Once you have inserted the templates, close the dialog by clicking *Close* button.


Now move the *Sales By State* map template by clicking to the map header and drag it to the lower left position in the dashboard. Then select the *Total Sales By Category* chart and drag it to the upper right position in the dashboard. Finally, move *Products Sales* report and drag it below the chart. You can also resize the chart/report/map by clicking

and dragging the sizing handle that appears in the lower right corner of the templates. After completing these steps, the dashboard should look like the following image:



Dashboard with Added Templates

Next, we will add dashboard shared parameters that allow us to group common parameters from the chart, report, and map into a single parameter. First open the Shared Parameters dialog by clicking on the *Shared parameters*

icon  on the toolbar. This will open the Share Parameters dialog that allows you to add shared parameters to the dashboard.

The screenshot shows the "Share Parameters" dialog box. It has a title bar with a close button (X). Inside, there is a section labeled "Shared Parameter" with a large empty text area. Below the text area are six buttons: "Add Share", "Edit Share", "List Values", "Add To Panel", "Set Range", and "Remove Share". At the bottom right is a "Close" button.

Share Parameters Dialog

From the dialog, click *Add Share* button. This will open another dialog that allows you to specify a shared parameter.

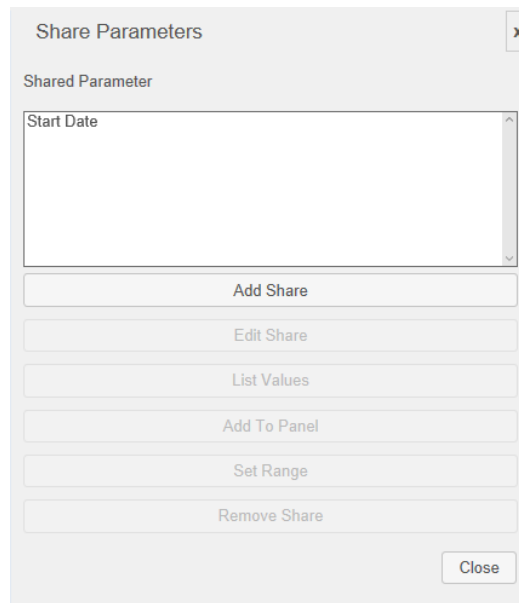
Shared Parameter Dialog

The left-hand side of the dialog shows all the available parameters from all the charts/reports/maps in the dashboard. You can select which parameters you would like to add to the selected parameters list by selecting them in the left-hand side and clicking the *Add* button. Note that all parameters in the selected parameters list must have the same data type.

In this example, we will create two shared parameters (*Start Date* and *End Date*) and put them into a date range panel. We will start with the first parameter (*Start Date*). First, enter the parameter name **Start Date** to the *Shared Parameter Name* text box at the dialog top. Enter the same text to the *Prompt Name* text field and then select all the *Start Date* parameters from the list on the left (you can select the parameters by **CTRL+Clicking** on the appropriate items in the list). After you have selected all the *Start Date* parameters from the list, click the *Add* button. This will add the selected parameters to the list on the right. At this moment the shared parameter dialog should look as follows:

Shared Parameter Dialog - Start Date Parameter

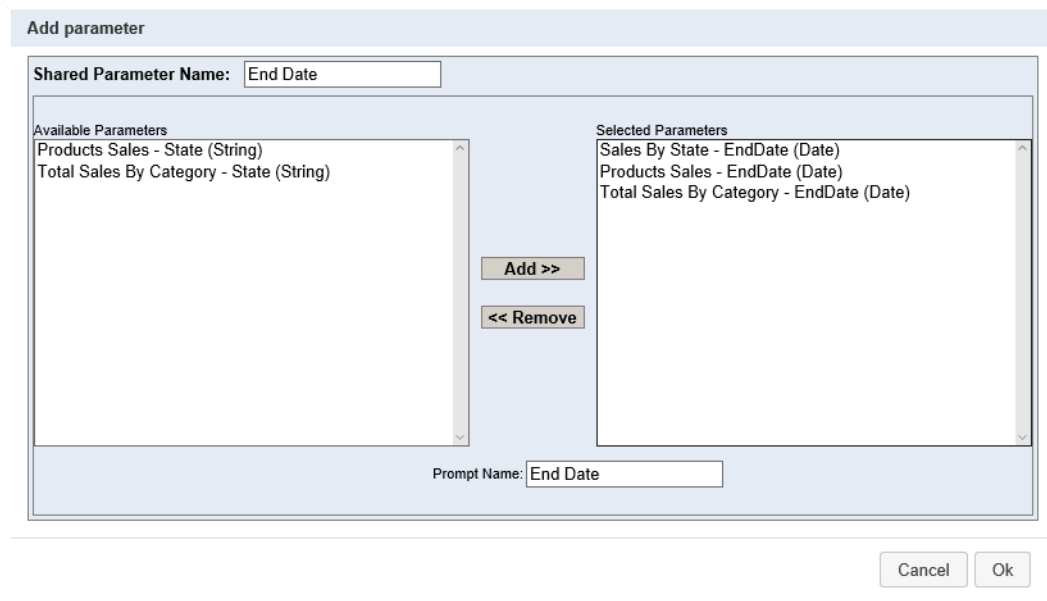
Now click the *OK* button. This will put you back to the shared parameters dialog. As you can see the new shared parameter *Start Date* has been added to the shared parameters list.



The 'Share Parameters' dialog box is shown. It has a title bar with a close button (X). Below the title bar is a section labeled 'Shared Parameter'. Inside this section is a text box containing 'Start Date'. Below the text box are several buttons: 'Add Share', 'Edit Share', 'List Values', 'Add To Panel', 'Set Range', and 'Remove Share'. At the bottom right of the dialog is a 'Close' button.

Share Parameters Dialog - Start Date parameter added

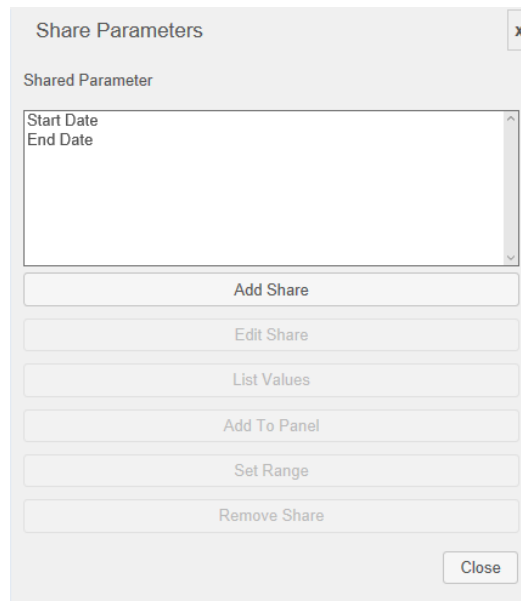
Next, we will add the second shared parameter **End Date** in the same way as mentioned above. From the dialog click the *Add Share* button again, which will open the shared parameter dialog. Enter the parameter name and the prompt name **End Date** and select all the **End Date** parameters from the list on the left. After you have selected all the parameters, click the *Add* button.



The 'Add parameter' dialog box is shown. It has a title bar. Below the title bar is a section labeled 'Shared Parameter Name:' with a text box containing 'End Date'. Below this is a section with two lists: 'Available Parameters' and 'Selected Parameters'. The 'Available Parameters' list contains 'Products Sales - State (String)' and 'Total Sales By Category - State (String)'. The 'Selected Parameters' list contains 'Sales By State - EndDate (Date)', 'Products Sales - EndDate (Date)', and 'Total Sales By Category - EndDate (Date)'. Between the lists are two buttons: 'Add >>' and '<< Remove'. Below the lists is a section labeled 'Prompt Name:' with a text box containing 'End Date'. At the bottom right of the dialog are 'Cancel' and 'Ok' buttons.

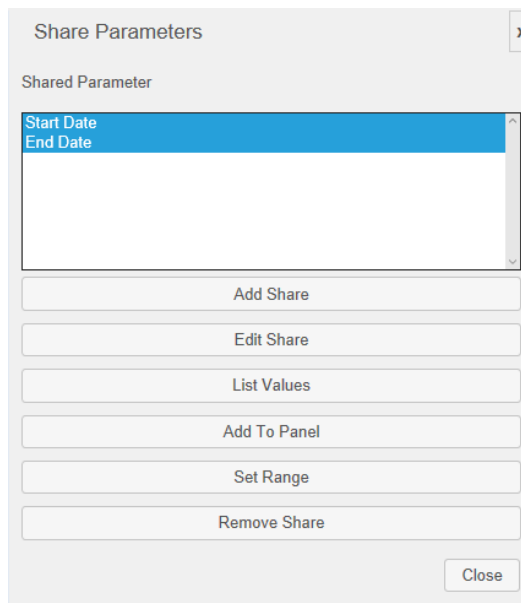
Shared Parameter Dialog - End Date Parameter

Click *OK* button. After adding both parameters you should see the same shared parameters dialog as follows:



Shared Parameters Dialog - Added Parameters

In the dialog, select both parameters by **CTRL+Clicking** on the parameters as shown in the image below.



Shared Parameters Dialog - Selected Parameters

After selecting the parameters, click *Set Range* button. This will open the *Range Param Attributes* dialog that allows you to specify ranges for the parameters.

Set Date Range Dialog

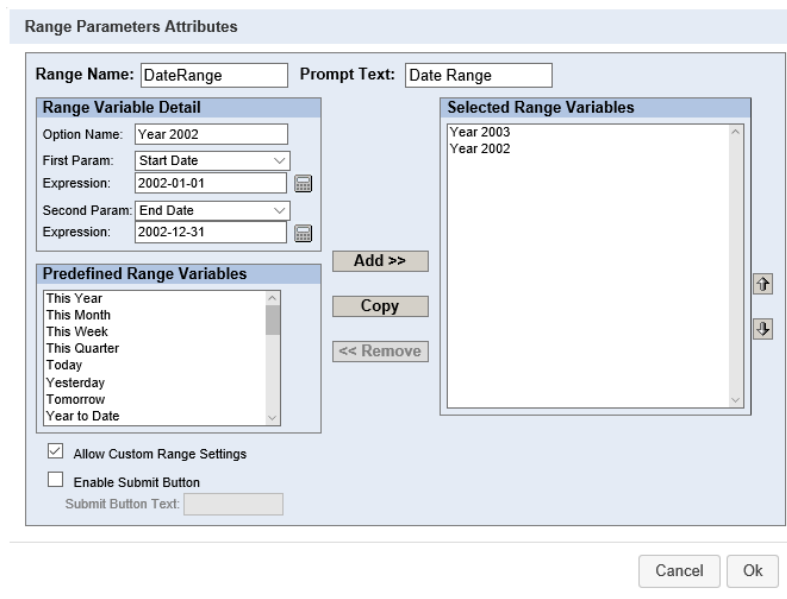
As you can see there are lot of options in the dialog. Please note that we will describe only the options necessary for the example. For more information about the rest of options, please see the Section 6.2.4.2 - Parameter Range.

First, specify the *Range Name* in the text box at the top of the dialog, e.g. **DateRange**. The name represents the title for the Date Range panel. Next, specify the *Prompt Text* to be **Date Range**. This text will be then displayed next to the selected date parameter drop-down menu in the Date Range panel. After entering the names, we will proceed with creating date ranges.

For simplicity we will add only two date ranges (Year 2003 and Year 2002) in this example. The range Year 2003 will include all the dates in year 2003 (dates from 2003-01-01 to 2003-12-31 in the Year-Month-Day format). Similarly, the range Year 2002 will include all the dates in year 2002 (dates from 2002-01-01 to 2002-12-31).

Now specify the name for the first range in the *Option Name* text box to be **Year 2003**. Next, enter the expressions for the *Start Date* and *End Date* parameters to be **2003-01-01** and **2003-12-31**. Once you have entered the *Range Name* as well as the expressions, click *Add >>* button. This will add the date range to the list of selected ranges on the right.

Similarly, add the second date range Year 2002. As mentioned above, first specify the range name to be **Year 2002** and enter the expressions for the *Start Date* and *End Date* parameters as **2002-01-01** and **2002-12-31**. After that, click the *Add >>* button to add the range to the list of selected ranges on the right.



Range Parameters Attributes

Range Name: Prompt Text:

Range Variable Detail

Option Name:

First Param:

Expression:

Second Param:

Expression:

Predefined Range Variables

- This Year
- This Month
- This Week
- This Quarter
- Today
- Yesterday
- Tomorrow
- Year to Date

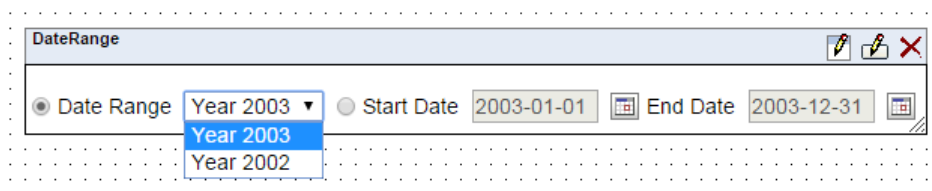
Selected Range Variables

- Year 2003
- Year 2002

☒ Allow Custom Range Settings
☐ Enable Submit Button
Submit Button Text:

Set Date Range Dialog - Specified Ranges

Once you have added both date ranges into the selected date ranges list on the right, click *OK* button. After that click somewhere to the Dashboard Builder, and the date range panel appears.

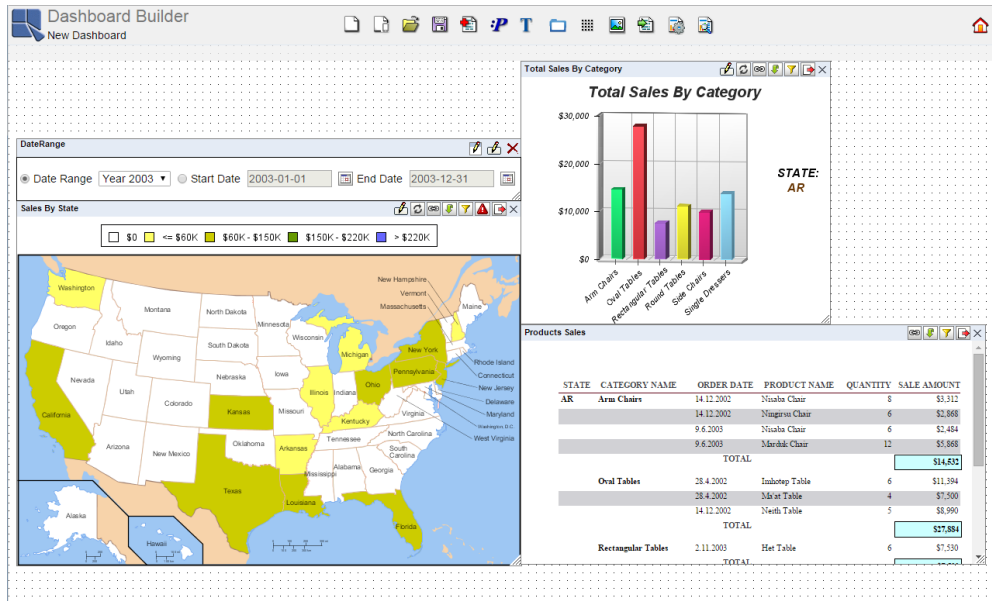


DateRange

☒ Date Range ☐ Start Date End Date


Date Range Panel

Now move the panel above the map as shown on the image below. To move the panel, just click on the panel header and drag it to the desired location. You can also resize the panel by clicking and dragging the lower right corner of the panel. The resizing rectangle will then appear, allowing you to adjust the size of the panel.




Dashboard With The Date Range Panel

The last thing we will do in this example will be adding links from the map to the chart and report in the dashboard. This will enable sending the `State` parameter from the map data points to the chart and report.

To set up links from the map, click the *Add/Modify Link* icon  from the map header. This will open the Set Up Link dialog below. There are three lists in the dialog. The first list contains all the available data columns and their data types in the source template (in our case, `Sales By State` map). The second list shows all the available destination templates in the dashboard (in our case, `Product Sales` report and `Total Sales By Category` chart). Finally, the third column contains all the available parameters in the selected destination template. Please note that the third column will show the parameters only after selecting the destination template in the second list.

Set up Links in Dashboard

 Sales By State

Column Name List:

- State (String)

Available Template(s):

- Products Sales
- Total Sales By Category

Available Parameter:

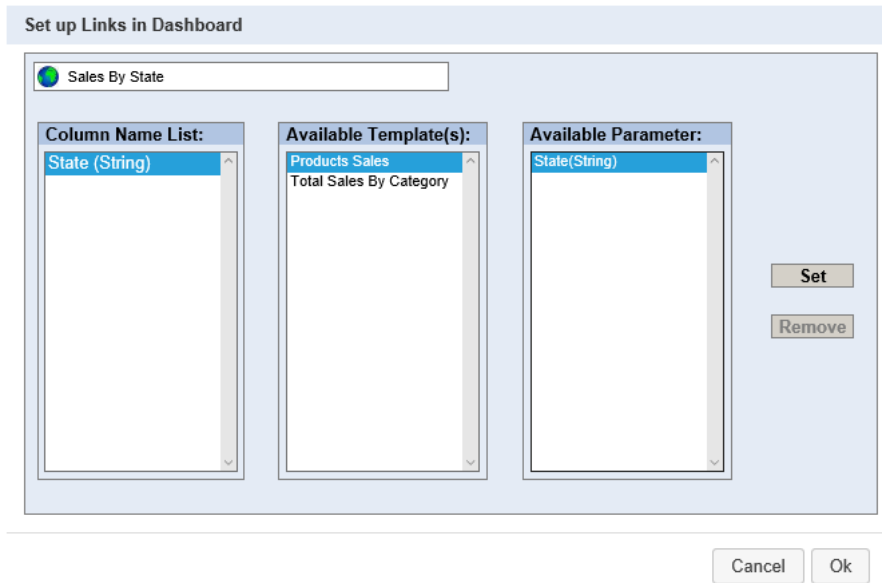
-

Set Remove

Cancel Ok

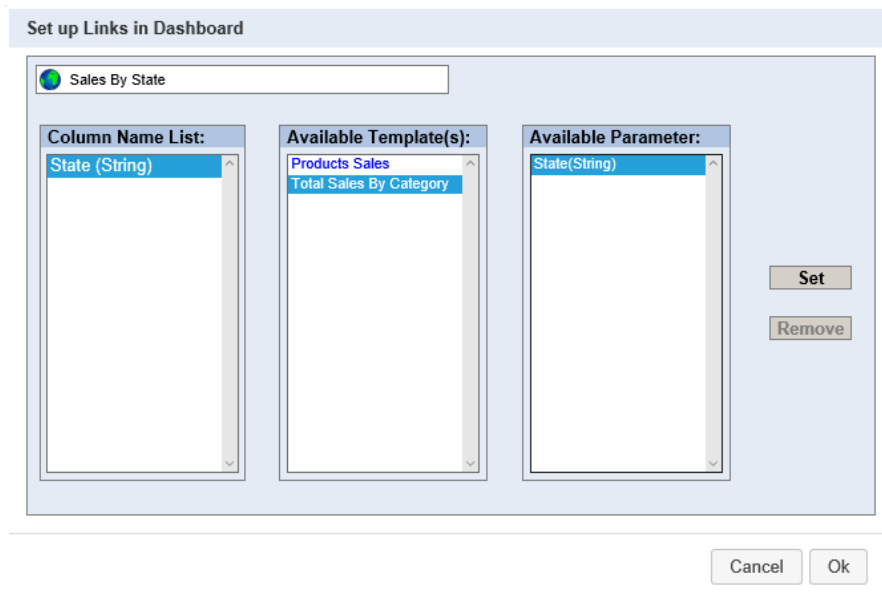
Set Up Links Dialog

To set up a link from the map to the `Product Sales` report, select `State (String)` item in the first list and the `Product Sales` item in the second list. This will cause showing the parameters from the report in the third list. After that select `State(String)` item from the third list. This should enable `Set` button as shown on the image below. Click the button to set up the first link.



Set Up Links Dialog - Link From The Map To The Report

Similarly, set up a link from the map to the Total Sales By Category chart. First, select *State (String)* item in the first list and the *Total Sales By Category* item in the second list. This will cause the parameters from the chart in the third list to be shown. From the third list select *State(String)* item. This should enable *Set* button as shown on the image below. Set up the second link by clicking the button.



Set Up Links Dialog - Link From The Map To The Chart

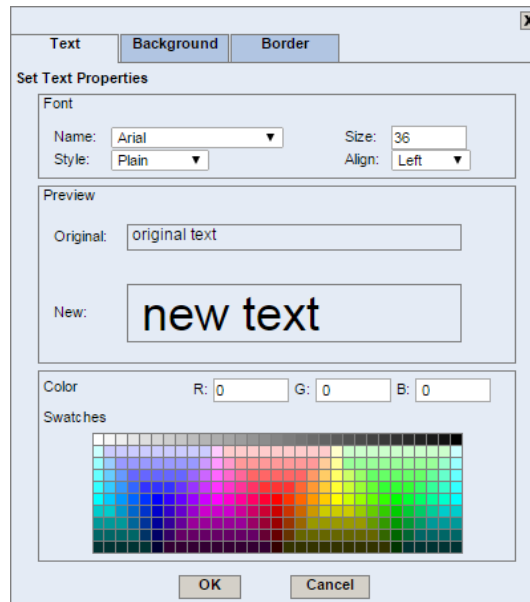
Once you have set up both links, click *OK* button that will close the dialog. At this moment the links from the map should be set up. So if you now click on a data point in the map, the chart and report should refresh according to the data point parameter value.

To finish our dashboard, we will just add a dashboard title and choose a dashboard background color. To add the

title, click the *Insert Label* icon **T** from the toolbar. A small rectangle will then follow your mouse cursor. Position the rectangle where you would like to insert the label and click. This should add the label panel into the dashboard. Next, double click to the label panel to edit the label text. Enter e.g. **SALES BY STATE DASHBOARD** text and click outside of the label panel. You may also want to resize the label because the label is now too small to be the



dashboard title. This can be done from the label properties dialog that opens after clicking on the *Edit* icon on the panel header.

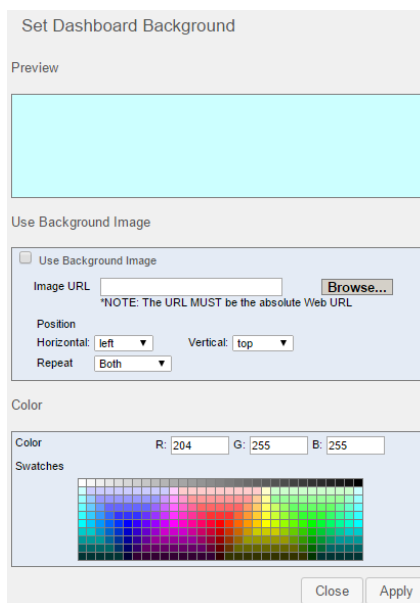


Set Label Properties Dialog

You can specify various properties for the label from this dialog. In the dashboard preview screen shot below, we just increased the label font size to **36px** in the *Text* tab, disabled the panel border in the *Border* tab (write **0** to the *Thickness* option), and set up the label background color to be transparent in the *Background* tab. Click *OK* to apply changes. Now, click and drag the bottom right corner of the label border to be the whole title in one line. For more information about the label properties, please see Section 6.2.5 - Insert Labels.




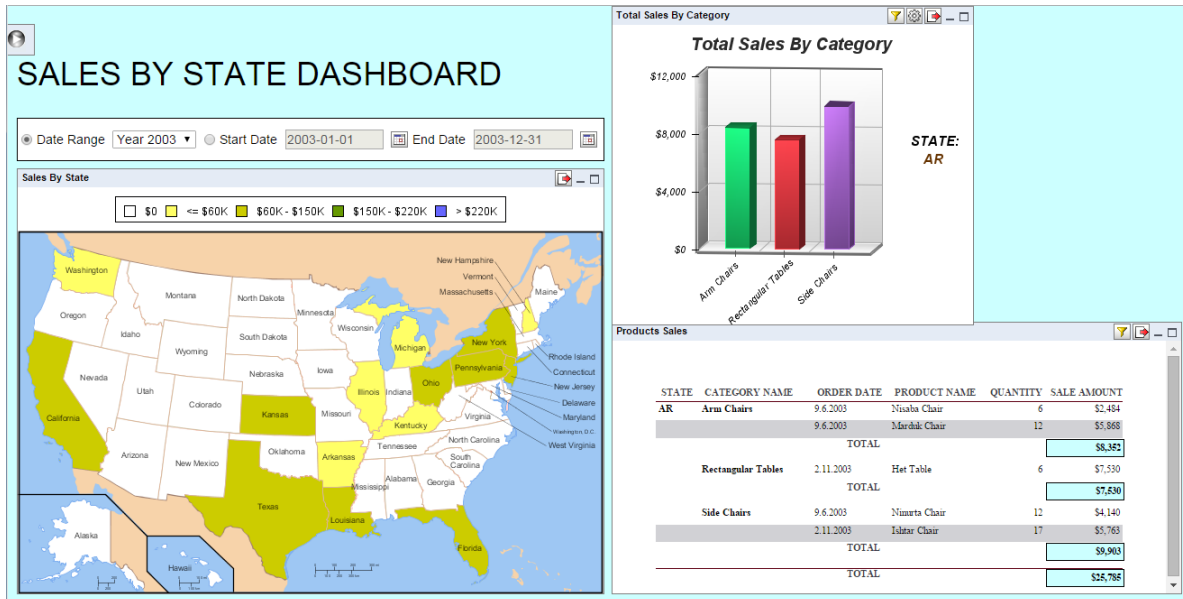
To change the dashboard background color, just click the *Add Dashboard Background* icon on the toolbar. This will open the *Set Dashboard Background* dialog from which you can choose the dashboard background color. Select the color and click *Apply*.



Set Dashboard Background Dialog


Once you have chosen the dashboard background color, you may preview the dashboard by clicking the *Preview*

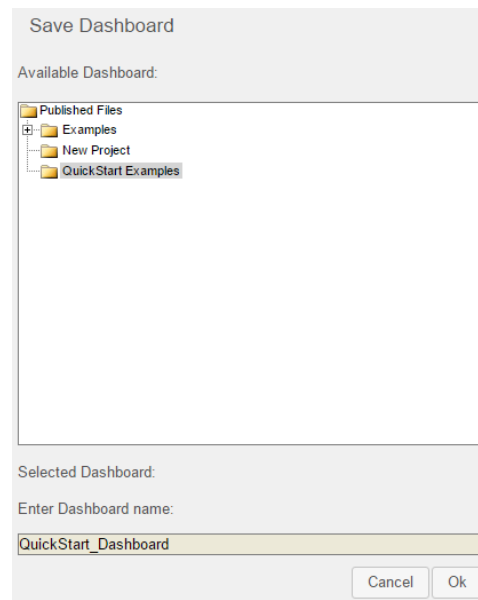
button  on the Dashboard Builder toolbar. The dashboard display will open in a new window. For more information about dashboard features, see Section 6.1 - Introduction to Dashboards.



Dashboard Preview

Q.9.1.2. Save the Dashboard





Close the dashboard preview window to return to the main Dashboard Builder interface. Click the *Save* button  on the toolbar. A dialog will open prompting you to specify a name for the dashboard.

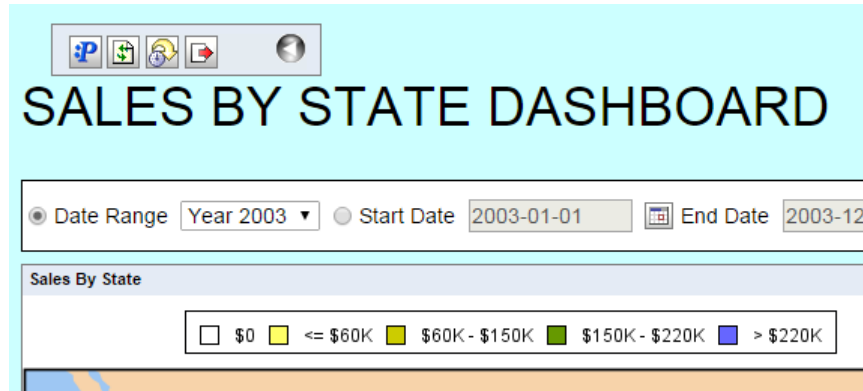


Save Dashboard Dialog

Enter a name for your dashboard and then from the treeview pane select the project that you created in Section Q.2.2.2 - Add a Project. Click *Ok* to save the dashboard. The window will then give you a message that the dashboard was saved successfully. Click *Ok* to close the dialog.

Q.9.1.3. Export Dashboard to PDF

Click the *Options* button  on the Dashboard Builder toolbar and check the option *Show toolbar in preview* (in the *Other* section). Click *Apply* to apply the setting. Open the dashboard preview window by clicking the *Preview* button . Then click the *Unpack* button  to open the preview toolbar. You can export dashboard to PDF file by clicking *Export Dashboard* button  from the preview toolbar.



Export Dashboard from Preview

All the reports, charts, SVG Map and other objects in the Dashboard will be exported to PDF file with the exception of Google Map due to licence restrictions.


Alternatively, you may open the same .dsb file in Published Files (for more information please see Section 7.1 - The Menu Page), open the Dashboard toolbar and Export PDF there.

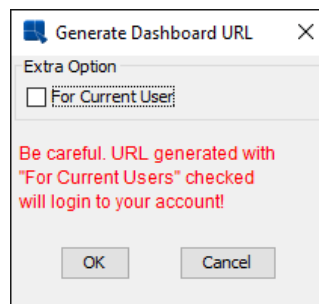
Note: There may be small differences in the appearance of elements in export (e.g. radio button shape can change from round to square).

Q.9.2. URLs

One automated deployment provided in EDAB is the ability to run dashboards via URL calls to the EDAB server.

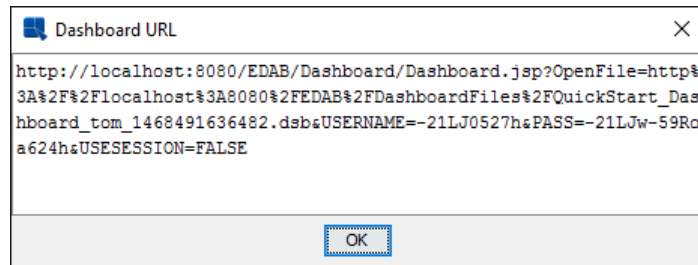
To generate a URL, first log into the Organizer, using the steps described in Section Q.2.2 - Start the Organizer. With the Organizer interface open, select the project you created in the left-hand side (Section Q.2.2.2 - Add a Project). You will see a list of files added from the previous exercises. Select the `QuickStart_Dashboard.dsb` file in

the Organizer (dashboard created above), and click the *Generate URL* button on the toolbar . A dialog will open, prompting you to specify option *For Current User*.



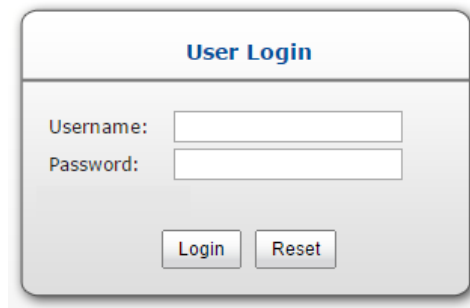
URL Options Dialog

In this dialog, disable the option marked *For Current User*. This will create the URL without encoding a username and password, prompting users to login when running the URL. Click *OK* and the URL will be generated in the *Dashboard URL* dialog.



Generated URL

Now, select the generated URL and copy the text to the browser's location dialog. Hit enter to run the URL. You will first be directed to a login dialog because you selected not to pass in the username and password in the URL.



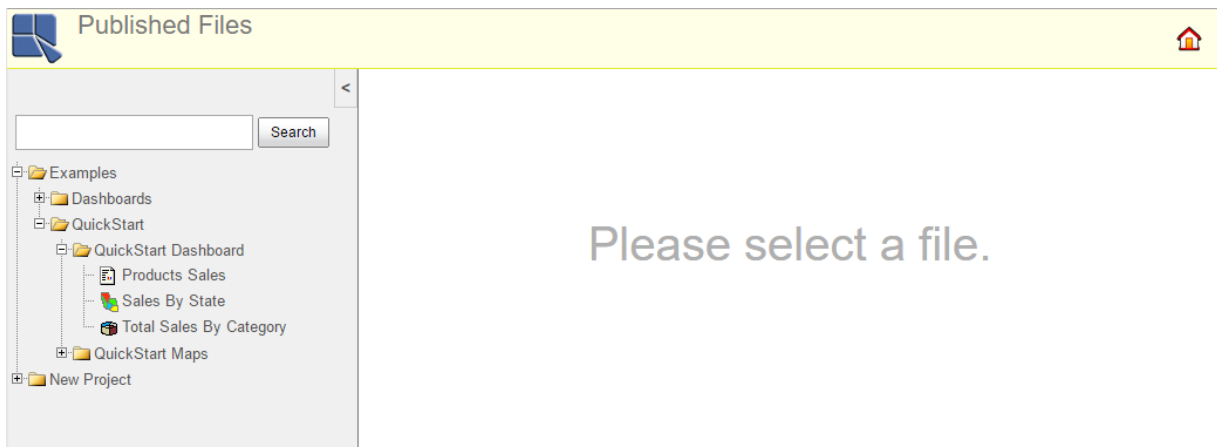
URL Login Dialog

Enter your *User Name* and *Password* in the dialog and click the *Login* button. The dashboard will then load in the browser.

Q.9.3. The Menu Page

Another way that EDAB automatically publishes reports, charts, maps, and dashboards is through the Menu Page. The Menu Page is a thin-client interface that allows users to run/view the reports, charts, maps, and dashboards to which they have access in the Organizer.

To launch the Menu Page, go to the EDAB Start page and click the link labeled *Published Files*. The Menu Page will load.



Menu Page

The page contains a list of all the reports, charts, maps, and dashboards in your project and some examples. The only option available is to run the file because you do not have any active schedule or archive jobs. To open a file, expand respective project/folder nodes (in the left-hand tree-list) to locate the file and click on the file name. The file will load in the right-hand DHTML Viewer panel.

For more information about the options and functions available in the Menu Page, see Section 7.1 - The Menu Page. After you have finished viewing your report/chart/map/dashboard, close the window that contains it, and click the *Home* button in the menu interface to return to the start page.

Q.10. Alerts

In previous sections, you learned how to create reports, charts, and maps (let's call them “objects”), and how to publish them in dashboards. This section describes how to add and manage alerts in objects, how to “watch” alerts in dashboards and how to monitor alerts automatically.

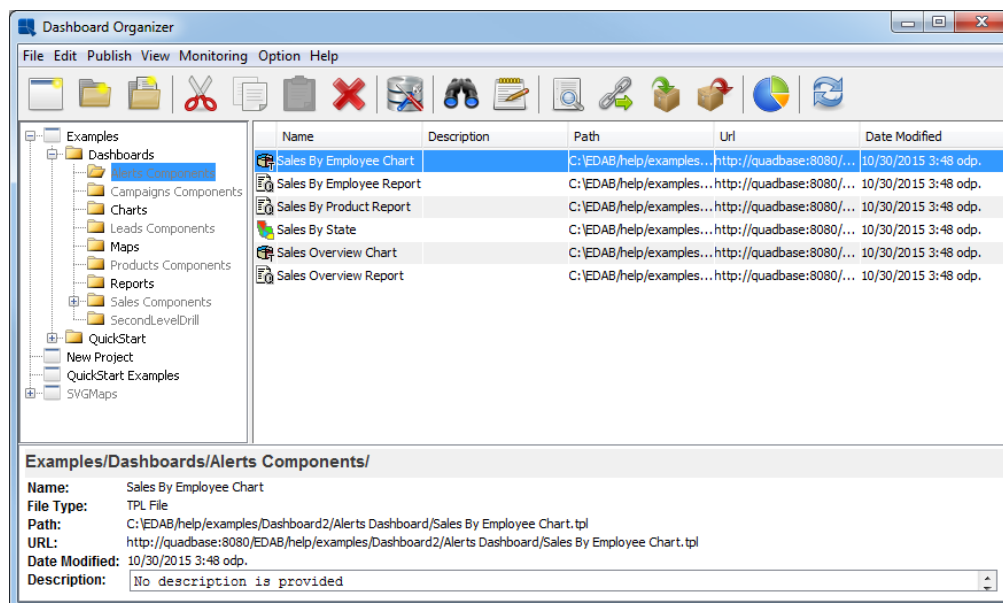
An alert is a range of some indicator (for example, profits/losses, sales, up-time, etc) you want or do not want the indicator to reach. If the indicator's value reaches into the alert range (in other words: if an alert was triggered), you will be instantly informed about that, so you can fix issues in a timely manner. More detailed information about alerts can be found in Section 11.1 - What is an Alert.

First of all, you have to create the alert. That means that you have to select which indicator you want to watch and what values are critical for you. Each object (report, chart, or map) can have its own alerts.

Q.10.1. Alerts in Charts

In Section Q.4.3.2 - Control Areas, you learned how to add a “control area” into a chart. That is all you have to know to add alerts into charts – just open a chart in Chart Designer and add one or more control areas into it. Basically, you can think of a control area as an alert in a chart.

Launch EDAB Organizer (as described in Section 2.1.1 - Starting the Organizer), expand *Examples/Dashboards/Alerts Components* and double-click on the *Sales By Employee Chart*.



Sales By Employee Chart Location

The *Enter Dialog* will show up. You do not have to change any parameter values, so just click *OK* to continue. You should be able to see a chart now. You may notice that at least one control area has been already added into the chart. The existing control area represents minimal sales limit – no employee should sell less than the limit. We will add another alert that will indicate that employee's personal sales are not optimal, although they are not critically low.

Open the *Insert* menu and choose the *Control Area* option. *Control Area Options* dialog has appeared.



Note

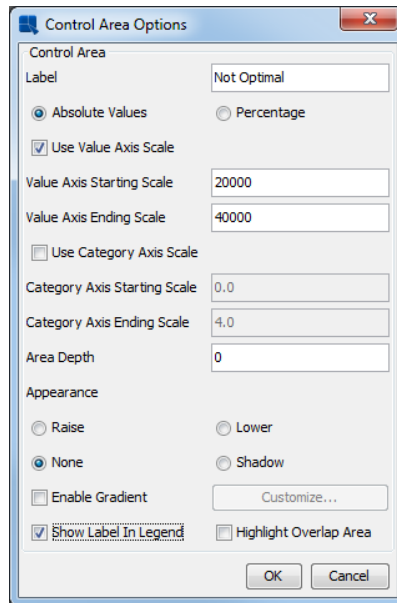
You may have noticed that the *Control Area Options* dialog contains two control areas, but you can see only one control area in the chart. That is because one control area (No sales, to be specific) is

so small that it cannot be seen on current axis scale. This does not affect the alert in any way, it will still work as if the control area was visible.

To add a new control area, click the *Insert* button. Now, it is time to set up our new alert. The first option is called *Label*. The *Label* is set to **New Range** by default. Delete the default label and set it to **Not Optimal** instead.

Now we will set up the alert range. Set the *Value Axis Starting Scale* to **20000** and the *Value Axis Ending Scale* to **40000**.

To include the control area in the legend, choose the *Show Label In Legend* option.



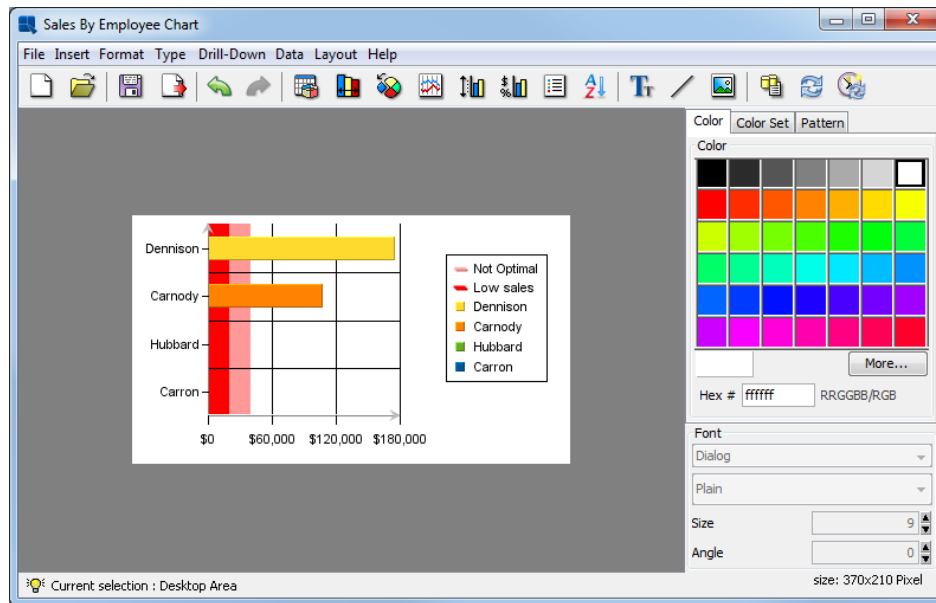
Control Area Options



Note

Complete description of this dialog can be found in Section 4.1.4.6.3 - Adding Control Areas



Click *OK* to close the dialog. As you can see, our new alert was created successfully. Now click the *OK* button again to go back to the Chart Designer. The new control area has appeared in the chart. It is highlighted by a random color. To change the color, click on the new area and then choose an indicative color (for example, light red) from the *Color* panel on the right side of the window. Save the chart and close the Chart Designer.




Sales By Employee Chart

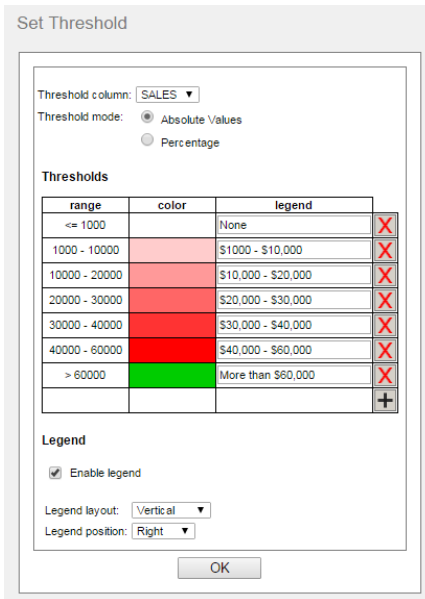
Q.10.2. Alerts in Maps

In the Section Q.8 - SVG Maps, you created a SVG map with thresholds. This means that you have also created a map with alerts because thresholds are equivalent to alerts in maps.

Let's add a threshold into an existing SVG map. Open EDAB main page and click on the *SVG Maps* link. Click on the  *Open Map* icon on the toolbar. Expand Examples/Dashboards/Alerts Components nodes in the *Open Map* tree-list, select the Sales By State map and click the *OK* button. Click *Submit* to use default parameter values. To manage thresholds, click on the .

To add a new threshold (alert), click on the  *Add Threshold* icon. In the *Enter threshold value* dialog, type **60000** and click *OK*. As new thresholds are added, some legend labels are not right. The legend labels are important for alerts because they also double as alert names. Set the *40000 – 60000* legend to **\$40,000 – \$60,000**. Now set the *> 60000* legend label to **More than \$60,000**. Click on the *> 60000* color field. *Select Color* dialog should show up. Choose an appropriate color (for example, green) and click *OK*.

Now, the Set Thresholds dialog should look like this:



Set Threshold

Threshold column: SALES ▼

Threshold mode: ☒ Absolute Values
☐ Percentage

Thresholds

range	color	legend	
<= 1000		None	X
1000 - 10000		\$1000 - \$10,000	X
10000 - 20000		\$10,000 - \$20,000	X
20000 - 30000		\$20,000 - \$30,000	X
30000 - 40000		\$30,000 - \$40,000	X
40000 - 60000		\$40,000 - \$60,000	X
> 60000		More than \$60,000	X
			+

Legend



☒ Enable legend

Legend layout: Vertical ▼

Legend position: Right ▼



OK

Set Thresholds Dialog

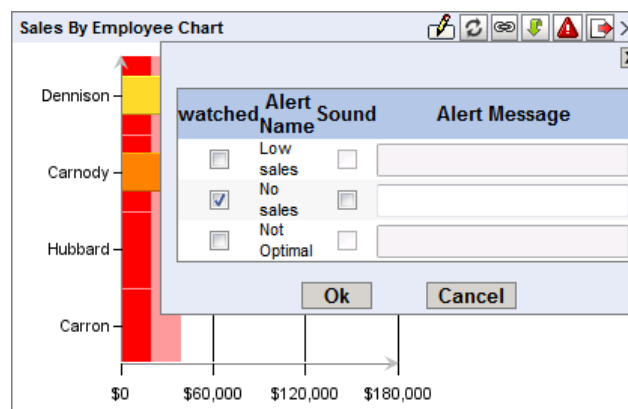
If it does, click *OK* to confirm the changes. The map has been changed since you added the threshold. Click on the  *Save* icon and click *OK* to save the changes. Click  icon in the top-right corner of the window to go back to the EDAB main page.

Q.10.3. Alerts in Dashboards

In previous paragraphs, you added alerts into few objects. Now we want to make use of those alerts. There are two ways to utilize alerts – alert watching and alert monitoring. This paragraph describes how to set up alert watching. Alert watching is set up in dashboards. The basic principle is quite easy: insert some objects (reports, charts or maps) into a dashboard and pick the alerts you want to watch.

Log on to the EDAB main page and launch *Dashboard Builder*. Click  *Open* icon. Expand *Examples / Dashboards* nodes, select *Alerts Dashboard* dashboard, and click *Ok*. All dashboard objects have  *Set Alert* icons in their header bars. That means that all dashboard objects have some alerts that can be watched.

Locate the *Sales By Employee Chart* chart and click on its  *Set Alert* icon. The following dialog should appear.



Sales By Employee Chart

watched	Alert Name	Sound	Alert Message
<input type="checkbox"/>	Low sales	<input type="checkbox"/>	
<input checked="" type="checkbox"/>	No sales	<input type="checkbox"/>	
<input type="checkbox"/>	Not Optimal	<input type="checkbox"/>	

Ok Cancel

Dennison

Carnody

Hubbard


Carron

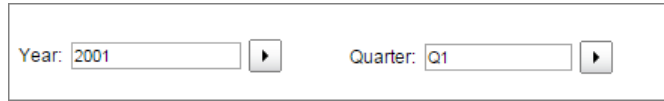
\$0 \$60,000 \$120,000 \$180,000

Sales By Employee - Watched Alerts

There is the list of all alerts that you saw in Section Q.10.1 - Alerts in Charts. One alert is set to be watched.

The list also includes the *Not optimal* alert you created previously. Check the *Not optimal* checkbox and uncheck the *No sales* check-box.

Click *Ok* to close the dialog and then click on the  *Preview* icon in the main menu. Notice that the chart's border is not flashing. That means that the *Not optimal* alert was not triggered. Now locate following panel:



The image shows a 'Parameter Panel' with two input fields. The first field is labeled 'Year:' and contains the value '2001'. The second field is labeled 'Quarter:' and contains the value 'Q1'. Both fields have a small arrow button to their right.

Parameter Panel


Change the *Quarter* parameter to **Q2**. The border started flashing, because the alert was triggered with current parameters. In other words: at least one *Sales By Employee Chart* chart bar ends in the *Not optimal* control area.


You may also notice that the *Sales Overview Report* header is flashing. That is because the report contains a watched alert which is triggered. You cannot see the report until you click on its tab header, but you can tell that an alert is triggered in the report.

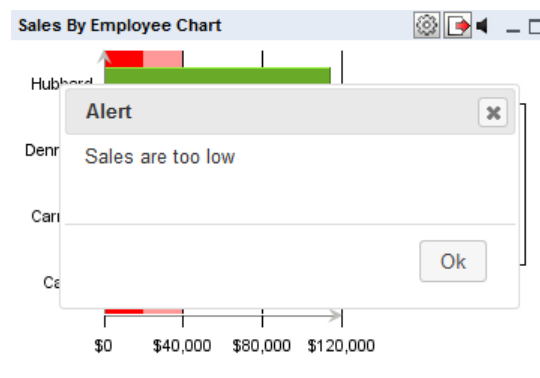
Please note that alert highlighting, blinking and alert messages are visible only in *Preview* of the Dashboard Builder.

Q.10.3.1. Sound alerts and alert messages

Border flashing is not the only way to alert you. You can also set alert messages and sound alerts. Close the *Preview*

window and click again on the  *Set Alert* icon for the *Sales By Employee Chart*. For our watched *Not Optimal* alert, check *Sound* and enter any message into *Alert Message* textbox. Click on *OK* and use *Preview* button to preview the dashboard.

Trigger the alert by changing the *Quarter* to *Q2* in the *Parameter Panel* and an alert message with your entered text will appear and you will also hear a beep sound. To mute the sound, click on the  *Mute* icon in upper right corner of the chart. Please note that each template has its own *Mute* button. To close the alert message, simply click on *OK* button.



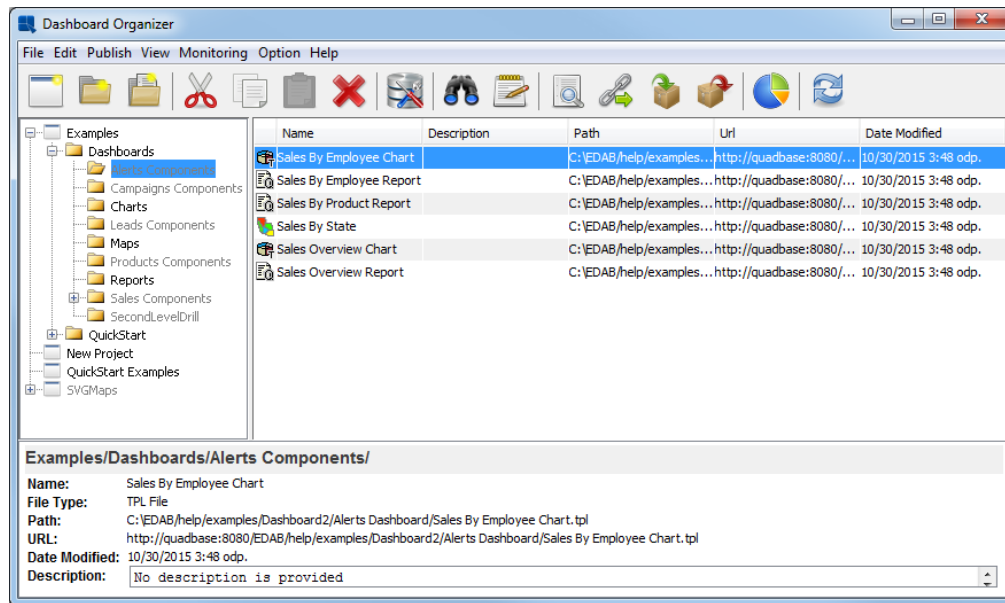
Triggered alert message

For more information about alerts, please refer to Section 11.3 - Alert types.

Q.10.4. Alert Monitoring

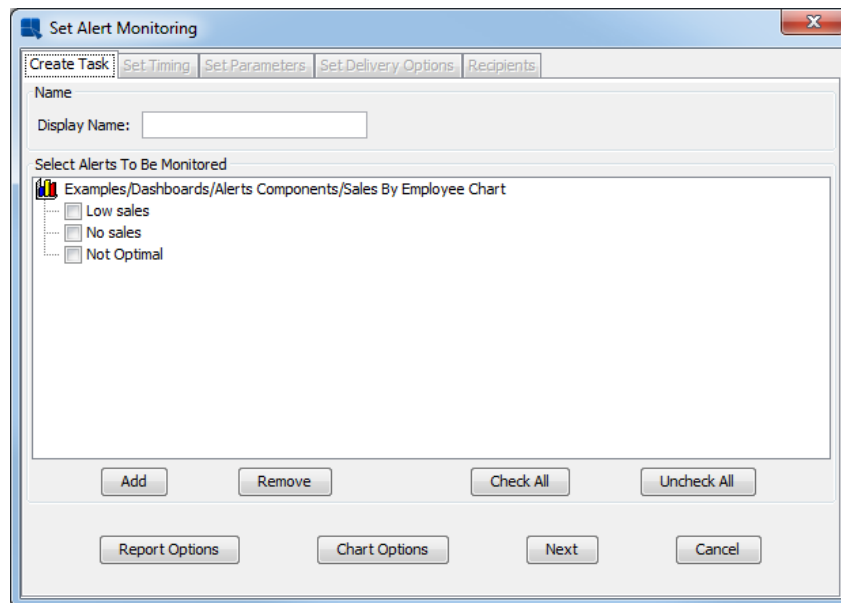
Another way of using alerts is alert monitoring. Unlike alert watching, alert monitoring checks alerts even when there is no dashboard running.

Lets set up a new alert monitoring job. Log on to the EDAB main page and click the *Organizer* button. Expand *Examples/Dashboards/Alerts Components* nodes and select the *Sales By Employee Chart* file.



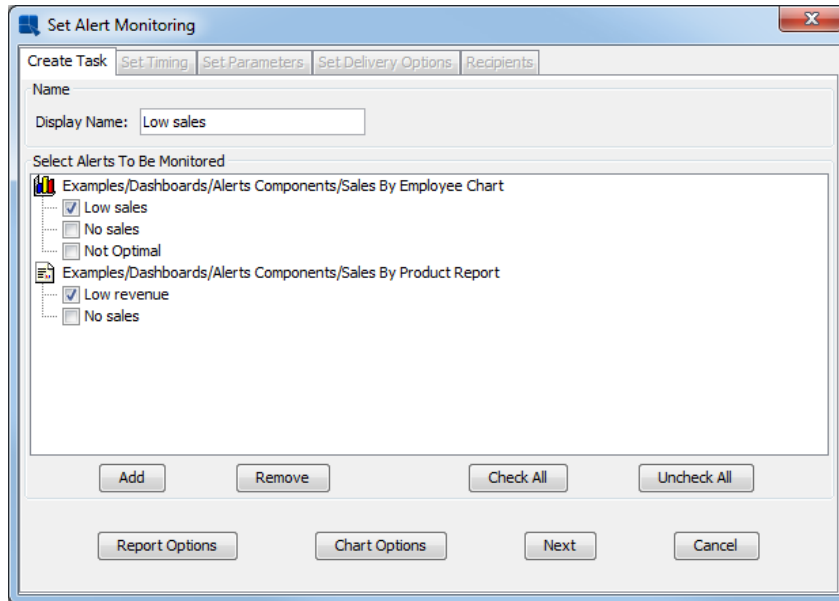
Sales By Employee Chart Location

Select *Set Alert Monitoring* from the *Schedule/Archive* menu. The following dialog should show up:



Setup Alert Monitoring Dialog

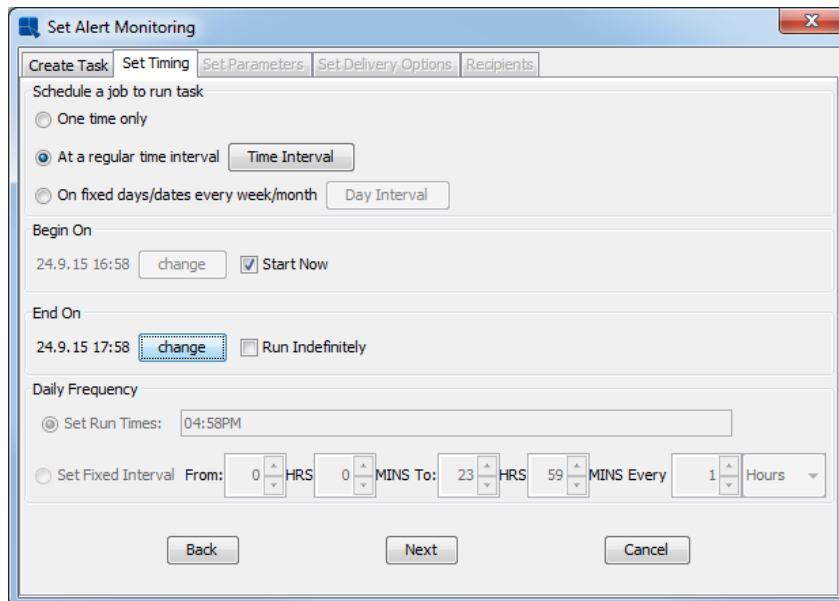
Click the *Add* button. Expand *Examples/Dashboards/Alerts Components* nodes, select the *Sales By Product Report* report and click *OK*. On the *Set Alert Monitoring* dialog select *Low sales* and *Low revenue* alerts. Type **Low sales** into the *Display Name* field. The dialog should look like this now:



Setup Alert Monitoring Dialog

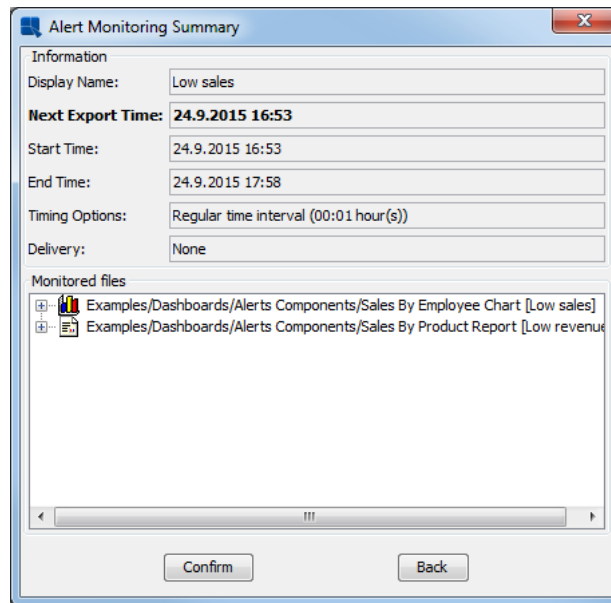
If it does, click *Next*.

You should be on the *Set Timing* tab now. By default, the alerts would be checked just once, few minutes after configuring it. Select the *At a regular time interval* option. Set Time Interval dialog should show up. Default time interval is **1** minute, which is fine for us, so just click *OK* to confirm it. Uncheck the *Run Indefinitely* checkbox and click the *change* button right next to it. Set the end time one hour to the future (so you do not have to delete the job manually after you are done with this QuickStart chapter) and click *Ok*. The dialog should look like this:



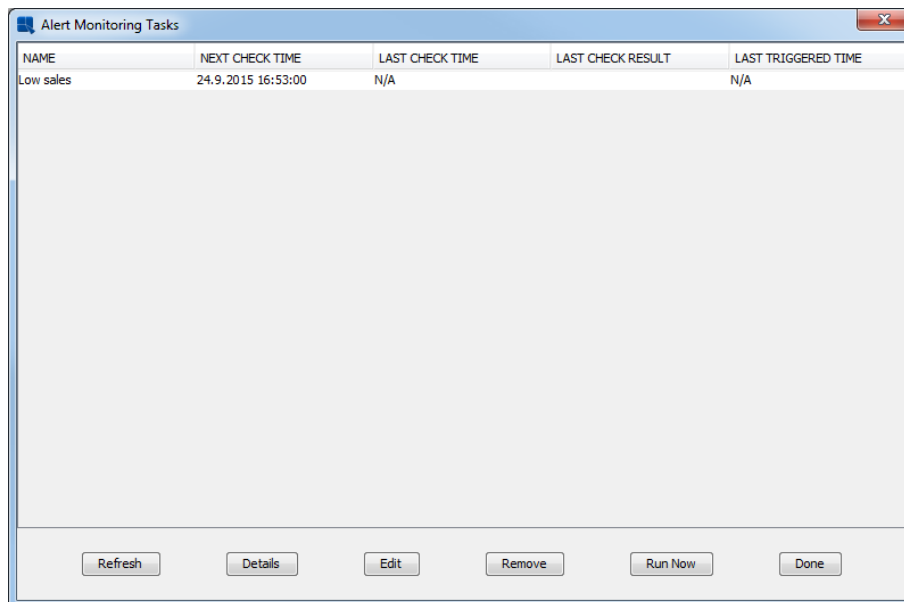
Set timing

Click *Next*. Click *Add* to add new parameter set to the *Sales By Employee Chart* chart. Click *OK* to use default parameters. Click on the *Sales By Product Report* tab header, click *Add* and *OK* again and then click *Next*. The next step is configuring delivery options. This requires further knowledge and we will skip this step right now. If you are interested in setting delivery options, see Section 11.4.3.1 - Create/Edit monitoring dialog. Click *Next*. The following dialog should appear allowing you to check your settings before saving the alert monitoring job. Click *Confirm* to save the alert monitoring job.



Alert Monitoring Summary

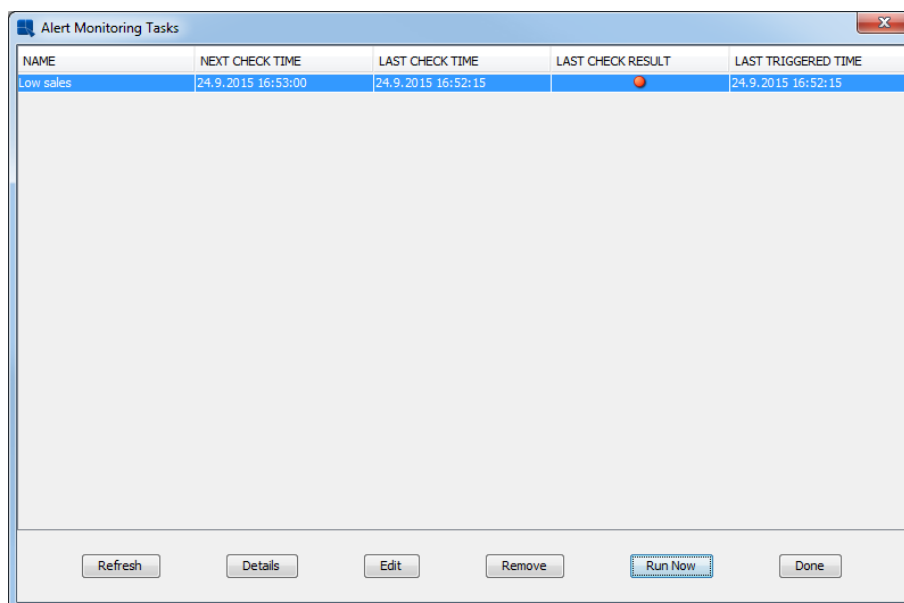
You should be back in Organizer now. Click on the *Schedule/Archive* pull-down option in the main Organizer menu and select *View Alert Monitoring Tasks* option. The following dialog should open:



Alert Monitoring Tasks Dialog

The *Low sales* task should be on the list. Check the *Last check time* value. If no check was performed until now, it will show *N/A*. You can either wait for next automatic check (look at the *Next check time* if you want to know when that's going to happen) and then click *Refresh*. You can also select the *Low sales* task and click *Run Now* and then click *Yes*.

After the task was checked, click *Refresh*. You should be able to see the results.

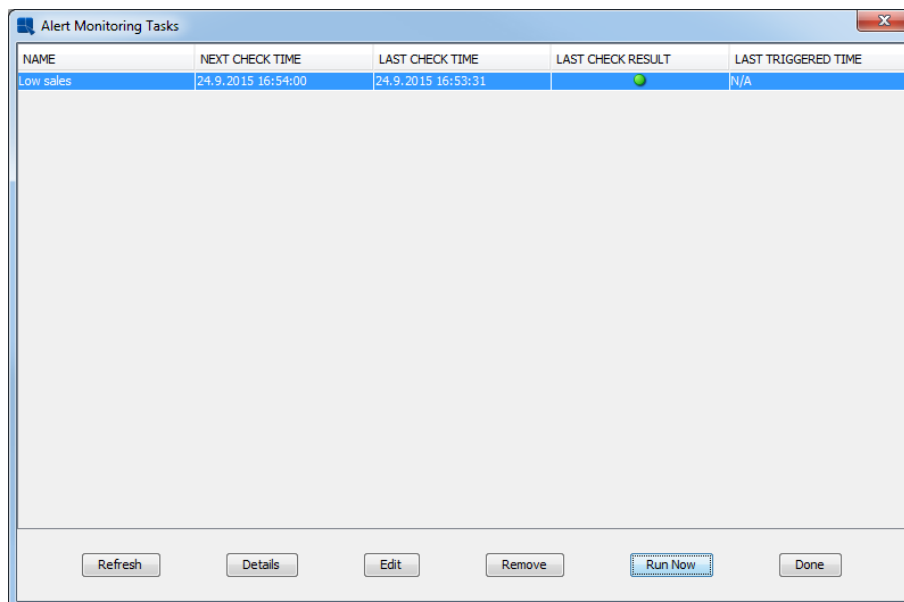


Alert Monitoring Tasks Dialog

The *Last check result* indicator is red. That means that at least one alert was triggered during last check. Also notice that the *Last triggered time* is now the same as *Last check time*.

Select the *Low sales* task and click *Edit*. Select the *Sales By Product Report* and then click the *Remove* button. Click *Done* and then click *Confirm* to save the changes.

Wait for the automatic check or click *Run Now* again. Click *Refresh*.



Alert Monitoring Tasks Dialog

Now, the *Last check result* indicator is green. That means that no alert was triggered during last check.



Note

Please, keep in mind that this is just a quick guide which should guide you through basic steps of configuring alert monitoring. In reality, data in the database would be changing constantly; therefore, you would not have to change alert monitoring job properties in order to trigger/not trigger an alert.

Chapter 1. Administration

1.1. Overview & Architecture

Welcome to EspressoDashboard (hereafter referred to as EDAB). EDAB is a powerful, server-based information presentation and delivery system. EDAB provides a fully secure environment where users can create and deploy feature rich dashboards.

With EDAB, users can easily draw data from a variety of sources and create a wide range of charts, reports, and maps using the integrated Chart Designer, QuickDesigner, and Map Builder. The Organizer interface provides a virtual file management and permission system for all reports, charts, maps, and dashboards.

There are two visual design interfaces for creating charts, namely the applet based Chart Designer and the zero-client QuickDesigner. Reports can be created and edited with the QuickDesigner only. Chart Designer is a full-featured chart creation and editing tool that empowers the user with complete control of the chart type and features in editing the chart, while the QuickDesigner provides a subset of the chart features and aims at zero learning curve and quick turn around time. Since reports in dashboards are typically quite simple in nature, QuickDesigner is the only interface for designing reports.

Deployment features allow users to automatically publish dashboards through the integrated menu features, and by generating URL requests.

1.1.1. EDAB Components

EDAB is composed of 8 main components:

EDAB Server:	The server is the back-end to EDAB. It deploys as a servlet collection within an application server or servlet container, and manages design and deployment for dashboards. The server handles user authentication and provides the data access and file I/O for clients running the Chart Designer and QuickDesigner components.
Organizer:	The Organizer is a powerful graphical user interface that acts as a virtual file management system. It allows users to organize charts, reports, maps, and dashboards in virtual folders, generate URLs deploying dashboards, and set file-level privileges. It also provides users with an interface for managing data sources, and designing database queries.
Chart Designer:	The Chart Designer is a graphical user interface, launched within the Organizer that allows users to design and customize full featured charts. Its point and click interface enables the swift and easy creation of charts with an almost infinite degree of flexibility.
QuickDesigner Reports:	The QuickDesigner Reports is a thin-client ad-hoc reporting tool. It allows users to quickly design data view queries, to use existing data sources, and build reports. Limited formatting options allow users to customize reports and save them back to the Organizer.
QuickDesigner Charts:	The QuickDesigner Charts is a thin-client ad-hoc reporting tool. It allows users to quickly design data view queries, to use existing data sources, and build charts. Limited formatting options allow users to customize charts and save them back to the Organizer.
Online Maps:	Online Maps is a thin-client tool that allows users to create online maps using OpenStreetMap or Google© maps. The easy to use interface allows users to create elegant maps with parameters, titles, legends, and drilldowns. For Online Maps maps, reports and charts can be directly embedded into the tooltips.
Dashboard Builder:	The Dashboard Builder is a thin-client tool that allows users to create custom dashboard presentations. The simple interface allows users to select charts, define parameters and drill-down using elements deployed in the Organizer.

Chart Viewer:

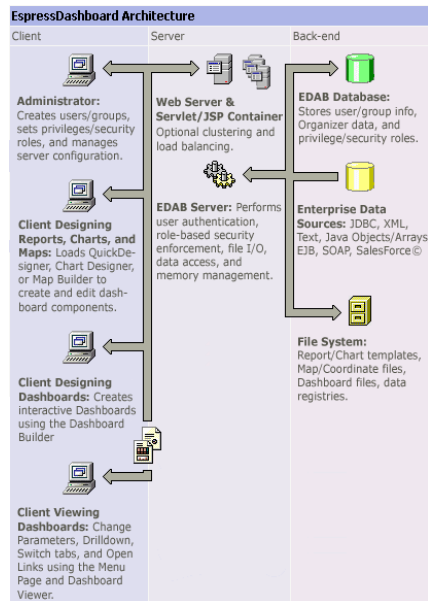
The Chart Viewer is a Java Web Start Application allows charts to be viewed remotely. It enables viewers to rotate, resize, zoom and pan a chart. Viewers can click on individual data points to either recover the data associated with the data point, or link to another chart.

EDAB API:

The EDAB API is a Java application programming interface that allows users to interface programmatically with the EDAB Server and Organizer. Users can get a handle to the Organizer to retrieve information about files and permissions.

1.1.2. EDAB Architecture

The following diagram illustrates the basic architecture of an EDAB implementation/installation:



EDAB Architecture Diagram

The EDAB server deploys as a servlet collection within an application server/servlet & JSP container. Other servlets/JSPs can be deployed in the server to connect to, and retrieve information from the EDAB server (like file names, schedule information, and schedule reports/charts).

On the back end, information for EDAB including user/group information, the Organizer structure and files, and security and privilege information are saved in a database. The server also performs data access and buffering for reports and charts, and file I/O on the server-side.

Clients can be administrators modifying users, privileges, and server settings from the remote Admin Console, or the Organizer. Clients can load the Chart Designer through the Organizer to create full featured charts, or load the thin-client QuickDesigners for creating basic reports and charts. Clients viewing dashboards can access the menu page and make dashboard URL calls to the EDAB server.

1.2. New Features List V7.0

1.2.1. New EspressDashboard Features

1. Unified data registry dialog in Quick Designers and Map Designers allows users to view query, query result, and test with different parameter values, so users can locate the desired query easily (Section 3.2 - Data in QuickDesigners and Maps).
2. Diagrams on start page of Quick Designers and Map Designers helps end-users to understand how to use the tools quickly (Section 5.2.3 - Start Online Maps).

1.2.1.1. Dashboard Builder (Chapter 6 - Designing Dashboards)

1. Being able to change chart type in dashboard preview gives end-users more control on dashboard contents (Section 7.4.2 - Preview Options).
2. Directly opening report/chart/map designer from dashboard builder makes it much more convenient to edit existing components or add new components (Section 6.2.3 - Add Charts, Reports and Maps).
3. Besides original static dashboard, responsive dashboard is added as a new dashboard type. Responsive Dashboard optimizes utilization of the display without exceeding the display width preset by user. Therefore, it's useful for mobiles (Section 6.2.2 - Responsive Dashboard).
4. Refresh Template icon on each report/chart/map makes it easier to test dashboard with real data (Section 6.2.3.3 - Chart/Report/Map Toolbar).

1.2.1.2. QuickDesigner Reports (Section 4.2 - QuickDesigner Reports)

1. Simplified UI includes only two steps: Select Datasource and Construct Report.
2. Constructing and formatting report in only one interface makes it intuitive and easy to see what you get and flexible to make changes (Section 4.2.4 - Format Report Elements).
3. Drag and drop to add report columns and order report column ease the work of report design (Section 4.2.4.1 - Add a Column, Section 4.2.4.5 - Column Order).
4. Right click on report column to do Aggregation/Format/Remove is straightforward (Section 4.2.6 - Aggregation/Group, Section 4.2.7 - Data Formating, Section 4.2.4.2 - Remove a Column).
5. 'Excel-like' search icon on each column header allows data filtering by column (Section 4.2.9 - Data Filtering).
6. Parameters pane on the right makes report testing easy (Section 4.2.5 - Parameter Setting).
7. Less items on Toolbar reduces confusion and saves time (Section 4.2.3 - Toolbar).
8. Collapse Sidebars allow more space for report design (Section 4.2.4.3 - Collapse Sidebars).

1.2.1.3. QuickDesigner Charts (Section 4.3 - QuickDesigner Charts)

1. Simplified UI includes only two steps: Select Datasource and Construct Chart.
2. Setting pane and chart preview pane are on the same window, so all design steps can be done on one window (Section 4.3.3 - Data Mapping and Ordering).
3. Improved toolbar shows Chart-Specific Options based on current chart type and combines settings to reduce icons (Section 4.3.5.5 - Chart-Specific Options, Section 4.3.4 - Toolbar).
4. Parameters setting on Chart Preview pane makes chart testing easy.
5. Collapse Sidebar allows more space for chart design.

1.2.1.4. Online Maps (Section 5.2 - Online Maps)

1. Online Map is a newly designed Google Map. It is more straightforward to create a new map (Section 5.2.3 - Start Online Maps).
2. Displaying setting on left pane, instead of popup dialog, makes UI more user-friendly.

3. Save dialog shows Organizer node tree. It makes it easier to locate the desired organizer folder to save map file (Section 5.2.7 - Save Map).
4. Heatmap feature provides visualization of data intensity (Section 5.2.6.6 - Heatmap).
5. Being able to set parameter values in Map Designer makes testing easier.
6. More map types satisfies different needs (Section 5.2.6.5 - Map Options).
7. Users can draw various shapes instead of simple map points. The shapes are drawn in Coordinates Editor (Section 5.2.4.2 - Create Coordinates, Section 5.2.4.2.3 - Manually).

1.2.1.5. SVG Maps (Section 5.3 - SVG Maps)

1. Redesigned UI is more straightforward to create a new map.
2. Allow users to change SVG image after a map is created (Section 5.3.4.3.1 - Change SVG Image).

1.3. Installation and Configuration

There are many important installation and configuration options available in EDAB, so it is important to refer to this section of documentation as you're starting out.

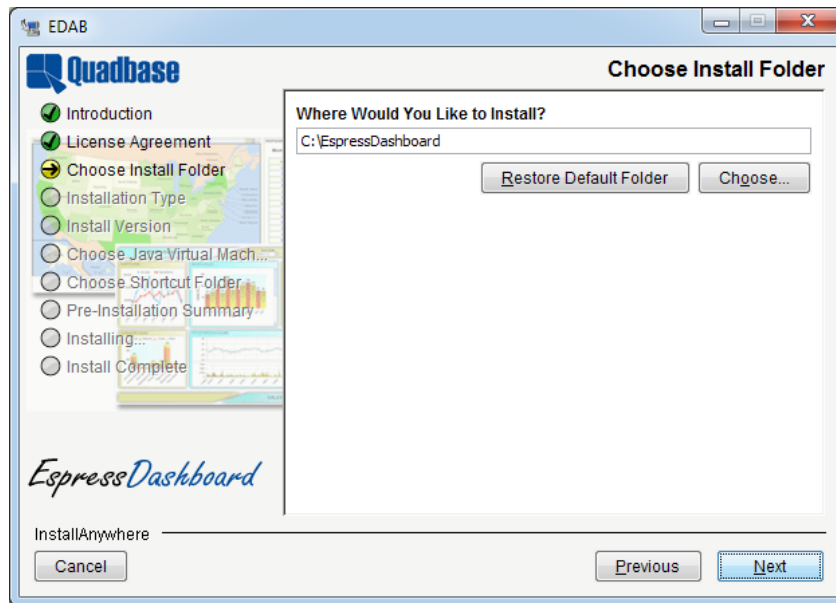
1.3.1. Installing EDAB

There are four versions of the installation program, one for Windows 10/8/7/Vista/XP/2003/NT/2000, one for Solaris/Unix, one for Mac OS X, and a pure Java version.

Windows 10/8/7/Vista/XP/2003/NT/2000:	To start the Windows installer, run the <code>installedEDAB.exe</code> file and the installer will launch.
Unix/Linux:	To start the Solaris/Unix installer, execute the <code>installedEDAB.bin</code> file, and the installer will launch.
Mac OS X:	To start the Mac installer, double click the <code>InstalledEDAB.zip</code> file to extract the <code>InstalledEDAB.app</code> file. Double-click on <code>InstalledEDAB.app</code> , and the installer will launch.
Pure Java:	To start the pure Java installer, a Java Virtual Machine, the equivalent of Java 1.8 or higher, must already be installed on the machine where EDAB is to be installed. Make sure that the JVM is included in your path (or move the <code>installedEDAB.jar</code> file to the same directory as the JVM). From a command prompt navigate to the directory where you have placed the <code>installedEDAB.jar</code> file, and type the following command: <code>java -jar installedEDAB.jar</code> . The installer will then launch.

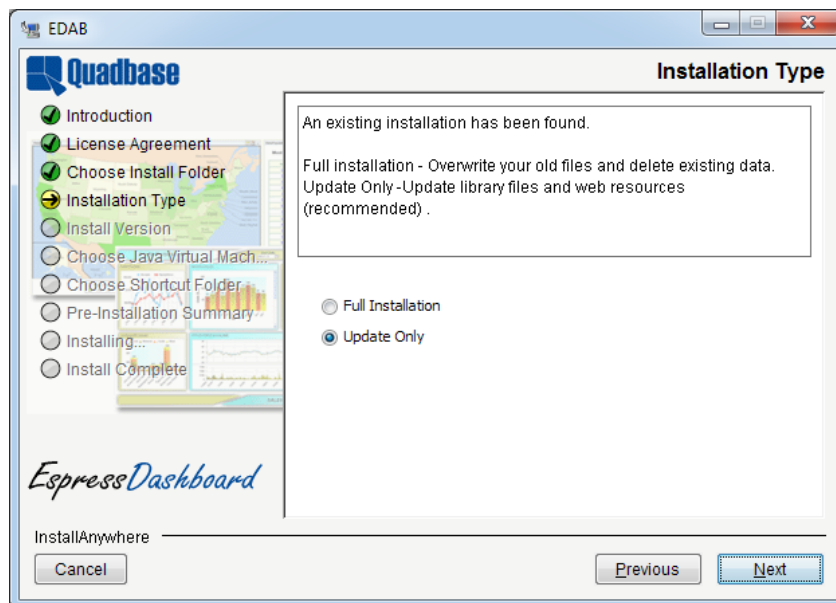
Each installer also comes with a console install mode where there is no graphical interface and the steps are performed through a terminal/console window. To launch any installer in console mode, add **`-i console`** to the end of the command. For example, for the Unix installer, you can execute the following command to run in silent mode: **`./installedEDAB.bin -i console`**

Once the installation program has launched and you agree to the license agreement, the first option that is presented asks you to specify the directory into which you would like to install EDAB.



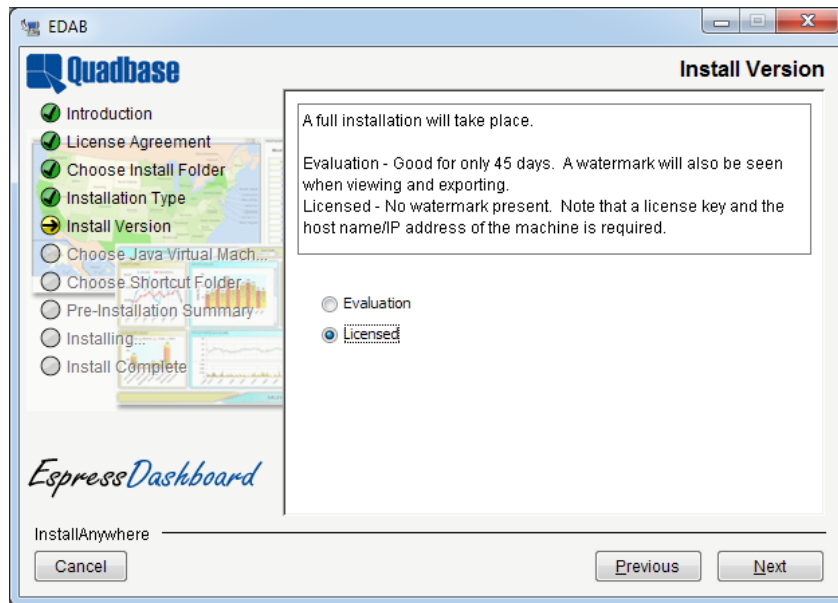
Choose EDAB Location Dialog

If the directory already exists and contains an installation of EDAB, the installer will give you the option to perform a full or update install. If the directory doesn't contain EDAB, the installation will always be a full install. For more information about update installs please see Section 1.3.1.1 - Re-Install/Update.



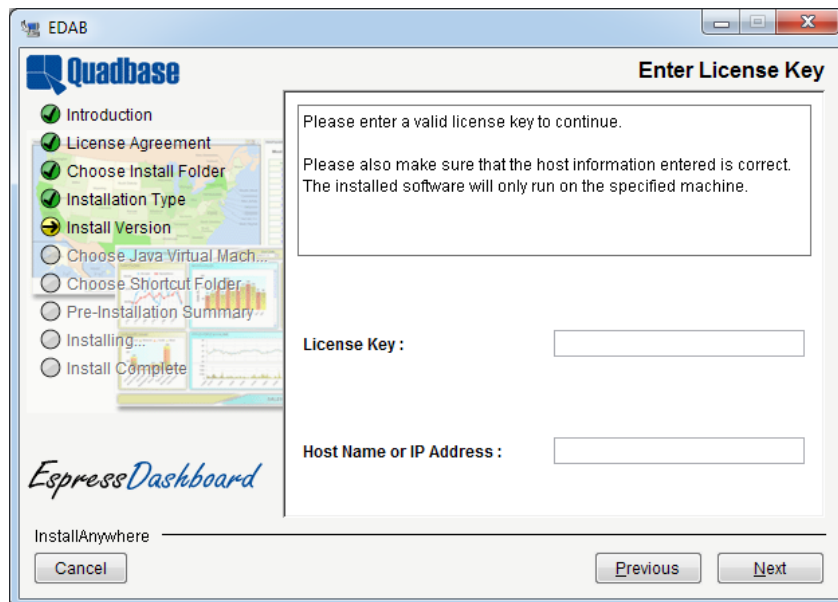
Install Type Dialog

For full installs, the next step will ask you whether you would like to install the evaluation or release version. The evaluation version is fully functional. However, there will be a water mark on any report, chart, map, and dashboard and the license will expire in 45 days. For update installs, it will always use an evaluation license. Please visit <https://www.quadbase.com/register/> to get the full license.



Install Version Dialog

If you select to install a release version, the next dialog will prompt you to enter your license key and verify the host name of the machine on which you're installing EDAB.



Enter License Key Dialog

After you have entered the information, the installer will attempt to register the license key with Quadbase. If the registration fails, you will be unable to continue installing the release version of EDAB. You will have the option to continue installing the evaluation version. After the installation completes, you can register your key online at <https://www.quadbase.com/register/>, or contact Quadbase Sales for additional help.

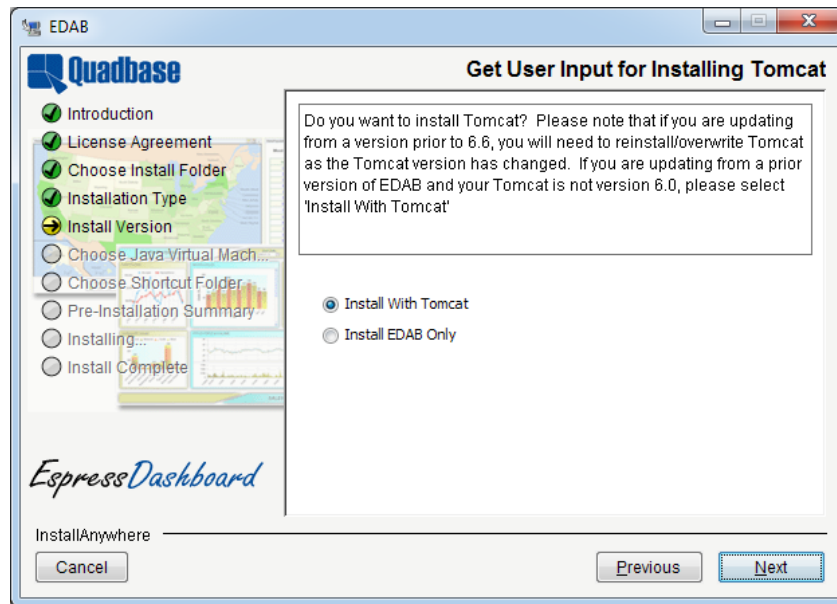


Note

After the installation completes, the release version will only run for the hostname specified in this dialog, so double-check to make sure the host name is correct. You can also use the IP address of the machine if you prefer.

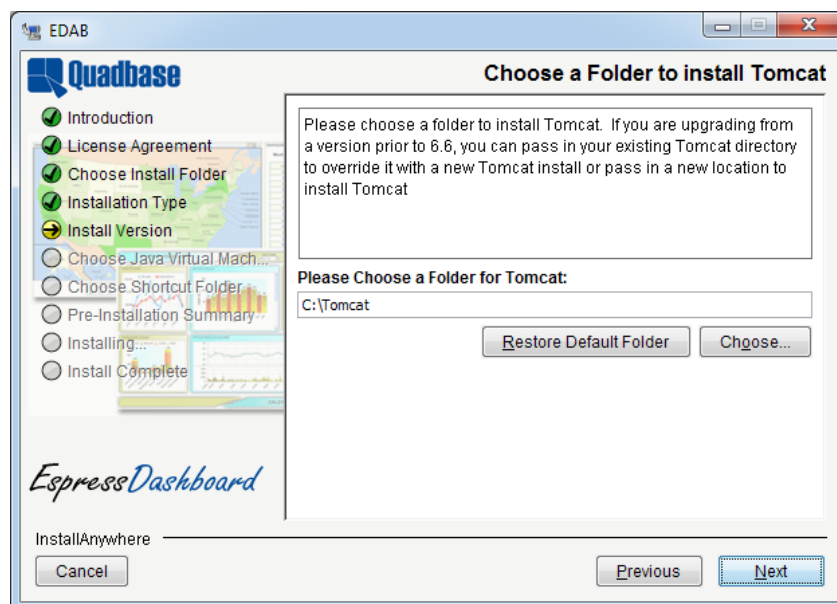
The next option that is presented is whether to install Tomcat with EDAB or not. As described in the previous chapter, the EDAB Server deploys as a servlet collection in an application server/servlet container. For out-of-the-box functionality, you can install the Apache Tomcat server with EDAB already deployed. If you are installing EDAB for the first time, or evaluating the product, it is recommended that you select this option.

If you select to install without Tomcat, the EDAB files will be installed in your system and you will need to manually deploy it within your application server. Instructions for different server platforms are in Section 9.3 - Using Other Application Servers.



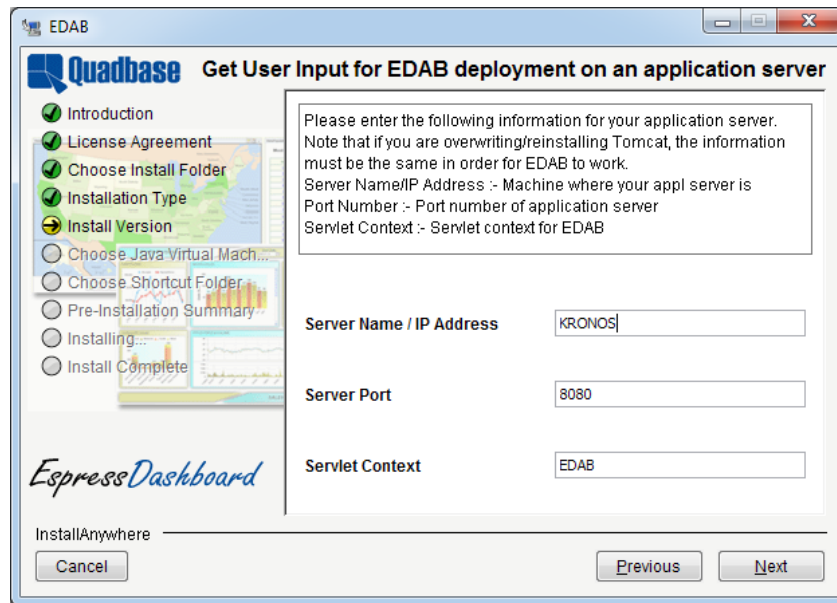
Use Tomcat Dialog

The next dialog prompts you to select the directory into which you would like to install Tomcat. If you select not to install Tomcat this option will be skipped. Tomcat and EDAB will be installed in different directories on the system, and the EDAB directory will be mapped as a virtual directory in Tomcat.



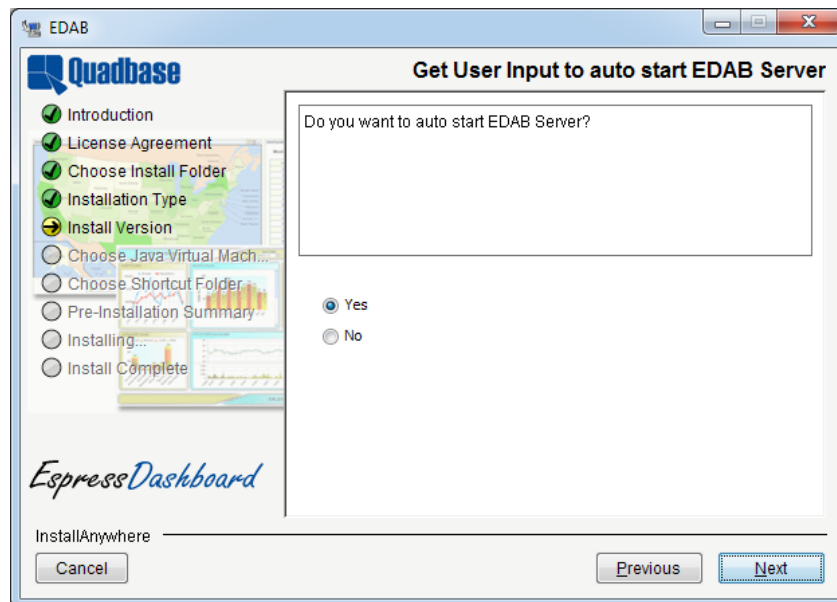
Choose Tomcat Location Dialog

The next dialog prompts you to specify the connection options for the application server. This dialog appears whether you choose to install EDAB with Tomcat or not. You will need to specify the machine name or IP address, the port number, and the context for the application server on which EDAB will be running.



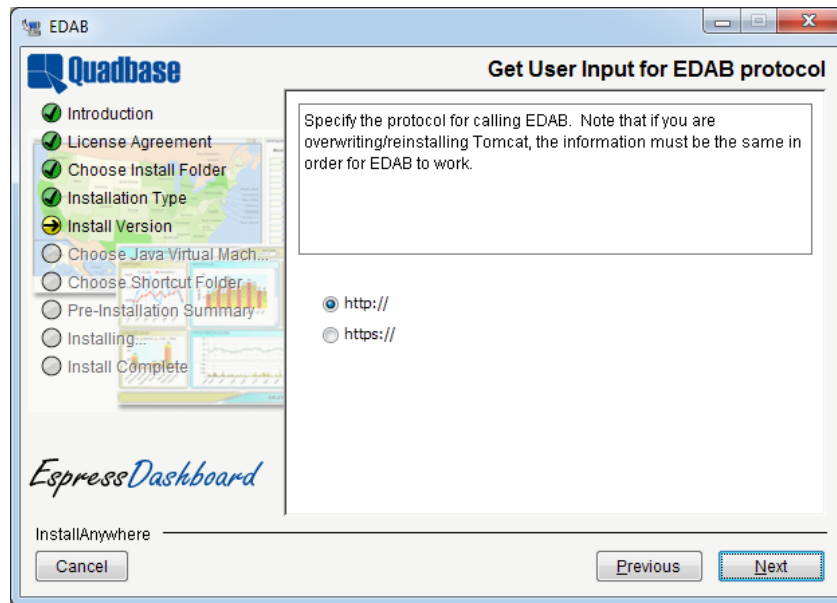
Choose Tomcat Settings Dialog

After specifying the connection details for the application server, the next dialog asks you to choose whether to auto start the EDAB Server, i.e., have the EDAB Server running whenever the application server is started.



Autostart EDAB Server

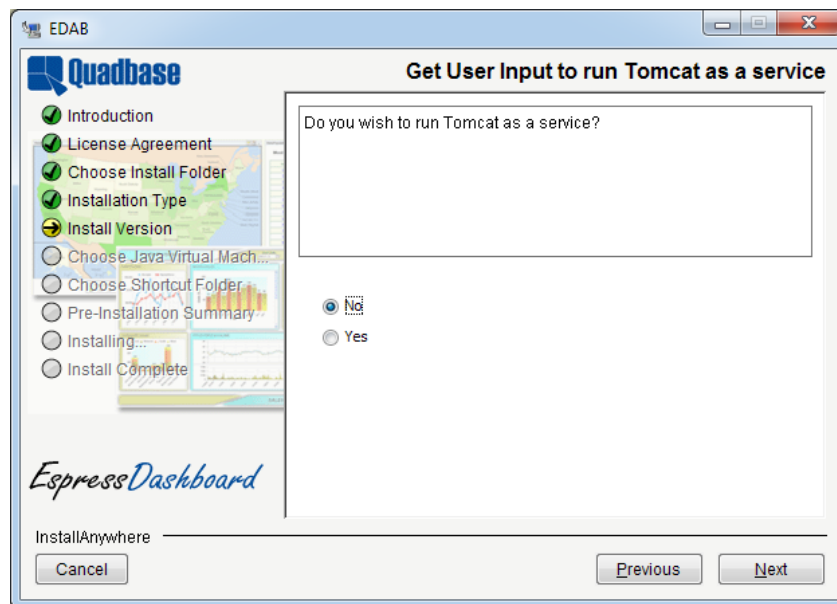
The next dialog asks you to choose the protocol you wish to use to access EDAB, i.e., do you wish to use `http://` or `https://`?



Specify Protocol for EDAB

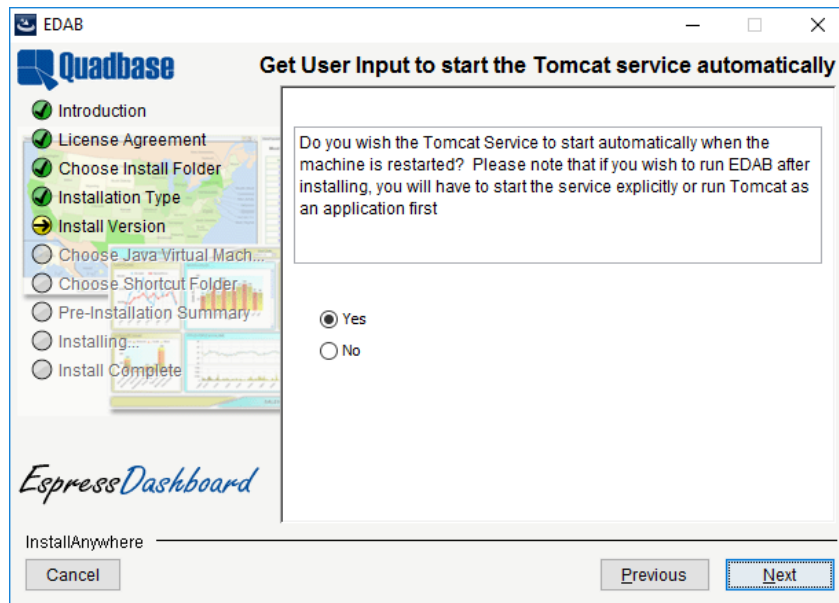
If you choose `https://` protocol and have selected to install Tomcat, the following two dialogs will appear. Note that if you have chosen not to install Tomcat, you are responsible for setting the https connection details and the certificate yourself.

The next dialog allows you to install Tomcat as a Windows service.



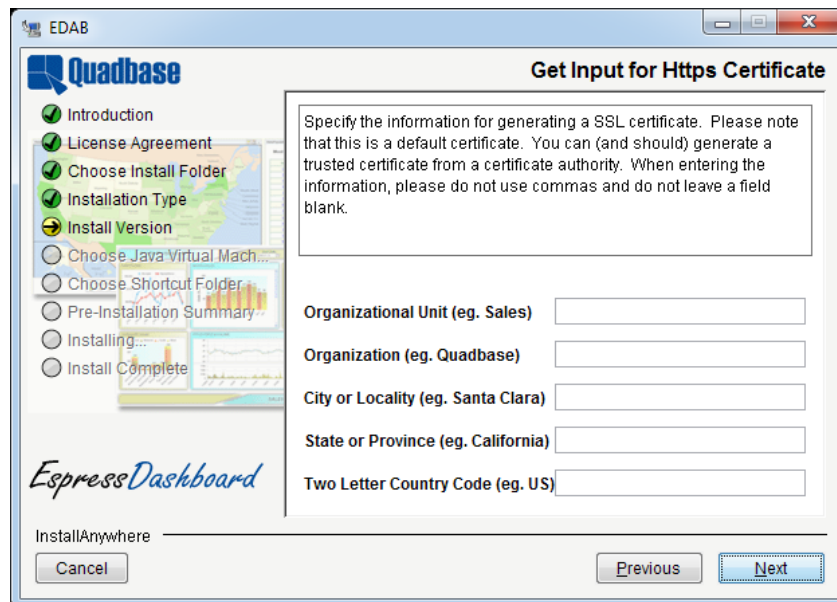
Install Tomcat as a service

If you choose to install Tomcat as a service, the next dialog will prompt you to select whether the service should start automatically when the machine is started.



Start Tomcat service automatically

The next dialog appears when the option to install Tomcat and the `https://` protocol is selected. This dialog obtains information to generate a generic SSL certificate for the installed Tomcat to use.



Specify User and Organization Information

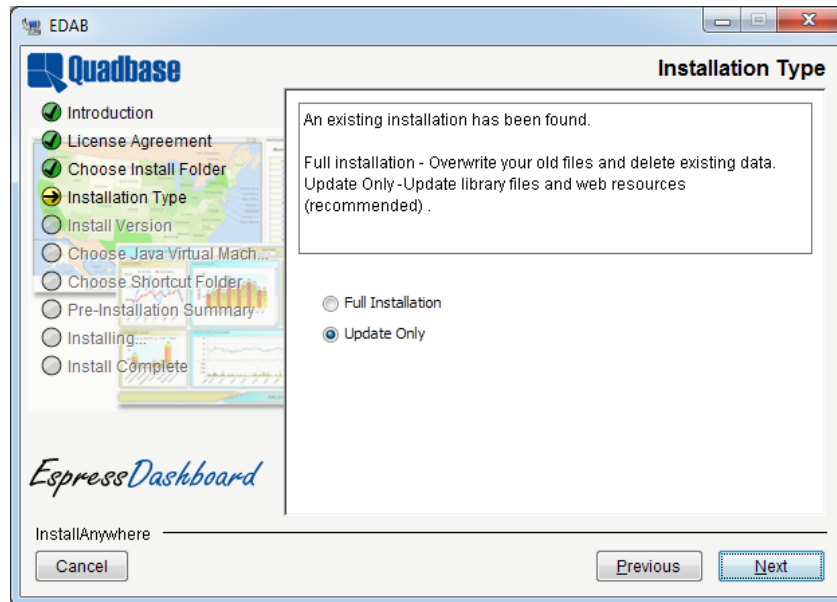
For this dialog, you cannot leave any information blank nor can you use a comma. If either of the two happens or if the two letter country code is not two characters in length, a error message is shown and you will be prompted to enter the information again.

The last option in the installer is only available for the Windows 10/8/7/Vista/XP/2003/NT/2000 installation. It allows you to specify where to create the program shortcuts in the Start Menu, on the Desktop, or both. Note that shortcuts will only be created if you select to install Tomcat.

After you complete the last option you will be shown a summary of the options you've selected. Next, the program will install.

1.3.1.1. Re-Install/Update

If you select a destination directory for the EDAB files that already contains an EDAB installation, you will be prompted as to whether you would like to update your existing installation, or create a new one.

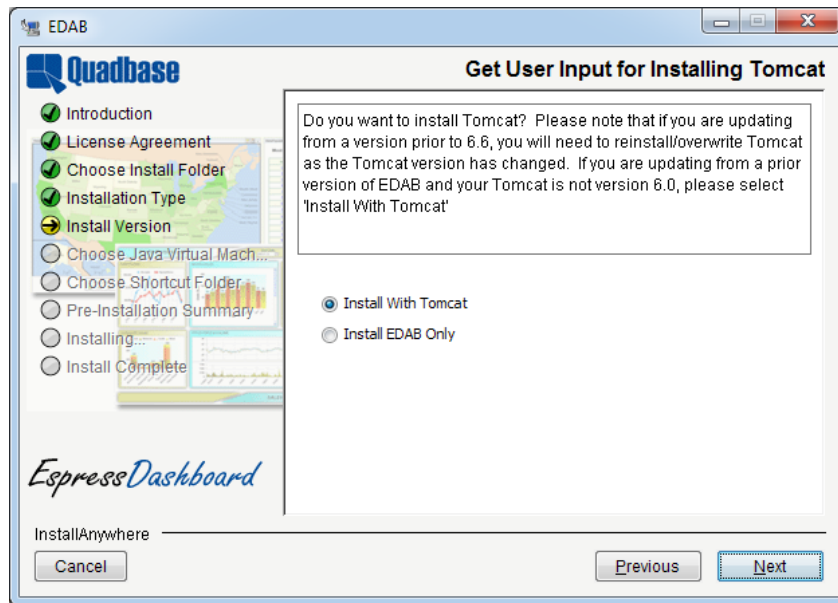


Install Type Dialog

If you select to create a new installation, then the entire existing installation will be overwritten (and all settings will be re-set to default values). If you select this option and click the *Next* button, the installation wizard will continue as if you were installing EDAB for the first time (as described in the previous chapter).

If you select to update the current EDAB installation, only program files will be updated (thus enabling new EDAB features and fixing corrupted files), but all EDAB settings and the EDAB database will be kept as they are (i.e. with your current settings).

If you select to install an update and you are updating from a version prior to 7.0 (in other words: you are not re-installing an existing EDAB 7.0 installation), you will be prompted whether you want to install EDAB with bundled Tomcat 8.0. Please note that **EDAB 7.0 requires Tomcat version 8.0 or higher**. EDAB versions prior to 7.0 were distributed with Tomcat version 6 and lower, so if you are currently using EDAB 6.6 or lower with bundled Tomcat, you will need to install EDAB 7.0 with Tomcat, or update your Tomcat manually (not recommended). If you're just re-installing EDAB 7.0, this step will be skipped, Tomcat will not be installed, and your deployment configuration will not be changed.



Install Tomcat Dialog

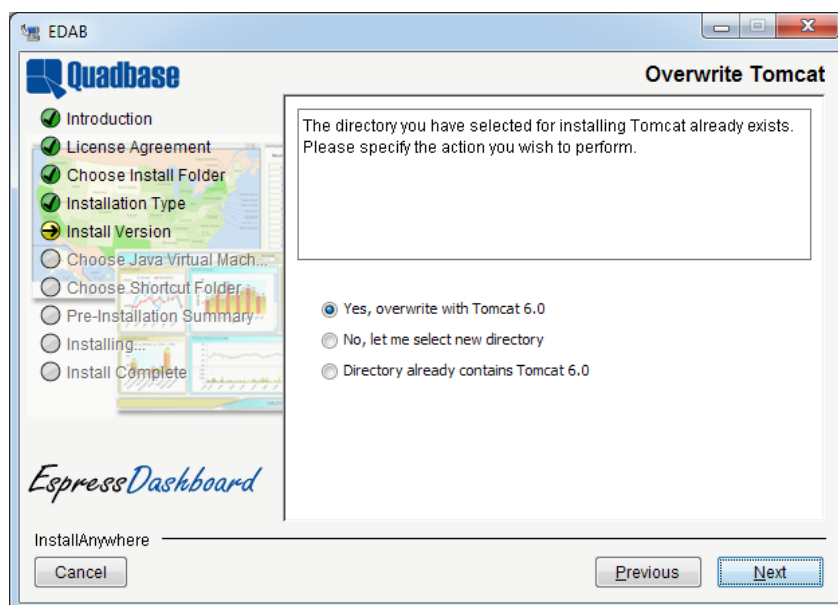
If you chose to install EDAB with Tomcat 8.0, the next step will be to select a directory for Tomcat. If you choose a directory that doesn't contain an existing Tomcat installation and click *Next* (recommended), you will be taken straight to the Tomcat configuration wizard step (as described in the previous chapter).

If you choose a directory that already contains the Tomcat installation, the following dialog will appear giving you three options: to overwrite the existing Tomcat installation with a new one, to go back to the previous step, or to tell EDAB to assume that the directory already contains Tomcat version 8.0 or higher.



Tip

If you want to use an existing Tomcat 8.0 installation for running EDAB, select the existing Tomcat installation directory and then select the *Directory already contains Tomcat 8.0* option.



Install Version Dialog

If you are upgrading from a previous version of EDAB, make sure to follow the steps in Section 1.3.2.1.3 - Upgrading EDAB Database from previous version of EDAB to update your existing EDAB database.

1.3.2. Configuration

After you have completed the installation of EDAB there are several configuration options available.

1.3.2.1. The EDAB Database

Information about users and groups, files in the Organizer, and security/privilege information is stored in a database. By default this is the HSQL Java application database which is included in the EDAB installation. Connection to this database is transparent to users and will work out of the box. This database works fine for development or evaluation environments, however, it is generally insufficient for production environments, as it will not scale to large deployments and provides no failure/recovery features.

1.3.2.1.1. Using a Different Database

EDAB provides users the option of using a different database than the HSQL database provided with the installation. To run with a different database, you will need to create the tables used by EDAB in your database. Create table scripts are provided in the EDAB installation in the data directory, and are available for most major databases.

The database connection that the EDAB server uses to connect to the database is specified in the Admin Console, under Setting Info → EDAB Repository. You can change the connection information to provide a JDBC connection to the database that you would like to use. The JDBC driver for the database you're using will need to be added/available to the classpath of the application server/servlet container where you have deployed EDAB. You can also make a connection to the EDAB database using JNDI. You can pass in the JNDI connection details after pointing your application server to the EDAB database.

Specific setup instructions for different databases are available in Section 9.1 - Using Other Databases.

1.3.2.1.2. Running HSQL in Client-Server Mode

Normally the HSQL database runs as an application process on the server-side and needs no user interaction to start or stop the database process. HSQL can also be run in client-server mode. This mode can improve performance and scalability for the database when run in a multi-user environment. To run the HSQL database in client-server mode, open a console window and navigate to the /data/ directory of your EDAB installation. In this directory run the following command:

```
java -classpath "../WEB-INF/lib/hsqldb.jar" org.hsqldb.Server -database
quadbasedb -port 2857 (or whatever port you would like the database to
listen on)
```

The database server process will then start. To configure the EDAB Server to connect to HSQL running in this mode, log on as Admin and enter the Admin Console → Setting Info → EDAB Repository and change the entry in Database URL to read `jdbc:hsqldb:hsql://machinename:port` where machinename is the name or IP address of the server, and the port number is the port you selected for the HSQL server.

1.3.2.1.3. Upgrading EDAB Database from previous version of EDAB

In order to support new features, the EDAB database structure generally needs to be modified.

If you are running the EDAB database on HSQL, MySQL, Oracle, MS-SQL, Informix or Postgres database, the database will be updated automatically during the EDAB installation process.

If you are using a different database, you need to run the upgrade program before you can start using the latest version of EDAB. The upgrade programs are located in the <EDAB-INSTALL>/data/dbupgrade directory.

Currently, the upgrade programs support MS-ACCESS and DB2 databases (besides the six databases that are also supported by the EDAB installer). If your database is not in the above list, please contact our technical support staff for more details.

There are two versions of upgrade programs, namely, DBUpgradeGUI and UpgradeAppl.

The former is the graphic user interface version, the latter the command line version.

Please note that EDAB log records are not preserved during the upgrade process.



Warning

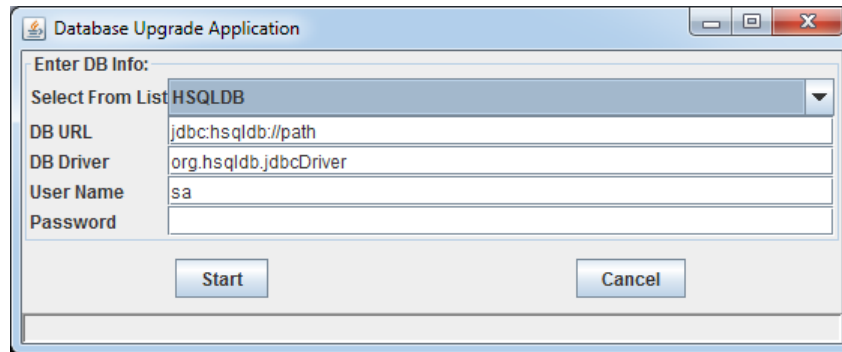
As a precaution, you are recommended to backup your EDAB database before you proceed with the upgrade. This is to safeguard the unlikely event that any abnormal situation may occur during the upgrade process.

Upgrade User Repository Database Using Graphic Interface

1. Change directory to <EDAB-INSTALL>/data/dbupgrade/
2. Enter command:

```
java -classpath "<DB_Driver>;DBUpgrade.jar;." quadbase.DBUpgradeGUI
```

The upgrade program graphic interface is shown below:



Upgrade DB Program GUI

You need to specify the database information so that the program can proceed to do the upgrade.

Please note that the default database connection information is provided only as a guideline, you need to fill in the exact connection information for the upgrade program to run. Also, please make sure that the database driver is included in the `-classpath` flag in the command.

For example, if you are using Oracle, you may run the command with classpath as follows:

```
java -classpath "c:\EDAB\WEB-INF\lib\jdbc_oracle.jar;DBUpgrade.jar;."
quadbase.DBUpgradeGUI
```

Press the *Start* button and the upgrade process will begin.

If you entered wrong database information, the program will prompt you with an error message complaining wrong database connection settings.

After the upgrade program is finished, the status bar at the bottom of the window will show *Done*.

If you click the *Cancel* button during the upgrade process, it will terminate the upgrade program. But it will NOT roll back to the original database. In such a case, you have to use your own backup data or use the backup SQL file generated by this program to restore your database.

Upgrading EDAB Database Using Command Line:

To upgrade using the command line version, simply type the command as follows:

```
java -classpath "<DB_Driver>;DBUpgrade.jar;." quadbase.UpgradeAppl
<EDAB_config_file>
```

The program will use the file specified by <EDAB_config_file> to obtain connection information. The config file should be written in the same format as the config.txt in <EDAB_INSTALL>/data/dpupgrade/.

1.3.2.2. Starting the Server

If you have selected the option to *Auto-Start Server*, then the EDAB Server is started the moment the application server (Tomcat by default) is started. If the option was not selected, or you change the option in the Administration Console → Server Options → Auto-Start, then you will have to start the EDAB Server manually.

The first step in running and administering EDAB is to start the server. To start the EDAB server, you will first need to start the application server/servlet container in which the server is deployed. If you installed EDAB with Tomcat, go to the /bin/ directory of the Tomcat installation and execute startup.bat / .sh to start Tomcat.

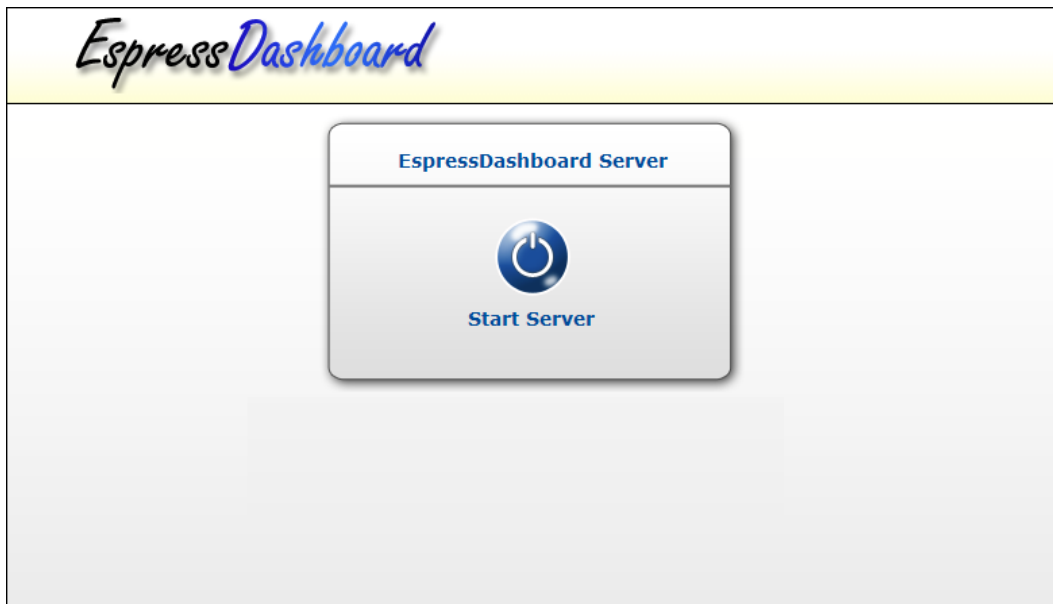
Next, load the start page for EDAB. This is the index.jsp page in the EDAB install directory. If you installed EDAB with Tomcat then the virtual directory mapped to the EDAB install directory is /EDAB/ so to reach the index page, you would use the following URL:

http://machinename:port/EDAB/index.jsp



Note

You must have cookies and javascript enabled in your browser in order to use the EDAB interfaces.



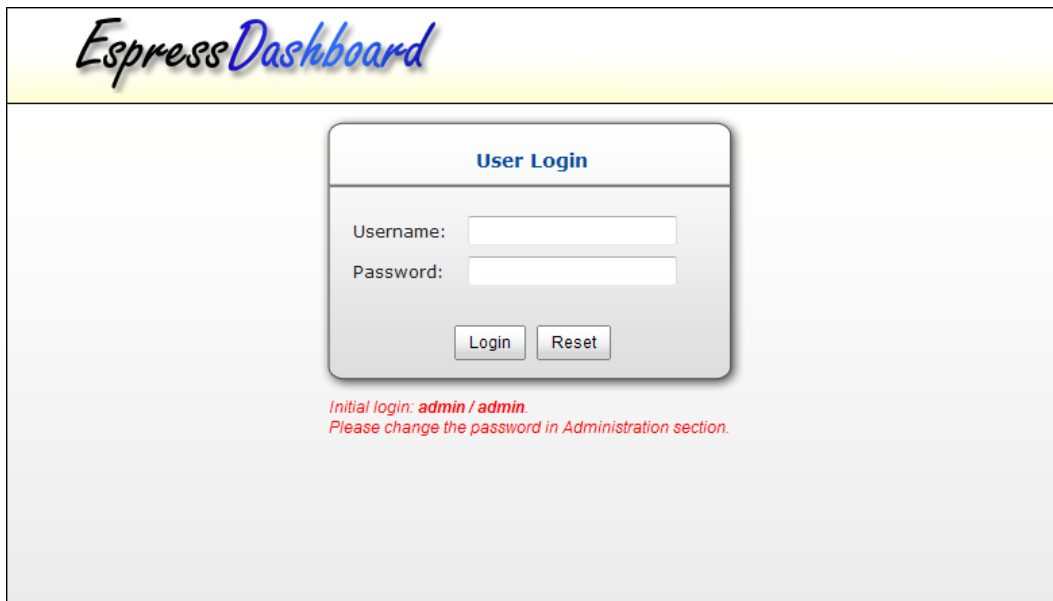
EDAB Start Page

You can start the server directly in the start page. If the “Autostart” feature has been enabled (see Section 1.3.1 - Installing EDAB for more details), the EDAB server should be already running. If it's not, click the *Start Server* button. Once the server is started, you can log-in to access other EspressDashboard functions (the default administrator login is username: **admin** password: **admin**).



Note

Any user can start the server, but only the administrator can shut down the server. Even with the server running, the *Shut Down* button will only become active when you login as the system administrator.



EDAB Server Running

1.3.2.3. Increasing maximum memory heap size for applets

All applets have a maximum memory heap size of 16 megs, by default. In Windows, this can be increased by going to Control Panel and selecting Java (or Java Plugin). Click on the *Java* tab and then *View* under *Java Applet Runtime Settings*. Enter **-Xmx256M** under *Java Runtime Parameters* and click *OK*.

1.3.3. Backward compatibility patches

If you upgraded from an older version, you may notice some changes in the default behavior. Although the backward compatibility is kept as much as possible, sometimes a new behavior is preferred. The new behavior should be better for most users, but if you already have some charts or reports from an older version, you may want to keep the old behavior, so your charts and reports look exactly the same as before. That is why we provide backward compatibility patches.



Caution

Patches are for advanced users only. Please apply them only if you need them and if you know what you are doing. If you are not sure, please contact support.

The patches can be found in `<EDAB_Installation_Directory>/lib/Patches` directory. They are stored in JAR archives. To apply a patch, you only need to add the appropriate JAR file to the classpath of your application as described below.

If you want to apply a patch to all designers (Quick Designer, Chart Designer, Dashboard Builder, etc.) and viewers (Dashboard Viewer, etc.), you have to copy the appropriate patch JAR file to `<EDAB_INSTALLATION_DIRECTORY>/WEB-INF/lib/` directory. Then you have to edit `Organizer.jnlp` file in the EDAB installation directory and add relative path to the patch JAR file to the *archive* attribute of the *applet* tag.

Below is a list of all available patches in the current version.

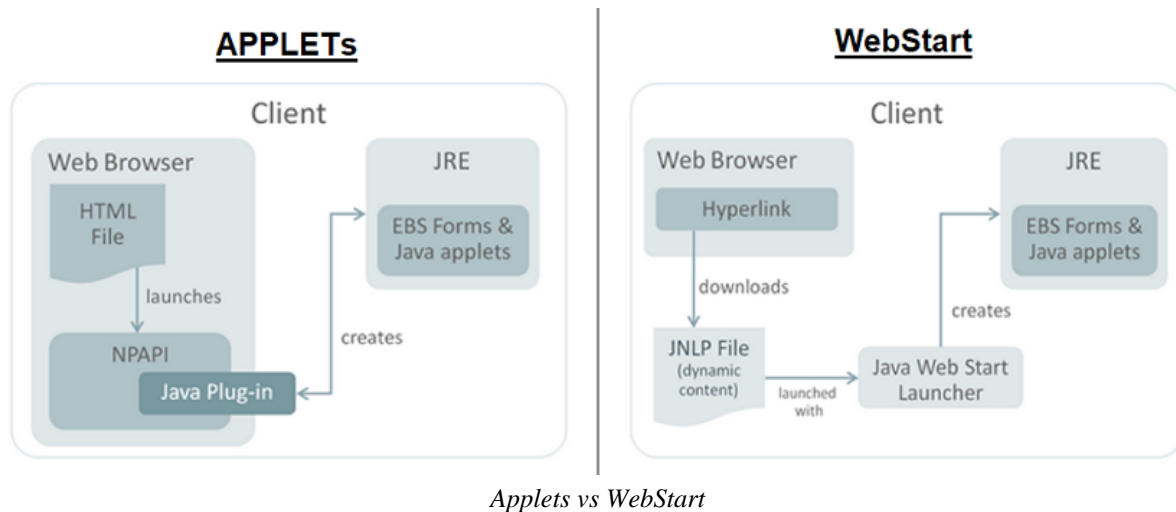
- patch1.jar** - turn off chart axis padding by default
- default behavior (without the patch) - axis padding is on by default
 - behavior with the patch - axis padding is off by default
 - new behavior has been introduced in version 4.0

- this feature can also be set in the Axis Scale dialog in the Chart Designer
- patch2.jar**
- add left margin for annotation text in charts
 - default behavior (without the patch) - annotation text does not have left margin
 - behavior with the patch - annotation text has left margin
 - new behavior has been introduced in version 5.0
 - annotation text is legend text, chart titles and any text inserted using Insert → Text in the Chart Designer
- patch3.jar**
- use 0 to 1 as min/max value when chart axis autoscale for axis pt less than 1
 - default behavior (without the patch) - maximum and minimum are always set according to data if autoscale is used
 - behavior with the patch - if max-min is < 1 and autoscale is used, min is set to 0 and max is set to 1
 - new behavior has been introduced in version 5.4
 - **It is not recommended to use this patch. The original behavior is a bug.**
- patch4.jar**
- turn off new pie chart label placement algorithm (calculate label placement based on pie sector position)
 - default behavior (without the patch) - new pie label placement algorithm is used
 - behavior with the patch - old pie label placement algorithm is used
 - new behavior has been introduced in version 6.0
- patch5.jar**
- always use integer value for chart axis auto scale
 - default behavior (without the patch) - integer is always used for axis auto scale
 - behavior with the patch - axis value data type is used for axis auto scale
 - new behavior has been introduced in version 6.0
- patch6.jar**
- disable minimum and maximum error check for chart axis scale
 - default behavior (without the patch) - the error check is enabled
 - behavior with the patch - the error check is disabled (so you can set maximum value lower than the one set in your dataset - same for minimum)
 - new behavior has been introduced in version 6.2
- patch7.jar**
- turn off Single color for categories feature for columnar and bar charts by default
 - default behavior (without the patch) - Single color for categories feature is turned on by default
 - behavior with the patch - Single color for categories feature is turned off by default
 - new behavior has been introduced in version 6.3
 - this feature can also be set in the Chart Options dialog in the Chart Designer
- patch8.jar**
- line chart end to end revert single point data to display on left axis

patch9.jar - display stack label despite not having enough space in the stack to render it

1.3.4. Run Applets in WebStart with JNLP file

Trying to run Java Applets in newer browsers and/or with newer Java versions can lead to problems with Applets being deprecated or not supported any more. Officially were Applets blocked to run in common browsers from about year 2016. However, it is still possible to run Applets in a similar use case as before (open a Java application from a remote server via your browser) thanks to Java WebStart technology that's supported by Java 8. Both Applets and Java Web Start are marked as deprecated technologies from Java 9.



In this case, EDAB is Java 8 compiled application and is best to use it with Java 8. For using Applets with WebStart in newer versions, you must use some different implementation of WebStart. Popular choices are OpenWebStart or IcedTea-Web and it is already included in some newer Java distributions.

To run a Java Applet via a Java Web Start JNLP file, you have to specify the `applet-desc` parameter in the JNLP file configuration file.

```
<applet-desc
  name="Chart Viewer"
  main-class="quadbase.chartviewer.Viewer"
  width="800"
  height="600">
<param name="filename" value="help/examples/ChartAPI/data/test.tpl"/>
<param name="preventSelfDestruct" value="true"/>
</applet-desc>
```

The `param` tags specify parameters for the applet. The parameters are different for each applet. The specific parameters are mentioned in the documentation chapter for each applet.

JNLP files can be either run locally when the required libraries (containing the compiled source code) are loaded as a local file or remotely when the libraries are loaded via HTTP/HTTPS.

When running JNLP files remotely via HTTP/HTTPS, the parameters `comm_protocol`, `comm_url` and `servlet_context` need to be added to the `applet-desc` element. You can either fill in the values manually (as the following code example shows) or you can have the values filled automatically in a JSP file.

```
<applet-desc
  name="Espress Report Designer"
  main-class="quadbase.reportdesigner.designer.ReportClient"
  width="380"
```

```

height="160">
<param name="comm_protocol" value="servlet"/>
<param name="comm_url" value="http://127.0.0.1:8080"/>
<param name="servlet_context" value="Espress70/servlet"/>
</applet-desc>

```

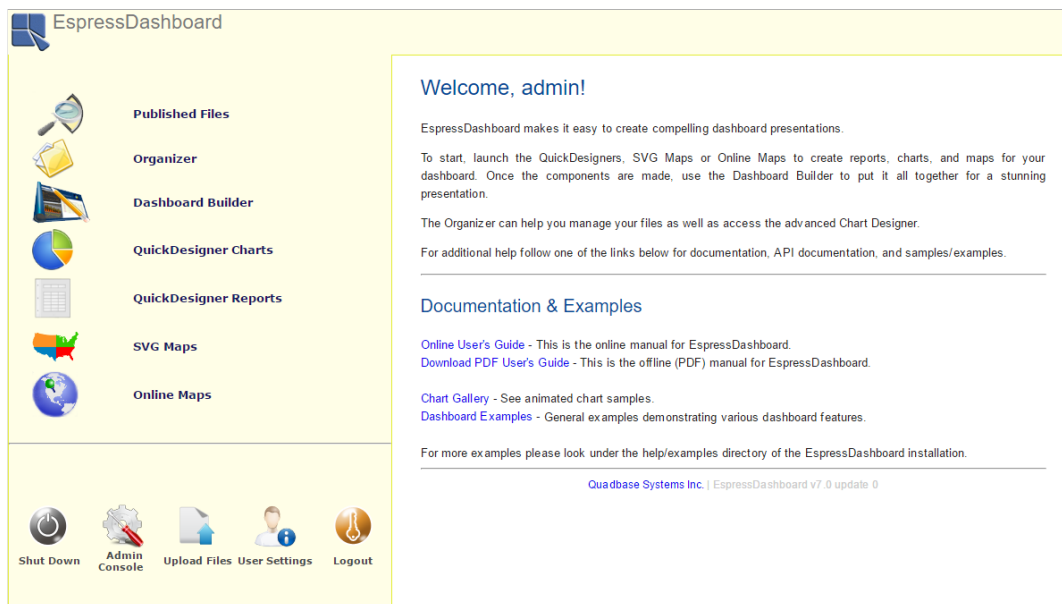


Note

If you want to pre-fill the values automatically in a JSP file, the actual values highly depend on your server configuration. The previous example is just illustrative.

1.4. Administration

Major administrative functions in EDAB are handled in the Admin Console. The admin console is a thin-client interface that can be launched from the EDAB start page. To start the Admin Console, start the server (see Section 1.3.2.2 - Starting the Server), and login as the administrator. The administrator username is **admin** and the default password is **admin**. This can be changed in the console.

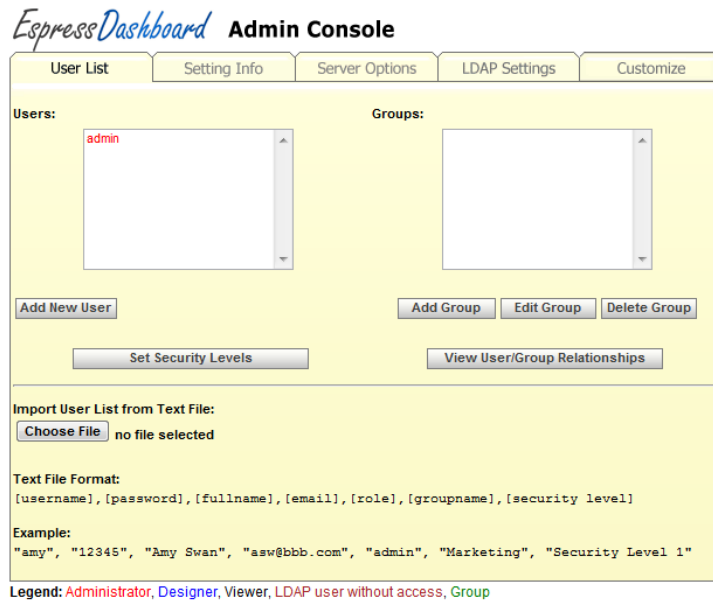


Admin Logged into Start Page

Once you log in as the administrator, an *Admin Console* button will be displayed at the top right portion of the page. Click this button to load the admin console.

1.4.1. The Console Interface

The top of the Admin Console interface contains five tabs. One is labeled *User List*, one *Setting Info*, one *Server Options*, one *LDAP Settings*, and one *Customization*.



Admin Console

1.4.1.1. User List

The *User List* tab of the Admin Console allows you to create and maintain users and groups that use EDAB to design or view reports and charts. The left-hand side of the dialog contains a list of all the defined users, and the right-hand side a list of all the defined groups. Normally these users and groups are stored in the EDAB database, however EDAB can be configured to retrieve the users and groups from existing systems including LDAP servers. To add a new user, click the *Add New User* button. This will bring up a new dialog allowing you to specify information for the new user.

Add User Dialog

For each user, you need to supply a username, an email address (for schedule delivery), and a password. Each user also has a primary role of a viewer or a designer which is assigned in this dialog. Viewers can see and run reports as well as build ad hoc reports using the QuickDesigner, but they do not have access to the core design/development tools, like Chart Designer, and Organizer. Designers have access to all the interfaces, but are restricted by licensing and can't change server settings and user privileges. Each designer user must be licensed by a developer seat.

Administrator can view all the interfaces, change server settings, view detailed logs and modify user privileges. Each administrator must be licensed by a developer seat.

In the user list, you can quickly distinguish user roles by their color. There is a legend for that below the admin console.

Once you have finished adding information for the new user, click *Ok* and the new user will be added.

You can edit or remove a user by selecting it in the user list, and clicking either *Edit User* or *Delete User(s)* button below the user list. If you select to edit the user, a dialog will open allowing you to change the information for that user. If you select to delete the user, it will be removed. The original administrator user (*admin*) can't be removed.



Note

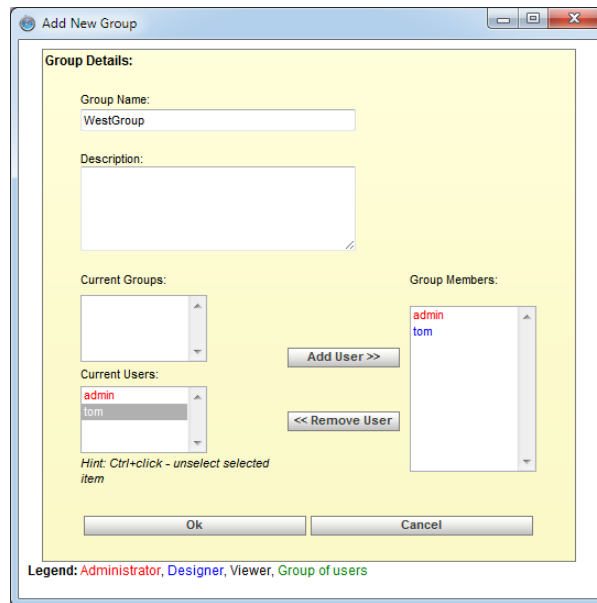
You can't edit users that have been read from LDAP (see the Section 1.4.1.4 - LDAP Settings chapter to learn more about LDAP users). If you select a LDAP user and click the *Edit* button, the following dialog will open.

Edit LDAP user dialog

You are allowed to edit only the user attributes, that have not been read from LDAP. The rest of the user settings can be edited on the LDAP server.

Also, LDAP user accounts can't be fully deleted. If you select a user that has been read from an LDAP server and click the *Delete User(s)* button, the user will be deactivated, but it will not be removed from the user list - it will be marked as *LDAP user without access*.

To add a new group, click the *Add Group* button under the group list. This will pop-up a new dialog prompting you to specify information for the group and select members.



Add Group Dialog

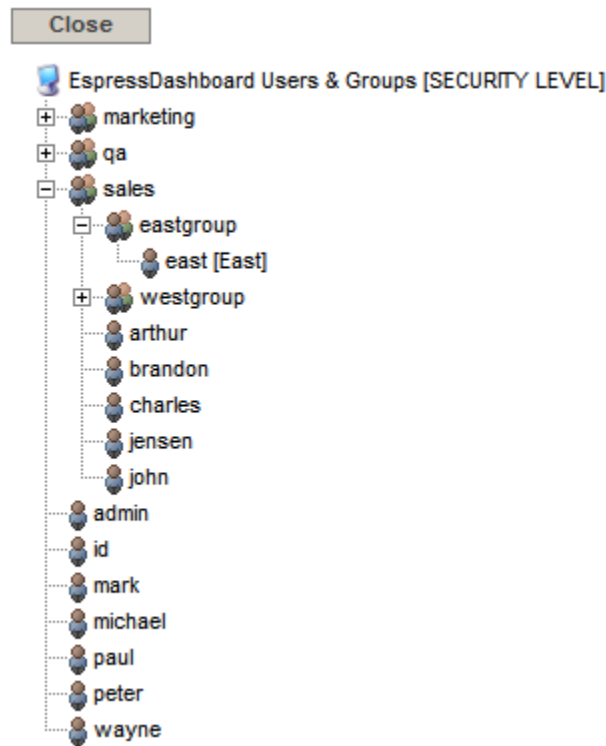
For each group, you need to name the group, and select members. Members can be individual users or other groups. To add group members, select users and/or groups from the left-hand side, and click the *Add* button. You can make multiple selections (or de-selections) by holding down the **CTRL** key when clicking on the user/group.

Once you have finished adding members to the group, click the *Ok* button and the group will be added.

You can edit or remove a group by selecting it in the group list, and clicking either *Edit Group* or *Delete Group* button below the group list. If you select to edit the group, a dialog will open allowing you to change the information for the group. If you select to delete the group, it will be removed.

There is also a button to set security levels for users and groups. Details for the security level dialog can be found in Section 1.4.1.1.3 - Setting User/Group Security Levels.

You can view all of the user and group relationships by clicking the *View User/Group Relationships* button. This will bring up a new dialog that shows all of the users and which groups they belong to in a tree format. If the user or group has a certain security level assigned to them, you will also see the name of the security level in brackets.



User/Group Relationships Dialog

1.4.1.1.1. Importing Users and Groups

In addition to adding users and groups via the Admin Console manually, you can directly import users from a delimited file. To import users, you must have a CSV file with the following structure:

```
[username],[password],[fullname],[email],[role],[groupname],[security level]
```

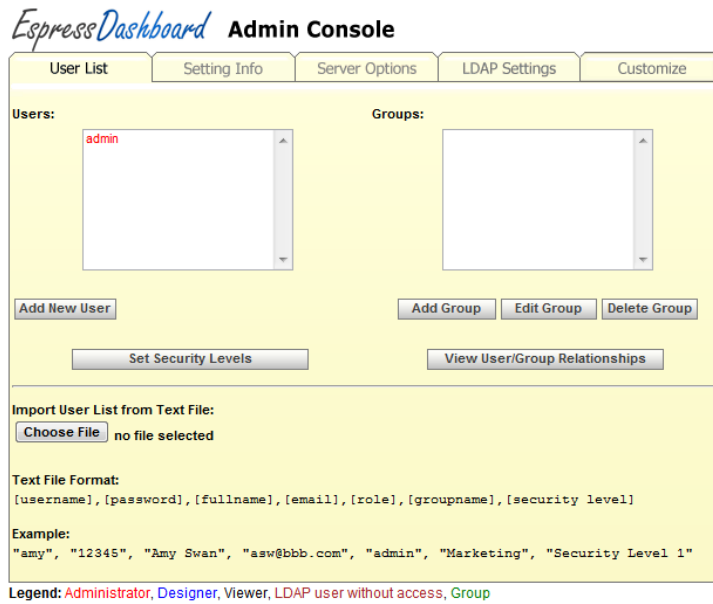
username is the user login name, password is the user password, fullname is the user's full name, email is the user's email address, and role indicates the user's primary role either *designer*, *viewer* or *admin*. The groupname field indicates which group the user belongs to. If that group has already been defined then the user will be added to the group. If the group has not been defined a new group will be created. A sample import file might look something like this:

```
"amy", "12345", "Amy Swansen", "aswansen@quadbase.com", "admin",
  "Marketing","Security Level 1"
"tom", "12345", "Tom Weeks", "tweeks@quadbase.com", "designer",
  "Executive","Security Level 2"
"sarah","12345", "Sarah Jensen", "sjensen@quadbase.com", "viewer",
  "Marketing","Security Level 1"
```

In the case of importing user list for LDAP users, add one line with only "LDAP" at the beginning of the user-list text file, then all the contents of the password column will be ignored, and the password of the individual user will be obtained from LDAP Active Directory at run time.

If you have a convenient delimited file with users that you want to import into the EDAB, log in as an administrator and then open the Admin Console. Make sure you are on the *User List* tab.

In the *Import User List from Text File* section, there will be a *Choose File* or *Browse* button allowing you to import the delimited file. Click on the button.



EspressoDashboard Admin Console

User List Setting Info Server Options LDAP Settings Customize

Users:

admin

Groups:

Add New User Add Group Edit Group Delete Group

Set Security Levels View User/Group Relationships

Import User List from Text File:

Choose File no file selected

Text File Format:
[username],[password],[fullname],[email],[role],[groupname],[security level]

Example:
"amy", "12345", "Amy Swan", "asw@bbb.com", "admin", "Marketing", "Security Level 1"

Legend: Administrator, Designer, Viewer, LDAP user without access, Group

User List tab of the Admin Console



Note

The *Choose File* button may be different in other web browsers (the screen shot was taken in Safari), but it will always do the same thing - if you click the *Choose File* (or *Browse*) button, a browse dialog will pop up allowing you to select a file.

Select the delimited file and confirm the browse dialog. The following message should show up confirming that the users, groups and security levels were imported successfully.

Import of 3 new users was successful!

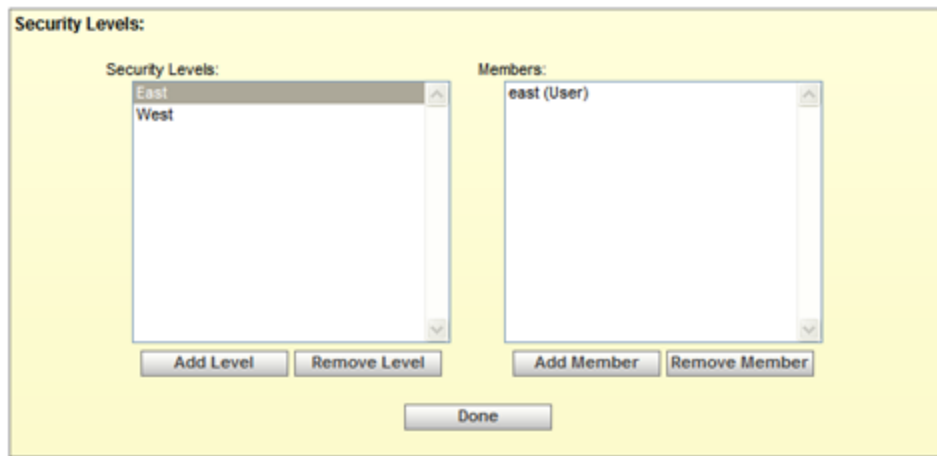
OK

1.4.1.1.2. Using Existing Users/Groups

Instead of creating new users and groups in EDAB, you can re-use existing users and groups. EDAB allows you to retrieve users and groups from an existing database, integrate with an LDAP server, or implement your own logic/methods to supply EDAB with users and groups. The security provider interface provides an API that allows you to implement any custom logic to return user's, groups, and user/group relationships. For more information about configuring EDAB to use existing users and groups, please see Section 9.2 - Integrating Existing Users/Groups. For more information about LDAP integration, please see Section 1.4.1.4 - LDAP Settings.

1.4.1.1.3. Setting User/Group Security Levels

On the User List page, there is also a button to *Set Security Levels*. When you click this button, a dialog will pop up allowing you to create, remove, and configure security levels within EDAB. For more information regarding security levels please see Section 2.2.3 - Security Levels.



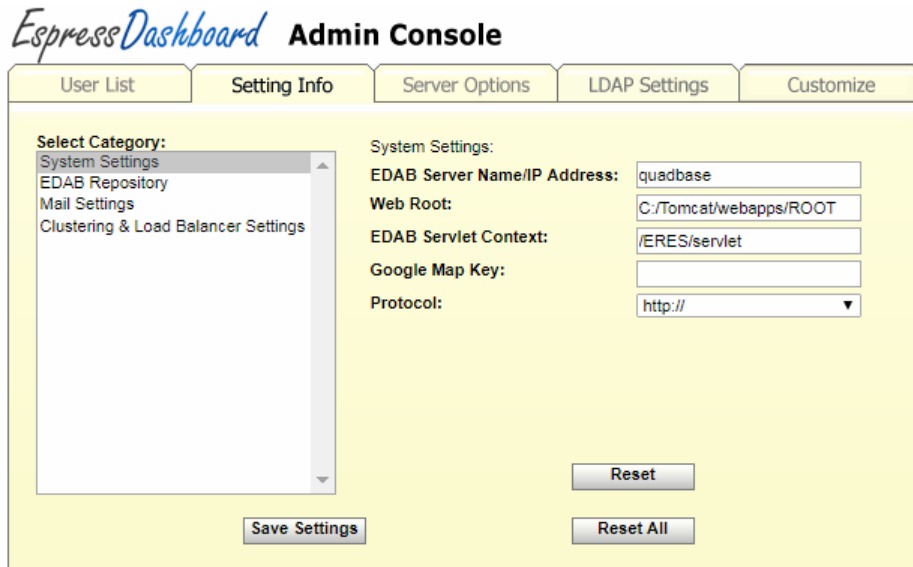
Set Security Levels Dialog

To add a new security level, click the *Add Level* button and provide a level name. You can add any number of users and groups into each security level. To do so, click on the *security level name* in the list box and click the *Add Member* button. This will present a dialog allowing you to select existing members and groups and add it to the security level. You can also remove members and security levels by selecting the item and using the *Remove Member* and *Remove Level* buttons respectively. Click *Done* when you are finished.

Please note that all security level changes will take effect immediately.

1.4.1.2. Setting Info

The *Setting Info* tab contains configurations and settings critical to the way EDAB functions.



Setting Info Dialog

The first category, *System Settings*, presents the server information.

EDAB Server Name/IP Address: This is the host name or IP address of the machine on which EDAB is installed. This is generally configured during installation, but can be changed here if necessary.

Web Root: This is the web root for the server in which you've deployed EDAB. If you installed EDAB with Tomcat, this should be configured automatically. However, if you use a different database than the default HSQLDB, this field

will need to be reconfigured. For tomcat, the default web root is `/webapps/ROOT/`.

EDAB Servlet Context:

This is the context in which the servlet collection for the EDAB server, and deployment options are deployed. If you installed EDAB with Tomcat, the default context is `EDAB/servlet`. If you change this option, you will need to re-start the server for it to take effect.

Google Map Key:

This is the key used for designing and viewing Google Maps. Please note that this must be changed to a key mapped for your machine, otherwise Google Maps will not appear. Please see Section 5.2 - Online Maps for Google Map details.

Protocol:

This tells the system whether to use HTTPS with SSL or regular HTTP to connect to the sever.

Please note that changing protocol here is not enough to setup the system to work with SSL. You also have to configure your application server and update URLs of the existing files in Organizer. The recommended way is to reinstall your product and use the HTTPS option in the installer that sets up everything automatically (see Section 1.3.1 - Installing EDAB for more details). It is not enough to do upgrade install, you have to perform full installation. Be aware that full installation overwrites internal EDAB database, so you loose all the files inserted in Organizer. Please back them up before making the full installation.

If you cannot or do not want to make the full installation, you can still configure SSL manually. To do that, please follow these steps:

1. Change the protocol in the Admin Console as described above.
2. Get (or generate) SSL certificate, install it on your application server (following instructions of your application server) and set up your application server to use HTTPS.
3. Start your application server and access EDAB start page using the HTTPS (if you changed port number, do not forget to use the new one).
4. Start Organizer and update your URL Mapping to use the new protocol (and new port if it has changed). See Section 2.1.5 - URL Mapping for more details about URL Mapping.
5. Run Repair Broken Links to update URL Mapping for all files in Organizer. See Section 2.1.6 - Repairing Broken Links for more details about this feature.
6. If you use the `EDABHome.html` shortcut to access EDAB start page, please also update the protocol (and port) in your `<EDAB_INSTALLATION_DIRECTORY>/EDABHome.html`.

If you are switching from HTTPS to HTTP, use similar instructions. Just replace HTTPS with HTTP and skip generating the SSL certificate.

EspressoDashboard Admin Console

User List | **Setting Info** | Server Options | LDAP Settings | Customize

Select Category:
 System Settings
EDAB Repository
 Mail Settings
 Clustering & Load Balancer Settings

☒ Database ☐ JNDI

Database Syntax Assistance: None

Database URL: jdbc:mysql://kronos:3304/edab

Database Driver: com.mysql.jdbc.Driver

Database Username: root

Database Password: ****

Re-enter Database Password: ****

Reset

Save Settings Reset All

Database Repository

The second category EDAB Repository describes the current database configuration options. You can use either a database connection or through JNDI. For database, you need to specify the following information.

Database URL: The JDBC URL to the database.

Database Driver: The JDBC Driver to the database.

Database Username: The username used to connect to the database.

Database Password: The password used to connect to the database. This field needs to be reentered in the next line to ensure validity.

You can use the Database Syntax Assistance drop down box to help in set up the connection by setting up a template of the URL and Driver.

Note that if you make a mistake while entering the database connection details, it may cause EDAB not to run properly. In that event, the only way to correct it would be to be open <EDAB Install Directory>/WEB-INF/classes/QB.properties file and edit the entries for DatabaseUrl, DatabaseDriver, DatabaseUserName and DatabasePassword. Depending on how the file was generated, the entries may not be grouped together.

The following is an image of the JNDI connection options. Not all components need to be completed in order to connect to the JNDI server.

EspressoDashboard Admin Console

User List | **Setting Info** | Server Options | LDAP Settings | Customize

Select Category:
 System Settings
EDAB Repository
 Mail Settings
 Clustering & Load Balancer Settings

☐ Database ☒ JNDI

JNDI Initial Context Factory:

JNDI Provider URL:

JNDI Name: java:com/env/jdbc/eresdb

Reset

Save Settings Reset All

JNDI Repository

- JNDI Initial Context Factory:** The JNDI initial context factory specifies the class used to connect to the JNDI server. If you wish to use the application server's default context factory, you can leave this field blank.
- JNDI Provider URL:** The URL to the JNDI Provider. If the provider URL is already set up in the application server environment, you can leave this field blank.
- JNDI Name:** The name for the JNDI connection. Frequently, the name contains the prefix `java:comp/env/`. This field is required.

Mail settings allow you to configure a SMTP server. You need to provide SMTP server details in order to enable *Email delivery* in alert monitoring (described in Section 11.4 - Monitoring).

The screenshot shows the 'EspressoDashboard Admin Console' interface. At the top, there are five tabs: 'User List', 'Setting Info', 'Server Options', 'LDAP Settings', and 'Customize'. The 'Setting Info' tab is selected. On the left side, under 'Select Category:', there is a list of categories: 'System Settings', 'EDAB Repository', 'Mail Settings' (which is highlighted in blue), and 'Clustering & Load Balancer Settings'. The main area displays the 'Mail Settings' configuration. It includes the following fields and controls:

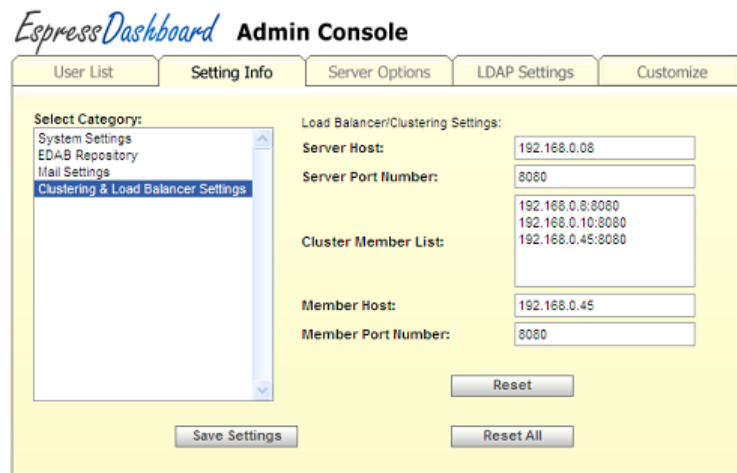
- SMTP Host:** A text input field containing 'smtp.someserver.com'.
- Use SSL:** A checkbox that is checked.
- Secured SMTP:** A checkbox that is checked.
- SMTP Port:** A text input field containing '25'.
- SMTP Username:** A text input field containing 'username'.
- SMTP Password:** A password input field with masked characters (dots).
- Re-enter SMTP Password:** A password input field with masked characters (dots).

At the bottom of the form, there are three buttons: 'Save Settings' on the left, 'Reset' in the center, and 'Reset All' on the right.

Mail settings

- SMTP Host:** This is the host or ip address of the SMTP server.
- Use SSL:** Enable this if the SMTP server uses SSL/TLS.
- Secured SMTP:** Enable this if username and password are required to send mail from this server.
- SMTP Port:** This is the port for the secured SMTP server.
- SMTP Username:** Specify the username to use to log into the server.
- SMTP Password:** Specify the password for the email account. Enter again on the next line to verify.

Clustering & Load Balancer Settings are used when you want to run EDAB in a Clustered environment.



The screenshot shows the 'EspressDashboard Admin Console' with the 'Setting Info' tab selected. On the left, under 'Select Category:', 'Clustering & Load Balancer Settings' is highlighted. The main area displays 'Load Balancer/Clustering Settings' with the following fields: 'Server Host' (192.168.0.08), 'Server Port Number' (8080), 'Cluster Member List' (a list containing 192.168.0.8:8080, 192.168.0.10:8080, and 192.168.0.45:8080), 'Member Host' (192.168.0.45), and 'Member Port Number' (8080). At the bottom are 'Save Settings', 'Reset', and 'Reset All' buttons.

Clustering & Load Balancing

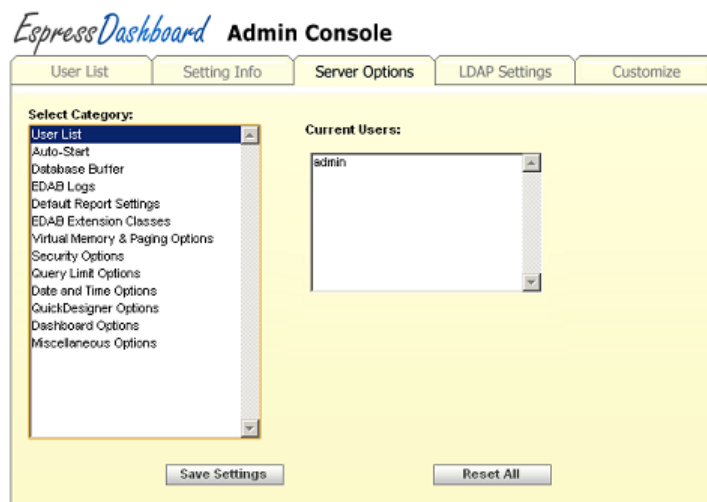
- Server Host:** This is the machinename or ip address of the load balancer.
- Server Port Number:** This is the port number for the load balancer.
- Cluster Member List:** This is the machinename or ip address of the cluster members. The first entry should always be the load balancer.
- Member Host:** This is the machinename or ip address of current node in the cluster.
- Member Port Number:** This is the port number for current node in the cluster.

1.4.1.3. Server Options

The *Server Options* tab contains a number of server configuration options.

The following configuration categories are available from this tab:

- User List:** This category shows you which users are currently logged into EDAB, either viewing or creating reports or charts.



The screenshot shows the 'EspressDashboard Admin Console' with the 'Server Options' tab selected. Under 'Select Category:', 'User List' is highlighted. The 'Current Users:' section shows a list with 'admin'. At the bottom are 'Save Settings' and 'Reset All' buttons.

User list Dialog

- Auto-Start:** These options allow you to specify whether the server should be started automatically and what configuration should be used.

Auto-Start Options

Database Buffer:

These options allow you to set the database connection buffering feature. If a database is being used as the data source for a report or chart, you can choose to buffer both the database connection and the data used for the report or chart using this feature. The number of connections and queries that will be stored, depends on the number of buffers from 1 to 999. The number of buffers is set in the *Buffer Number* dialog.

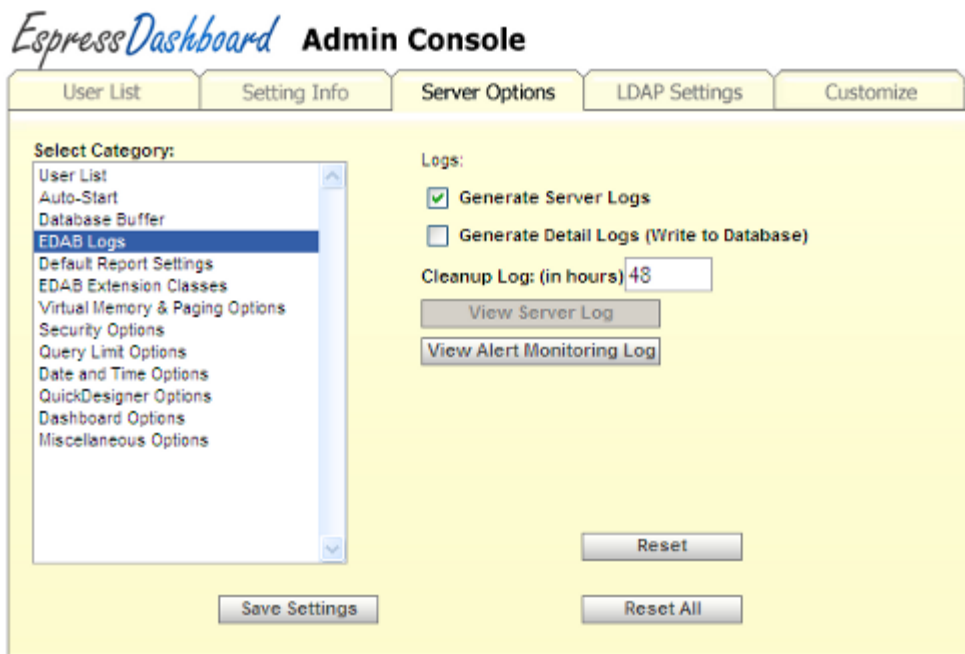
Database Buffer Options

Buffer Number: The number of buffers to use ranging from 1 to 999.

Buffer Cleanup: You can also specify the clean-up interval for database buffers in the Server Monitor. This interval indicates how often the buffers will be flushed, and the queries re-executed when reports/charts are run. You can specify the interval in days, hours, and minutes, by specifying the values in the appropriate dialogs.

Clean Buffers: You can immediately clean the database buffers by clicking the *Clean Buffers* button.

EDAB Logs: EDAB provides two levels of logging activities.



EDAB Logs Options

Generate Server Logs:

The Generate Server Logs option allows you to turn on/off the EDAB Server logs. When this option is enabled, EDAB writes a log file on the server-side that monitor the EDAB installation. EDABServer.log monitors all of the client-server communication for the system, as well as the report generation. Any errors or exceptions are printed.

Generate Detail Logs (Write to Database):

The Generate Detail Logs (Write to Database) option creates an in depth report of all activities related to EDAB. A full list of logged activities are described in Appendix 1.A - EDAB Details Log. To view this log, click on the *View Log* icon. This will pop up a report of the recent EDAB activities. You can adjust the refresh rate of this report by entering a number in minutes for the refresh interval. By default, the report will maintain the last 48 hours of events, but you can adjust this duration by changing the Cleanup value.

View Alert Monitoring Log:

Pops up alert monitoring log. Monitoring log is described in Section 11.4.4 - Monitoring log.

Default Report Settings:

These options allow you to specify global formatting, and font mapping XML files that will apply by default to new reports.

EspressoDashboard Admin Console

User List Setting Info Server Options LDAP Settings Customize

Select Category:

- User List
- Auto-Start
- Database Buffer
- EDAB Logs
- Default Report Settings**
- EDAB Extension Classes
- Virtual Memory & Paging Options
- Security Options
- Query Limit Options
- Date and Time Options
- QuickDesigner Options
- Dashboard Options
- Miscellaneous Options

Default Report Settings:

Default PDF Font Mapping:

Export character encoding:

Default HTML DPI (System default: -1):

Paper Size:

Excel Export Fit Cell: ☐

CSV Newline Delimiter:

Report Setting Options

Default PDF Font Mapping:

The font mappings will be applied by default anytime the report is exported to PDF. For more information about PDF font mapping, see Section 4.1.6.2.1 - PDF Font Mapping.

Export character encoding:

Character encoding that will be used for created reports. Default is *UTF-8*.

Default HTML DPI (System default: -1):

This argument allows you to set the screen resolution that should be used when reports are exported to various format. By default, the system resolution is used, however this can cause some discrepancies when reports are generated on a Linux or Unix server and viewed on a Windows client. Generally in these scenarios, setting the DPI to 96 will produce a consistent export. Setting -1 will use the system default resolution.

Paper Size:

This argument allows you to set the default paper size. Available options are LETTER or A4

Excel Export Fit Cell:

This argument allows you to fit numeric values into single cells when exporting your report to Excel from the Menu Page, DHTML Viewer, or Dashboard. By default, this option is unchecked.

CSV Newline Delimiter:

This option allow you to set a default delimiter for all CSV exports.

EDAB Server Extension Classes:

These options allow you to specify classes that invoke the server extension options for EDAB.

EDAB Server Extension Options

EDAB Listener Class:

The EDAB Listener Class allows you to implement listeners for server-driven report and chart execution. This includes reports and charts run from the Menu Page repository or via Dashboard URL calls. With these listeners in place, reports and charts will be passed to the custom code prior to final execution. For more information about the EDAB Listener, see Section 7.3 - Menu Page Listener.

EDAB Login Listener Class:

The EDAB Login Listener Class allows you to modify the database users connect to when they run reports and charts in the dashboard. For more information please see Section 8.1.4 - Login Listener.

User Security Provider Class:

The User Security Provider class allows you to specify the class which implements the methods to return your users, groups, and security relationships for use with EDAB. For more information about the security provider interface, see Section 9.2.2 - Implementing UserSecurityProvider. Note that you do not have to setup the security provider if you are connecting to an LDAP server.

Virtual Memory/Paging Options:

For reports with large result sets, the paging feature reduces the amount of memory needed by using a virtual memory paging system. This system will write the data to temporary files on the hard drive once the specified memory usage is reached. You can specify the parameters for the paging feature using the options listed below. Please note that the older record set options (**Max Records in Memory**, **Max Characters Per Field**, and **Record Paging Buffer Size**) are no longer available.

The screenshot shows the 'EspressoDashboard Admin Console' interface. At the top, there are tabs for 'User List', 'Setting Info', 'Server Options', 'LDAP Settings', and 'Customize'. The 'Server Options' tab is selected. On the left, a 'Select Category:' list includes options like 'User List', 'Auto-Start', 'Database Buffer', 'EDAB Logs', 'Default Report Settings', 'EDAB Extension Classes', 'Virtual Memory & Paging Options' (which is highlighted), 'Security Options', 'Query Limit Options', 'Date and Time Options', 'QuickDesigner Options', 'Dashboard Options', and 'Miscellaneous Options'. Below this list are 'Save Settings' and 'Reset All' buttons. The main area on the right is titled 'Virtual Memory/Paging Options:' and contains the following settings:

- Enable Paging:** A checkbox that is currently unchecked.
- Paging Threshold: (in MB)** A text input field containing '-1'.
- Max Field Size: (in Byte)** A text input field containing '500'.
- Paging Buffer Size: (in MB)** A text input field containing '20'.
- Total Paging Buffer Size: (in MB)** A text input field containing '256'.
- Cleanup Paging Buffer: (in hour)** A text input field containing '12'.

Below these settings are 'Reset' and 'Reset All' buttons.

Virtual Memory/Paging Options

Paging Threshold:

This argument specifies when the paging feature will be activated. If this argument is set to “-1” then data will never be paged. This value is independent of Page Buffer Size option. If this value is larger than the Paging Buffer Size, then the system will begin paging when the threshold is reached, not when the Paging Buffer Size is reached. Therefore each report or chart might use more memory than the amount specified in the Page Buffer Size. If this value is equal to or lower than the Paging Buffer Size, the paging feature will begin when the Paging Buffer Size is reached. This value is set in Megabytes.

Max Field Size:

This argument specifies the expected maximum field size for large field types such as varchars. Fields that are larger than this value are **not** necessarily truncated. The behavior depends on the current memory usage in relation to the Total Paging Buffer Size allowed. As the amount of available memory decreases the amount of data stored for these large fields will decrease until it reaches this value. For example, a user is running a report with a field size that is larger than the specified Max Field Size value. When the server is not busy and there is ample memory available, the report will generate the full field without any truncation. However, when the server is heavily loaded and the available memory is near 0,

the field in the report will be truncated to the Max Field Size. For numeric, boolean, date, time, and character datatypes, the data will never be truncated. For those database fields that are defined with a size limit smaller than the Max Field Size (e.g. Max Field Size = 500, but the field is defined as `varchar(20)`), the database limit will be used in place of the Max Field Size. This value is set in Bytes.

Paging Buffer Size:

This argument allows you to set the amount of memory that each report or chart will use when the paging feature is invoked. The size of the buffer affects performance. The larger the buffer size, the faster the report or chart is generated, but more memory is used. When the amount of total memory used by the paging system approaches the **Total Paging Buffer Size**, the Paging Buffer Size provided to new reports will begin to diminish in order to avoid exceeding the specified total amount. This value is set in Megabytes.

Total Paging Buffer Size:

This argument allows you to set the total amount of memory used by the paging feature across all reports and charts. The memory allocated to each report will diminish as the memory usage approaches this value. This value is set in Megabytes.

Cleanup Paging Buffer:

This argument allows you to specify the cleanup interval for the temporary files used in the paging feature. Typically, these files are deleted when the reports are finished, however under rare circumstances (e.g. server crashed), these files are left on the server. The cleanup thread takes care of these rare scenarios. This value is set in hours.

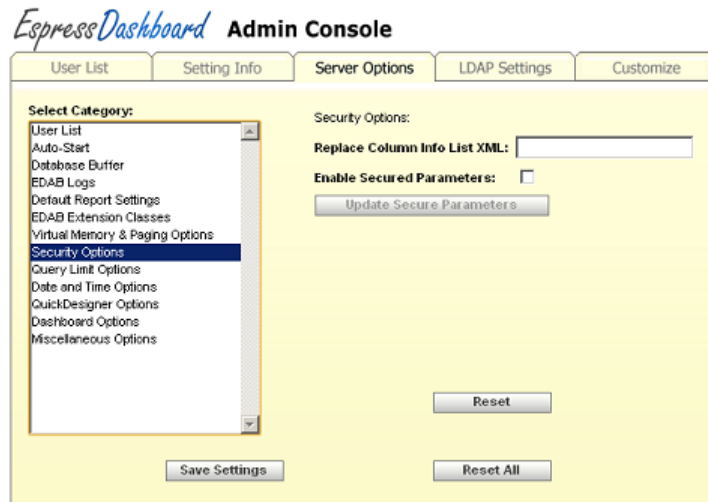
Best Practice:

Generally, if you have a lot of large reports, make sure to enable this feature by setting an adequate paging threshold. Specify the options based on the amount of memory available to your application server. For example, if you are running Tomcat with 512 MB memory, start with 256 MB for the Total Paging Buffer Size and around 10-20 MB for the Paging Threshold. This way, between 10-20 users can be running the large reports without crippling the performance. The reason you should not use 100% of the Tomcat memory is because there are other components that will require memory as well. For instance, the POI libraries will require additional memory when exporting a report to Excel to generate the actual xls file. Each environment will be different, the number of large reports, the size of the result sets, and the number of concurrent users will all affect the optimized option settings. However,

er, as a general starting point, shoot for around 50% of application server memory for the Total Paging Buffer Size. Then set the Paging Buffer Size to be around 2-10% of the Total Paging Buffer Size. The Paging Threshold should not be much greater than the Paging Buffer Size.

Security Options:

EDAB also provides a set of sophisticated security options.



Security Options

Replace Column Info List XML:

This argument allows you to turn on the automatic decryption feature in QuickDesigners and menu page. The XML file must specify all the information for the column to be decrypted, including database information, table name, and column name. You can specify either an absolute path to the XML file or a relative path to the EDAB install directory (i.e. `help/examples/DataDecryption/QBReplaceColumnInfoList.xml`). You can find more details as well as an example for this feature in Section 3.2.2.1.2 - Querying Encrypted Data.

Enable Secured Parameters:

This argument allows you to turn on the secured parameters feature. This feature activates a global security structure allowing you to specify the segments of data that each security level is entitled to view. You can find more details as well as interface for configuring Secured Parameters in Section 1.4.1.3.1 - Secured Parameter.

Query Limit Options:

These options allow you to set strict limits to the all query execution.

The screenshot shows the 'EspressoDashboard Admin Console' with the 'Server Options' tab selected. On the left, a 'Select Category:' list has 'Query Limit Options' highlighted. The main area displays 'Query Limit Options' with two input fields: 'Max Query Records: (Disable: -1)' and 'Query Timeout (Sec): (Disable: -1)', both containing the value '-1'. At the bottom are 'Save Settings', 'Reset', and 'Reset All' buttons.

Query Limit Options

Max Query Records:

This argument allows you to set a maximum limit for the number of records the server will retrieve from a database when executing a query. Once the maximum is reached, the server will stop running the query. If this argument is set to **-1** then no query record limit will be applied.

Query Timeout (Sec):

This argument allows you to specify a timeout interval in seconds for report/chart queries. Once the query execution time passes the timeout argument, EDAB will abort the query. This feature prevents users from accidentally creating run-away queries.

Date and Time Options:

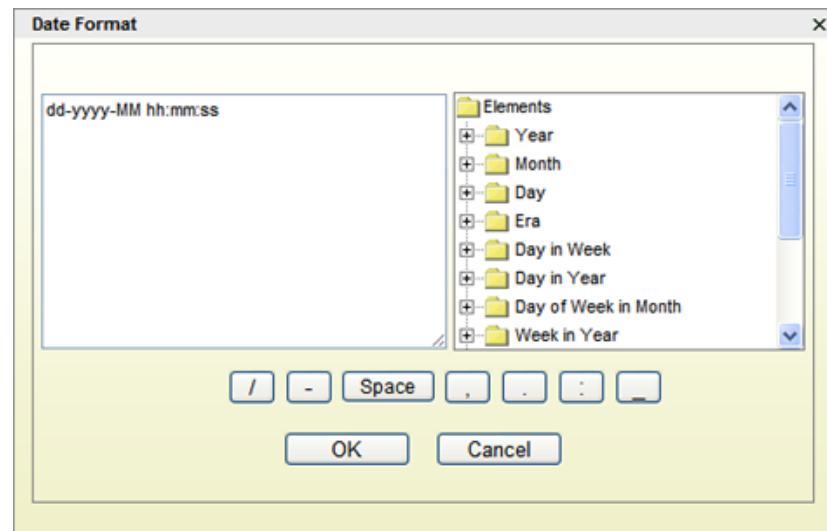
This section allows you to specify a default custom date, time, and timestamp format used when parameters are mapped to a column in a database. When the *Custom Date Format* option is checked, this is the default format presented to the user.

The screenshot shows the 'EspressoDashboard Admin Console' with the 'Server Options' tab selected. On the left, a 'Select Category:' list has 'Date and Time Options' highlighted. The main area displays 'Default Date Format:', 'Default Time Format:', and 'Default Timestamp Format:' with input fields and 'Build' buttons. The input fields contain 'yyyy-MM-dd', 'hh:mm:ss', and 'dd-yyyy-MM hh:mm:ss' respectively. At the bottom are 'Save Settings', 'Reset', and 'Reset All' buttons.

Date and Time Options

Click the *Build buttons* to build the custom format. The date/time representations are listed on the right and optional spacers and symbols are shown as a

collection of buttons on the bottom. Once you have finished creating the format, click *OK* to save.

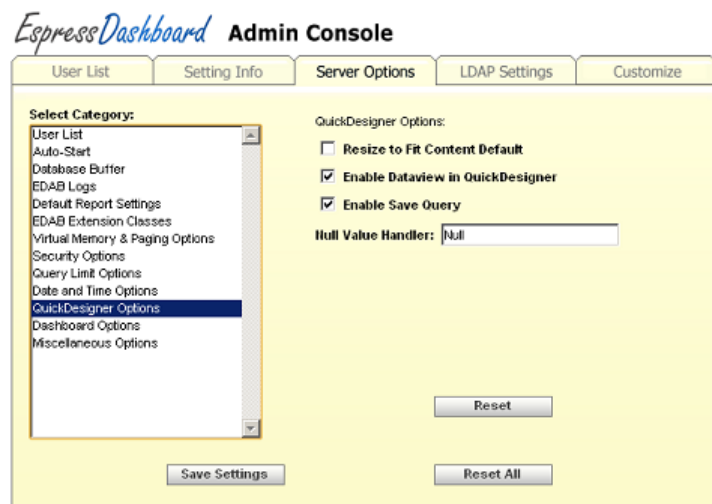


Custom Timestamp Dialog

QuickDesigner Options:

Please note, these settings apply only to the old QuickDesigner (from version 6.6). To see how to enable the old QuickDesigner, visit Section 1.4.1.2 - Setting Info.

This section provides several defaults for the QuickDesigner.



QuickDesigner Options

Resize to Fit Content Default:

While resize to fit content can be changed in any report within the QuickDesigner, this option allows you to set a default behavior. If you enable this option, all cells in the QuickDesigner will have Resize to Fit Content enabled by default.

Enable Dataview in QuickDesigner:

This option allows you to disable the data views in quick designers such that the user can only make ad hoc reports based on data view queries, but not from da-

ta views. This feature can prevent users from bypassing the data view secured parameters.

Enable Save Query:

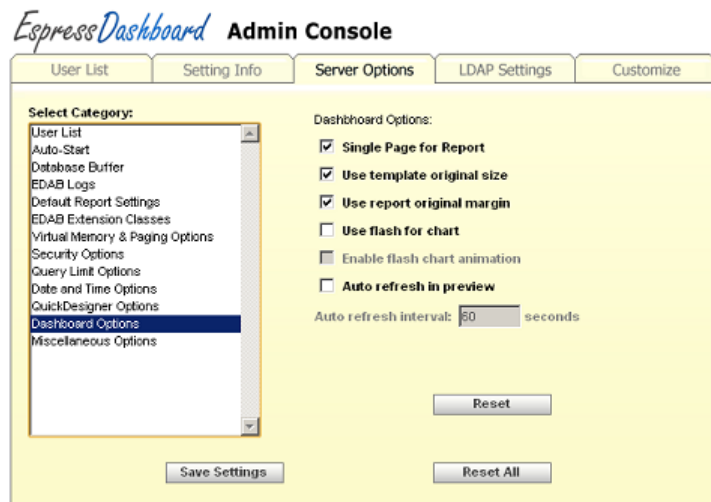
To prevent users from creating too many custom dataview queries, you can disable this option so that users are forced to work from the list of dataview queries that you have provided.

Null Value Handler:

This option allows you to control the display for null data in the QuickDesigner. The value can be any string that does not contain spaces. To insert a space into the null data handler, use the word *SPACE* in uppercase. For example: - *SPACE*- will replace all null values in the QuickDesigner with - -.

Dashboard Options:

This section provides several defaults for the Dashboard Builder.



Dashboard Options

Single page for report:

While the single-page or multi-page layout can be changed in any report within the Dashboard Builder, this option allows you to set a default behavior. If you enable this option, all reports in the Dashboard Builder will have the single-page layout enabled by default.

Use template original size:

This option allows you to configure the original size of the template to be used by default.

Use report original margin:

This option allows you to configure the original margin of the report to be used by default.

Use Flash for charts:

This option allows you to specify whether Flash should be used in charts by default.

Enable Flash chart animation:

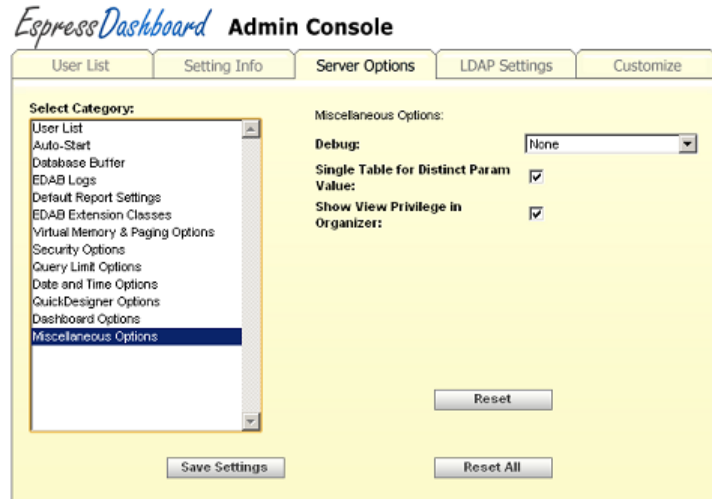
This option allows you to specify whether Flash animation should be turned on in charts by default.

Auto refresh in preview:

This option allows you to specify whether the previewed dashboard should be refreshed automatically, and if it should be refreshed automatically, you can configure the time interval for refreshing the dashboard preview.

Miscellaneous Options:

This section contains a number of general options available for EDAB.



Miscellaneous Options

Debug:

When setting up the EDAB system, you might need to enable the debug option for some components to troubleshoot problems. This option makes it easy to enable the debug statements for Secure Parameter, Replace Column, or Virtual Memory features. Once you restart the server, whenever the feature is invoked, informative outputs will be printed to the system console.

Single Table for Distinct Param Value:

When you enable this argument, the parameter dialog for parameterized charts and reports will be drawn differently. When you map the parameters to a database column, the distinct list will be drawn by running a select distinct on only the mapped field from the table that contains that field. The default behavior (without this argument) will use the joins and conditions in the original query to constrain the distinct parameter list.

Show View Privilege in Organizer:

By default, regular users (with designers role) will not be able to configure file privileges, but they can view them. Disable this option if you do not want users to see the privilege list for organizer files and folders.

Once you've specified any changes to the options on this tab, click the *Save Settings* button to save the changes.



Note

The changes will not take effect, nor will your changes display in the *Server Options* tab until the server is re-started.

These options can also be directly entered into a Server Command text file. For more information for the Server Commands, please see Appendix 1.B - Server Commands.

1.4.1.3.1. Secured Parameter

EDAB contains a feature which allows you to filter data based on a user's security level within the QuickDesigners, Dashboard Builder, Dashboard Viewer and Published Files.

To set up and configure this feature, go to *Server Options* in the *Administration Console*, select the *Security Options* category, *Enable Secured Parameters* (if it is not checked), and click the *Update Secure Parameters* button. This will take you to the following dialog.

Secured Parameter Dialog

By default, there will be a secured parameter set up for *Customers.Region* in the HSQL database. You can add more secured parameters at any time by using the *Add* or *CopyDB* option at any time. The *Copy DB* option will copy the database connection information so you don't have to type it in again. Once you have created a new secured parameter or selected an existing one, the options on the right will enable. Fill in each of the options.

Parameter Name:	This is just a name for the Secured Parameter, you can give it any name.
Database Type:	This is a syntax assistance drop down box. Selecting the database will place default URL syntax and default JDBC Driver in the corresponding fields. You can then go into the URL and fill in the server specific information.
Database URL:	Specify the URL for the database you would like to apply the Secured Parameter on. Use the Database Type drop down to assist in the syntax.
Database Driver:	Specify the driver or use the Database Type drop down to use the default database driver.

Table Name	Specify the table you would like to apply the Secured Parameter.
Column Name:	Specify the column you would like to apply the Secured Parameter.
Data Type:	This field will automatically be configured and is only displayed for verification.
Allow Multiple Values:	This will determine if security levels will each be able to access only one value or multiple values. If you select multiple values, the Values drop down box will become a list box allowing you to select multiple values.
Security Level:	Go through the security levels on the system and specify a value or values for each security level. The first time you select a security level, you will need to click the <i>Update Values List</i> button to retrieve a list of available values for that column.
Values:	The <i>Update Values List</i> button will launch a prompt for username and password to this database the first time it is run. Once you have supplied the information, it will retrieve a list of available values for the specified column. You can map a security level to one or more (if multiple is checked) values by clicking and highlighting the row of data. You can also use the <i>Grant All</i> option which allows that security level to view all data in that column.

Once you have set up mappings for secured parameters, click the *Done* button. That should bring you back to the Admin Console. Click the *Save Settings* button. You will be reminded to restart the EDAB server. Confirm the message, go back to the EDAB start page, shut down the server and then start it again.

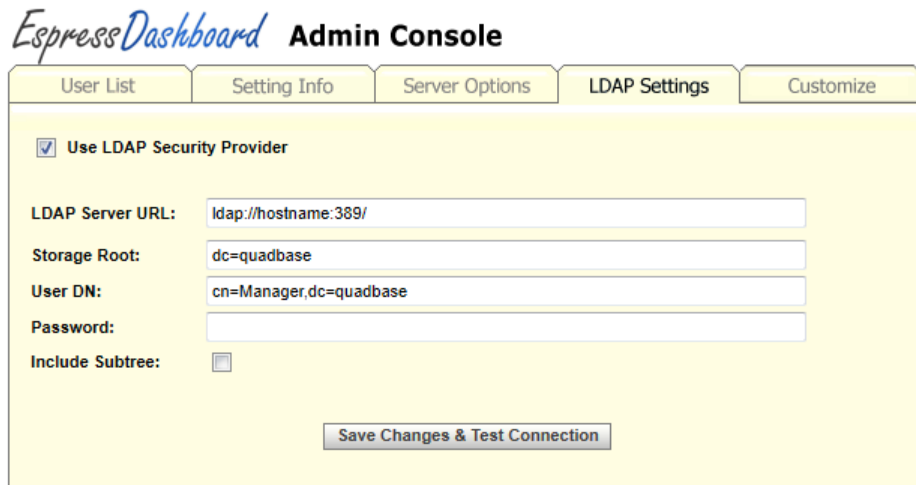
After the server has been restarted, the secured parameters settings will carry through to most components in EDAB. To illustrate the concept, let us walk through a simple example.

Suppose you have a secured parameter set up for the *REGION* column from the *CUSTOMERS* table in the Woodview sample database. It specifies the behavior of the filter for six different security levels. For level *East* the query will be automatically filtered so that only rows where region is *East* will be returned. For level *South* the query will be automatically filtered so that only rows where region is *South* will be returned. For the level *Midwest* only rows where the region is *Midwest* will be returned. For the level *West* only rows for *West* region will be returned. The *Allow Multiple Values* attribute allows you to indicate whether multiple parameter values can be specified for the filter, like mapping the *EastSouth* security level to both *East* and *South*. For level *Executive*, *Grant All* is selected and all values from the region column will be available.

With the options set up, any time a user creates or runs a query, data view, or data view query in the QuickDesigners that includes the *CUSTOMERS* table, the filter defined for their security level will be applied. These settings reside on the server, and are not saved into the report, chart, or dashboard. Therefore if the user then creates and saves a report in QuickDesigner Reports and transfers the report to another machine, the security will not be apply. Instead, it will follow the security settings sepecified on the new Server.

1.4.1.4. LDAP Settings

The *LDAP Settings* tab in the Admin Console provides options that allow you to configure EDAB to retrieve users from an LDAP server. To set the LDAP connection, select *Use LDAP Security Provider* option. The following dialog should open.



The image shows the 'LDAP Settings' tab in the 'EspressDashboard Admin Console'. The 'Use LDAP Security Provider' checkbox is checked. The 'LDAP Server URL' field contains 'ldap://hostname:389/'. The 'Storage Root' field contains 'dc=quadbase'. The 'User DN' field contains 'cn=Manager,dc=quadbase'. The 'Password' field is empty. The 'Include Subtree' checkbox is unchecked. A 'Save Changes & Test Connection' button is at the bottom.

LDAP Settings Dialog

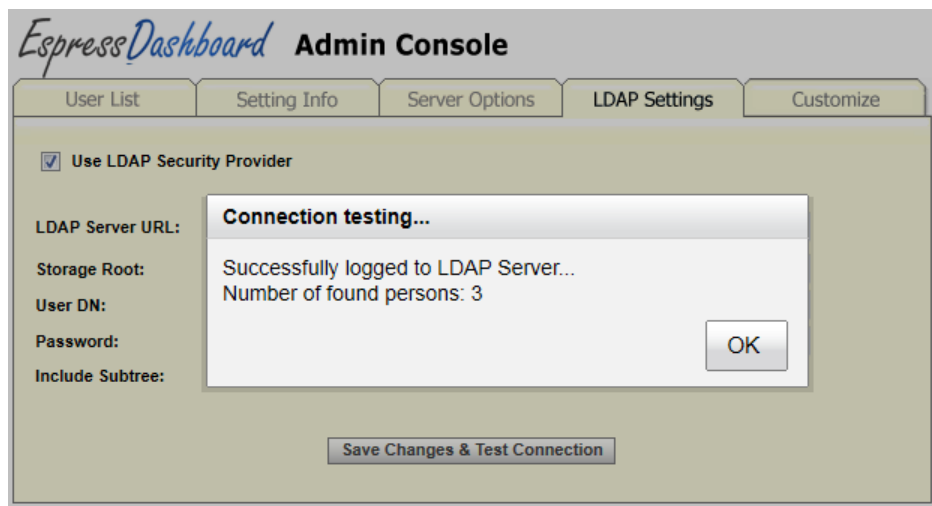
The above dialog contains example LDAP settings of Microsoft Active Directory.

To the *Storage Root* field, enter the distinguished name of the AD node that contains all the users you want to import.

By default, only the given AD node (i.e. the *Storage Root* node) will be searched for users, excluding it's sub-nodes. If you want to search for users in the *storage root* node and all of it's sub-nodes, select the *Include Subtree* option.

In order to connect with EDAB you will need to have an user in your LDAP server that can read from the Active Directory. Enter the user's distinguished name in to the *User DN* field and then type the user's password to the *Password* field.

Click the *Save Changes & Test Connection* button. If you set the LDAP correctly, the following dialog will show up.



The image shows the 'LDAP Settings' tab in the 'EspressDashboard Admin Console' with a 'Connection testing...' dialog box overlaid. The dialog box contains the text 'Successfully logged to LDAP Server...' and 'Number of found persons: 3'. There is an 'OK' button in the dialog box. The 'Save Changes & Test Connection' button is visible at the bottom of the settings dialog.

Click *OK*. The LDAP users were imported, but haven't been activated yet. To activate the users, click on the *Users List* tab.

EspressoDashboard Admin Console

User List | Setting Info | Server Options | LDAP Settings | Customize

Users:

- admin
- patrik
- leos
- alois

Add New User

Groups:

Add Group | Edit Group | Delete Group

Set Security Levels | View User/Group Relationships

Import User List from Text File:

Choose File | no file selected | [learn more...](#)

Legend: Administrator, Designer, Viewer, LDAP user without access, Group

There are some users marked as *LDAP users without access* in the *Users* list (user roles are distinguished by their colors - there is a legend for that below the AdminConsole).

Select one or more LDAP users without access. The *Insert LDAP User(s)* button will show up'. Click on it.



Note

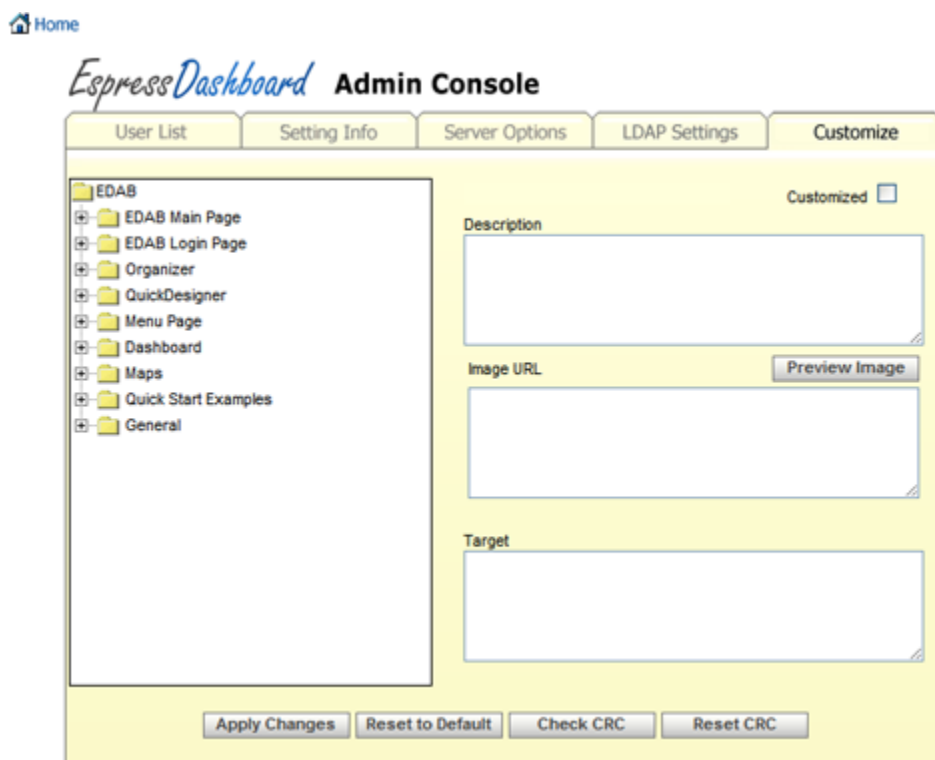
If you include a non-LDAP user in the selection, the 'Insert LDAP User(s)' button will disappear.

A new window dialog will show up allowing you to select user role for the selected user(s). To learn more about user roles, visit Section 1.4.1.1 - User List. Select a user role and click *Ok*.

The users will not be marked as *LDAP Users without access* any more. Their color changed from dark red to light red (if you gave them admin privileges), blue (if they are designers) or black (if you gave them viewer rights). They are valid EDAB users and they can start using the EDAB now.

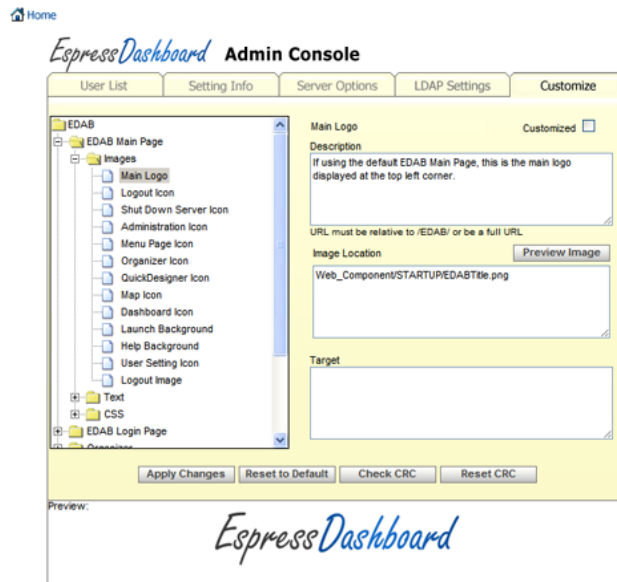
1.4.1.5. Customization

EDAB can be customized to maintain a consistent look and feel for your web application. The customization tab allows the administrator to change the appearance of many components in EDAB. When customizing, it is recommended that you save your images and files under different names rather than overwriting existing files in the EDAB. This way, you will not have to worry about future updates overwriting your changes.

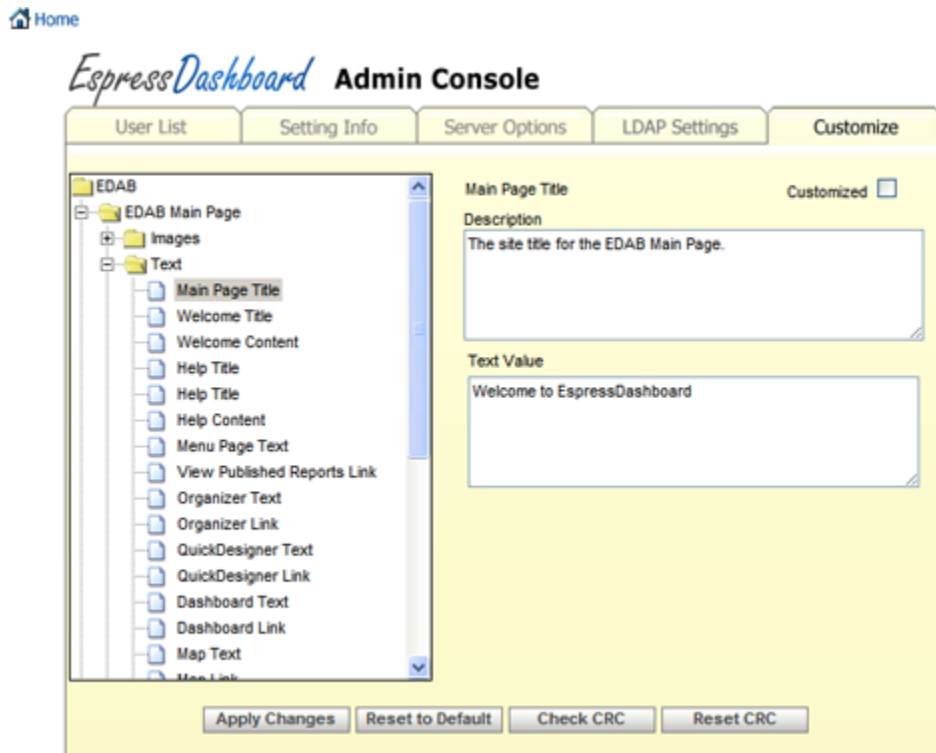


The Customization Interface

The left window provides a tree of the different elements that can be customized. The customizable elements are broken up into sections based on the components in the EDAB application. Most sections contain three types of elements: images, text, and CSS files. By clicking on an image element, the page will display the description, an image URL, and a target URL if available. You can change the image file by providing a different URL. Clicking on the *Preview* button will display the image at the bottom of the screen. The preview window is only available for images, clicking on a text or CSS element will disable the preview window. The target URL is only available for certain elements. If the selected element allows for a target to be set, this text box will appear beneath the image URL box. Enter a URL in this window if you would like to add a hyperlink for this image. If this link is provided, when users click on the image, the browser will be redirected to the target URL.

*Preview Image*

If you select a text node, the page will display the description and a text value. Change the text value to adjust the message or link for this element. The text will be used directly in the EDAB application, so you can enter any html tags in the tool. Certain elements also contain a link. You can utilize this to redirect the user to another URL if you are not using the default components. For example, if you are using a customized Menu Page, you can modify the link for the Published Dashboards element to point to your customized page. Editing the text value also gives you the ability to add even more CSS to the page.

*Customizing Text*

Changing the CSS file is quite straightforward, simply enter the url for the new CSS file. It is recommended that you make a copy of the original CSS file and edit the copied version so that future updates will not overwrite your

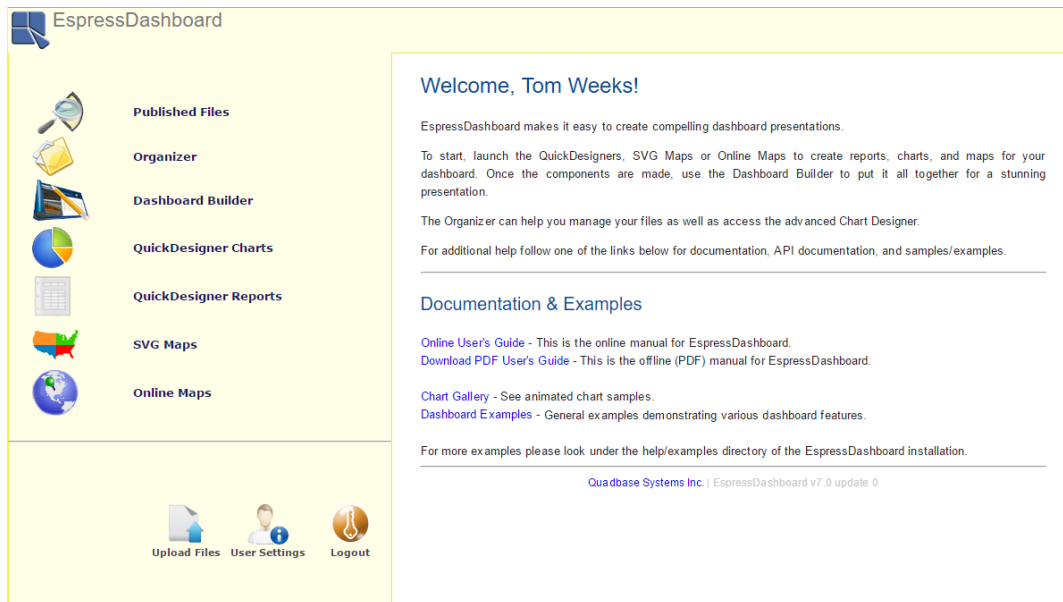
changes. As mentioned earlier, you can specify new CSS elements in customized text elements and enter the style in your CSS file.

When you change the values for any element, the *Customized* box for that element will automatically check. If you wish to revert back to the default value for this element, simply uncheck the *Customized* box and choose *Ok* from the pop up dialog. If you wish to reset all elements to their default values, click on the *Reset to Default* icon at the bottom.

Once you are finished with your changes, make sure to click *Apply Changes* to save all changes. To see the changes appear in the EDAB application, you will need to shut down the EDAB Server and restart it.

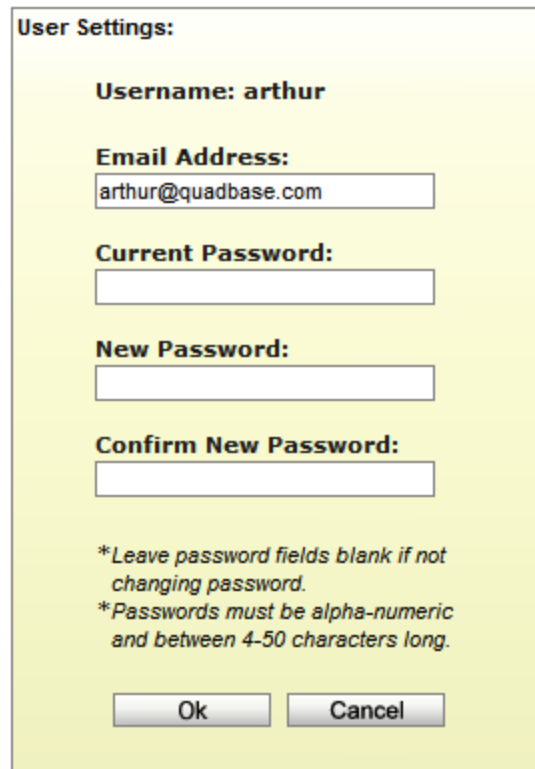
1.4.2. User Settings

In addition to the administrator, users can modify some of their own information/settings in EDAB. When a user other than the administrator logs in in the Start page, the *User Settings* link will appear at the bottom on the left side.



User Logged into EDAB Start Page

When the link is clicked, a new window will open that allows the user to modify the email address and password associated with his/her account.



The image shows a 'User Settings' dialog box with a yellow background. It contains the following fields and labels: 'Username: arthur', 'Email Address: arthur@quadbase.com', 'Current Password:', 'New Password:', and 'Confirm New Password:'. Each label is followed by a text input field. At the bottom, there are 'Ok' and 'Cancel' buttons. Below the buttons, there are two lines of italicized text: '*Leave password fields blank if not changing password.' and '*Passwords must be alpha-numeric and between 4-50 characters long.'

User Settings Dialog

After making the appropriate changes, clicking *Ok* will save the user changes and close the window.

1.A. EDAB Details Log

The EDAB Details Log Report maintains a record of many key events that take place. Administrators can use this information to track what changes were made, which user made the changes, and when the changes took place. This feature is also useful for troubleshooting issues and backtracking through events.

Here is a list of all events recorded in the EDAB Detailed Log Report:

EDAB Events

- Login
- Timeout
- Logout
- Set Webroot
- Set Server Name
- Set Protocol
- Set Port Number
- Set Web Server Port
- Set JSP Default Directory
- Set Servlet Context
- Set SMTP Hostname

- Repair Broken Links
- Replace Directory
- Add New Node
- Remove Node
- Copy Node
- Move Node
- Set Name
- Set Node Description
- Set Node Filename
- Update Node Owner Id
- Add Schedule Task
- Add Report Schedule Task
- Add Chart Schedule Task
- Remove Schedule Task
- Modify Schedule Task
- Run Schedule Task
- Add Data Registry
- Remove Data Registry
- Set Data Registry Permission
- Delete Data Registry File
- Set User Data File
- Set User Permission
- Set Group Permission
- Insert Security Level
- Remove Security Level
- Set Security Levels
- Set Node Permission
- Set File Permission
- Set Data Registry Invisible Node
- Table Setvalue
- Table Add Item
- Table Remove Item
- Table Create Item

- Urlmap Setvalue
- Urlmap Add Item
- Urlmap Insert Item
- Urlmap Move Item
- Urlmap Remove Item
- Urlmap Create Item
- Schedule Export
- Write Dataview
- Write Queryfile
- Menu Viewpage
- Menu Login
- Open Report
- Open Chart
- Export
- Set Cleanup Log Interval
- Get Cleanup Log Interval

Language Events

- User Language

Misc Events

- Get OS Name
- Data Registry

Report & Chart Designer Events

- Default Files
- Export
- Pageviewerhtml
- Stream
- Stream Read Only
- Pageviewer Stream
- Load View Page Slave
- Load View Page Export
- File Info
- License Info

- Pageviewer File Info
- Fileinfo Root
- XML Report
- XML Report Data
- XML Data Parser
- XML Data Sheet
- XML Global Format
- XML Font Mapping
- Data File
- EJB Access
- EJB Metadata
- EJB Home Method Access
- XML Verify
- SQL
- Test Db Connection
- Url
- Rename
- Replace
- Export Barcode
- Server Directory
- Db Distinct
- Db First Value
- Db Param Meta
- Db Param Function Meta
- Db Param Type
- Db Query Meta
- Data File Meta

Query Builder Events

- Qb Product Name
- Qb Numeric Function
- Qb System Function
- Qb String Function

- Qb Time Date Function
- Qb Table List
- Qb Table Info
- Qb SQL

Chart Generator Events

- Chart Export
- Filter Data File
- XML Chart
- XML Template
- Creat Chart
- Filter PPSQL
- Img Export
- Cht Export
- Delete
- Delete Temp File
- Release File
- Copy File
- Working Directory
- Outputstream Initialize
- Outputstream Write
- Outputstream Close
- Classfile

Here is a list of the possible products represented in the log.

Loginpage

Adminpage

Menupage

Dashboard

Organizer

Report

Chart

Scheduler

Lookupservlet

Dhtmlviewer

Dashviewer

Soap

Designer

1.B. Server Commands

The configurations specified in the Server Options tab can also be set in a configuration text file. Under the `<EDABInstallDir>/WEB-INF/classes` directory there is a file called `QB.properties` that contains several arguments that control server settings. The `QB.properties` is a INI file. Server commands belong to the "ServerCommands" property. Arguments in the text file assume the format of a dash "-" followed by a command. Arguments should be added in a single line (starting with `ServerCommands=`) in the file separated by spaces.

-log:	When you type the -log argument, this turns on the basic server logging. This has the same effect as enabling the option in the Admin Console.
-writeLogToDatabase:	This turns on the detailed server logging feature. This has the same effect as enabling the <i>Generate Detail Logs</i> option in the Admin Console.
-cleanupIntervalForLog:hh:	This is the cleanup interval for the detail logs. The value following the colon is specified in Hours.
-recordLimit:nn:	This argument allows you to set a maximum limit for the number of records the server will retrieve from a database when executing a query. This has the same effect as setting the record limit in the Admin Console
-queryTimeout:sss:	This argument allows you to specify a timeout interval in seconds for report/chart queries. This has the same effect as setting the query timeout in the Admin Console.
-DBBuffer:nnn:	This allows you to set the number of database buffers for the server to use. This argument is the same as the buffering feature available in the Admin Console. The argument allows you to specify the number of buffers from 1 to 999.
-DBCleanAll:ddhhmm:	<p>This allows you to set the clean up interval for the database buffers. This argument is the same as the clean up feature in the Admin Console. The value of <code>ddhhmm</code> means "dd" days, "hh" hours and "mm" minutes. You can also use omitted format, meaning that the server will translate the value with the assumption that the omitted digits are the higher digits. For Example:</p> <p>DBCleanAll:101010 means clean buffer every 10 days, 10 hours and 10 minutes</p> <p>DBCleanAll:1010 means clean buffer every 10 hours and 10 minutes</p> <p>DBCleanAll:10 means clean buffer every 10 minutes</p>
-RequireLogin:	This argument is used in conjunction with the <code>QbDesignerPassword</code> argument to apply security to Report Designer and Chart Designer when they are launched via the API. Normally when Report or Chart Designer is called via the API there is no user authentication. This allows users to apply their own authentication to the programs, but can also allow unauthorized users access to the server. To prevent this, users can turn on this argument (values are true/false) to force

	authentication for Report and Chart Designer when they are called from the API. This option cannot be set in the Admin Console
-QbDesignerPassword:	This argument allows you to set the password to use when the <i>-RequireLogin</i> argument is turned on. The password specified here needs to be passed in via a method call when calling Report or Chart Designer via the API. This option cannot be set in the Admin Console.
-globalFormat:<xmlfile>:	This argument allows you to supply a global format XML file to set the default look and feel of report elements. This argument is the same as the default global formatting option in the Admin Console. For this argument specify a file path to the XML file relative to the EDAB install directory i.e. (<i>-globalFormat:ReportFiles/FormatFile.xml</i>).
-fontMapping:<xmlfile>:	This argument allows you to supply a font mapping XML file to set the default font mapping for PDF export. This argument is the same as the default font mapping option in the Admin Console. For this argument specify a file path to the XML file relative to the EDAB install directory i.e. (<i>-fontMapping:Templates/FontFile.xml</i>).
-htmlDpi:nn:	This argument allows you to hard-code the screen resolution that should be used when reports are exported to DHTML format. This argument is the same as the HTML DPI option in the Admin Console.
-ListenerManagerClass:<classfile>:	This argument allows you to specify a class for the EDAB listener mechanism. This argument is the same as the EDAB listener class option in the Admin Console.
-LoginListenerClass:<classfile>:	This argument allows you to specify a class for the EDAB login listener mechanism. This argument is the same as the EDAB login listener class option in the Admin Console.
-ScheduleCallBackClass:<classfile>:	This argument allows you to specify a class for the scheduler callback mechanism. This argument is the same as the scheduler callback class option in the Admin Console.
-UserSecurityProviderClass:<classfile>:	This argument allows you to specify a class file for the user security provider interface. This argument is the same as the user security provider class option in the Admin Console.
-singleTableForDistinctParamValue:	When you use <i>-singleTableForDistinctParamValue</i> argument, the parameter dialog for parameterized reports and charts will be drawn using a select distinct on a single table. This argument is the same as the <i>Single Table for Distinct Param Value</i> option in the Admin Console.
-showViewPrivilege:true/false:	This option determines if non-admin users will be able to view the file and folder privilege in the organizer. By default this option is true.
-schedulerThreadLimit:nn:	This argument allows you to set the maximum number of schedule jobs that can run concurrently. This argument is the same as the max scheduler threads option in the Admin Console.
-runMissedScheduleJob:	When you use the <i>-runMissedScheduleJob</i> argument, any schedule jobs that were missed due to server down-time will be run when the

	server is re-started. This argument is the same as the re-run missed schedule jobs in the Admin Console.
-batchDeleteForScheduler:TRUE/ FALSE:	This argument controls the process by which schedule job information is removed from the EDAB database for expiring schedule jobs. Setting this option to true can improve schedule performance if you have large numbers of jobs that will expire at the same time. The effects of this feature may vary depending on the EDAB database platform. This option cannot be set in the Admin Console.
-xmlEncoding:encoding:	This argument allows you to specify the encoding that EDAB should use when writing XML. This includes data registry files, XML report and chart templates, XML exports, and XML global formatting, and font mapping files. This parameter needs to be set both in the server and in the Organizer. For more information about XML encoding, see Section 10.1.2.4 - XML Encoding.
QuickDesignerSecuredParameters:xmlfile:	This argument allows you to supply a data view security XML file to enable/apply security filters to data views and data view queries. You can specify either an absolute path to the XML file or a relative path to the EDAB install directory i.e. (-QuickDesignerSecuredParameters:qDesigner/WoodviewParameters.xml). For more information about secured parameters, see Section 1.4.1.3.1 - Secured Parameter.
-debug:DVW_SEC_PARAM:	Use this argument to troubleshoot the <i>-QuickDesignerSecuredParameters</i> option. This debug statement will print multiple debug statements to the server console specifying whether there is a match for the column listed in the xml file. If you specify multiple debug options, only the last one will take effect.
-disableDataViewInQuickDesigner:	This argument allows you to disable the data views in quick designers such that the user can only make ad hoc reports based on data view queries, but not from data views. This feature can prevent users from bypassing the data view secured parameters.
-disableSaveQueryInQuickDesigner:	This argument allows you to disable the save query option in quick designers. This feature prevents users from creating and saving too many queries in the data registry.
-ReplaceColumnInfoList:xmlfile:	This argument allows you to turn on the automatic decryption feature in QuickDesigners and Menu Page. The XML file must specify all the information for the column to be decrypted, including database information, table name, and column name. You can specify either an absolute path to the XML file or a relative path to the EDAB install directory i.e. (-ReplaceColumnInfoList:help/examples/DataDecryption/QBReplaceColumnInfoList.xml). You can find more details as well as an example for this feature in Section 3.2.2.1.2 - Querying Encrypted Data.
-debug:REPLACE_SQL_COLUMN:	Use this argument to troubleshoot the <i>-ReplaceColumnInfoList</i> option. This debug statement will print multiple debug statements to the server console whether there is a match for the column listed in the xml file. If you specify multiple debug options, only the last one will take effect.
-debug:MEMORY_PAGING:	Use this argument to troubleshoot the virtual memory/paging feature. This debug statement will print debug statements to the serv-

	er console regarding the amount of memory used, and the paging process. If you specify multiple debug options, only the last one will take effect.
-ExcelExportFitCell:true/false:	Set this argument to true if you want to fit numeric values into single cells when exporting your report to Excel from the Dashboard. By default, this option is false.
-paperSize:LETTER/A4:	Set this argument to define the default paper size. If this option is not set, reports will default to Letter size.
-ExportNewlineDelimiter:windows/mac/others/system:	Set this argument to specify a default newline delimiter for CSV export. Use windows for any Windows version, it will use \n\r for the delimiter. Use mac for any version of Mac, it will use \r for the delimiter. Use others for any Unix/Linux variants, it will use \n for the delimiter. You can also use system to allow the server to pick up the system default value.
-QDResizeToFitContentDefault:true/false:	<p>Please note, this command applies only to the old QuickDesigner (from version 6.6). To see how to enable the old QuickDesigner, visit Section 1.4.1.2 - Setting Info.</p> <p>When you set this argument to true, reports created in QuickDesigner will automatically have the <i>Resize to Fit Content option</i> turned on for all columns. By default this option is turned off.</p>
-QDNullValueHandler:"<NullValue>":	<p>Please note, this command applies only to the old QuickDesigner (from version 6.6). To see how to enable the old QuickDesigner, visit Section 1.4.1.2 - Setting Info.</p> <p>This argument allows you to control the display for null data in the QuickDesigner. <NullValue> can be any string that does not contain spaces. To insert a space into the null data handler, use the word "SPACE" in uppercase. For example:</p> <pre>-QDNullValueHandler: "-SPACE- "</pre> <p>will replace all null values in the QuickDesigner with "- -".</p>

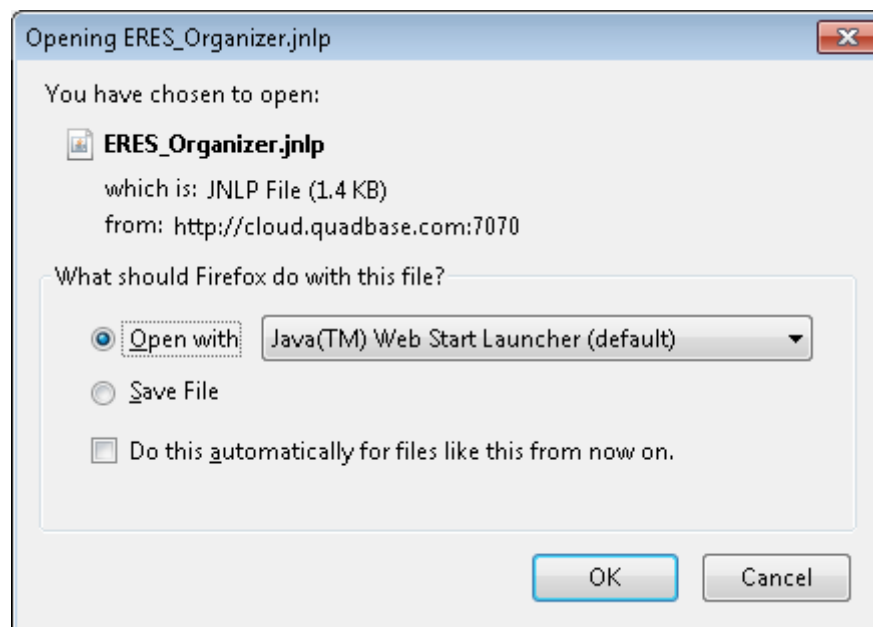
Chapter 2. Organizer

2.1. Using the Organizer

The EspressoDashboard Organizer is a powerful graphical user interface that allows users to design charts and reports and organize them with other files in virtual folders. The Organizer contents are automatically published to the Menu page and users can automatically generate URLs to run reports and charts. In addition, schedule and archive jobs are created and maintained through the Organizer. The Organizer also includes the user permissions and security level setting options.

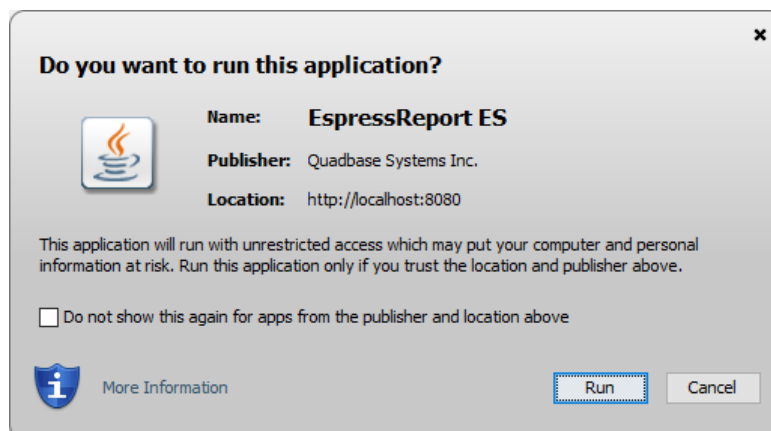
2.1.1. Starting the Organizer

To start the Organizer, click the link labeled *Organizer* in the Start page. This will open a pop-up window. Confirm `ERES_Organizer.jnlp` file open by Java Web Start Launcher. Then the Organizer will open.



Opening Organizer

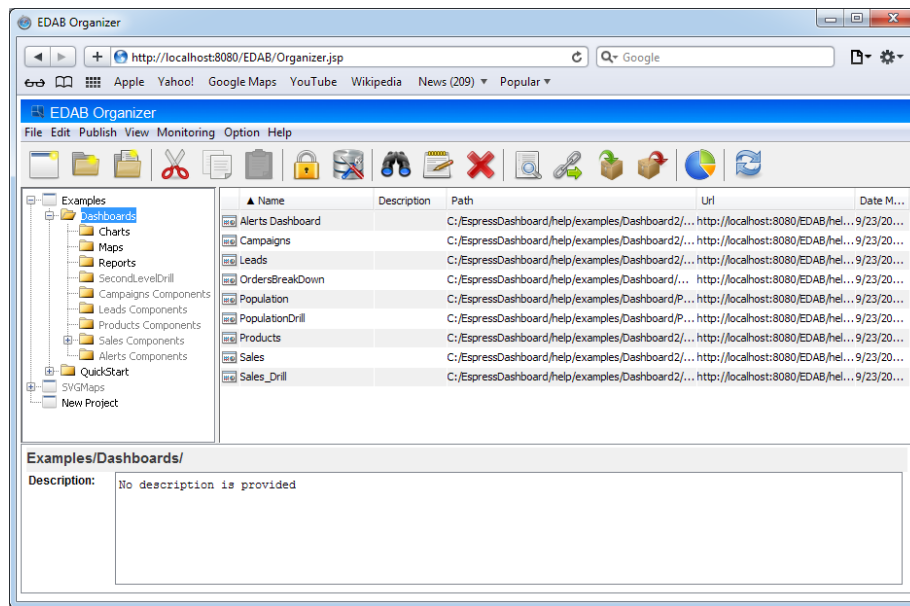
If you use Chrome as a browser, you have to save `EDAB_Organizer.jnlp` at first, and then to open it from your download directory. You will see the next dialog. Click *Run* and Organizer will open.



Opening Organizer in Chrome

2.1.2. The Organizer Interface

Since the Organizer is at heart a file management system, it is no coincidence that the application window resembles a file browser. The left-hand side of the window contains a list of projects and their associated sub-folders, while the right-hand side lists the contents of a folder in the upper pane and the project/folder descriptions in the lower pane.



Organizer Interface

2.1.2.1. The Organizer Menus

Most of the Organizer functions can be controlled using the drop-down menus located at the top of the Application window.

2.1.2.1.1. File Menu

This menu performs most of the file operations. Options in this menu include:

New	If you select this option, another menu pops up.
Project	Creates a new project in Organizer.
Folder	Creates a new folder in current project/folder.
Chart	Launches ChartDesigner. (see Section 4.1.1 - Introduction to Chart Designer).
Dashboard	Launches DashboardBuilder (see Chapter 6 - Designing Dashboards).
Manage Data Sources:	This option allows you to set-up and manage data registries, their associated data files, and database queries.
Open File:	This will open the currently selected file. If the file is a chart then the Chart Designer will open, allowing you to edit the chart. If the file is a report file then the Report Designer will open allowing you to edit the report.
Create PAK:	Pack a GXML (Online Map), SXML (SVG Map) or DSB (Dashboard) file and its components (Templates, Tooltips etc.) into a single file (see Section 2.1.4.4 - Dashboard and Map Packages).
Unpack File:	Unpack a GPAK (Online Map), SPAK (SVG Map) or DPAK (Dashboard) pack file into Organizer (see Section 2.1.4.4 - Dashboard and Map Packages).

Delete:	This will delete the currently selected project, folder, or file.
Refresh:	This will refresh your Organizer window, allowing it to reflect the latest changes/modifications performed by other users.
Exit:	This will close the Organizer.

2.1.2.1.2. Edit Menu

This menu allows you to edit, and cut/paste files, folders and projects. Options in this menu include:

Insert File(s):	This allows you to insert a report or chart into the selected project/folder.
Edit:	This allows you to edit the currently selected project, folder, or file. For projects and folders, you can change the project/folder name as well as its description. For files, you can change the display name, the location path or URL, and the description.
Set User Privileges:	This option is only available to the administrator. It allows you to set owner privileges for defined users and groups.
Set Security Levels:	This option is only available to the administrator. It allows you to setup and maintain user/group security levels that are applied to reports.
Change Ownership:	This option is only available to the administrator. It allows you to change the owner of the selected file, folder, or project.
Cut:	This removes the selected project, folder, or file, and places it on the clipboard.
Copy:	This makes a copy of the selected project, folder, or file, and places it on the clipboard.
Paste:	This pastes the current clipboard project, folder, or file into the currently selected project or folder.

2.1.2.1.3. Publish Menu

This menu allows you to setup and view published dashboards. Options in this menu include:

Preview Menu Page:	This will load the EDAB Menu page in a new browser window.
Generate Dashboard URL:	This option generates a dashboard URL for the currently selected dashboard.

2.1.2.1.4. View Menu

This menu allows you to search for files or edit charts. Options in this menu include:

Search:	This allows you to search through projects, folders, and files.
Chart Designer:	This will open the Chart Designer, allowing you to design or modify charts.

2.1.2.1.5. Monitoring Menu

Set up and manage alert monitoring tasks.

Set alert monitoring	Create new alert monitoring task. See the Section 11.4 - Monitoring chapter to learn more about alert monitoring.
View alert monitoring tasks	Shows list of all active monitoring tasks and allows you to edit them. To learn more, visit the Section 11.4.3.2 - Monitoring list chapter.

2.1.2.1.6. Option Menu

This menu allows you to set and view several of the Organizer options. Options in this menu include:

Set Designer:	This allows you to specify whether to open every chart in the same Chart Designer window (thus closing the current chart), or to open a new Chart Designer window for each chart opened.
Set URL Mapping:	This allows you to set up and modify virtual directories.
Repair Broken Links:	This allows you to search for, repair, or delete any broken links to files in the Organizer.
Update Directory:	This allows you to update the path of certain items in the Organizer. This is helpful if you are moving files between installations or moving complete installations.

2.1.2.1.7. Help Menu

This menu allows you to view a version of the program and to open a documentation. Options in this menu include:

About: This brings you an information about the version of the program.

Contents: This opens the EspressDashboard User's Guide.

2.1.2.2. The Organizer Toolbar

The toolbar at the top of the Organizer window offers easy access to some of the most commonly used features. The buttons perform the following functions:



Create a new project



Insert a new folder into the current project



Insert a new file into the current project or folder



Cut the currently selected project, folder, or file, and place on the clipboard



Copy the currently selected project, folder, or file, and place on the clipboard



Paste the current clipboard project, folder, or file into the currently selected project or folder



Delete the currently selected project, folder, or file



Set user privileges (Administrator only)



Manage Data Registries, files, queries, etc.



Search for projects, folders, or files



Edit the currently selected project, folder, or file



Preview the menu page



Generate a URL for the currently selected file



Pack a map or a dashboard and its components into a single file (see Section 2.1.4.4 - Dashboard and Map Packages)



Unpack a GPAK (Online Map), SPAK (SVG Map) or DPAK (Dashboard) pack file into Organizer (see Section 2.1.4.4 - Dashboard and Map Packages).



Go to Chart Designer



Refresh the Organizer window

2.1.3. Projects and Folders

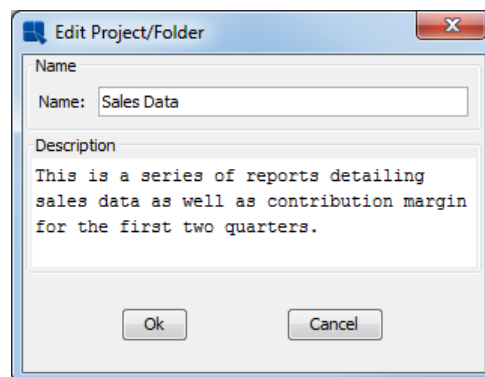
Projects and their associated folders are the central part of the Organizer interface. They provide the primary means of organizing charts and reports. Projects are the top level of organization. You can use different projects for different users, for different groups in your organization, or to group charts and reports representing different data or information. You can insert many layers of sub-folders into the top-level project, creating just about any organizational structure.

It is important to note that all folders in the Organizer are virtual folders, meaning that they do not necessarily have a physical location on the server machine or mirror an actual directory structure. Projects and folders in the Organizer are pointers to real directories and files that may be stored in a variety of locations.

2.1.3.1. Creating and Modifying Projects and Folders

To create a new project, you can do one of the following: click the *New Project* button on the toolbar, select *New Project* from the File menu, right click in the left-hand pane and select *New Project* from the pop-up menu, or press **Ctrl+P**. A node labeled *new project* will appear at the top of the file tree in the left-hand pane.

To edit the project name and description, first select it, then you can do one of the following: click the *Edit* button on the toolbar, select *Edit* from the Edit menu, press *F2* key, or right click, and select *Edit* from the pop-up menu. A dialog will then appear, prompting you to change the project name and description. Once you have entered your desired changes, click *Ok* and the new project name and description will be displayed in the lower right-hand pane.



Edit Project/Folder Dialog

To insert a sub-folder into a project or folder, first select the project or folder in which you would like to place the new sub-folder. Then you can either click the *New Folder* button on the toolbar, select *New Folder* from the File menu, or right-click in the left-hand pane and select *New Folder* from the pop-up menu. A node labeled *new folder* will appear under the currently selected project or folder.

To edit the folder name and description, first select it, and then you can either click the *Edit* button on the toolbar, select *Edit* from the Edit menu, press *F2* key, or right-click, and select *Edit* from the pop-up menu. A dialog will

then appear prompting you to change the folder name and description. Once you have entered your desired changes, click *Ok*, and the new folder name and description will be displayed in the lower right-hand pane.

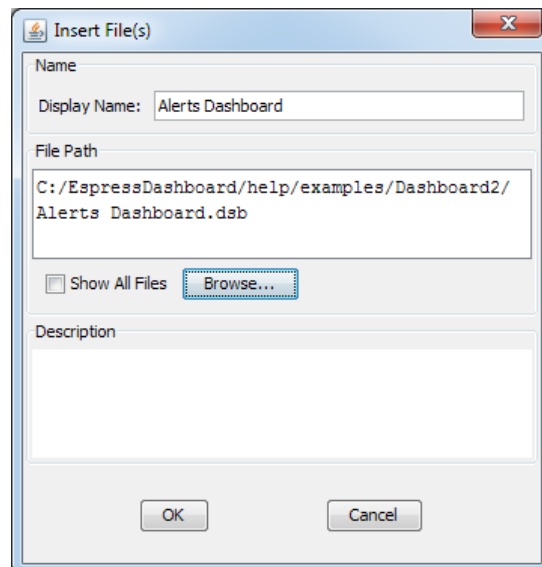
2.1.4. Files

You can insert files of any type into projects and folders. However, if you want your files to be accessible to other users or visible within the Menu, then files should be viewable within a web browser (i.e. HTML pages, image files, etc.). They can also be chart, report, or dashboard files (CHT, TPL, RPT, EZR, QPT, QCH, DSB, or XML format). Files added to the Organizer must be accessible via http protocol, and may need to be mapped via URL mapping. For more about URL mapping in Organizer see Section 2.1.5 - URL Mapping.

2.1.4.1. Adding and Modifying Files

To add a file, first select the folder or project in which you would like to place the file. Then you can either click the *Insert File* button on the toolbar, select *Insert File* from the Edit menu, or right click in the upper right-hand pane, and select *Insert File* from the pop-up menu. A dialog box will appear, prompting you to enter a display name for the file (by default the file name is used), the location of the file (a direct path) and a description of the file. (If you select a path that is not the server root or a mapped virtual directory, you will be prompted to map one.) Click *Ok* and the file will be inserted into the current directory or folder.

To insert more than one file at a time, click the *Browse* button on the insert file dialog, and use **CTRL+Click** to select multiple files in the browse dialog. Then when you click *OK* all of the files will be added to the Organizer.



Insert File Dialog

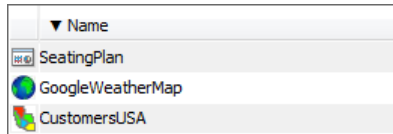
The list of files in the currently selected folder will be displayed in the upper right-hand pane of the Organizer. Each entry shows the file's display name, the file type, the file description, path, and URL.

To edit the file's display name, location, or description, first select the file that you would like to edit. Once you have selected a file, you can either click the *Edit* button on the toolbar, select *Edit* from the Edit menu, press *F2* key, or right click, and select *Edit* from the pop-up menu. A dialog box will appear allowing you to make changes to the display name, location, and description. Click *Ok* and the changes will be reflected in the file list.

To open a file, double click on it, or first select it, and then you can click the *Open File* button on the toolbar, select *Open File* from the *File* menu, or right click and select *Open File* from the pop-up menu. If the file is a chart (CHT, TPL, or QCH) format, it will be opened in the Chart Designer. If the file is a report (RPT, QPT, or XML) format, it will be opened in the Report Designer. If the file is a dashboard, the Dashboard Builder interface will open in a new window. All other file types will be opened in a web browser window.

2.1.4.2. Arranging Files

To arrange files within folder or project, open the folder (click on it in the left-hand side tree-list) and then click on a column header in the right-hand side. Files in the folder/project will be arranged by the content of the selected column. To switch between ascending and descending order, click on the column header again.



A folder arranged by the file names in descending order.

2.1.4.3. File Identification Icons

Each file type can be easily identified by its icon. You can see these icons next to file names in Organizer and all dialogs in EDAB designers where you need to select some file.



Chart file created in Chart Designer (.cht). For more information see Section 4.1.2.3.1 - Saving Chart Definitions.



Chart template created in Chart Designer (.tpl). For more information see Section 4.1.2.3.1 - Saving Chart Definitions.



Chart file created in Chart Designer that contains supplementary files associated with it (.pac). For more information see Section 4.1.2.3.1 - Saving Chart Definitions.



Report file created in QuickDesigner Reports (.qdr). For more information see Section 4.2 - QuickDesigner Reports.



Chart file created in QuickDesigner Charts (.qch). For more information see Section 4.3 - QuickDesigner Charts.



Map file created in Online Maps (.gxml). For more information see Section 5.2 - Online Maps.



Packed map file created in Online Maps and packed in Organizer (.gpak). For more information see Section 2.1.4.4 - Dashboard and Map Packages.



Coordinates file for Online Maps (.cxml). For more information see Section 5.2.2 - Data Sources.



SVG map file created in SVG Maps (.sxml). For more information see Section 5.3 - SVG Maps.



Packed SVG map file created in SVG Maps and packed in Organizer (.spak). For more information see Section 2.1.4.4 - Dashboard and Map Packages.



SVG map image (.svg). For more information see Section 5.3.1 - Introduction to SVG Maps.



Dashboard file created in Dashboard Builder (.dsb). For more information see Section 6.1 - Introduction to Dashboards.




Packed dashboard file created in Dashboard Builder and packed in Organizer (.dpak). For more information see Section 2.1.4.4 - Dashboard and Map Packages.


2.1.4.4. Dashboard and Map Packages

A single map or a single dashboard usually consists of several files (for example, a dashboard needs several chart/report/map templates, a svg map needs a svg image, drill-down template, etc) which could make moving maps or dashboards to another machine very difficult, because you would have to move all the linked files as well. Fortunately, you do not have to move each file manually. A dashboard or a map can be packed into a single file along with all the linked files (dashboard templates, drill-down templates, etc) without even knowing which exact files are needed for the dashboard or map to work. The packed files can be moved or archived quickly and easily.

To pack a dashboard or a map (both SVG and Online maps can be packed), select the dashboard (.dsb) or map

(.sxml or .gxml) file in Organizer and then click on the  *Create PAK* icon on the main Organizer toolbar. A dialog will pop up prompting you to select the pack file name. Enter a file name or use the default one and click *OK*. Another dialog should show up confirming that the package was created successfully and showing you the full file path. The packed file will be also automatically inserted into the current Organizer folder/project.

Packed files can be viewed in the Menu page without being unpacked, but they can not be modified in the SVG and Online Maps designers or in the Dashboard Builder. In order to modify a packed dashboard or map, you need to unpack the gpak, spak, or dpak file first.

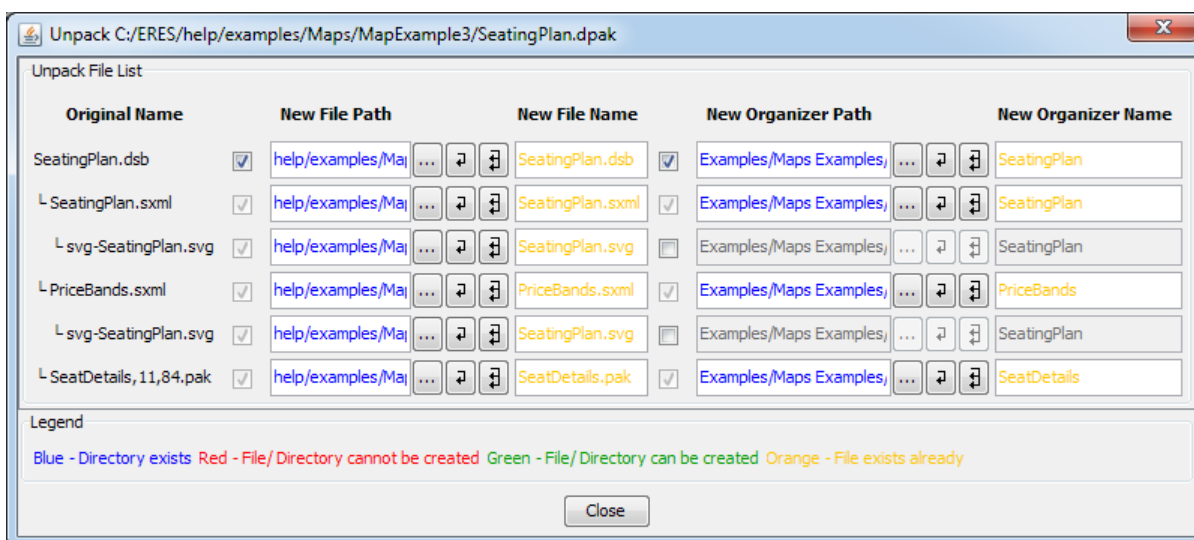
To un-pack a dpak, spak, or gpak file that has not been inserted into Organizer, click on the  *Unpack File* icon on the main Organizer toolbar. A dialog will pop up, allowing you to choose the file on your hard drive and to enter a name which will be used for the main unpacked file (.dsb, .sxml or .gxml file).

If you want to unpack a file that has been inserted into Organizer, select the file in Organizer and then click on the



Unpack File. The same dialog will pop up but the file path and the file name fields will be filled automatically.

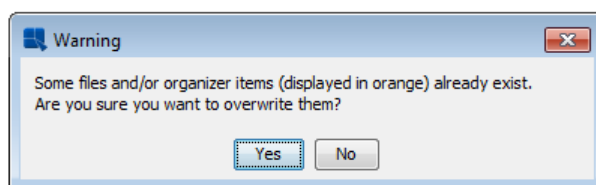
To view the list of all files that are stored in the package, click on the *Details* button. A new window will open.



Unpack File List Dialog

In this window, you can edit both physical and Organizer file path of all files that can be un-packed from the package (which also allows you to resolve potential file name conflicts). After you are done editing the file paths, click the *Close* and then click *OK* to unpack the package.

The EDAB will now try to unpack all files from the packed file to their original locations. If there are any filename conflicts (i.e. there already is a file with the same name in the same path on your hard drive or in Organizer), the following dialog will show up:



If you click *Yes*, all existing files will be overwritten by the files from the package without no further questions. If you click *No*, the *Unpack File List* dialog will open again allowing you to edit the filename conflicts. Then click *Unpack* to unpack the files into their locations and to insert them into Organizer.

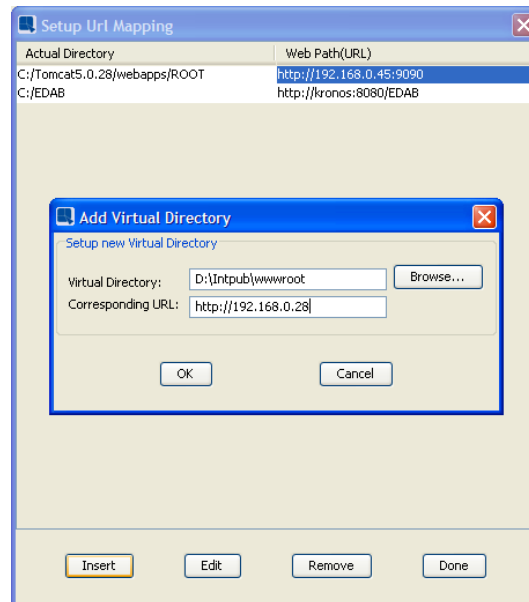
2.1.5. URL Mapping

Any report or chart that is added to the Organizer needs to be accessible via URL. This is so that reports can be run/viewed through the menu page, using URLs, or retrieved through the API. In order for the Organizer to know the URL path to charts and reports, you need to set the URL mapping.

The first, and most important, thing to map is your EDAB installation. You should have a virtual directory set-up in the app server/servlet container where you have deployed the server that points to your installation. If you selected to install Tomcat with EDAB then a virtual directory has automatically been setup where `http://MachineName:Port/EDAB` points to the EDAB installation directory. In order for Organizer to read and write files to your installation directory you will need to make sure that this mapping is correctly set.

URL mapping can also be used to set the URL path for other servers or virtual directories on your server whose report/chart files you would like to import to the Organizer. Please note that only the admin can create and change the URL Mapping entries. You will have to log in as the admin user to make changes to the URL Mapping.

To set up a virtual directory, select *Set URL Mapping* from the Option menu. A dialog will appear listing all of the virtual directories that have been set up. (By default, the Web root for the app server/servlet container where you have deployed the server is mapped to the URL `http://machinename:port/` depending on your machine name and specified port number. To add a virtual directory, click on *Insert*. A dialog will appear allowing you to enter or browse to a direct path. Once you have specified the path, then enter the corresponding URL. Click *Ok* and the virtual directory will be added. You can also edit and remove mapped URLs from the main dialog.



URL Mapping List & Add Mapping Dialog

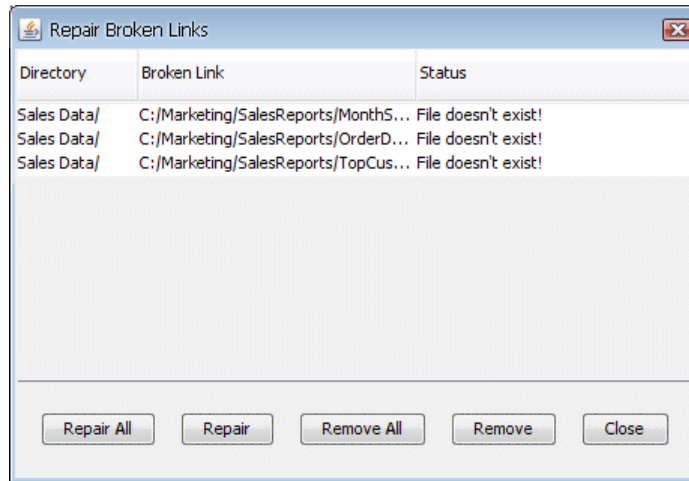
You will also be prompted to specify URL mapping any time you add a file from a new location that has not been mapped with a corresponding URL.

2.1.6. Repairing Broken Links

Because Organizer uses virtual directories, it is possible to have files in many different physical locations listed in the Organizer. While this makes it easy to publish reports and charts, it can be difficult to manually keep track of all the files. Specifically, if the files are moved or the directory structure is changed, then the Organizer will not be able to find the files and the links within the Menu page may no longer work.

You can use the *Repair Broken Links* feature in the Organizer to automatically check the status of the listed files. The Organizer will check all of the files within your projects and folders to check whether the path/URL specified is still correct.

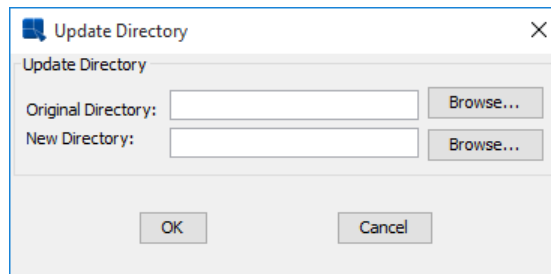
To check for broken links, select *Repair Broken Links* from the Option menu. The Organizer will then check all of your files and pop up a list of any broken links found. You can then choose to repair any broken links by specifying a new path or file/filename, or you can remove the files from the Organizer.

*Broken Links List*

2.1.7. Update Directory

Sometimes you may want to update the directory path of certain items in the Organizer. The *Update Directory* feature is helpful if you are moving files between installations or if you are moving complete installations.

In order to update the directory paths of items in the Organizer, select *Update Directory* from the Option menu. The update directory dialog will then open.

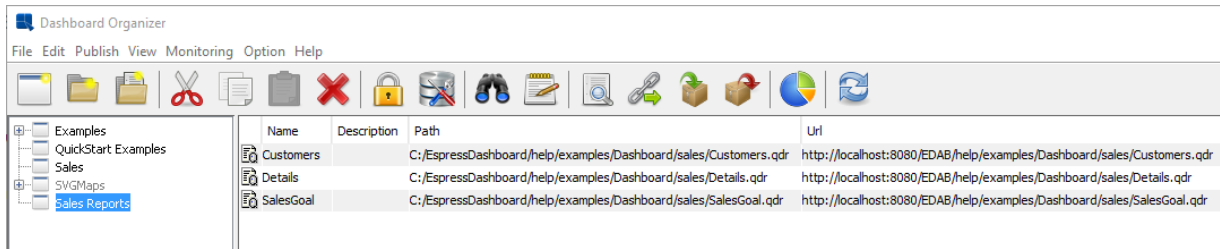
*Update Directory Dialog*

The dialog allows you to specify the original directory (the directory you want to update) and the new directory. You can either use the *Browse* buttons to select directories or just type directory paths manually.

Please note that when you are updating directories using the update directory feature, all the Organizer items that contain the original path in their directory paths will be updated. For example, if you specify the original path as C:/EDAB and the new path as D:/Install/EDAB, all the items in the Organizer that contain C:/EDAB in their paths such as (C:/EDAB/Reportfiles/SalesReport.qrp, C:/EDAB/ChartFiles/MonthSales.cht, C:/EDAB/DashboardFiles/Dashboards/SalesDashboard.dsb, ...) will be updated to (D:/Install/EDAB/Reportfiles/SalesReport.qrp, D:/Install/EDAB/ChartFiles/MonthSales.cht, D:/Install/EDAB/DashboardFiles/Dashboards/SalesDashboard.dsb, ...). Also note that the new directory (the updated directory) needs to be added to your URL mapping. Otherwise, you won't be able to repair broken URL links after the update, so the affected Organizer items won't work correctly. For more information about URL mapping and repairing broken URL links, please see Section 2.1.5 - URL Mapping and Section 2.1.6 - Repairing Broken Links.

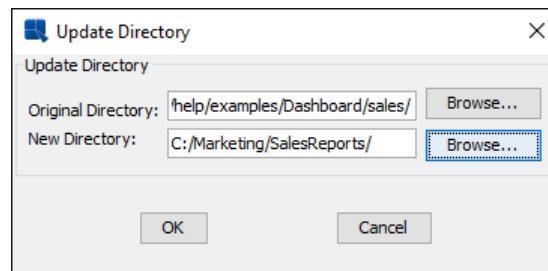
Example:

Imagine that you have three reports Customers.qdr, Details.qdr, and SalesGoal.qdr in the C:/EspressoDashboard/help/examples/Dashboard/sales directory, and assume that you have moved them to the C:/Marketing/SalesReports directory. The following image shows the reports in the Organizer before the updating process:



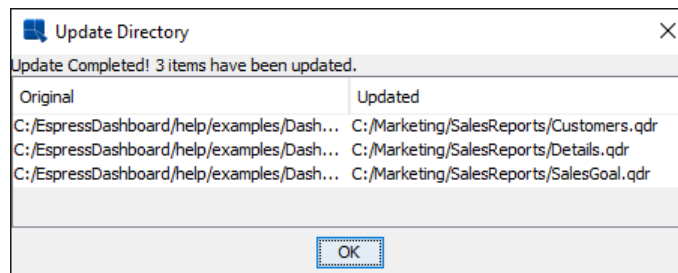
Example - Reports in the Organizer - Before Directory Updating

Select *Update Directory* from the *Option* menu in the Organizer to open the update directory dialog. In the dialog, specify the original directory as C: /EspressDashboard/help/examples/dashboard/sales and similarly the new directory as C: /Marketing/SalesReports.



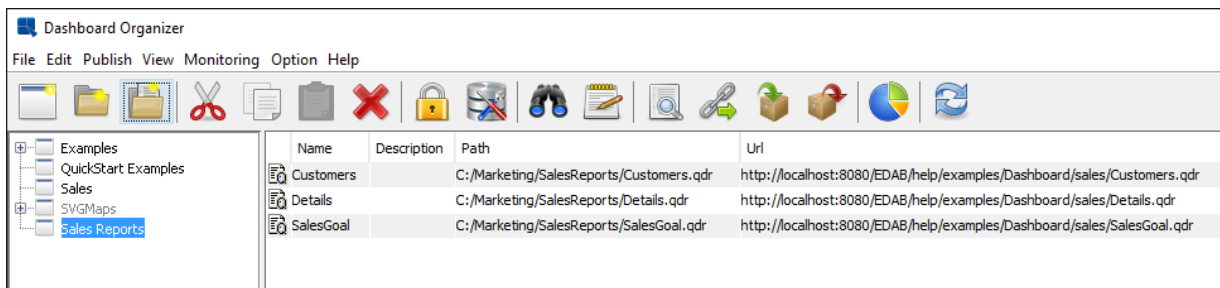
Example - Update Directory Dialog

Once you have properly specified the directories, press the *OK* button. The following dialog will then appear showing you the list of updated Organizer items as well as number of updated files.



Example - Update Results Dialog

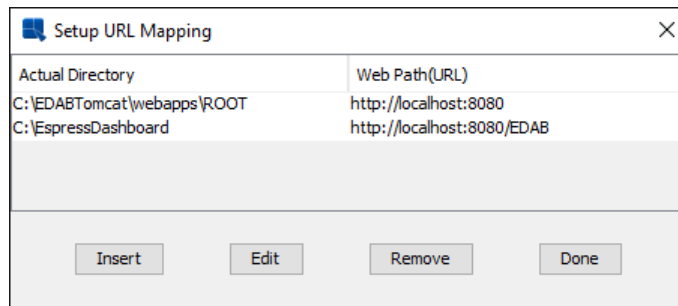
Next, click the *OK* button in the dialog. This will bring you back to the Organizer. The image below shows updated reports paths after the updating process.



Example - Reports in the Organizer - After Directory Updating

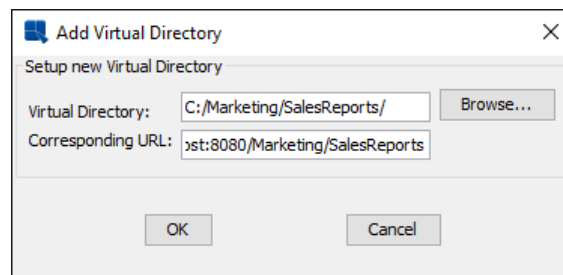
Now you have to repair the broken URL links. Otherwise, URLs of the affected Organizer items will remain mapped to the original directory. In order to repair broken URL links properly, you have to add the new directory (the directory you have specified in the Update directory dialog C: /Marketing/SalesReports) as new URL link

to your URL mapping. To open the URL mapping dialog, select *Setup Url Mapping* from the *Option* menu in the Organizer. The dialog will then appear.

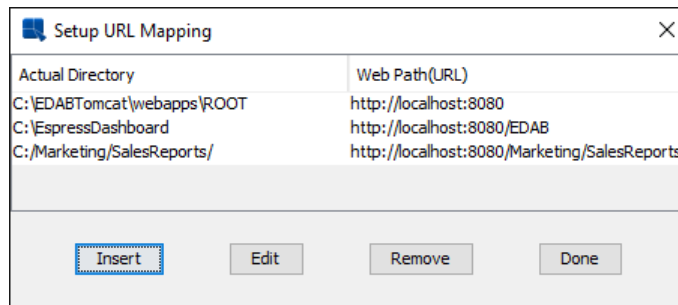


Example - Setup URL Mapping Dialog

In order to add a new URL link, click the *Insert* button, another dialog will pop-up prompting you to specify the *Virtual Directory* and *Corresponding URL*. Specify the virtual directory as C: /Marketing/SalesReports and the corresponding URL e.g. as http: // <MachineName> : <Port> /Marketing/SalesReports. After that, click *OK*. For more information about URL mapping, please see Section 2.1.5 - URL Mapping.

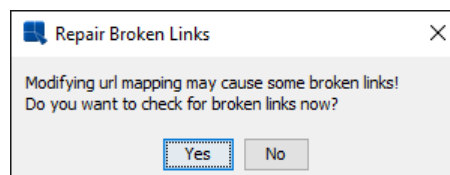


Example - Inserting New URL Link



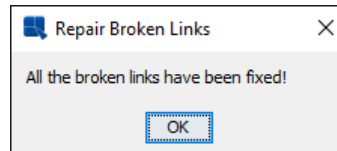
Example - Setup URL Mapping Dialog - New URL Link Added

Once you have specified the URL link, click the *Done* button again. The repair broken links dialog will then appear prompting you whether you want to check for broken links now. Click *Yes*. Please note that if you click *No*, you may check for broken URL links by selecting *Repair Broken Links* from the *Option* menu in the Organizer. For more information about repairing broken URL links, please see Section 2.1.6 - Repairing Broken Links.



Example - Repair Broken Links Dialog

If you have correctly specified the URL mapping, you should get the following dialog that will inform you that broken links have been successfully fixed. Clicking the *OK* button will bring you back to the Organizer.

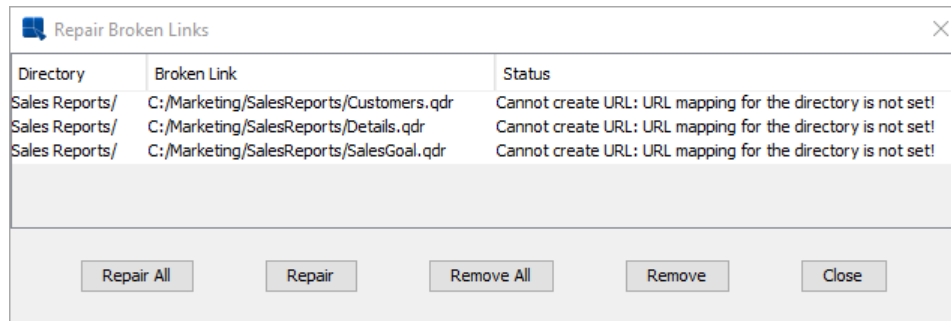


Example - Repair Broken Links - Links Successfully Fixed



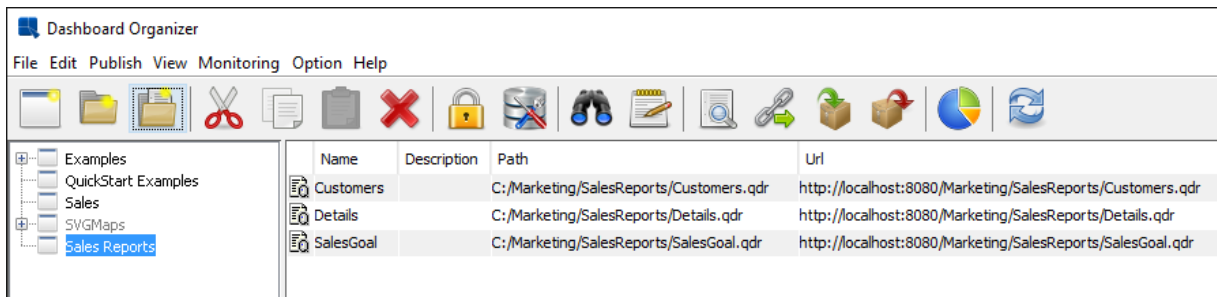
Note

If you did not specify the URL link in the URL mapping or did not specify the URL link correctly, the dialog below will appear informing you that broken URL links cannot be repaired. If you got this dialog, return back to the URL mapping dialog and add/modify the URL link as described in previous steps. After that try to repair broken links again.



Example - Repair Broken Links - Error During Links Fixing

Now you may check whether the URL fields of the organizer items have been correctly repaired. The URL fields should be similar as the ones shown on the image below.



Example - Reports in the Organizer - After Repairing URL Links

2.1.8. Searching in Organizer

Often, if you have many files in many folders in the Organizer, it can be hard to find a specific file. To locate files



or folders, you can use the search function. To search, you can either click the *Search* button on the toolbar, select *Search* from the View menu, press **Ctrl+F**, or right click in the upper right-hand panel and select *Search* from the pop-up menu. The search dialog will then appear.

Search Dialog

The search dialog prompts you to enter a search string in the project or folder in which you would like to search. By default, the currently selected project or folder is listed. If you leave the *Look In* field blank, it will search all projects.

Other options allow you to include sub-folders in your search, and to match the search string to filenames, descriptions, and URLs. You can also match case in your search by selecting the *Case Sensitive* option.

2.1.9. Limiting Browse Directories

In EDAB, you can limit the open and insert dialogs in Organizer, Report Designer, and Chart Designer by adding a parameter to the Organizer. If launching the Organizer through the Java Web Start Launcher, edit `/EDABOrganizer.jnlp` and add the following line:

```
<argument>-BrowseRootDir:<Directory></argument>
```

For instance, the follow limits the browse directory to the EDAB root directory located under `C:\EDAB`

```
<argument>-BrowseRootDir:C:\EDAB</argument>
```

If starting the Organizer using the bat file, edit `EDABOrganizer.bat` and add the following option at the end of the command.

```
... quadbase.reportorganizer.manager.OrganizerClient -  
BrowseRootDir:<Directory>
```

2.2. Security and Security Administration

This chapter covers security concepts in EDAB and the administration of user privileges. Other security features for data registries and deployment (menu page, URLs, and the API) are discussed in later sections.

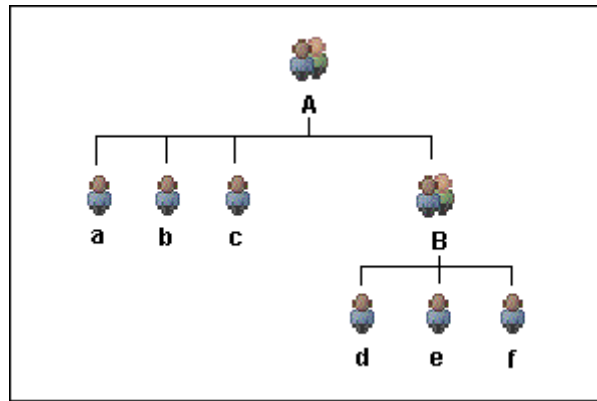
There are two security concepts in EDAB:

- | | |
|-------------------------|--|
| User privileges | This is the default security concept. User privileges are derived from relations between users and groups (will be described later). Relations between users/groups can be modified. |
| File permissions | You can disable the “User Privileges” mode for certain files/folders and set custom permissions for those files/folders. |

2.2.1. Security Concepts

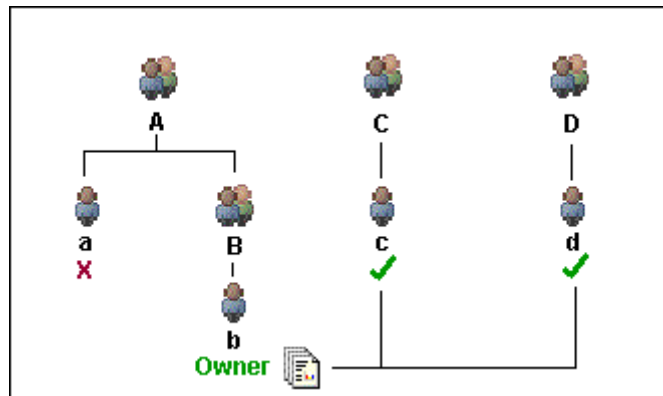
In EspressoDashboard, privileges are based on the concept of ownership by default. This means that the privileges or access to a particular folder, dashboard, maps, report, or chart stem from who is the owner or creator of that folder, dashboard, maps, report, or chart. Each user (or group) has a set of privileges that apply to other users or groups when accessing dashboards, maps, reports, or charts created by that user.

In this ownership approach, users belonging to the same group or subgroups of the same group have full access privilege to items (folders, dashboards, maps, reports, charts) created by each other. For example, group A (group structure A) has users a,b,c and sub-group B. Group B (as a sub-group of A) has users d,e, and f. The default behavior is such that users a,b,c,d,e,f will have full access privilege to items created by each other. At the same time, users outside of the group structure A have no access to the items created by users in this group structure.



Users from group A and B will have access to each other's files.

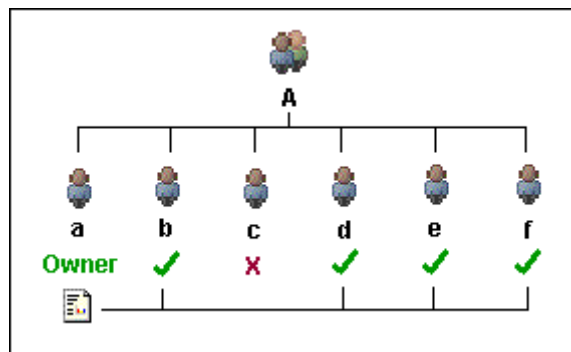
With the “user privileges” setting functionality, you (as the administrator) can modify the default privilege granting scheme. For example, you can grant users from group C and D read or read/write access to group B users' items whereas group A's users have no permission to access group B users' items. Meanwhile, group B users still have same access right to other group A users' items if group A is still using default privileges.



After setting group permission, groups C and D have access to group B's files, but group A does not.

In addition to the “user privileges” ownership model, the privileges for individual items can be modified to further enhance the security settings. This function is called the “File permissions”. In this scenario, individual items can be made accessible to groups and users independent of the ownership assigned scheme.


For example, an item created by user a is automatically fully accessible to users b,c,d,e, and f. But you can specifically modify the permission setting such that user c has no access right to that particular item while the rest of the users in the group structure maintain their access rights.

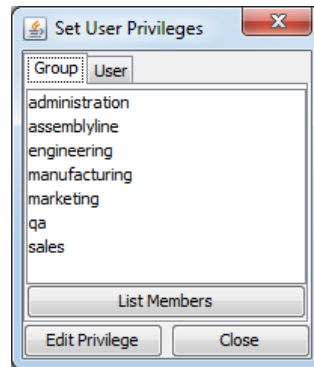


After setting file permission, user c does not have access to this particular report.

2.2.2. Setting User Privileges

Privileges are set by the administrator in the Organizer interface (with the exception of the basic designer/viewer role for users set in the Admin Console). To set user privileges in Organizer, select Edit → Set Privileges or click

the  *Set Privileges* button on the toolbar. This will bring up a dialog, allowing you to set owner permissions for users and groups.



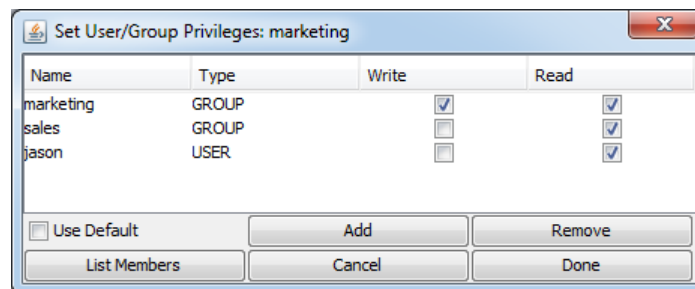
Permission Setting Dialog

The dialog contains two tabs one listing all of the groups currently defined for EDAB and the other listing all the users. From either of these tabs you can select a user or group for which you would like to set owner permissions by selecting it and clicking the *Edit Privilege* button.



Note

When you select a user or group, you are setting privileges for folders, reports, and charts that this user, or users in this group create (or become owners of). You are not setting access levels for that particular user or group, rather you are setting privileges for other groups or users who access reports created by this user or group.



Group/User Permissions

Every group or user will automatically have read and write permissions for their own report and charts. You can give other groups and users access, by clicking the *Add* button. This will bring up a list of users and groups allowing you to select which users and/or groups you would like to add. For each additional user or group, you can set their access to read-only, or read-write.

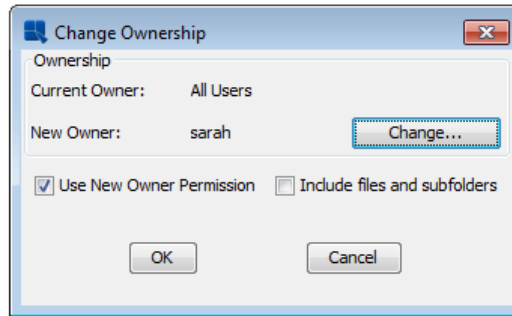
Read-only access allows the user or group to view the file using various EDAB tools. For example, when the user opens the file from the EDAB Organizer, the program will launch the corresponding viewer (Chart Viewer for charts, Page Viewer for reports, Dashboard Viewer for dashboards) rather than opening the Designer. The user can also create schedules and archives for this file and access the file through the EDAB menu page. With read-only permission, the user will not be able to edit the file using the Designers, nor will they be able to generate URL for the file. In addition, the user cannot see the file and URL path in the Organizer.

Read-write access grants full privilege to the user. They can edit the file using the corresponding designer as well as generate URL and see the file and URL path in the organizer.

If the *Use Default* checkbox is checked then the user or group will inherit permissions from the group to which he/it belongs. If the user or group does not belong to any other groups then they will have read and write permissions for their reports but no other users will have access. This option is used by default in EspressoDashboard. There users will automatically inherit group permissions unless settings for that user are specifically modified.

2.2.2.1. Changing Ownership

Generally, a user is the owner of any folder, chart, or report he/she creates. However, the administrator can change the owner of any folder or file in the Organizer. To change the owner of a folder or file, right click on it, and select *Change Ownership* from the pop-up menu. This will bring up a dialog showing the current owner of the file or folder.

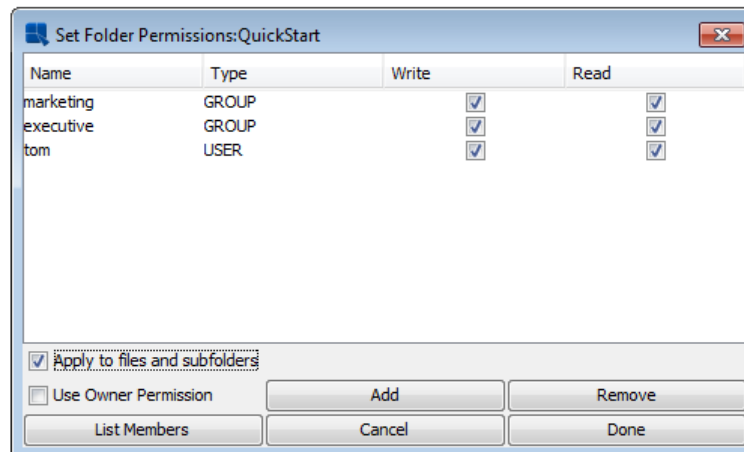


Change Ownership Dialog

To change ownership, click the *Change* button. This will bring up a list of all the defined users. Select the user to which you would like to assign ownership and click *Ok*. The new owner will then be reflected in the dialog. Click *Ok* again to apply the changes. Note that changing the owner of the file will generally effect which users can access and edit the file.

2.2.2.2. Folder/File Permissions

By default, user privileges for files and folders in the Organizer are derived from the owner of that file or folder. However, in certain circumstances you may want to set privileges for specific files or folders to be different from those associated with the owner. To apply a different set of permissions to a file or folder, right click on it and select *Change Folder/File Permissions* from the pop-up menu.



File/Folder Permissions Dialog

The *Use Owner Permission* checkbox indicates, if checked, that the folder or file is currently using owner permissions. To add different levels of access for that particular object, first un-check this option, and click the *Add* button. This will bring up a list of users and groups, allowing you to select which users and/or groups you would like to add. For each additional user or group, you can set their access to read-only, or read-write. Once you have finished adjusting access for the object, click *Ok* and the changes will be applied.

Organizer

Here is a practical example that illustrates how the file/folder permissions function. Suppose John, a member of the marketing group, can only see the demo project, which is used during project demonstrations. All other existing projects that John does not have access to are automatically hidden from view.

Name	Description	Path	Url	Date ...
AccountStatement		C:/WORK/HO...	http://localhost:8080/ERES/ReportFiles/AccountStatement...	4/27/2...
CategorySales		C:/WORK/HO...	http://localhost:8080/ERES/ReportFiles/CategorySales.qrp	4/27/2...
ProductList		C:/WORK/HO...	http://localhost:8080/ERES/ReportFiles/ProductList.qrp	4/27/2...
DrinksSold		C:/WORK/HO...	http://localhost:8080/ERES/ChartFiles/DrinksSold.pac	4/27/2...
SalesWest		C:/WORK/HO...	http://localhost:8080/ERES/ChartFiles/SalesWest.pac	4/27/2...
SalesEast		C:/WORK/HO...	http://localhost:8080/ERES/ChartFiles/SalesEast.pac	4/27/2...
SalesSouth		C:/WORK/HO...	http://localhost:8080/ERES/ChartFiles/SalesSouth.pac	4/27/2...

demo/
Description: Used during product demonstration, including report and chart examples

John's Organizer

Suppose the administrator would like for John to edit and incorporate a dashboard into his presentations. To do so, the administrator gives John read and write permissions for one dashboard in the Dashboard/DashDemo/ project. It is not necessary for administrator to set folder privileges since the change will be propagated through the parent folders.

Name	Description	Path	Url	Date Mod...
DemoDashboard	Dashboard	C:/WORK/HOME...	http://localhost:8080/ERES/DashboardFiles/DemoDash...	4/27/2012 5...
WeatherForecast	Dashboard	C:/WORK/HOME...	http://localhost:8080/ERES/DashboardFiles/WeatherF...	4/27/2012 5...

Set File Permission:

Name	Type	Write	Read
admin	USER	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
john	USER	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>

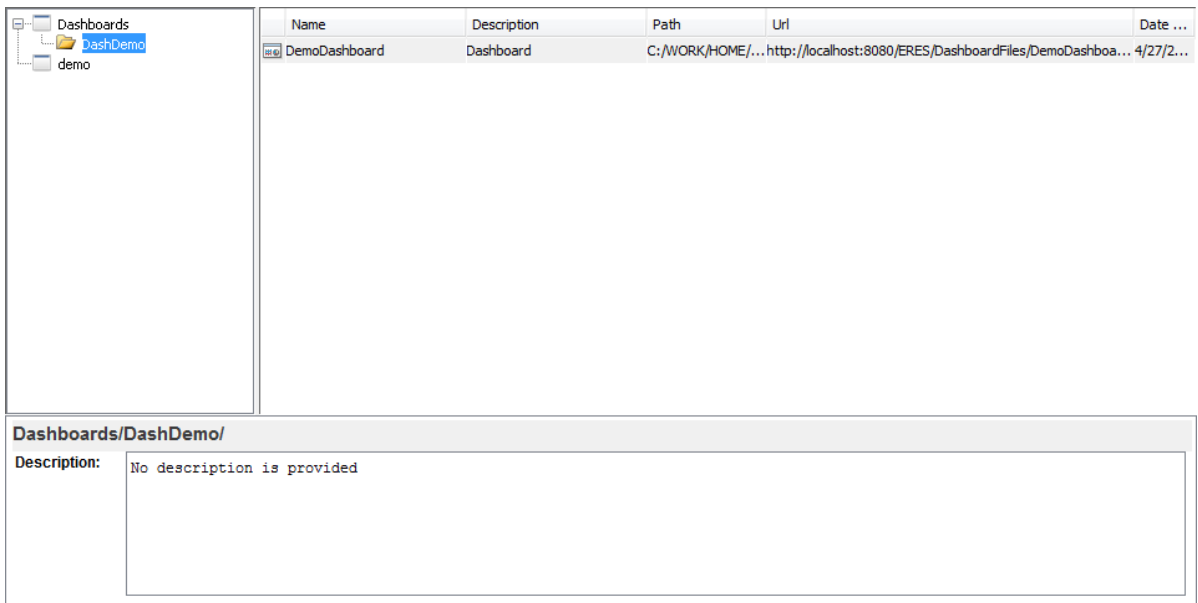
☐ Use Owner Permission Add Remove

List Members Cancel Done

Dashboards/DashDemo/
Name: DemoDashboard
File Type: DSB File
Path: C:/WORK/HOME/git/ER_ANT_66/dist/eres/DashboardFiles/DemoDashboard_admin_1335539843486.dsb
URL: http://localhost:8080/ERES/DashboardFiles/DemoDashboard_admin_1335539843486.dsb
Date Modified: 4/27/2012 5:17 odp.
Description: Dashboard

Setting File Permission

The next time John launches the Organizer, he will see the dashboard file that he has been given permission to view and edit. However, John will not be able to see any other files in this folder nor any files in the Dashboard folder (the parent folder).

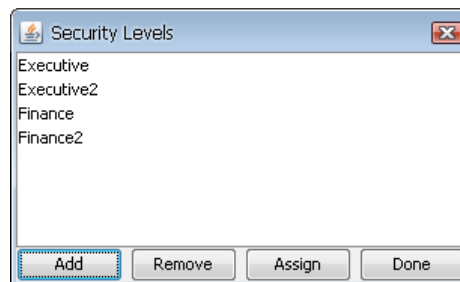


Dashboard File is Accessible

2.2.3. Security Levels

In addition to setting permissions and access for different dashboards, the administrator can also apply security settings to the EDAB system, such that users and groups can get different views of the same report when accessing it. This security feature is administered through security levels.

Security levels are created by the administrator in the Admin Console or in the Organizer, and applied to the server using the Secured Parameter configuration in the Admin Console. To create or modify security levels, select *Set Security Levels* from the *Edit* menu. This will bring up a dialog listing all the security levels that have been defined. For more details regarding secured parameters, please see Section 1.4.1.3.1 - Secured Parameter.

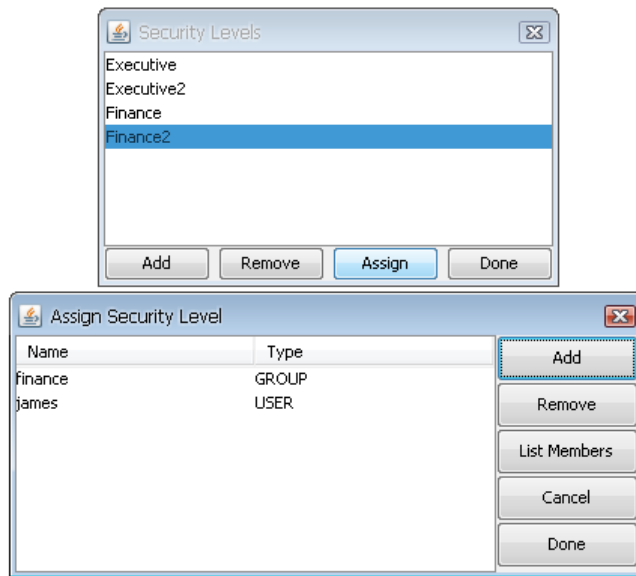


Security Levels Dialog

To add a new security level, click the *Add* button. This will bring up a dialog prompting you to specify a name for the new security level. There is no limit to the number of different security levels that can be defined.

2.2.3.1. Associating Security Levels with Users and Groups

In EDAB, users and groups can be associated with defined security levels to provide end-to-end security. When a user accesses a report, the security level to which they are associated will be applied, and the view defined by that security level will be presented to the user. To associate users and groups with security levels, select a security level in the list and click the *Assign* Button in the Security Level Dialog.



Assign Security Level Dialog

In the dialog that opens, click the *Add* button to associate a user or group with the selected security level. This will bring up a dialog that contains a list of all the defined users and groups. From this dialog you can select groups and users to associate with the security level. When you have finished, click *Done* to return to the security levels dialog.

You can assign more than one user or group to a security level, but users cannot be assigned to more than one security level at a time. Security level associates, like permissions are applied at the lowest level. This means that if an individual user is associated with a security level, it will take precedence over settings for their group.


Chapter 3. Data Sources

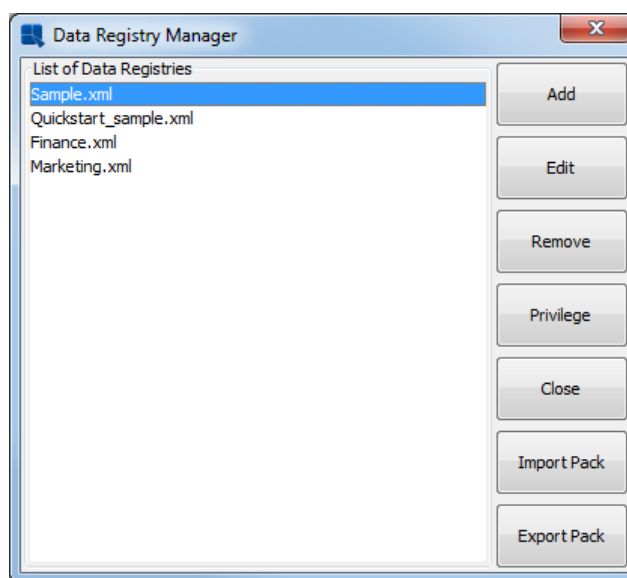
3.1. Data in Organizer

EspressDashboard can draw data from JDBC compliant databases, text files, XML files, EJBs, and even bring in object/array data through class files. Prior to creating reports, charts, or maps, users must first set up data sources they want to use. Data source information, including database connection information, database queries, text file location, XML file/DTD/Schema location, Java class location, and EJB connection information is stored in XML registry files that are set up and defined within the Organizer interface.

3.1.1. Managing Data Registries

To edit or create a data registry, you can either select the *Manage Data Sources* option from *File* menu, or click on

the  *Manage Data Sources* icon on the toolbar. This will launch the Data Registry Manager dialog.



Data Registry Manager Dialog

From this dialog, you can add, remove, and edit registries, as well as assign user privileges for data registries.

To add a registry, click the *Add* button and a dialog will appear prompting you to specify a filename for the new registry. Click the *OK* button and a new Data Source Manager window will open, allowing you to set up data sources within the registry.

To edit a registry, first select it from the list on the left side and click the *Edit* button. This will bring up the Data Source Manager window for the selected registry file.

3.1.1.1. Exporting/Importing Data Registries

A data registry consists of several files such as database queries, text files, xml files, etc. If you export a data registry, all files will be packed in a single *rpak* file that can be archived or imported to another EDAB installation/server.

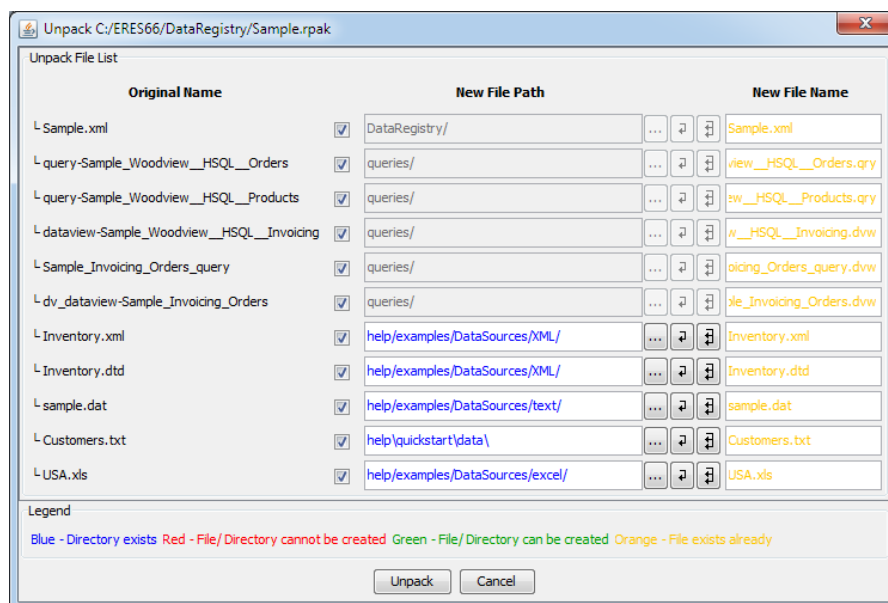
To export a data registry, select it and then click the *Export Pack* button. Enter a file name and click the *OK* button.



Note

Only administrators can export or import data registries.

To import a data registry, click the *Import Pack* button and then select a *rpak* file. You can insert the data registry using its original name or you can change it in the *New registry name* field. If you want to choose which files should be unpacked or change their path or file name, click the *Details* button.



Unpack dialog

In this dialog, you can choose which files you want to unpack and you can also change their names and paths.

If you don't want to unpack any file, uncheck 'unpack file to disk' checkbox.

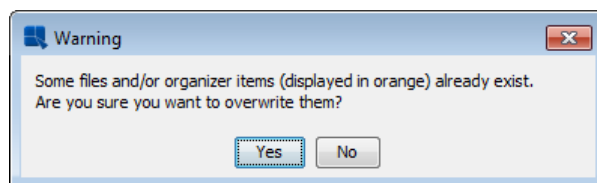


Note

File paths of some type of files (for example: sql queries) can't be changed. In such cases, the *New file path* field and buttons will be disabled.

Click the *Cancel* button to get back to the *Import Data Registry* dialog.

Click the *Unpack* button. The data registry will now unpack all contained files. It is possible that some files from the *rpak* file might have the same file names and paths as other existing files on your hard drive. If there are any filename conflicts, the following dialog will pop up.



Data Registry Manager Dialog

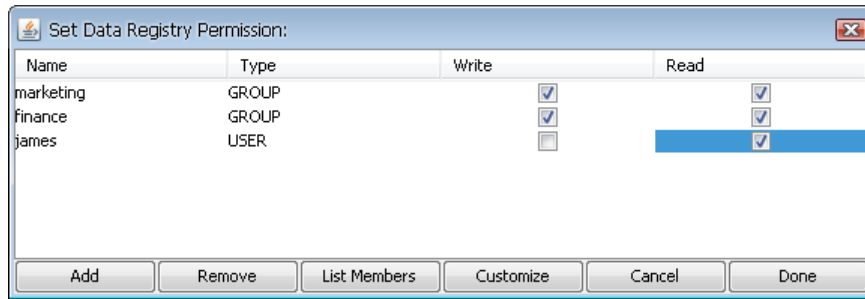
If you click *Yes*, the conflicting files on your hard drive will be overwritten with the files from the *rpak* file.

If you click *No*, the *Unpack dialog* will open again allowing you to solve the filename conflict.

3.1.1.2. Data Registry Privileges

As with other objects in EspressoDashboard, the administrator can also assign privileges to Data Registries. These include access permissions (read/write) as well as content filtering (restricting which sources in the registry users or groups can see).

To set privileges for a registry, select it in the Data Registry Manager and click the *Privilege* button. This will bring up a dialog listing permissions for the file.



Registry Privileges Dialog

To give users or groups access to the registry, click the *Add* button. This will bring up a list of all the defined users and groups. From this dialog, select the groups and users you want to add. You can set access rights for each user or group.

For each user or group that has access to the data registry, you can also customize their view of the registry. This allows you to restrict access of a group of users to specific data sources or types of data sources. To customize the registry display, select one of the users or groups in the registry privileges dialog and click the *Customize* button. This will bring up a dialog showing all the data sources that have been defined in the registry.



Customize Registry Dialog

Each node (data source) in the registry can be set to visible or invisible by selecting it and checking the option at the bottom of the dialog. Invisible nodes will be grayed out. Note that if you render a parent node invisible, all the sub-nodes will also be invisible, so you can't make Queries node invisible and individual queries visible at the same time.

Once you finish, click the *Ok* button and the settings will be associated with the user or group. When they access the registry, only the visible nodes will appear.

3.1.2. The Data Source Manager

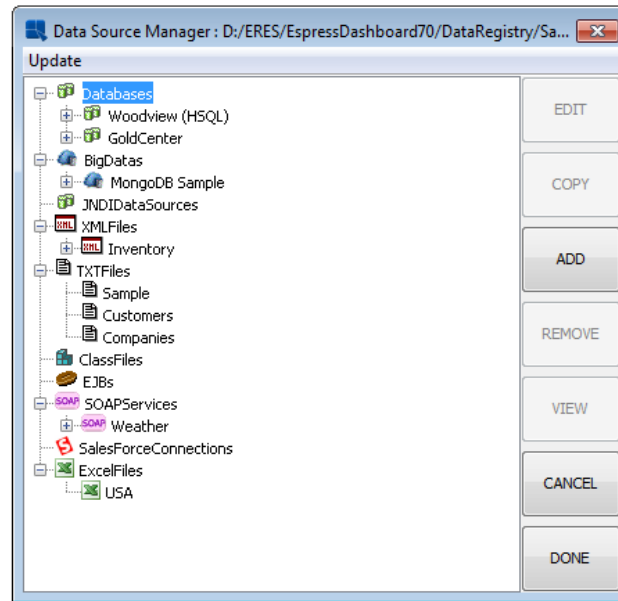
When you edit or create a new data registry, the Data Source Manager window will open allowing you to add and edit data sources. The Data Source Manager is a visual representation of the registry XML file.



Note

The XML data source repository only stores location and connection information, not the actual data. It is not an XML file that contains data to be used in a report or chart.

3.1.2.1. Using Data Source Manager

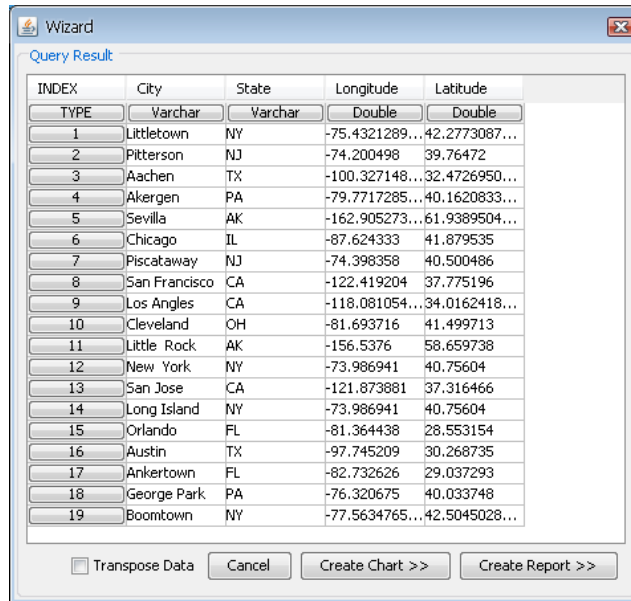


Data Source Manager

The left side of the window contains a tree listing all of the data sources in the registry file. Grouped under *Databases* are individual databases and their associated queries and data views. Grouped under *JNDIDataSources* are database sources that use JNDI (Java Naming and Directory Interface) name to connect instead of JDBC. Grouped under *XMLFiles* are all XML files and their associated queries. Grouped under *TXTFiles* are all specified Text files. Grouped under *ClassFiles* are all specified class files, and grouped under *EJBs* are all specified EJB connections.

Right side of the window contains a series of buttons controlling all of the functions of the Data Source manager. Each button performs the following function:

- Edit:** This option allows you to modify attributes of a data source. For a database, it allows you to change the connection information and modify queries/data views. For XML files, it allows you to change the file and its location, and modify XML queries. For text files, it allows you to change the display name and file location. For class files, it allows you to change the display name and modify the location. And for EJBs, it allows you to change the display name as well as parameter values.
- Copy:** This option is not only available for database nodes and bigdata nodes but also for the specified query or data view.
- Add:** This option allows you to add a data source. It will create a new source depending on which node is selected on the left side of the window. Hence, if you select *TXTFiles* and click the *Add* button, you will be prompted to add a new text file data source.
- Remove:** This option will remove the selected data source.
- View:** This will open a new window showing the tabular data from the selected data source, either query, XML file, text file, class file or EJB. For data views, the data view query interface will start. From this window, you can select to create a new report or a new chart by selecting the appropriate button.



View Data Source Dialog

Cancel: This will cancel the wizard process.

Done: This will save any changes you've made in the registry and return you to the registry list.

3.1.2.2. Data Source Node Locking

The data registry has an automatic node locking system designed to block multiple users from editing related nodes at the same time. The system utilizes your IP address, username, and a special security token to determine whether you are allowed to add, remove, or edit a specific node. If you are currently editing a node, other users will not be able to edit it. However, if you disconnect from the server or move to another computer and try to open the same node from the same username, an option box will appear allowing you to override your previous lock.

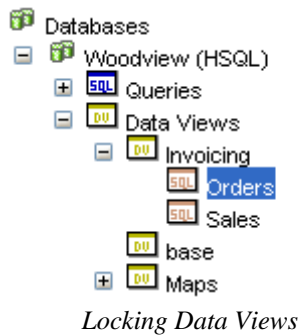
Please note that administrator is capable of unlocking any node, even if they did not open it in the first place.

The data registry is a tree where data type is the root and queries and files are the leaves. Changing any node within a particular branch will lock all parent nodes and all child nodes in that branch. Here are some examples of how the node locking system works:



Locking Query

In the above image, you can see that the database *Woodview (HSQL)* and query *Orders* are part of the same branch. So if you edit *Orders* query, it will lock both *Orders* and database *Woodview (HSQL)*. If you edit *Woodview (HSQL)*, the database will be locked and all child nodes connected to this database will be locked as well, which includes *Orders*, *Products*, *State*, and *Region*.

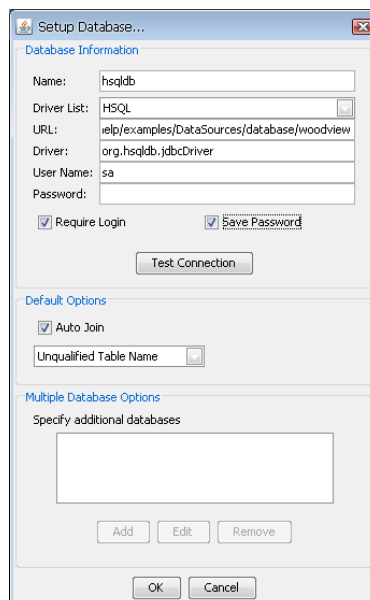


In the case of data views, there are three levels in the tree. In the above image, editing *Orders* data view query will lock the query, data view *Invoicing*, and database *Woodview (HSQL)*. If the data view *Invoicing* is being edited, both queries *Orders* and *Sales* will be locked, as well as database *Woodview (HSQL)*. However, in both of these scenarios, other data views and queries can still be edited. If the database *Woodview (HSQL)* is being edited, all data views as well as their queries will be locked.

3.1.3. Data from a Database

EDAB can draw data from any JDBC compliant database. In order to connect to a database via a 3rd party driver (other than the JDBC bridge), you will need to add the classes for that driver to the classpath of the application server/servlet runner where you have deployed the EDAB server. If you installed EDAB with Tomcat, then you need to modify `setclasspath.bat` / `.sh` file in `/bin/` directory of your Tomcat installation to add the driver classes, or copy them to `<EDABInstallDir>/WEB-INF/lib` directory. Note that JDBC drivers for MS SQL Server, MySQL, Oracle, Informix and PostgreSQL databases are included as a convenience. Other database JDBC jar files are not included due to licensing, multiple drivers, and/or other concerns, although support for these databases exist and the jar files can be explicitly added.

The first step in using a database as the data source is to set up a database in the registry and specify the connection information. To add a database, click on the *Databases* node and click the *Add* button. This will bring up a window prompting you to specify the connection information for that database. You can choose a database to connect to from the Driver List or specify the information manually. Fields to enter are database name, URL, and driver. You can also select whether the database requires a login and whether you want to save username and password information. If you select to save login and password information, you can then enter these informations in *User Name* and *Password* textboxes. Then click the *Ok* button and the new database will be added to the Data Source Manager window.



Add Database Dialog

In order for EDAB to create a connection to the database, the following information must be provided:

URL: This JDBC URL specifies the location of the database to be used. A standard JDBC URL has three parts, which are separated by colons:

`jdbc:<subprotocol>:<subname>`

The three parts of a JDBC URL are broken down as follows:

1. `jdbc` - the protocol. The protocol in a JDBC URL is always `jdbc`.
2. `<subprotocol>` - the name of the driver or the name of a database connectivity mechanism, which may be supported by one or more drivers. A prominent example of a subprotocol name is `odbc`, which has been reserved for URLs that specify ODBC data source names. For example, to access a database through a JDBC-ODBC bridge, you have to use a following URL:

`jdbc:odbc:Northwind`

In this example, the subprotocol is `odbc` and the subname `Northwind` is a local ODBC data source, i.e. `Northwind` is specified as a system DSN under ODBC.

3. `<subname>` - a way to identify the database. The subname can vary, depending on the subprotocol, and it can have a subsubname with any internal syntax the driver writer chooses. The function of a subname is to give enough information to locate the database. In the previous example, `Northwind` is enough because ODBC provides the remainder of the information.

Databases on a remote machine require additional information to be connected to. For example, if a database is to be accessed over your company Intranet, the network address should be included in the JDBC URL as part of the subname and should follow the standard URL naming convention of

`//hostname:port/subsubname`

Assuming you use a protocol called `VPN` for connecting to a machine on your company Intranet, the JDBC URL you need to use can look like this:

`jdbc:vpn://dbserver:791/sales` (similar to `jdbc:dbvendorname://machine-Name/SchemaName`)

It is important to remember that JDBC connects to a database driver, not the database itself. The JDBC URL that identifies the particular driver is determined by the database driver vendor. Usually, your database vendor also provides you with the appropriate drivers. It is highly recommended that users contact their database driver vendor for the correct JDBC URL that is needed to connect to the database driver.

Driver: This is the appropriate JDBC driver that is required to connect to the database. If you are using the JVM included within the installation (or Oracle's J2SE), use the following driver specification to connect to an ODBC data source:

`sun.jdbc.odbc.JdbcOdbcDriver`

You can also specify a JDBC driver name specific to your database if you are NOT using the JDBC-ODBC bridge. For example, the Oracle database engine will require `oracle.jdbc.OracleDriver`.

User Name: This is the login used for the database.

Password: Password for the above user.

Once you specify the connection information, you can test the database connection by clicking the *Test Connection* button. This will test the connection using the information you've provided and report any problems.

The *Default Options* portion of the dialog allows you to specify some properties for queries generated through the Query Builder interface or data views. You can specify whether to auto-join selected tables. Auto-join will attempt to join primary and foreign keys defined in the database. You can specify the table name format that should be used for queries either unqualified (only table name), or 2-part or 3-part qualified. Properties specified here will become the default setting for new queries and data views. They can also be modified for individual queries.

The *Multiple Database Options* portion of the dialog allows you to specify additional databases (i.e. additional database URLs) to obtain data from within the query. This option is only available when the database (original and

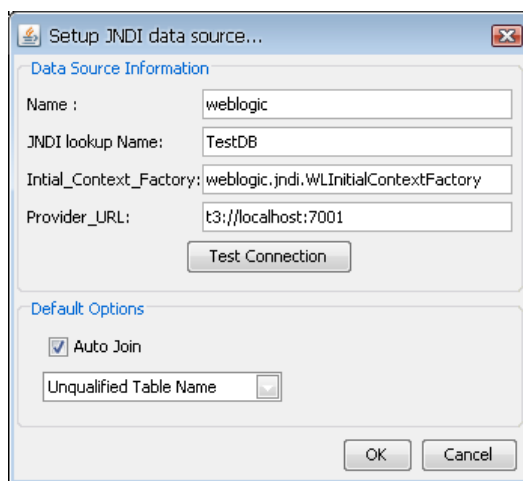
any additional database) is MS SQL Server and 3-Part Qualified Table Name option is chosen. Note that the same login details as well as the same driver (as defined in the original connection) are used to connect to the specified additional databases as well. The query can obtain data by referencing a column in the additional database using a 3-Part table nomenclature.

There are two sample databases included within the EDAB installation. One is a HSQL (a pure Java application database) database and the other one is a MS Access database. Both contain the same data and are located in `help/examples/DataSources/database` directory. For details about how to set up connections to these sample databases, please see Section Q.3.1.1 - Setup Database Connections of the Quick Start.

3.1.3.1. JNDI Data Sources

In addition to connecting to databases via JDBC, EDAB lets you use the JNDI (Java Naming and Directory Interface) to connect to data sources. In EDAB, JNDI data sources are treated just like database data sources and support the same functionality (queries, parameters, data views, etc.). The advantage of using a JNDI data source is that it potentially makes it easier to migrate reports between environments. If data sources in both environments are set up with the same lookup name, reports can be migrated without any changes.

To connect to a JNDI data source in EDAB, you must have a data source deployed in the same Web application environment as you are running the EDAB server. To set up a JNDI data source, select the *JNDIDataSources* node in the Data Source Manager and click the *Add* button. This will bring up a dialog allowing you to specify the connection information.



JNDI Setup Dialog

The first option allows you to specify a display name for the data source. The second option allows you to specify the JNDI lookup name for the data source. The third option allows you to specify the initial context factory for the data source, and the last option allows you to specify the provider URL. This information will vary depending on the application server you're using as different vendors implement JNDI data sources differently. You can test the connection by clicking the *Test Connection* button.

3.1.3.2. Queries

Once you add a database, a new node for your database will appear in the Data Source Manager window. When you expand the node, you will see two more nodes, one called *Queries* and one called *Data Views*. These are the two ways to retrieve data from your database. To create a new query, select the *Queries* node and click the *Add* button. A dialog will come up prompting you to specify a query name and select whether you would like to enter SQL statement or launch the Query Builder.

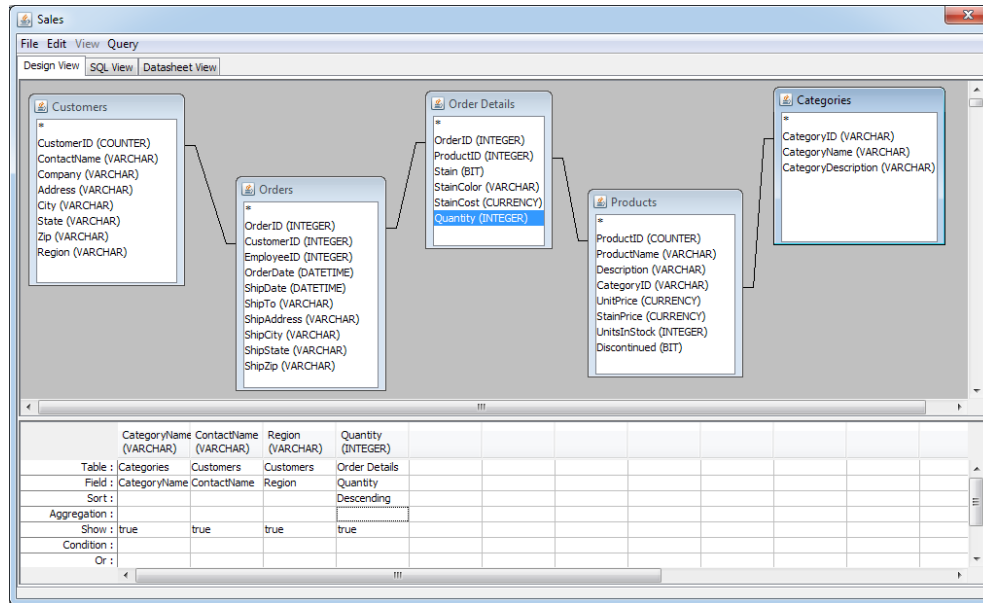
If you select to enter an SQL statement, a dialog box will come up allowing you to type in your SQL statement. From this dialog, you can also load a QRY or text file containing SQL text, or execute a stored procedure. If you select to launch the Query Builder, the Query Builder will open in a new window, allowing you to construct the query visually. After you finish building or entering the query, you will return to the Data Source Manager window and the query will appear as a new entry under the *Queries* node for your database.

3.1.3.2.1. Using Query Builder

The Query Builder is an integrated utility that allows you to construct queries against relational databases in a visual environment. To launch the query builder, add a new query within the Data Source Manager and select the *Open*

Query Builder option. The Query Builder will then open in a new window. You can also launch the query builder to modify an existing query by double clicking the query name in the Data Source Manager.

The main Query Builder window consists of two parts. The top half of the window contains all the database tables selected for the queries and their associated columns. The top window also shows what joins have been set up between column fields. The lower half of the main window or QBE (query by example) window contains columns that have been selected or built for the query and their associated conditions.



Query Builder Window

There are three tabs at the top of the Query Builder window. These allow you to toggle between different views. The *Design View* tab is the main designer window described above, the *SQL View* tab shows the SQL statement that is generated by the current query and the *Datasheet View* tab shows the query result.

When you finish constructing the query, select *Done* from the File menu to return to the Data Source Manager.

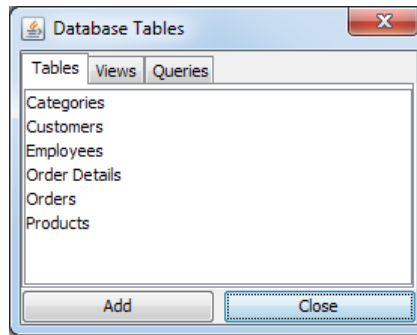
3.1.3.2.1.1. Tables

When the Query Builder launches for the first time, a tabbed window will appear containing a list of all the tables within the database. A second tab contains a list of all the views in the database, and the third tab contains a list of other queries you have designed for the database under a heading called *Queries*. From this window, you can select the tables/views/queries from which you would like to build the query. You can also load a previously designed query as a table. To add a table, select it and click the *Add* button or double click on the table name. When a table is added, it will appear in the main Query Builder window and will display all columns within that table. To remove a table, right click within the table and select *Delete* from the pop-up menu. You can also specify a table alias and sort the fields alphabetically from this menu. You can close the tables window by clicking the *Close* button. To re-open it, select *Show Tables* from the *Query* menu.



Note

By default, the tables will appear using the name format you specified when setting up the database connection. You can change the naming by selecting *Table Name Format* from the Query menu.



Query Builder Tables Window

3.1.3.2.1.2. Joins

When you select database tables for the query, the Query Builder can auto-detect joins between column fields based on primary key-foreign key relationships in the database. Auto-joins will be added depending on which option you selected when setting up the database connection. Auto-joins will create a standard join between tables. A join is represented by a line drawn between two fields in the top half of the design window. To remove a join or edit its properties, right click on the line and select your choice from the pop-up menu. To add a join, click and drag one column field to another in a different table. A join will then appear. You can change the auto-join settings by selecting *Auto Join* from the Query menu.

Join Properties:

Selecting *Join Properties* from the pop-up menu will bring up three options allowing you to select the type of join used between the column fields. Query Builder only supports equi-joins. Inequality joins can be easily accomplished using the *conditions* field. You can specify inner joins, left outer joins, and right outer joins. See the examples below for an explanation of the different join types.

Suppose you have the following two tables: Customers and Orders

CustomerID	CustomerName
1	Bob
2	Ivan
3	Sarah
4	Randy
5	Jennifer

OrderID	CustomerID	Sales
1	4	\$2,224
2	3	\$1,224
3	4	\$3,115
4	2	\$1,221

An inner join on **CustomerID** on the two tables will result in combining rows from the **Customers** and **Orders** tables in such a way that each row from the **Customers** table will be “joined” with all the rows in the **Orders** table with the matching **CustomerID** value. Rows from the **Customers** table with no matching **CustomerID** fields from the **Orders** table will not be included in the query result set.

Now suppose you create a query by selecting the **OrderID**, **CustomerName**, and **Sales** fields with an inner join on the **CustomerID** field. The select statement generated by the Query Builder would look like this:

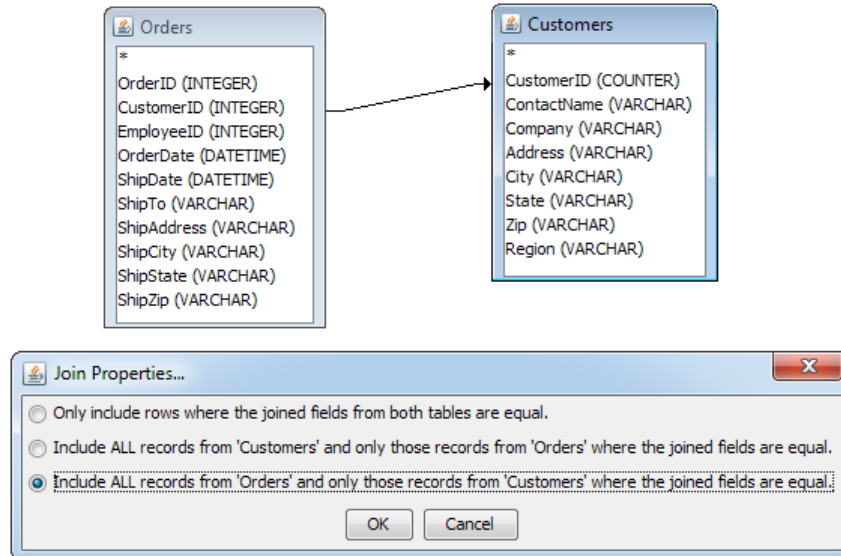
```
Select Orders.OrderID, Customers.CustomerName, Orders.Sales
From Customers, Orders
Where Customers.CustomerID = Orders.CustomerID
Order by Orders.OrderID;
```

The result of the query is shown below:

OrderID	CustomerName	Sales
1	Randy	\$2,224
2	Sarah	\$1,224
3	Randy	\$3,115
4	Ivan	\$1,221

As you can see, the **CustomerName** entries “Bob” and “Jennifer” do not appear in the result set. This is because neither customer has placed an order. There are situations where you may want to include all the records (in this example customer names) regardless whether matching records exist in the related tables(s) (in this case the **Orders** table). You can achieve this result using outer joins.

The Query Builder gives you the option of either right or left outer joins. The keywords “right” and “left” are not significant. It is determined by the order that the tables are selected in the Query Builder. If the outer table (the one that will have all records included regardless of matching join condition) is selected first, then Query Builder will use a right outer join. If the outer table is selected after the other join table, a left outer join is used. In our example, the **Customers** table has been selected before the **Orders** table, hence to select all of the records from the **CustomerName** field, the Query Builder will use a right outer join on the **CustomerID** fields.



Join Properties Dialog

Now, using the previous example, suppose you create the same query as before, except this time you specify to include all records from the **Customers** table. The select statement generated by the Query Builder would look like this:

```
Select Orders.OrderID, Customers.CustomerName, Orders.Sales
From Orders right outer join Customers on Orders.CustomerID =
    Customers.CustomerID
Order by Orders.OrderID;
```

The result of the new query is shown below:

OrderID	CustomerName	Sales
	Jennifer	
	Bob	
1	Randy	\$2,224

OrderID	CustomerName	Sales
2	Sarah	\$1,224
3	Randy	\$3,115
4	Ivan	\$1,221

As you can see, all of the customer names have now been selected and null values have been inserted into the result set where there are no corresponding records. If you specify an outer join, the join line connecting the two tables in the Query Builder will become an arrow in the direction of the join.

3.1.3.2.1.3. Columns

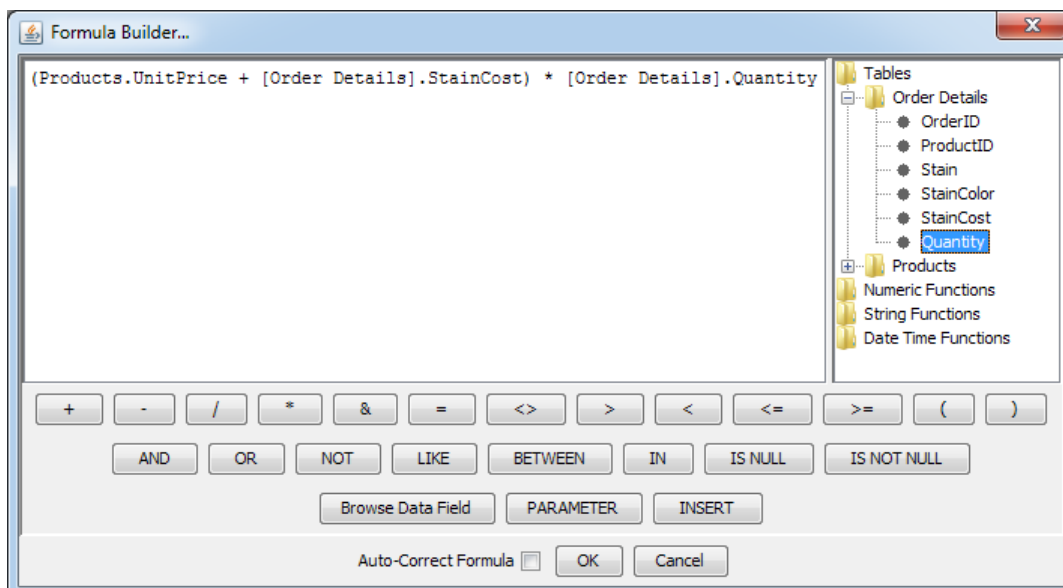
The QBE window contains information on column fields selected for the query, as well as any conditions for the selection.

Selecting Column Fields:

You can add column fields to the query from any table that has been selected in one of two ways. You can double-click on a field name within a table to add it to the query, or you can double-click on the *Table* or *Field* fields to bring up a drop-down menu with field choices. You can remove a column from the query by right clicking in the lower window and selecting *Delete Column* from the pop-up menu, or by selecting *Delete Column* from the Edit menu. Once you select a column field, you can specify how you want to sort the column, in ascending or descending order, by double clicking on the *Sort* field. You can also specify group by or column aggregation by double clicking on the *Aggregation* field. Aggregation options include: *Group By*, *Sum*, *Average*, *Min*, *Max*, *Count*, *Standard Deviation*, *Variance*, *First*, *Last*, and *Where*. If you select group by for one column, then you are required to specify group by (or aggregation) for all of the other columns. To specify a column alias, right click on the column and select *Alias* from the pop-up menu. You can perform a *SELECT DISTINCT* operation by selecting the *Select Distinct* option from the query menu.

Building Columns:

To build your own column, right click on a blank column in the QBE window and select *Build* from the pop-up menu. This will launch the Formula Builder. The Formula Builder allows you to construct columns in a visual environment using the tables that you have selected and the formula library for the database that you are using. You can click the *Browse Data Field* button to see the first few records of data for any field in your query.



Formula Builder Window

Conditions: You can place conditions on the query selection by entering them in the *Condition* or *Or* fields. A condition placed in the *Condition* field creates an AND clause within the generated SQL. A condition placed in the *Or* field creates an OR clause within the SQL. Right clicking in either field and selecting *Build* from the pop-up menu will bring up the Formula Builder. In the Formula Builder, you can specify standard conditions, =, <, >, BETWEEN, LIKE, NOT, etc., as well as construct formulas to filter the query. You can also specify a query parameter here.



Note

EDAB can auto-correct items entered as query conditions by appropriately appending the field name and encasing string arguments in quotes. For example, if you enter = ARC, EDAB will change the query condition to `Categories.CategoryName='ARC'`. If you're using complex functions (i.e. database functions that take multiple string arguments), EDAB may not be able to properly parse the function. You can turn off the auto-correct feature by un-checking the box at the bottom of the formula builder window.

3.1.3.2.1.4. Using Database Functions

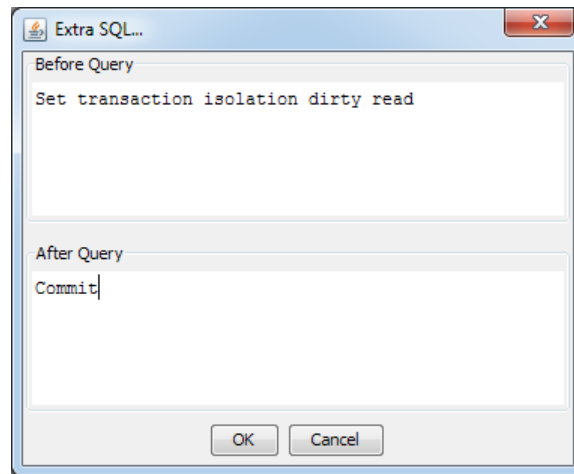
The formula builder component in the query builder allows you to use database specific functions when building a column or condition for the query. You can use the functions that are supplied or add your own to the interface.

EDAB comes with the function libraries for Oracle, Access, MS SQL, and DB2 pre-loaded. They are stored in XML format in `DatabaseFunctions.xml` file in `userdb` directory. For databases with functions not stored in XML, EDAB will use default ones. You can specify different database functions by editing the XML file or creating a new one based on `DatabaseFunctions.dtd` file in `userdb` directory. A sample database functions file might look like this:

```
<DatabaseFunctions>
  <Database ProductName="ACCESS">
    <FunctionSet Name="Numeric Functions">
      <Function>Abs(number)</Function>
      <Function>Atn(number)</Function>
    </FunctionSet>
  </Database>
</DatabaseFunctions>
```

3.1.3.2.1.5. Adding Extra SQL

Sometimes it is necessary to add extra SQL statements to run before or after a query. For example, you may need to set a transaction level or call a stored procedure before executing a query and/or commit a transaction or drop a temporary table after executing a query. The query builder allows you to specify these extra SQL statements by selecting *Extra SQL* from the Query menu. This will bring up a window allowing you to write statements to execute before and/or after a query.



Extra SQL Dialog

You can enter any SQL statements you would like to run before and/or after the query in the appropriate boxes. Once you finish, click the *Ok* button and the statements will be added to the query.

3.1.3.2.1.6. Query Output

The *SQL View* and *Datasheet View* tabs allow you to see two different views of the query.

SQL View:

The *SQL View* tab shows you the SQL statement generated by the query in the design view. It allows you to see how the Query Builder is translating different operations into SQL. You can edit the generated SQL if you want, however, if you change the SQL and then return to the *Design View*, all changes you made will be lost. If you save a query after changing the SQL, the query will re-open in the *SQL View* tab if you select to edit it.

Datasheet View:

The *Datasheet View* tab shows you the query result in data table form (this tab is also available in the Enter SQL dialog). The datasheet view will show you all the data that is drawn as a result of executing the query. Going to the datasheet view will also test the query to check for design errors. You can navigate the query result by using the toolbar at the bottom of the window.



Go to the first page of the data table



Go to the previous page of the data table



Go to a specific row of data



Go to the next page of the data table



Go to the last page of the data table



Set number of rows to display per page (default is 30)

Exporting Queries:

You can export queries in one of two ways. You can output the SQL statement as a text or you can output the query result as CSV file. To export a query, select *Export* from the File menu. A second menu will appear giving you the option to *Generate SQL* or *Generate CSV*. Select the desired option and a dialog box will appear prompting you to specify the filename and location.



Note

To save the query and exit the Query Builder, select *Done* from the File menu.

3.1.3.2.2. Parameterized Queries

You can also use the Query Builder to design parameterized queries. This feature allows you to filter the data at run-time.

Query parameters can be defined when typing an SQL statement or using the Query Builder. They can also be defined when running data views (this is covered in the next section). A parameter is specified within an SQL statement by the " : " character. Generally, the parameter is placed in the WHERE clause of an SQL Select statement. For example, the following SQL statement

```
Select * From Products Where ProductName = :Name
```

specifies a parameter called Name. You can then enter a product name at run-time and only retrieve data for that product.

Within the Query Builder, you can specify a query parameter by right clicking on the *Condition* field and selecting *Build* from the pop-up menu. The Formula Builder will open, allowing you to place a condition on the column.



Specifying a Parameter in the Formula Builder

You can insert a parameter by clicking the *PARAMETER* button. A second dialog will appear prompting you to specify a name for the parameter. Type the parameter name, click *OK* and then click *OK* again to close the formula builder. You can specify as many different parameters for query as you like.

3.1.3.2.2.1. Multi-Value Parameters

EDAB supports a special kind of parameter that takes an array of values as input rather than a single value. Multi-value parameters are useful when you want to filter the result set based on an unknown number of values. For example, say a report is run to return a list of customers for a specific state/province. Users could select as many different states/provinces as they wanted and return the relevant information.

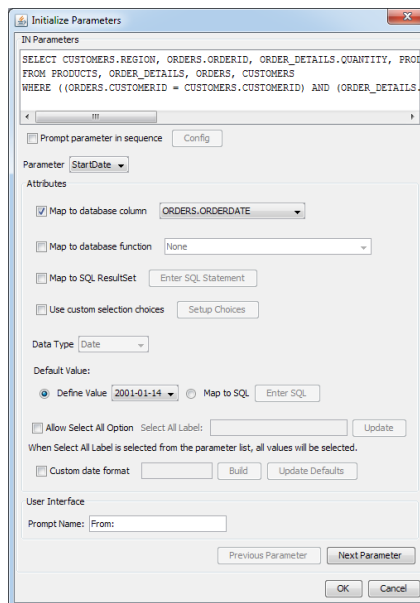
To create a multi-value parameter, place a parameter within an `IN` clause in an SQL statement. For example, the following query

```
Select Customers.Company, Customers.Address, Customers.City,
       Customers.State, Customers.Zip
From Customers
Where Customers.State IN (:State);
```

will create a multi-value parameter named `State`. Multi-value parameters will only be created if there is only one parameter in the `IN` clause. If you place more than one parameter in the `IN` clause, i.e. `Customers.State IN (:State1, :State2, :State3)`, it will create three single value parameters instead.

3.1.3.2.2.2. Initializing Query Parameters

When you attempt to save (by selecting *Done* from the File menu) or preview (by clicking the *Datasheet View* tab) a parameterized query, you will first be prompted to initialize the parameter. You can also initialize it by selecting *Initialize Parameters...* from the Query menu or by clicking the *Initialize Parameters* button in the Enter SQL Dialog.



Initialize Parameter Dialog

From this dialog you can specify the following options:

Map to database column:

This option allows you to specify a column from the database whose values will be used for the parameter input. Selecting this option modifies the parameter prompt that the end user will get when previewing or running the report in the Report Viewer. If you map the parameter to a database column, then the user will be prompted with a drop-down list of distinct values from which to select a parameter value. If you do not map, the user will have to type in a specific parameter value.

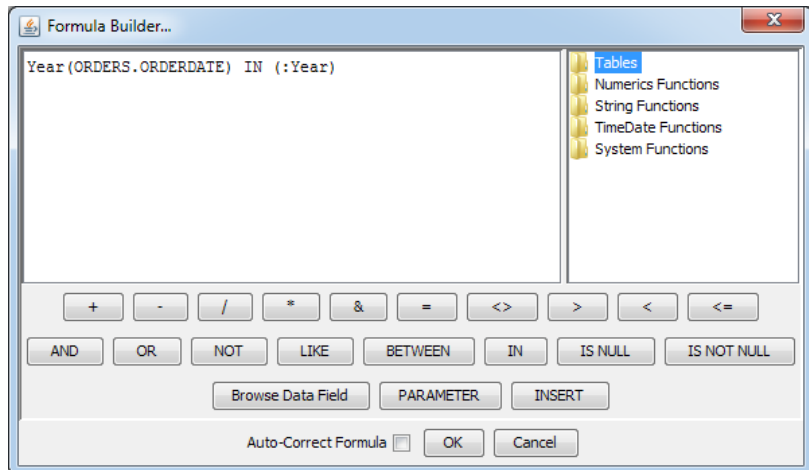


Tip

Normally, this drop-down list is populated by running a select distinct on the column while applying the joins and conditions from the query. If you prefer to get all data from the column without constraints (sometimes this can improve performance of the parameter prompts), you can set the *Distinct Parameter List Selection* option in the Admin Console. This will result in selecting the distinct values of the mapped column from the table that contains the column. For more information about EDAB server options, see Section 1.4.1.3 - Server Options.

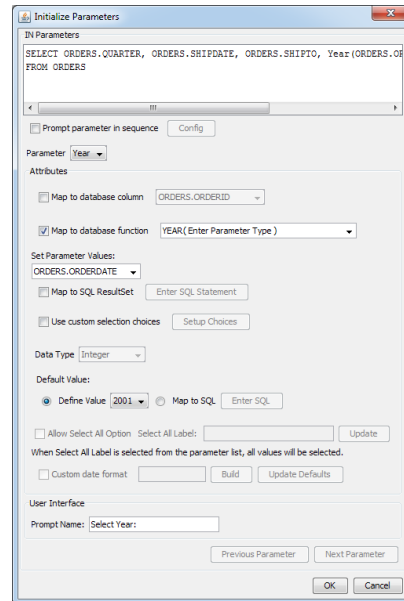
Map to database function:

The *map to database column* feature is very handy if you want to enter a valid value for a parameter from a list box, but sometimes you rather want a computed value or a derived value from a database column. For example, you want to find all orders from year 2007. However, OrderDate is a date. What you want is to apply the *Year* function to the OrderDate column. This is the impetus behind this feature. Mapping a parameter to a database function is very similar to mapping to a column. In the formula builder, enter a condition comparing a function result to a parameter as shown below:



Condition for Mapping to Database Function

In the initialize parameter dialog, check the *Map to database* function checkbox and the values will be automatically filled in.



Map Parameter to Database Function

The list of custom functions is extracted from DatabaseFunctions.xml file located in <EDAB Install>/userdb/ directory. Modify the .xml file if you wish to add a new database or custom functions. The new functions will appear in this list after you restart the program.

If your database is not listed in the .xml file, the function list will be populated with functions listed in the JDBC driver. However, the function parameters are not provided. For example, the HSQL database will not be listed in the .xml file.

An interesting example using the HSQL database is as follows. Suppose you would like to create a report for orders that were delayed. You can utilize the HSQL DateDiff function to find the number of days for the order to ship.

```
DATEDIFF('dd', ORDERS.ORDERDATE, ORDERS.SHIPDATE)
>= :ShipDelay
```

This function finds the difference between the order date and the ship date and displays the result in terms of days. If you initialize the parameter and check the map to database function, the following window will be displayed:

No Parameter Types for HSQL Function

The DateDiff function requires a string and two date values for the parameters. Enter these parameter types in the parentheses. This will bring up three set parameter value lists. Enter **dd** (day) for the first parameter, select `Orders.OrderDate` from the list for the second parameter, and select `Orders.ShipDate` from the list for the third parameter. The default values will be updated with the function results.

Map Parameter to HSQL Function

Map to SQL ResultSet:

A parameter mapped to a database column will give you a list of distinct values in a drop-down list box for the user to choose from when running the report. However, to produce the list of values, a select distinct on the column with the joins and conditions from the query will be run. In some cases, this can be a time-consuming process. To obviate this problem, and in fact gain complete control as to what and how to populate the drop-down list box, you can write your own select statement to populate the drop-down

list. An added bonus is that parameters that are in the query can be included in this query. With proper joins and parameters included, you can use this feature to facilitate cascading parameters (See Section 3.1.3.2.2.3 - Cascading Parameters). Below is an example:

Suppose you have two parameters in the query. So, your query is as follows:

```
SELECT CATEGORIES.CATEGORYID,
       PRODUCTS.PRODUCTNAME, PRODUCTS.UNITPRICE,
       PRODUCTS.UNITSINSTOCK
FROM PRODUCTS, CATEGORIES
WHERE ((PRODUCTS.CATEGORYID =
        CATEGORIES.CATEGORYID))
AND (((CATEGORIES.CATEGORYID =:category) AND
        (PRODUCTS.PRODUCTNAME =:product)))
```

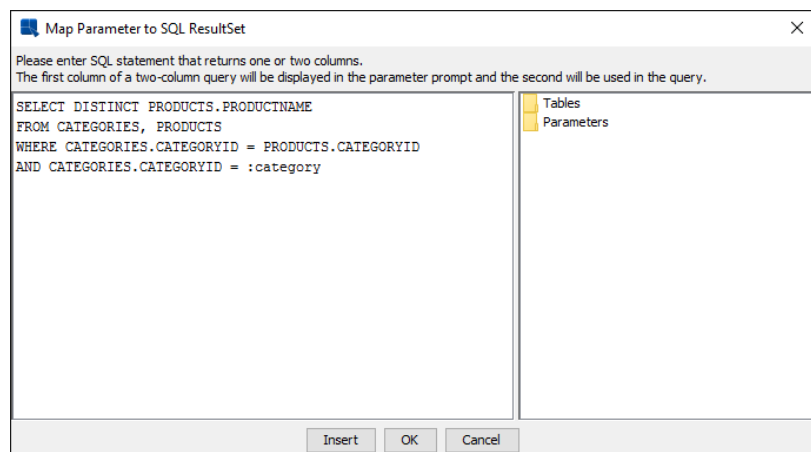
In the config prompt in `initialize` parameter, set the order for parameter prompting to `category` first, then `product`.

The select statement for parameter `category` can be the following:

```
SELECT DISTINCT CATEGORIES.CATEGORYID
FROM CATEGORIES
```

The select statement for parameter `product` will be as shown below:

```
SELECT DISTINCT PRODUCTS.PRODUCTNAME
FROM CATEGORIES, PRODUCTS
WHERE CATEGORIES.CATEGORYID = PRODUCTS.CATEGORYID
AND CATEGORIES.CATEGORYID = :category
```



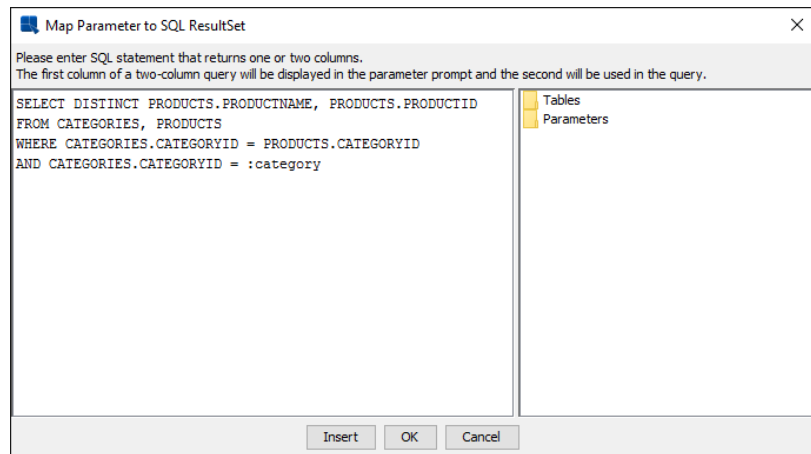
Select Statement for Product

When the user runs the template, `category` will be prompted first. Then the value of `category` chosen will be used to filter for `product`.

The select statement mapped to a parameter can have either one or two columns in the select list. It is clear that if one column is in the select list, it must be the column that supplies list of distinct values for the parameter. Another useful feature provided here is that you can actually select two columns

in the select list such that the first one of the columns will supply values for the drop-down list while the second column will be the actual parameter value for the filter condition. Consider the following example.

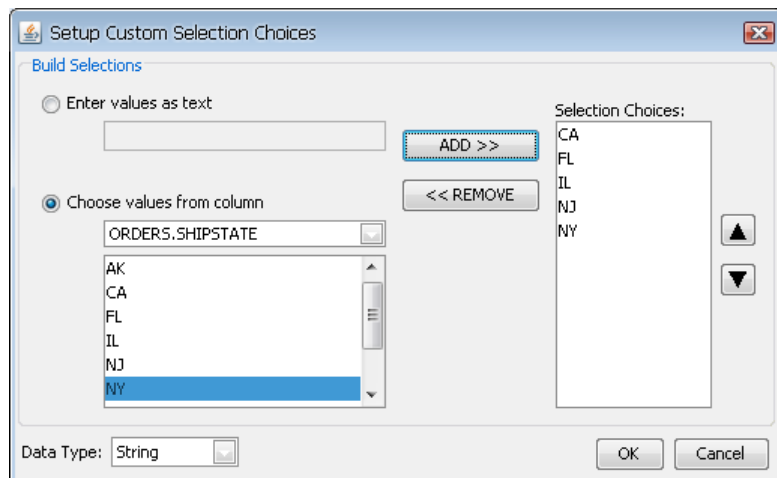
Suppose your database has a table with product ID as the primary key. When your end user wants to search for products from the database, they want to use the *product name* as parameter (therefore it is listed in the query as first) since a *product ID* could be just a cryptic code (therefore it is listed in the query as second). Using this feature, you can choose product name for the values in the drop-down list and product ID as the actual value filter condition.



Select Statement with Two Columns

Use custom selection choices:

Rather than having a drop-down menu with all the distinct column values, you can also create a custom list of parameter values. To create this list, select this option and click the *Setup Choices* button. This will launch a new dialog allowing you to create a list of choices.



Custom Parameter List Dialog

In this dialog, you can either enter custom values or select values from the distinct values of a column in the database. Once you finish specifying the values for the list, click the *OK* button and the choices will be saved.

Default Value:

This allows you to specify a default value for the parameter. Although you don't have to specify a default value, it is recommended that you do so. If

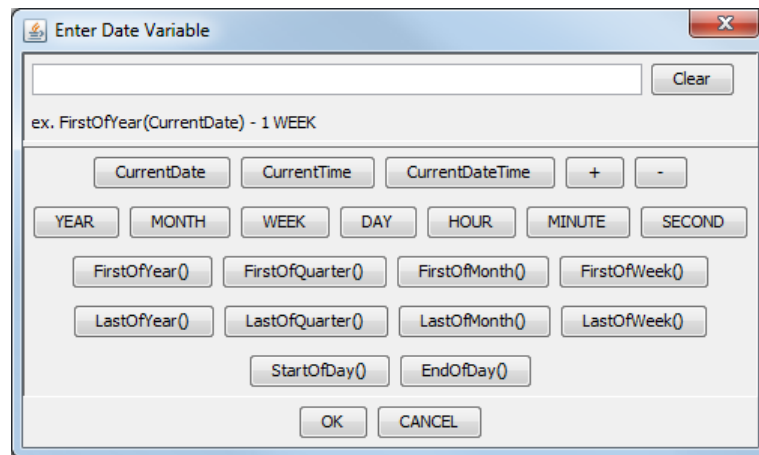
you do not supply a default value, you cannot open or manipulate the report template without the data source present.

You can either select a single value manually (either choose it from a list or type it manually, it depends on the mapping method you chose) or map the default value to a SQL query.

For multi-value parameters (see Section 3.1.3.2.2.1 - Multi-Value Parameters), the SQL query can return more than one value. In such case, several values will be chosen as default parameter values.

Date Variable:

This option is only available when the parameter is not mapped to a database column or function, or it's mapped to a SQL resultset and not set to a custom selection choice. This option is only intended for parameters with variable type date/time. When you click this button, the following panel will pop up, listing all the supported keywords.



Enter Date Variable Dialog

This dialog allows you to select one of the three keywords: `CurrentDate`, `CurrentTime`, and `CurrentDateTime`. You can add or subtract units of time from the current date/time, allowing you to have a dynamic date range. For example, a report can have the following default values:

```
StartDate: CurrentDate - 1 WEEK
EndDate: CurrentDate
```

This would indicate that every time the report is run, the default prompt should be one week ago to the current date. Other supported time units are `YEAR`, `MONTH`, `DAY`, `HOUR`, `MINUTE`, and `SECOND`. This feature only supports a single addition or subtraction, it does not support multi-value parameters.

You can also use functions to define the parameter value:

FirstOfYear()

Argument format: `CurrentDate`, `CurrentDateTime`,
e.g.
`FirstOfYear(CurrentDate)`

This function returns a date of the first day of the year from the argument. For example, when the argument evaluates to `2012-08-14`, the function returns `2012-01-01`.

LastOfYear()	<p>Argument format: CurrentDate, Current-DateTime, e.g. LastOfYear(CurrentDate)</p> <p>This function returns a date of the last day of the year from the argument. For example, when the argument evaluates to 2012-08-14, the function returns 2012-12-31.</p>
FirstOfQuarter()	<p>Argument format: CurrentDate, Current-DateTime, e.g. FirstOfQuarter(CurrentDate)</p> <p>This function returns a date of the first day of the quarter which includes the date from the argument. For example, when the argument evaluates to 2012-08-14, the function returns 2012-07-01.</p>
LastOfQuarter()	<p>Argument format: CurrentDate, Current-DateTime, e.g. LastOfQuarter(CurrentDate)</p> <p>This function returns a date of the last day of the quarter which includes the date from the argument. For example, when the argument evaluates to 2012-08-14, the function returns 2012-09-30.</p>
FirstOfMonth()	<p>Argument format: CurrentDate, Current-DateTime, e.g. FirstOfMonth(CurrentDate)</p> <p>This function returns a date of the first day of the month from the argument. For example, when the argument evaluates to 2012-08-14, the function returns 2012-08-01.</p>
LastOfMonth()	<p>Argument format: CurrentDate, Current-DateTime, e.g. LastOfMonth(CurrentDate)</p> <p>This function returns a date of the last day of the month from the argument. For example, when the argument evaluates to 2012-08-14, the function returns 2012-08-31.</p>
FirstOfWeek()	<p>Argument format: CurrentDate, Current-DateTime, e.g. FirstOfWeek(CurrentDate)</p> <p>This function returns a date of the first day of the week which includes the date from the argument. For example, when the argument evaluates to 2012-08-14, the function returns 2012-08-12 (Sunday is considered as the beginning of the week).</p>
LastOfWeek()	<p>Argument format: CurrentDate, Current-DateTime, e.g. LastOfWeek(CurrentDate)</p> <p>This function returns a date of the last day of the week which includes the date from the ar-</p>

gument. For example, when the argument evaluates to 2012-08-14, the function returns 2012-08-18 (Saturday is considered as the end of the week).

StartOfDay()

Argument format: CurrentTime, Current-DateTime, e.g.
StartOfDay(CurrentDateTime)

This function returns a time of the start of the day from the argument. For example, when the argument evaluates to 2012-08-14 12:15:03, the function returns 2012-08-14 00:00:00.0.

EndOfDay()

Argument format: CurrentTime, Current-DateTime, e.g.
EndOfDay(CurrentDateTime)

This function returns a time of the end of the day from the argument. For example, when the argument evaluates to 2012-08-14 12:15:03, the function returns 2012-08-14 23:59:59.999.

Data Type:

This allows you to specify data type for the parameter value(s). If you mapped the parameter to a column, the data type is set automatically.

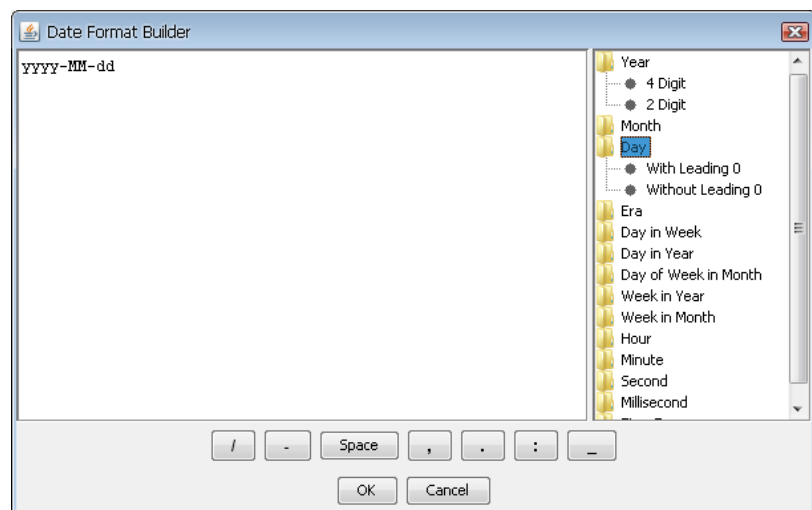
Allow Select All Option:

Use this to add an option to the parameter prompt dialog that will allow users to select all parameter values even for single-value parameters. See the Section 3.1.3.2.2.4 - All Parameters for more details.

Custom Date Format:

This allows you to set the format in which the date parameter should be entered. This option is only available when the data type is either date, time, or timestamp and the parameter is either mapped to column or not mapped at all.

When you check this option, the default custom date format is shown. These defaults can be changed in the admin console (See Section 1.4.1.3 - Server Options). Please note that the default date format only applies to database data sources. The date format is built using a combination of characters that represent date/time elements. You can build the format easily using the date format builder by clicking the *Build* button.



Date/Time Format Builder

The builder contains a list of elements available on the right. You can mouse over the elements to see an example of each presentation. The bottom section contains a set of separators available for use.

You can also type in the format by hand either in the builder or directly in the Custom Date Format input box. Formatting for this option is the same as for the format argument of the function.

Prompt Name:

This allows you to specify the prompt that is given to the user in the parameter dialog.

If you map the parameter, the user will see either a drop down box (single value parameter) or a list box (multi-value parameter) containing various options. If you choose not to map the parameter, the user will see a textbox to enter their own value. In case of a multi-value parameter, it is recommended to let the user inform in the parameter prompt that this parameter accepts multiple values. Users can separate multiple values using a comma (e.g. ARC, DOD, TRD). If the text requires the use of comma, the user can use quotes to include the comma within the filter string (e.g. "Doe, John", "Smith, Mike").

Clicking the *Previous Parameter* and the *Next Parameter* buttons allows you to initialize each of the parameters that have been defined in the query.

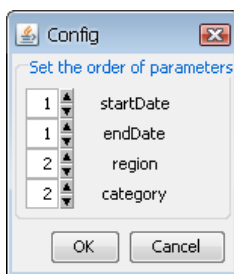
When you select to use a parameterized query to design a report, or open a report that uses a parameterized query, the report will load/start with the default values. You will be prompted to provide parameter values when you preview the report.

3.1.3.2.3. Cascading Parameters

By default, the user is prompted to enter all report parameters at once in the prompt dialog. This configuration, however, may not be the best approach if some parameters are mapped to database columns with a significant number of distinct values. It can be difficult to select from a very large list and depending on the parameter combination, users may be able to select parameters that do not return any data.

To assist with these problems, EDAB provides a feature that allows the user to configure the order in which the parameters should be entered. With this feature enabled, the user enters parameters in the dialog in a pre-defined order. As such, each selection will be applied as a filter to the next parameter prompt(s). Using cascading parameters can limit the number of distinct values presented to the user and can prevent the user from selecting invalid parameter combinations.

To enable cascading parameters, check the option marked *Prompt parameter in sequence* in the parameter initialization dialog. Then click the *Config* button to set the order of the parameter prompts. A dialog will open showing all the parameters defined in the query.



Parameter Sequence Dialog

Using the spin boxes, you can set the sequence for the query parameters. User will be prompted to start with the lowest numbered parameter and work his/her way up to the highest one. If two or more parameters share the same number, the user will be prompted to enter those parameters at the same time (in the same dialog).

By default, the parameter values for the next level are generated by rerunning the entire query with the previously prompted parameters filled in. If the original query is slow to execute, you can improve performance by mapping higher order parameters to SQL Queries. You can even include previously selected parameter values in the mapped query. For more information, please see Section 3.1.3.2.2.2 - Initializing Query Parameters.

Please note that this feature can only be used with reports.

3.1.3.2.2.4. All Parameters

Sometimes you want to select all parameter values at once. The *All Parameters* feature allows you to do so.

Single-value parameters

It is possible to select all parameter values at once, even for parameters that do not allow multi-value selection.



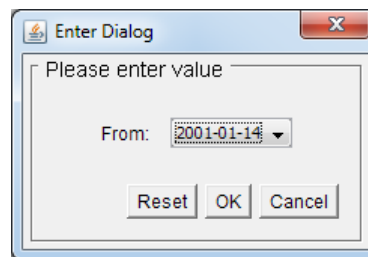
Note

There is a difference between multi-value selection and all-value selection. See the *Inner Workings* chapter to learn more.

For example: Let's assume you have a condition like this:

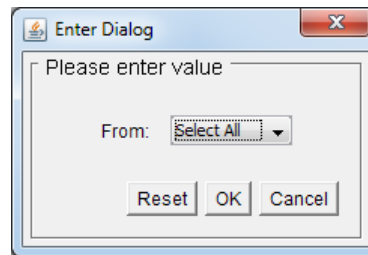
```
WHERE column = :Parameter
```

In such case, the parameter prompt dialog will not allow you to select more than one value.



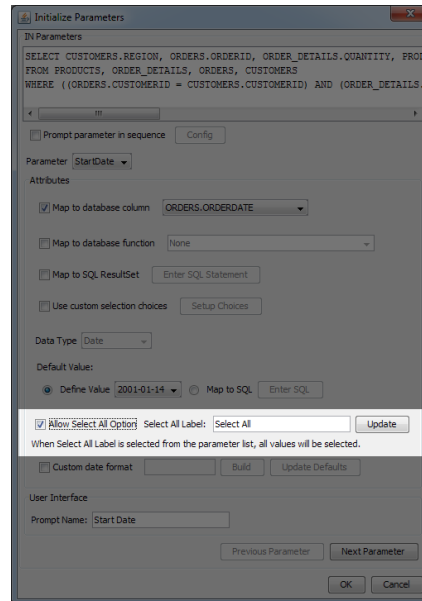
Typical single-value parameter

But you can use the *Select All Values* feature to add an option to the parameter value list that will allow viewers to select all parameter values at once (even if the parameter does not allow multi-value selection).



Single-value parameter with the Select All functions enabled

The *Select all* feature can be enabled in the *Initialize Parameters* dialog (see Section 3.1.3.2.2.2 - Initializing Query Parameters for more details) by selecting the *Allow Select All Option* checkbox.



This option is only available for parameters that meet the following requirements:

1. The parameter uses one of the following operators. If there are multiple occurrences of this parameter in the query, all parameter comparison operators have to be one of these:

< less than

<= less than or equal to

> greater than


>= greater than or equal to

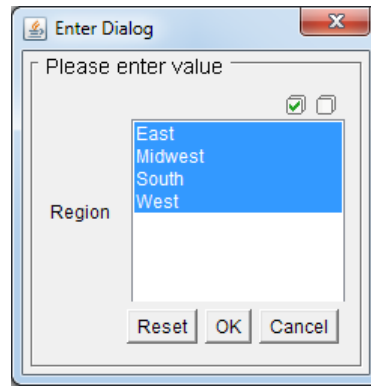
= equal to

2. The parameter is mapped to the same column as the column from the parameter condition **or** the parameter isn't mapped to anything.

After you select the *Allow Select All* option, the *Select All Label* field activates, allowing you to enter a text that will be used for selecting all data from the query. For parameters that are mapped to a column, this text will be displayed in the parameter value list in 1st place. For parameters that aren't mapped to anything, entering this text as the parameter value will result in selecting all data.

Multi-value parameters

Unlike single-value parameters, the *Select All* feature is enabled for all multi-value parameters by default (in fact, it can't be disabled, because disabling it for multi-value parameters would make no sense). All you have to do to use this feature is to click on the  *Select All* icon in the parameter prompt.



However, multi-value parameters can work in two modes:

1. If the parameter meets the conditions from the previous paragraph and the parameter is on the first cascading level (i.e. parameter cascading is disabled or the parameter is on the first cascading level), it is parsed by the SQL parser and the parameter condition is nullified. Nullifying the parameter optimizes the query and prevents it from causing performance issues or even errors. See the Inner Workings to learn more about how it works.
2. If the parameter doesn't meet the conditions from the previous paragraph or if it's not on the first cascading level, selected values will be injected to the query as a list of values separated by comma. If there is a large amount of values injected to the query as a list, the query can become quite long. Long queries can cause performance issues or even errors, so it is **not recommended** to use this option for parameters with many values.

Inner Workings

If a report/chart/map viewer chooses to select all parameter values for a single-value parameter or for a multi-value parameter that meets the conditions for parameter disabling, the query is then automatically parsed and a special condition is added to the parameter which basically disables the parameter.

For example: The following query

```
select *
from table
where column > parameter_value
```

Would be parsed and passed to the database as:

```
select *
from table
where ((column > parameter_value) OR (1 = 1))
```

This example also demonstrates another important thing: selecting all values for the < (less than) or > (greater than) operators returns all values from the table (if there are no other conditions) rather than returning no data at all (because condition like WHERE <all data from the Date column> > Date would return no data...).

Because EDAB allows you to use many database systems, parsing may fail for certain complex queries in certain databases. In such case a warning dialog will be displayed.

In such situations, you have the following three options:

1. Try to modify the query so it can be parsed by our parser.
2. Add your own *Select all parameters* condition to the query.

For example:

```
WHERE ((column = :Parameter) OR (:Parameter
LIKE 'selectall'))
```



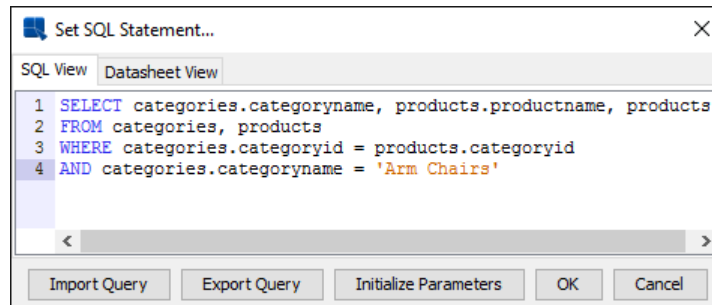
Note

If you embed all parameters directly to the query, leave the *Allow Select All Option* option **disabled**.

3. Contact Quadbase support [<https://www.quadbase.com/support-overview/>].

3.1.3.2.3. Entering SQL Statements

Typically, the Query Builder is recommended for creating queries. However, there are cases when it is necessary to enter SQL statements directly: for example if the query is already created in a QRY file, if the query is built into a stored procedure/function, or if the query requires commands not supported by the Query Builder. In these situations, select *Enter SQL statement* to open the Set SQL Statement window. Here you can enter SQL statements directly into the text area as shown below or you can load an existing QRY File.



Enter SQL Statement Dialog

To preview the result set, click on the *Datasheet View* tab.

3.1.3.2.3.1. Calling Oracle Stored Procedures

Compared to other database systems, Oracle uses a different approach when it comes to stored procedures and functions. For example, on MS SQL Server, using the EXEC command will return a result set. However, Oracle requires the use of an OUT parameter with a REF CURSOR type to return the result set. In addition, Oracle will not accept multiple statements from a single query. Therefore, it is necessary to store the query within a stored function and use special syntax to access the existing Oracle stored procedures.

To access your Oracle stored procedures, the first step is to define a weakly typed REF CURSOR using the following PL/SQL statement:

```
CREATE OR REPLACE PACKAGE types
AS

    TYPE ref_cursor IS REF CURSOR;

END;
```

This `ref_cursor` type will be used to store the query result set and return as an OUT parameter. The next step is to create a function which calls your stored procedure and executes your query. The following skeleton code will return a simple query using the `ref_cursor` type.

```
CREATE OR REPLACE FUNCTION my_function()

    RETURN types.ref_cursor

AS

    result_cursor types.ref_cursor;

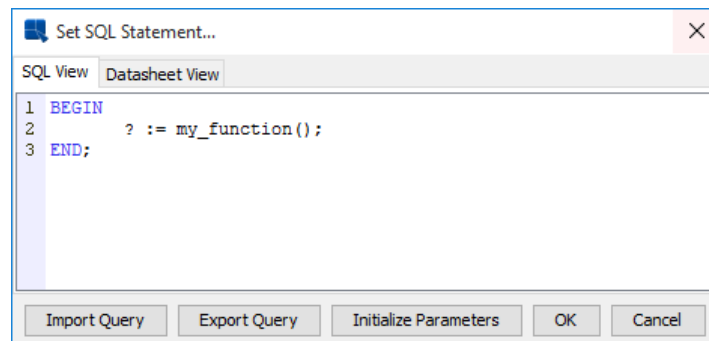
BEGIN

    do_stored_procedure();
    OPEN result_cursor FOR
        SELECT * FROM Categories

    RETURN result_cursor;

END;
```

Now that the Oracle stored function is set up, it can be easily called from ChartDesigner using a special PL/SQL like syntax. In the Set SQL Statement window, enter the following syntax to call the Oracle stored function:



Calling simple Oracle stored function

The `BEGIN ... END;` syntax alerts the system that the user is trying to access an Oracle stored function. And the “?” notifies the ChartDesigner that a variable is reserved for the OUT parameter. The JDBC syntax for calling Oracle stored procedures is as follows:

```
( call ? := my_function() )
```

However, EDAB does not support this format. Preview the results by clicking the *Datasheet View* tab.

Here is a more practical example to illustrate how stored procedures can be used with EDAB to develop useful solutions. Suppose you have a table called *employee_table* that stores an organization's location hierarchy such as the one shown below:

ID	NAME	PARENT	EMPLOYEE
1	All	NULL	0
2	America	1	0
3	Europe	1	0
4	New York	2	20

ID	NAME	PARENT	EMPLOYEE
5	Santa Clara	2	30
6	Dallas	2	12
7	London	3	14
8	Paris	3	11

The table lists the various corporate locations in a tree structure. The numbers of employees are stored in the leaf nodes (e.g. New York, London, etc.) and each node contains information about its immediate parent. Suppose you want to create a report that displays the number of employees in a certain region and information about the separate branches within that region. For example, if the user inputs ID = 2 (America), you want the report to display the total number of employees in America along with the branch locations. Using Oracle's CONNECT BY and START WITH clauses, the problem can be solved with two simple Oracle Stored Functions:

```
CREATE OR REPLACE FUNCTION sum_employees(locID IN NUMBER)

RETURN NUMBER

AS

sum_emp NUMBER;

BEGIN

SELECT sum(employee) INTO sum_emp
FROM employee_table
CONNECT BY PRIOR id = parent
START WITH id = locID;

RETURN sum_emp;

END;

CREATE OR REPLACE FUNCTION regional_employees (locID IN NUMBER)

RETURN types.ref_cursor

AS

result_cursor types.ref_cursor;

BEGIN

OPEN result_cursor FOR
SELECT id, name, sumEmployees(id) AS Employees
FROM employee_table
CONNECT BY prior id = parent
START WITH id = locID;

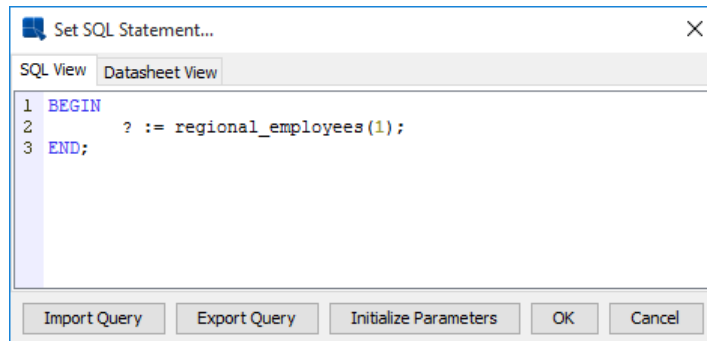
RETURN result_cursor;

END;
```

The function sum_employees takes the starting node as an argument and finds the sum of all leaf nodes that are descendants of that node. For example, sum_employees (3) returns 25 because there are 25 employees in Europe (14 in London, 11 in Paris). The second function, regional_employees, traverses through the tree structure

starting with the locID and builds a result set from the ID, Name and the result from the sum_employees function. The result set is then returned through a REF CURSOR.

To call a stored function that requires an argument, enter the following statements in the Set SQL Statement window:



Calling regional_employees function

Preview the results by clicking the *Datasheet View* tab.

SQL View	Datasheet View	
ID	NAME	EMPLOYEES
1	All	87
2	America	62
4	New York	20
5	Santa Clara	30
6	Dallas	12
3	Europe	25
7	London	14
8	Paris	11

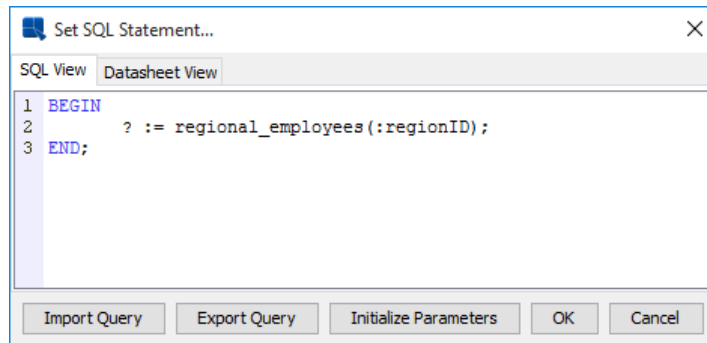
Result set from regional_employees

As seen in the results, the CONNECT BY clause traverses the tree recursively listing the American nodes together before listing the European nodes. If the user is only interested in European locations, they can enter 3 for the parameter and the following result set would return this:

SQL View	Datasheet View	
ID	NAME	EMPLOYEES
3	Europe	25
7	London	14
8	Paris	11

Result set from regional_employees in Europe

To create a parameterized report, use the :param_name syntax. The SQL parser in EDAB will be able to differentiate between the colon used for parameters and the one used for the assignment operator (:=). Here is an example of using the parameters:



Calling Oracle Stored Function using Parameter

When using IN parameters, it is necessary to initialize the parameters prior to executing the query. It is especially important to set the correct default data type for executing stored procedures because the parameters cannot be mapped to existing columns. You can find more information about initializing parameters in Section 3.1.3.2.2 - Parameterized Queries.

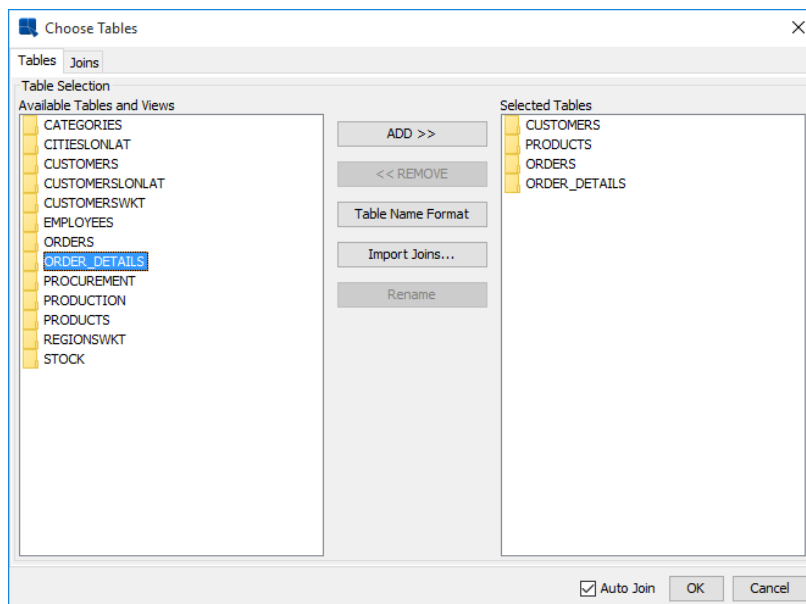
To try this example, <EDABInstall>\help\examples\DataSources\database\locationHierarchyExample.sql contains SQL commands to create employee_table as well as two stored functions.

3.1.3.3. Data Views

In addition to the query interfaces, EDAB provides another way of retrieving database data - data views. Data views provide a simplified view of the database in which users can design queries by simply selecting fields without using the Query Builder or having any knowledge of the underlying database structure. Using data views, administrators can predefine tables, joins, and fields, creating a local schema for the user to select from.

For example, an administrator could set up a data view for the sales department. The appropriate database tables and fields are pre-selected and grouped in a manner congruent with business users' logic. For example a group called *invoices* would have the appropriate customer and order fields. End users would then select this data view, pick the pertinent fields, specify a date range, and then begin designing a report or chart.

To create a data view, select the *Data Views* node in the Data Source Manager window and click the *Add* button. A new window will open allowing you to select the database tables you want to use for the data view.

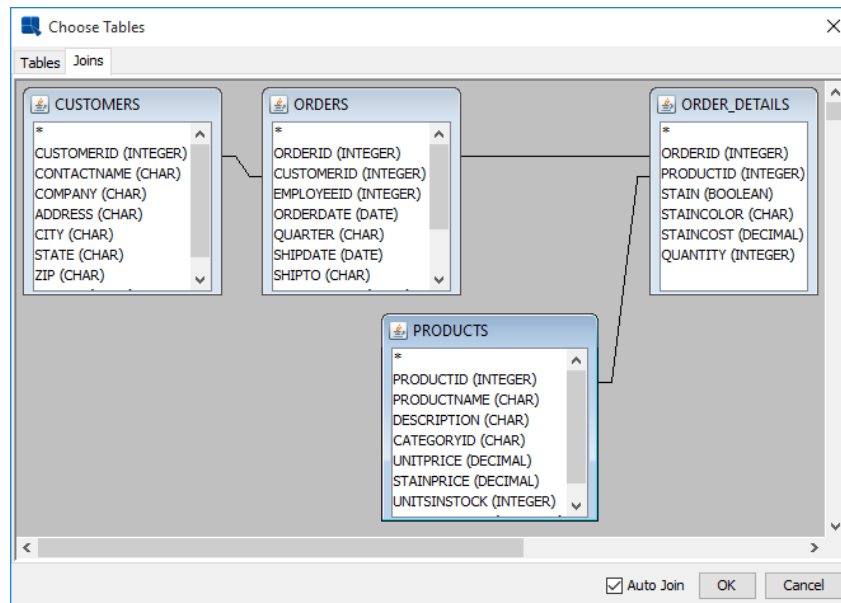


Data View Choose Tables Dialog

Left window contains all available database tables and views. You can add a table by selecting it in the left window and clicking the *ADD>>* button. By default, the data view will use the name format you specified when setting up the database connection. You can change the naming by clicking the *Table Name Format* button, or specify a

table alias by clicking the *Rename* button. You can also import selected tables and joins from another data view by clicking the *Import Joins...* button.

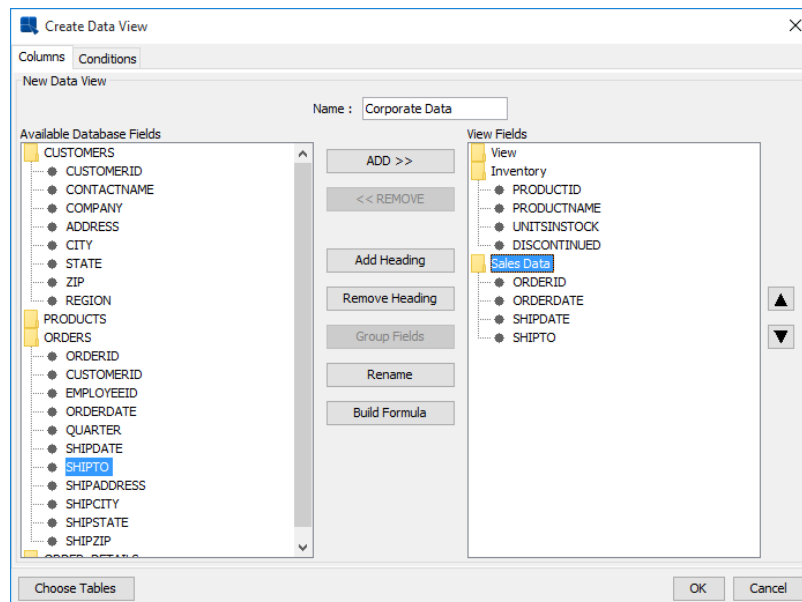
The *Joins* tab of this window allows you to specify the joins between the selected tables.



Data View Joins Dialog

The *Joins* tab shows all selected tables and their associated fields. The tables will be auto-joined depending on which option you selected when setting up the database connection. These auto-joins create a standard join between tables represented by a line. To remove a join or edit join properties, right click on the line and select your choice from the pop-up menu. To add a join, click and drag one column field to another in a different table. A join will then appear. Data views use the same join properties as the Query Builder. For more information about join properties, please see Section 3.1.3.2.1.2 - Joins.

After you finish selecting and joining tables, click the *OK* button and a new window will open allowing you to construct the data view.



Create Data View Dialog

The left window contains a list of tables you have selected and their associated fields. Each folder represents a table and can be opened and closed by double clicking. The right window contains fields that have been selected

for the data view. To add a field to a data view, select it in the left window and click the *ADD>>* button. Fields can be removed from the data view in the same way by selecting a field in the right window and clicking the *<<REMOVE* button. You can create a calculated column by clicking the *Build Formula* button. This will open the formula builder, allowing you to build the column. You can also define an alias by selecting any of the view fields in the right window and clicking the *Rename* button.

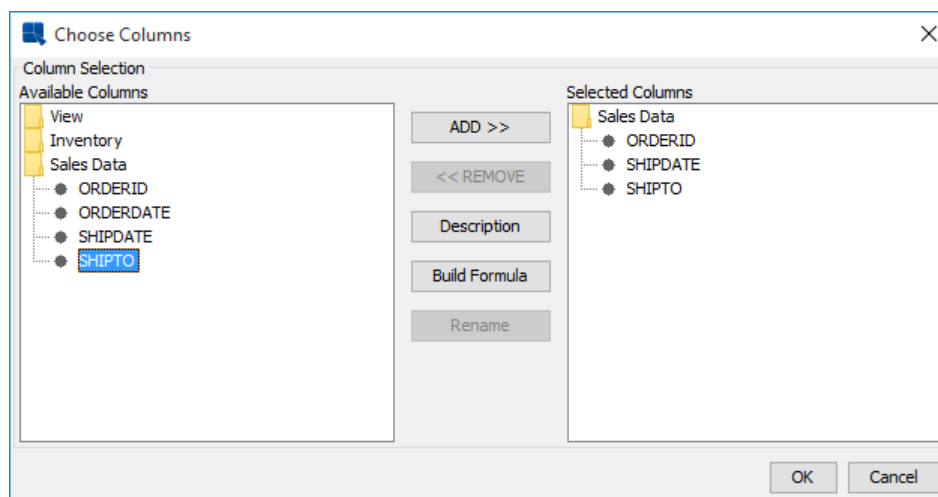
You can also group fields within the data view by adding headings. This allows you to create your own organizational structure of *virtual tables* that group data from different database tables under one heading. To create a heading, click the *Add Heading* button. You will then be prompted to specify a name for the heading. The new heading will then appear as a folder in the right window. To add fields under a heading, first select the fields you want to add from the right window and click the *Group Fields* button. You will then be presented with a drop-down menu, allowing you to select the heading under which you would like to add the fields.

The *Conditions* tab contains a formula builder window that allows you to specify certain filtering criteria for end users. Anything added in this window will be added to the *Where* clause of the generated SQL. For more information about using the formula builder, please see Section 3.1.3.2.1.3 - Columns.

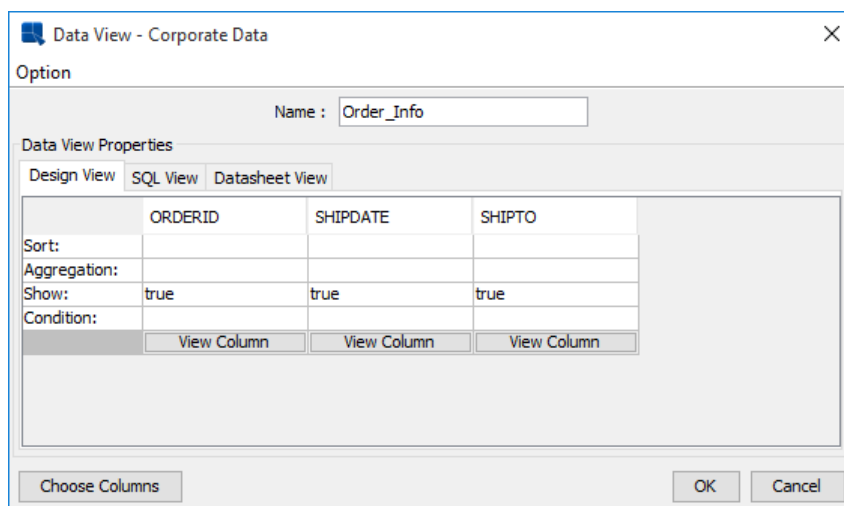
Once you finish creating the data view, click the *OK* button and the data view will be added to the Data Source Manager. Users can now use this view to construct ad-hoc queries.

When you design a report or chart using a data view as the data source (by selecting the data view and clicking the *Create Report* or *Create Chart* button), a window will open allowing you to select which fields in the view you want to use for the report. From this dialog, you can also build computed fields based on the available view columns.

After you select the fields, click the *OK* button and a new window will open allowing you to specify sorting, aggregation, and filtering conditions for the data view.



Data View Choose Fields Dialog



Data View Conditions Window

You can specify sorting, aggregation, and conditions for every field in the data view by double clicking on the respective field. Sorting and aggregation can be selected from drop-down menus. Double clicking on the *Conditions* field brings up a new window that allows you to specify simple selection criteria like $>$, $<$, $=$, and *between*. Users can build more advanced filtering criteria by right clicking on the *Conditions* field and selecting *Build* from the pop-up menu. This will open the Formula Builder window allowing you to build a condition. You can also display all of the unique values in the column by double clicking on the *View Column* button.

The Option menu in the upper left corner of the conditions window allows you to select a vertical/horizontal view for the conditions window, initialize any parameters in the data view, or save the query.

The selection set and conditions that you specify will be saved as a data view query with the name that you specify in the name field. Data view queries are saved under the node for the data view. A report created from the data view will refer to the data view query for updating/modification.

3.1.3.3.1. Data View Parameters

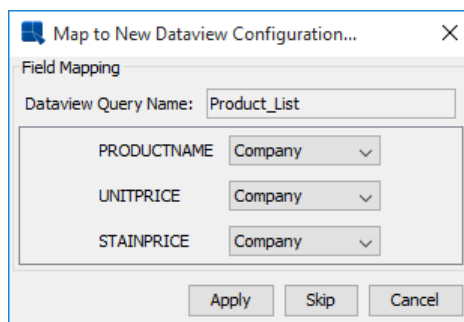
As with Query Builder, users can specify query parameters in Data Views. To add a parameter to a data view, select a data view in the Data Source Manager and click *View* to run the data view. After you select fields for the data view and you are in the conditions window, right-click in the *Condition* field for a column and select *Build* from the pop-up menu. This will bring up the formula builder, allowing you to specify a parameter in the same way as in Query Builder. For more information about this, please see Section 3.1.3.2.2 - Parameterized Queries.

Once you enter the parameter, you will be prompted to initialize it if you go to the *Datasheet View* tab and then click *Ok* to continue with the report wizard, or if you save the selections as a query. You can also initialize the parameter by selecting *Initialize Parameters* from the Option menu.

3.1.3.3.2. Updating Data View Queries

Sometimes you may need to make changes to the structure/make-up of the data view as your data model or requirements change. Changes could include adding/removing fields or re-naming them. You can propagate changes from the data view to its associated queries by selecting it in the data source manager and selecting *Data View Queries* from the Update menu.

All of the queries associated with the view will be scanned and any inconsistencies in fields or field names will be presented for you to update.



Update Query Fields Dialog

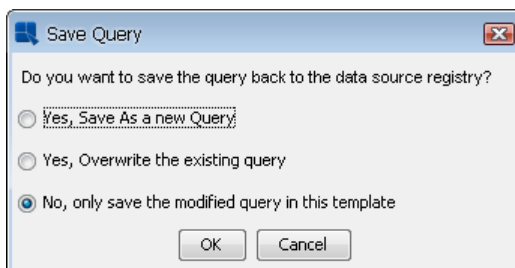
For each query, you will be prompted to change any fields that no longer match the data view structure. For each field, you can select a field from the data view to map it to, or remove the field from the query. If you do not want to change anything in the query, you can click the *Skip* button. The query will continue to run, but it will refer to the old data view structure. Click the *Apply* button to save the changes to the data view query.

3.1.3.4. Editing Queries

If you have selected to build a report using database data either by designing a query in the Query Builder, writing an SQL statement, or running a data view, you can modify the query directly from the ChartDesigner without having to go back to the Data Source Manager.

To modify a report or chart's query, select *Modify Query* from the Data menu in ChartDesigner. If you have designed a query in the Query Builder, then the Query Builder interface will re-open allowing you to modify the query. If you have entered an SQL statement, a text box will open allowing you to modify the SQL. If you have used a data view, the data view conditions window will re-open allowing you to change the filters or pick additional fields.

Once you specify all the changes, you will be given an option to modify the query in the data registry, save a new query in the data registry, or modify only the query in the template.

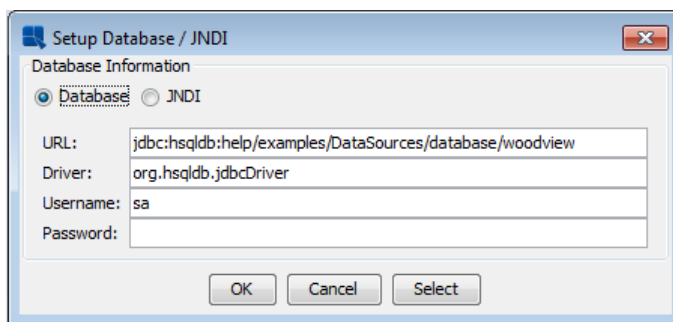


Saving Query Options

Once you specify the save options, the modified query will be applied to the report or chart. Note that if you made significant changes to the query, you may need to perform the data mapping again.

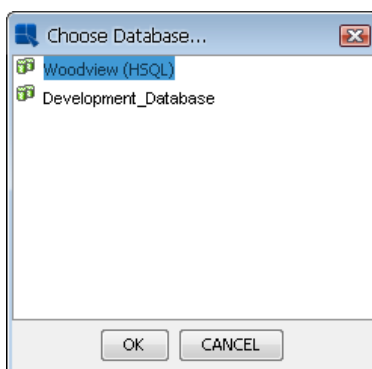
3.1.3.5. Editing Database Connections

If you have selected to build a report or chart using database data, you can also directly edit a template's database connection information from within the Chart Designer. To modify the connection information in the chart, select *Modify Database* from the Data menu. This will bring up a dialog allowing you to specify a different setting for the chart to use when connecting to the database or JNDI data source.



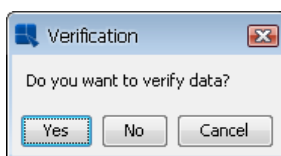
Change Database Connection Dialog

In addition to manually entering the database information, you can retrieve the database connection information from a data registry. To do this, click the *Select* button on the Database Connection dialog. This will allow you to browse to XML registry file from which you want to pull the database connection. When you select a registry file, you will be presented with a list of databases defined in the registry.



Select Database from Registry

Select the database that you want to use and click the *Ok* button. The connection information for that database will be automatically applied to the connection dialog. After you set the connection information for the template, click the *Ok* button to apply the changes. A dialog will open asking you if you would like to verify the new connection information.



Verify Connection Dialog

Unless you know that data source isn't present, it's generally a good idea to check that the supplied connection information is correct.

Unlike the modify query feature, changes to a template's database connection will only saved be to the template. It will not be saved to the data registry.

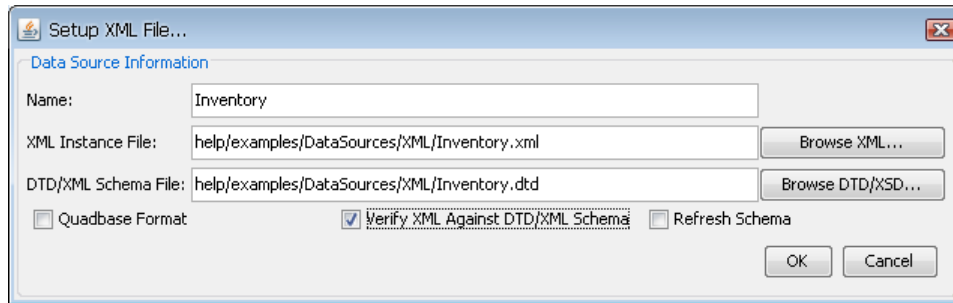
3.1.3.6. Troubleshooting Database Connections

If you're having difficulty connecting to your database via JDBC, EDAB provides a small utility you can use to help troubleshoot the problem. To launch the utility, point your Web browser to `http://machinename:port/EDAB/TestConnection.jsp`. This page opens a small form that allows you to enter database connection information as well as sample query.

The utility will directly contact the database (outside of any EDAB components) and execute the query, if provided. Any connection/query errors and suggested fixes will be printed on the page.

3.1.4. Data from XML and XBRL Files

In addition to relational databases, EDAB allows you to retrieve data and query XML files. XML data can be in virtually any format, but you need to specify a DTD file or an XML Schema (XSD) along with the XML data. To set up an XML data source, select *XMLFiles* node in the Data Source Manager and click the *Add* button. A dialog will open prompting you to specify options for the new XML source.



Setup XML Data Source Dialog

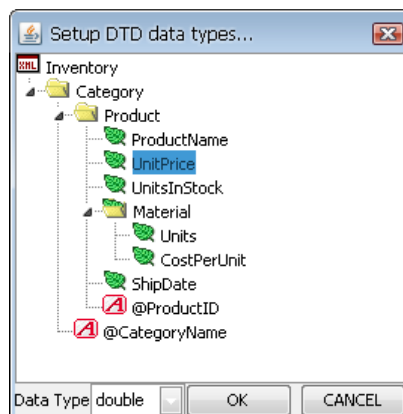
The first option allows you to specify a display name for the XML data source. The second option allows you to specify the location of the XML file from which you want to retrieve data. Note that you can also specify an XBRL file in this field. You can set up a data source that retrieves XML data from an HTTP server here as well, by adding the appropriate URL as the file location. The third option allows you to specify the location of a valid DTD or XML schema file for the XML file.

The *Quadbase Format* checkbox allows you to indicate whether the XML file is in the form of an XML export from EDAB. For example, if you choose to export a report's data in XML format, you can read it back in using this format. When you use a file in this format, you do not have to specify a DTD or XML Schema.

The *Verify XML against DTD/XML Schema* checkbox will make sure that the supplied XML file/source complies with the layout specified in the DTD or XML schema file. Because queries are designed based on the structure of the DTD/XML Schema file, a non-conforming XML source could produce unexpected results. If the XML does not conform to the DTD or XML schema, you will be given a warning. You can, however, continue setting up the data source.

The *Refresh Schema* checkbox will only appear if you choose to edit an existing XML data source. Checking this option will reload the schema or DTD definition to incorporate any changes to the structure in the XML data source in the registry. This option is only necessary if the DTD or schema definition has changed since the data source was first created.

Once you finish setting up the XML file and the DTD/XML Schema, a new dialog will open allowing you to specify the data type for all the selectable elements in the XML data source. The dialog will be different depending on whether you are using a DTD file or XML Schema.



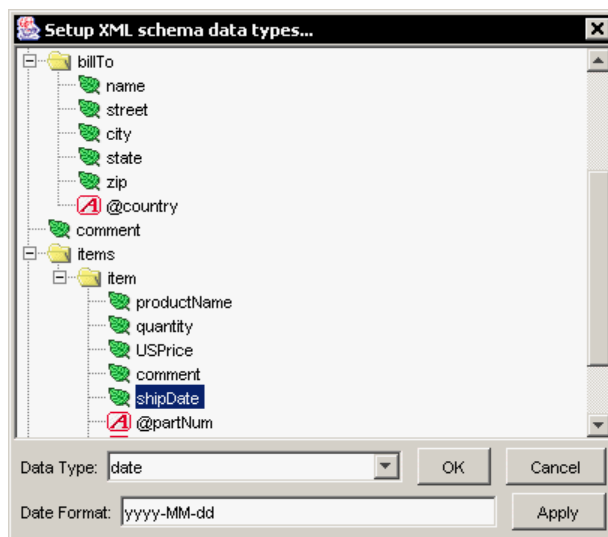
DTD Data Type Selection Dialog

Because DTD files do not specify a correct data type for elements, all elements are considered to be Strings by default. To change the data type of an element, you have to select the element and pick a data type from the drop-down window at the bottom of the dialog. To ensure proper results when you query the XML file, you should set the data type for all selectable elements. This includes leaf nodes, parent nodes that contain data, and attribute elements. The following data types are supported:

- String
- Integer
- Double
- Date (If you specify date as the data type, you will also be required to specify the date format.)
- Boolean

Once you finish specifying the data types, click the *OK* button and the XML source will be added to the Data Source Manager.

If you're using an XML schema, a different data types dialog will open.

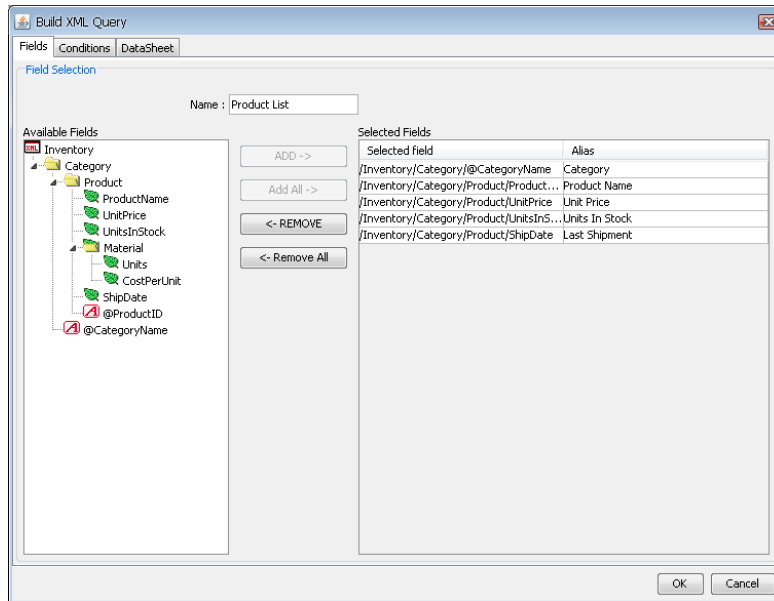


XML Schema Data Type Selection Dialog

Generally, the data types should already be defined in the XML schema file, but you can make any changes in this dialog. Once you finish specifying the data types, click the *OK* button and the XML source will be added to the Data Source Manager.

3.1.4.1. XMLQueries

Once you set up an XML data source, you can then create queries to select nodes, specify filtering conditions, and transform the tree structure into the tabular form used by EDAB. To add a query, select the node for your XML source and click the *Add* button. This will launch the XML query builder interface that allows you to construct a query.



XML Query Field Selection Tab

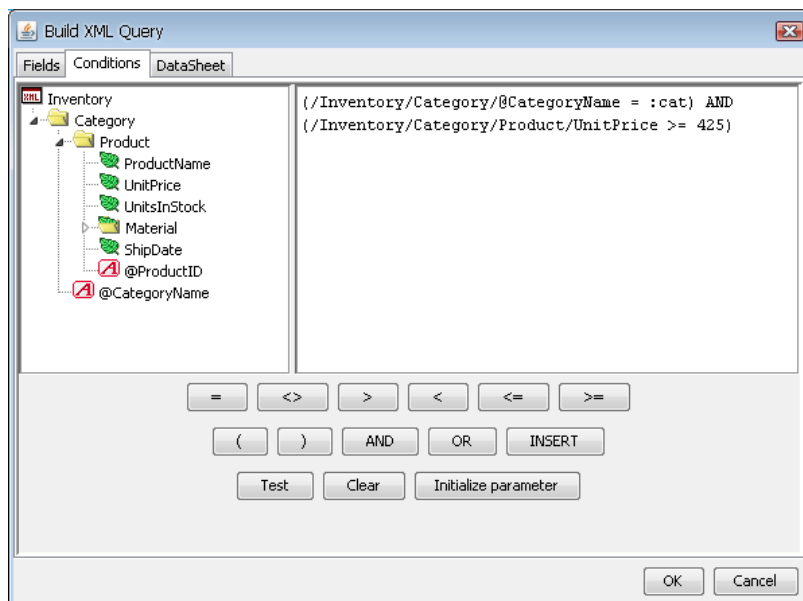
The first tab in the XML query builder allows you to select the fields/nodes from the XML file that you would like to use. The left side of the window contains the tree structure from the DTD or XML schema file. You can pick any selectable elements and add them to the query by clicking the *Add* button. Selected fields will appear in the right side of the window. You can specify an alias for any field by double clicking the *Alias* field for a column and typing the new column alias.



Note

Each selected element will become a column in the report or chart data. For results where a one-to-one relationship cannot be determined, the tabular structure is built using all available permutations in the data (similar to a cross-join in SQL). For best results, it is recommended that you select fields for a query where a clear hierarchical relationship is present.

The *Conditions* tab of the XML query builder allows you to specify some basic filtering criteria for your selection.



XML Query Conditions Tab

You can specify an equal, not equal, greater than, less than, less or equal to, or greater or equal to condition for any selectable element in the XML file. You can also use the AND and OR operators to build compound conditions. Fields are specified using a direct path down the XML tree. Currently, only direct path is supported. You cannot use more complex XPath expressions. To add a field, you can double click on it in the left side, or you can select it and click the *Insert* button.

After you finish writing the conditions, click the *Test* button to verify that the syntax is correct.

The *DataSheet* tab allows you to preview the query result and see how the XML data is converted to tabular form. You can navigate through the result set the same way as in Query Builder (Section 3.1.3.2.1.5 - Adding Extra SQL).

Once you select the fields and specify the appropriate conditions, click the *OK* button. The query will then be added as a new node under your XML source in the Data Source Manager and can now be used to create a report or chart.

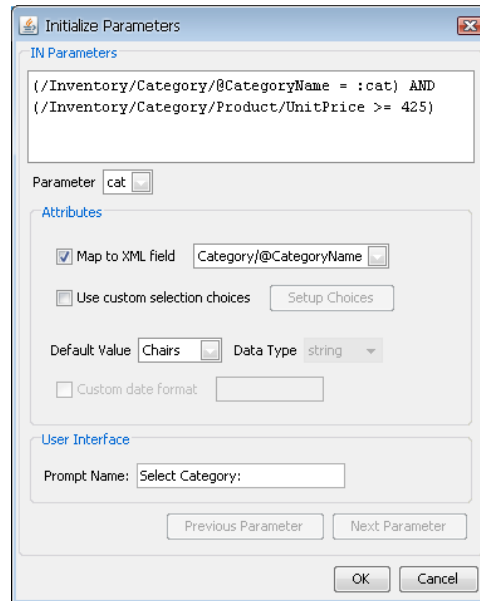
There is a sample XML file with DTD and XML schema descriptions included in the EDAB installation. The files are located in `help/examples/DataSources/XML` directory of your installation and are called `Inventory.xml`, `Inventory.dtd` and `Inventory_XSD.xml`, `Inventory.xsd` for the XML schema example. There is also a sample servlet that allows you to stream XML data to the Chart Designer. The servlet code and instructions are located in `help/examples/DataSources/XML/servlet` directory.

3.1.4.1.1. XML Parameters

As with database queries, you can also specify parameters for XML queries. The same syntax “:” is used to denote a parameter in the XML condition as it is in a query condition. So the following XML condition:

```
/Inventory/Category/@CategoryName = :category
```

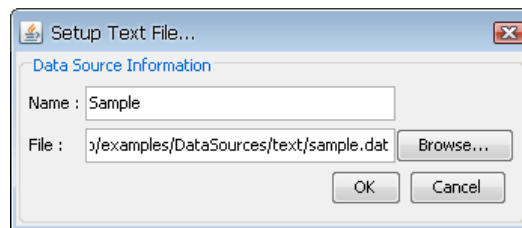
would place a dynamic filter on the query for the *CategoryName* attribute. XML parameters are initialized in the same way as query parameters. The initialization dialog will appear if you try to preview or close the query, or you can trigger it by clicking the *Initialize Parameters* button. The only difference is that instead of mapping to a database field, the parameter prompt can be mapped to a node in the XML file.



XML Parameter Initialization Dialog

3.1.5. Data from Text Files

EDAB also allows you to retrieve data from flat text files. To add a text file as a data source, select the *TXTFiles* node in the Data Source Manager and click the *Add* button. A dialog will open prompting you to specify a display name and the location of the text file you want to use.



Add Text Data Source Dialog



Tip

The text file can be also retrieved from URL. To do so, enter the URL to the *File* text field. You have to enter a full URL with protocol etc. (e.g. `http://www.quadbase.com/textfile.txt`).

After you specify the information, click the *OK* button and the text source will appear under the *TXTFiles* node of the Data Source Manager window.

3.1.5.1. Formatting Requirements for Text Files

There are certain formatting requirements for the data within a text file in order make it readable by EDAB. Generally, data is expected to be in a form similar to the following:

```
String,date,decimal
Name,day,Volume
"John", "1997-10-3", 32.3
"John", "1997-4-3", 20.2
"Mary", "1997-9-3", 10.2
"Mary", "1997-10-04", 18.6
```

The above data file is a plain text file. The first row specifies the data types and the second row specifies the field names. The third row and so on are the records. Every text file must consist of these three parts. There are four records, with three fields each in the example data file. The delimiter between the fields may be one of the following characters: " , " , " ; " , or " " (that is comma, semi-colon, or space). Each field can be put in quotes (single or double).

3.1.5.2. Data Types and Format for Text Files

In text data files, the data type is specified using a keyword. The following is a list of recognized keywords and their corresponding JDBC type and Java type.

Data File Keywords (Not Case Sensitive)	JDBC Type	Java Type in EDAB
Boolean, logical, bit	BIT	Boolean
tinyint	TINYINT	byte
smallint, short	SMALLINT	short
int, integer	INTEGER	int
long, bigint	BIGINT	long
float	FLOAT	double
real	REAL	float
double	DOUBLE	double
numeric	NUMERIC	java.math.BigDecimal
decimal	DECIMAL	java.math.BigDecimal
date	DATE	java.sql.Date
time	TIME	java.sql.Time
timestamp	TIMESTAMP	java.sql.Timestamp
string	CHAR	String
varchar	VARCHAR	String
longvarchar	LONGVARCHAR	String

For certain data types, the data in a text file must be presented in a specific format. The following is a list of the data types that require specific formatting:

Data Type	Format	Example
Date	yyyy-mm-dd or yyyy-mm	2001-06-12 or 2000-06
Time	hh:mm:ss	12:17:34
Timestamp	yyyy-mm-dd hh:mm:ss	2001-06-12 12:17:34
Boolean	true/false, t/f, 1/0 (case insensitive)	true

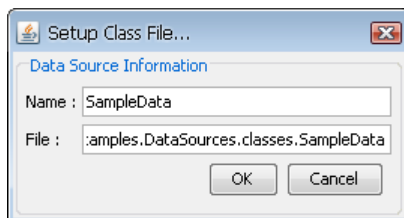
There is a sample text file included in the EDAB installation. The file is located in `help/examples/Data-Sources/text` directory of your installation and is called `Sample.dat`.

3.1.6. Data from Class Files

For maximum flexibility, EDAB allows you to design reports and charts using object or array data by providing an interface to pass data to the Chart Designer as an argument. Using the API, you can implement `IDataSource` to return an `IResultSet` object similar to the `java.sql.ResultSet` interface used for JDBC result sets. Users can provide their own implementation of `IResultSet` or use one provided by EDAB.

For more information about this feature, please see Section 8.5 - Class File Data Source.

To add a class file as a data source, select the *ClassFiles* node in the Data Source Manager and click the *Add* button. A dialog will open prompting you to specify a display name and the location of the class file you want to use.



Add Class File Dialog

There is sample code available in `help/examples/DataSources/classes` directory of the installation. The compiled code will generate a class file that passes the data array into Chart Designer. Note that in order to use a class file as a data source, you must have the file or directory containing the package in the classpath of your application server/servlet runner.

Here is a simple guide to use `SampleData.class` file available as datasource in EDAB Data Registry:

The most simple solution is to copy `SampleData.class` file from `<EDAB install dir>/help/examples/DataSources/classes` into `<EDAB install dir>/WEB-INF/classes/help/examples/DataSources/classes` directory (you will have to create `help/examples/DataSources/classes` under `<EDAB install dir>/WEB-INF/classes/` before copying the file).



Note

You can also change the app server CLASSPATH to include `<EDAB install dir>` instead. (Note that in Tomcat, you have to include all the jars, as setting the CLASSPATH in Tomcat will not pick up the jars under `EDAB/WEB-INF/lib`.)

Start or restart your Tomcat server and open EDAB Organizer - DataRegistry

Select *ClassFiles* and add the new classfile by filling in the dialog (as showed in the picture above):

Name: SampleData

File: help.examples.DataSources.classes.SampleData

Now you can view your ClassFile data and use it as datasource for reports and charts.

3.1.6.1. Parameterized Class Files

EDAB provides an additional API interface, `IPParameterizedDataSource`, that allows you to define report/chart parameters within the context of a class file data source.

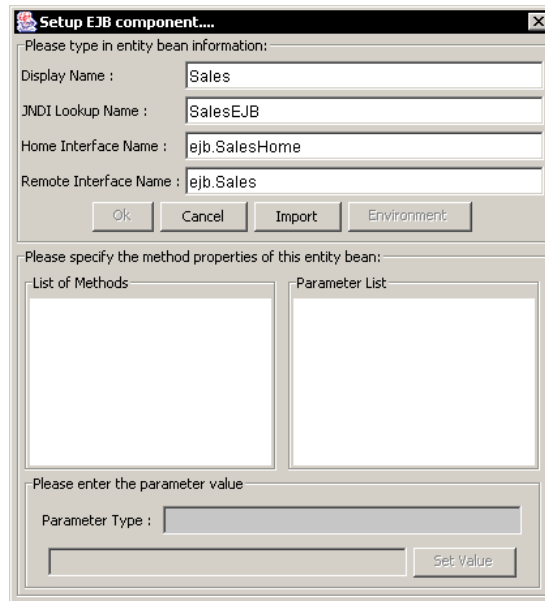
Parameters that are defined within the context of a class file work in the same way as query parameters. For more information about setting up a parameterized class file data source, please see Section 8.5.2 - Parameterized.

3.1.7. Data from EJBs

Using Enterprise JavaBeans™ technology, developers can simplify the development of large enterprise applications. With EJB technologies, developers can rely on building business logic and allow the application server (EJB container) to manage all system level services.

When working in the Java EE™ environment, persistent data interfaces are provided by entity beans. Although the underlying storage mechanism might be a relational database, the application data model is the EJB and it may not be desirable to have a reporting tool making redundant database connections. For this situation, EDAB allows users to query data directly from an entity bean.

To add an EJB as a data source, the EJB must first be deployed in the application server and the client JAR file containing the appropriate stub classes must be added to your application server/servlet runner's classpath. Select the *EJBs* node in the Data Source Manager and click the *Add* button. This will bring up a dialog allowing you to specify a display name and the connection information for the bean.



Setup EJB component....

Please type in entity bean information:

Display Name : Sales

JNDI Lookup Name : SalesEJB

Home Interface Name : ejb.SalesHome

Remote Interface Name : ejb.Sales

Ok Cancel Import Environment

Please specify the method properties of this entity bean:

List of Methods

Parameter List

Please enter the parameter value

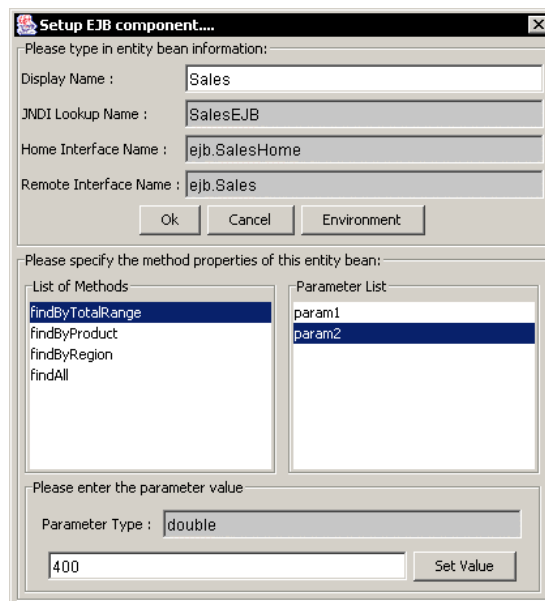
Parameter Type :

Set Value

Add EJB Dialog

To connect to an EJB data source, you must provide the JNDI lookup name for the bean (this is specified when you deploy the bean). For EJB 1.1 users, specify the name for the home interface and the remote interface. For EJB 2.0 users, specify the local home interface and the local interface. Once you specify all the information, click the *Import* button, which will analyze the home interface and retrieve all of the finder methods. Any methods found will be populated in the *List of Methods* section in the dialog.

The same dialog can be used to filter the data being retrieved based on parameters that are present in the finder methods. When you select a method in the left dialog, any parameters present will appear in the *Parameter List* section. You can then click on a parameter to set its value.



Setup EJB component....

Please type in entity bean information:

Display Name : Sales

JNDI Lookup Name : SalesEJB

Home Interface Name : ejb.SalesHome

Remote Interface Name : ejb.Sales

Ok Cancel Environment

Please specify the method properties of this entity bean:

List of Methods

Parameter List

Please enter the parameter value

Parameter Type : double

400 Set Value

Specifying EJB Parameter Values Dialog

When you select a parameter, the data type of the parameter will appear in the lower portion of the window. Below that you will be able to specify a value for the parameter. Be sure to enter a correct value for the data type, and then click the *Set Value* button. This will fix the parameter values. Once you finish setting up all parameter values, click the *Environment* button. This will bring up a new dialog allowing you to specify environment properties for your application server. This information is necessary for the EDAB Server to connect to the EJB.

Please enter environment information:	
APPLET	
AUTHORITATIVE	
BATCHSIZE	
DNS_URL	
INITIAL_CONTEXT_FACTORY	weblogic.jndi.WLInitialContextFactory
LANGUAGE	
OBJECT_FACTORIES	
PROVIDER_URL	t3://192.168.0.4:7001
REFERRAL	
SECURITY_AUTHENTICATION	
SECURITY_CREDENTIALS	
SECURITY_PRINCIPAL	
SECURITY_PROTOCOL	
STATE_FACTORIES	
URL_PKG_PREFIXES	

Ok Cancel

EJB Environment Setup

The fields here are the available environment properties for the JNDI context interface. You don't have to specify all values, only the information necessary for your environment (application server). Once you finish specifying the environment variables, click the *OK* button. You will be returned to the EJB setup window. Click *OK* again to finish setting up the EJB data source. A new node will appear under *EJBs* with your EJB.

You can modify the parameter values by selecting your EJB source, and clicking the *Edit* button.

There is a sample EJB data source included in the installation in `help/examples/DataSources/EJB` directory. The directory contains `sales.ear` and `salesClient.jar` files, as well as the source code for the Sales entity bean. The `sales.ear` file is designed to be deployed in Java 2 Reference Implementation and uses the Cloudscape database as the underlying storage mechanism. You can use `SalesClient.java` program to populate the Cloudscape database. The `salesClient.jar` file contains the stub classes to connect to the deployed Sales EJB and needs to be in the classpath.

3.1.8. Data from SOAP with WSDL support

EDAB also allows you to retrieve data using SOAP (Service Oriented Architecture Protocol). To connect to a SOAP data source using WSDL, you don't need to know any URLs for the services, SOAP actions, operation names or parameters. All you need to know is a location of WSDL file, which contains all the necessary information.

To set up a SOAP data source, select the *SOAPServices* node in the Data Source Manager and click the *Add* button. A dialog will open prompting you to specify options for the new SOAP data source.

Setup SOAP service...

Data Source Information

Name : WeatherForecast

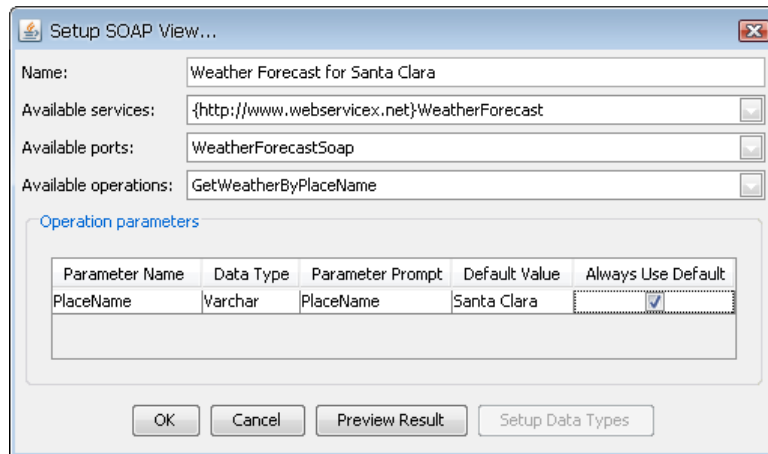
WSDL file URI: http://www.webservice.net/WeatherForecast.asmx?WSDL

OK Cancel Test

SOAP data source setup

The first option allows you to specify a display name for the data source. The second option allows you to specify a location of the WSDL file. The location can be either absolute path on the server or path relative to your EDAB installation directory or URL. Once you specify the connection information, you can test the connection to the WSDL file by clicking the *Test* button. This will test retrieving the WSDL file from the URI you've provided, check the file for any supported SOAP operations and report any problems. After clicking *OK*, a new SOAP data source node will be added to the data registry.

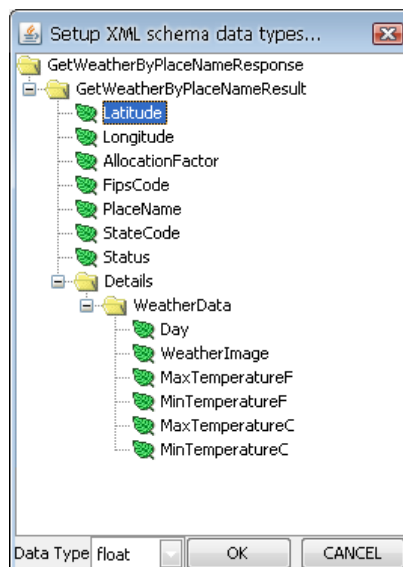
To add a new SOAP View, select an existing SOAP data source and click the *Add* button. A dialog will open prompting you for all the parameters necessary to make a SOAP query.



Setup SOAP View dialog

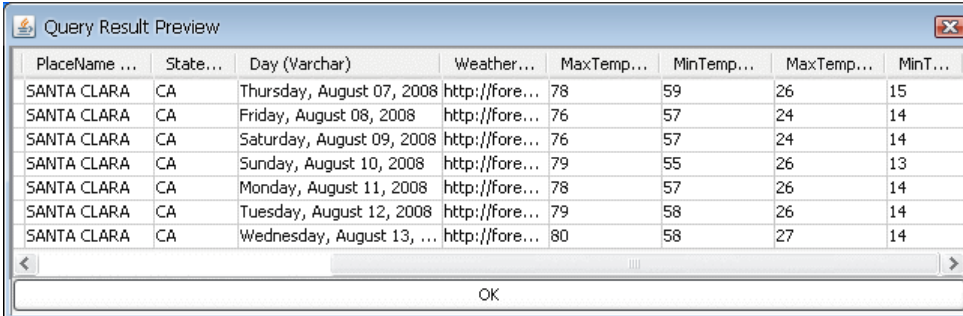
The first option in the dialog allows you to specify a display name in the Data Source Manager. Next, there are three drop-down menus in the dialog. The first drop-down menu contains all the SOAP services described in the WSDL file. Once you specify a service, the second drop-down list will be populated with all the ports of this service. Selecting a port will populate the last drop-down list with all the operations of this port. If you move the mouse over any drop-down list, a hint will appear with documentation for the service/port/operation (if the documentation is provided in the WSDL file). After specifying the service, port and operation, all the parameters of the operation will be read. If there are some parameters, they will be displayed in a table. The first two columns of the table are not editable. They are read from the WSDL file. The next 2 columns are editable and allow you to specify parameter prompts and default values. The last column contains a checkbox which allows you to choose whether the default parameter value will always be used or not. This means that this parameter value will be fixed and you will not be prompted for it. All the parameters that don't have this checkbox checked will be used as report/chart parameters.

The *Setup Data Types* button is only available when editing the SOAP View from the Data Source Manager and allows you to adjust data types when necessary. In order to verify result from the SOAP response, click the *Preview Result* button. All the default values will then be tested to see if they have proper data type or not. In case they do not match, you will be prompted to adjust them. After that, you will get the setup data types dialog (the same as for XML data source). If the data source is parameterized, you will get the parameter prompt dialog before specifying data types. Please note that in order to generate the XML schema properly, you have to specify existing parameter values.



Setup XML Schema Data Types dialog

You can setup data types from this dialog. The behavior is exactly the same as for XML data source with DTD schema (see Section 3.1.4 - Data from XML and XBRL Files). Once you finish specifying data types, click the *OK* button. A dialog will open showing you the result preview.



PlaceName ...	State...	Day (Varchar)	Weather...	MaxTemp...	MinTemp...	MaxTemp...	MinT...
SANTA CLARA	CA	Thursday, August 07, 2008	http://fore...	78	59	26	15
SANTA CLARA	CA	Friday, August 08, 2008	http://fore...	76	57	24	14
SANTA CLARA	CA	Saturday, August 09, 2008	http://fore...	76	57	24	14
SANTA CLARA	CA	Sunday, August 10, 2008	http://fore...	79	55	26	13
SANTA CLARA	CA	Monday, August 11, 2008	http://fore...	78	57	26	14
SANTA CLARA	CA	Tuesday, August 12, 2008	http://fore...	79	58	26	14
SANTA CLARA	CA	Wednesday, August 13, ...	http://fore...	80	58	27	14

Query Result Preview dialog

Clicking the *OK* button in this dialog will take you back to the Setup SOAP View dialog. Once you finish specifying all necessary information, click the *OK* button in the dialog. A new node will be added under your SOAP data source in the Data Source Manager and can now be used to create a report or chart.

3.1.9. Data from Salesforce

The Salesforce data source is designed for existing Salesforce users who want to display their Salesforce data in EDAB. The connection to the Salesforce server is established via SOAP using Salesforce Partner WSDL (version 13.0). Users communicate with Salesforce server by SOQL (Salesforce Object Query Language) queries. Please note that users must have valid Salesforce accounts with username and password to work with this data source. Moreover, users who use the EDAB Salesforce data source must have access to Salesforce account from trusted networks. To add your IP address to the trusted IP list, you have to activate your computer as described below.

For more information about SOQL queries and activating Salesforce user's accounts from trusted networks, please visit the following Salesforce sites:

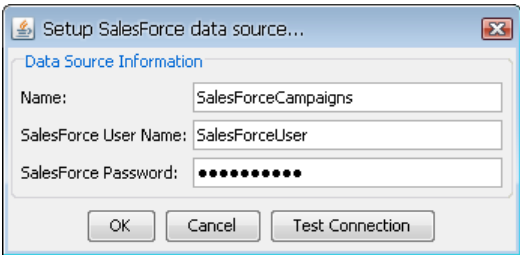
SOQL queries

https://developer.salesforce.com/docs/atlas.en-us.soql_sosl.meta/soql_sosl/sforce_api_calls_soql_sosl_intro.htm

Activating Salesforce user's accounts

https://help.salesforce.com/s/articleView?id=sf.security_networkaccess.htm

To set up a Salesforce data source, select the *SalesForce* node in the Data Source Manager and click the *Add* button. A dialog will open prompting you to specify a display name for the data source, user name and password to your Salesforce account. Once you specify the connection information, you can test the connection to your Salesforce account by clicking the *Test Connection* button. This will test the connection using the information you've provided and report any problems.



Setup Salesforce data source...

Data Source Information

Name: SalesforceCampaigns

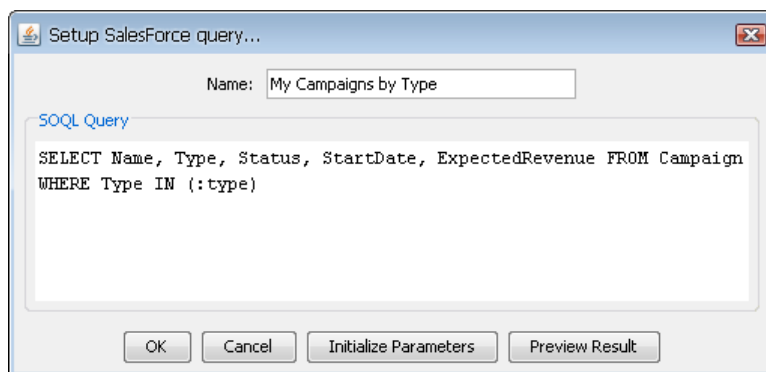
SalesForce User Name: SalesforceUser

SalesForce Password: ••••••••

OK Cancel Test Connection

Setup Salesforce Data Source Dialog

Once you add a Salesforce data source, a new node will appear in the Data Source Manager window. To add a new Salesforce query, click the *Add* button. A new dialog will open prompting you to specify a query name and SOQL query.



Setup Salesforce Query Dialog

Please note that only child-to-parent relationship queries are supported in the current EDAB version. You cannot use parent-to-child queries (using nested SOQL queries). For more information about Salesforce relationship queries and their syntaxes, please visit the following Salesforce site: https://developer.salesforce.com/docs/atlas.en-us.soql_sosl.meta/soql_sosl/sforce_api_calls_soql_relationships.htm

Moreover, this dialog allows you to initialize query parameters in case that your query contains single value or multi value parameters. A parameter is specified within an SOQL statement using the ":" character. Generally the parameter is placed in WHERE clause of an SOQL Select statement. For example, the following SOQL statement

```
Select Name, Type, Status, ActualCost From Campaign Where Name
= :CampaignName
```

specifies a single value parameter called *CampaignName*. You would then be able to enter a campaign name at run-time and only retrieve data for that campaign.

Another example of SOQL statement shows using of multi value parameters that take an array of values as the input rather than single values.

```
Select Name, Description, Type, LeadSource, Probability From Opportunity
Where Type IN (:OpportunityType) And LeadSource IN (:OppLeadSource)
```

The statement specifies two multi value parameters called *OpportunityType* and *OppLeadSource*. You would then be able to specify opportunity types and lead sources at run-time and you will only retrieve data according to specified parameters values.

In order to initialize SOQL query parameters, click the *Initialize Parameters* button. The initialize parameters dialog will then appear allowing you to specify parameters mapping.

Initialize Parameters Dialog

From this dialog, you can specify the following options:

Map to field:

This allows you to specify a field from the Salesforce data source whose values will be used for the parameter input. Selecting this option modifies the parameter prompt that you will get when previewing or running the report/chart. If you map the parameter to a Salesforce field, you will be prompted with a drop-down list of distinct values from which you can select a parameter value. If you do not map, you will have to type in specific parameter value.

Use custom selection choices:

Rather than having a drop-down menu with all the distinct column values, you can build a custom list of parameter values. To set up the list, select this option and click the *Setup Choices* button. This will launch a new dialog allowing you to create a list of choices.

The rest of the options are basically same as for database query parameters. For further information about initializing database query parameters, see Section 3.1.3.2.2.2 - Initializing Query Parameters. Once you specify mapping for all available parameters, click the *OK* button and you will be taken back to the Setup Salesforce Query dialog.

From the Setup Salesforce Query dialog, you can also preview the query result using the *Preview Result* button to verify output from your query. In case you have a parameterized query, the parameter prompt dialog will appear prompting you to specify parameter values. Once you specify the parameter values, click the *OK* button and the query result preview dialog will appear.

*Parameter Prompt*

Name (Varchar)	Type (Varchar)	Status (Varchar)	StartDate...	Expected...
GC Product Webinar - Mar 28-3...	Webinar	Completed	2007-03-28	800000.0
User Conference - Jul 7-8, 2007	Conference	Completed	2007-07-07	5500000.0
Global Energy Conference 2008	Conference	In Progress	2008-01-04	4200000.0
2nd Better Living Trade Show & ...	Conference	Completed	2007-03-04	2200000.0
4th Clean Tech Innovation Conf...	Conference	Planned	2009-07-18	300000.0
Smart Energy 2009 Conference	Conference	Planned	2009-03-20	500000.0
GC Product Webinar - Jun 17-1...	Webinar	Planned	2008-06-17	830000.0
AD Sales References	Advertisement	In Progress	2008-05-01	1300000.0
3rd Clean Tech Innovation Conf...	Conference	Completed	2008-03-28	900000.0
Smart Energy 2008 Conference	Conference	In Progress	2008-03-21	800000.0
GC Product Webinar - Jan 7, 2007	Webinar	Completed	2007-01-07	2800000.0
AD Advertisement Campaign	Advertisement	Planned	2008-07-02	500000.0
GC Product Webinar - Jan 7-8, ...	Webinar	Completed	2008-01-07	40000.0
Generals Advertisement Campaign	Advertisement	In Progress	2008-06-03	700000.0
GC Product Webinar - Aug 10-1...	Webinar	Planned	2008-08-10	1160000.0

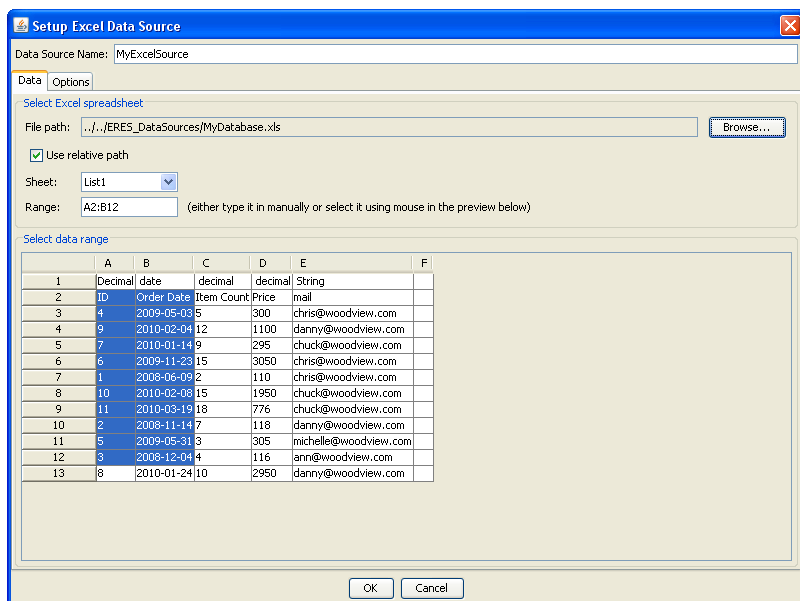
Query Result Preview Dialog

From this dialog, you can verify the query output. Clicking the *OK* button will take you back to the Setup Salesforce Query Dialog.

Once you specify the query, click the *OK* button. The query will then be added as a new node under your Salesforce data source in the Data Source Manager and can now be used to create a report or chart.

3.1.10. Data from Excel files

EDAB also allows you to design reports and charts using data retrieved from Excel files. To add an Excel file as a data source, select the *ExcelFiles* node in the Data Source Manager and click the *Add* button. A dialog will open prompting you to specify the data source name and to select the Excel file from which the data should be imported.



Setup Excel Data Source dialog - Data

After selecting the Excel file, the imported data will be previewed in the dialog. If the checkbox *Use Relative Path* is checked, the file path to the selected Excel file will be set as relative to the EDAB installation directory. Otherwise the full path will be used. In case the Excel file is stored on a different disk drive from the one which EDAB is installed on, this option is not available. You can select the sheet (if there is more than one in the file) and the cells which are relevant for the data source being designed using your mouse or by specifying the range in the *Range* box (for instance, you can specify that your source will use the data from the columns A and B and from the rows 2 to 12 by typing **A2:B12** in the *Range* box; this is the format which is also used for ranges in MS Excel). You can also select the data by clicking the row header or the column header. Hold **Ctrl** or **Shift** to select more rows or columns. Click the top left corner to select all cells. Again, this behavior is similar to MS Excel.

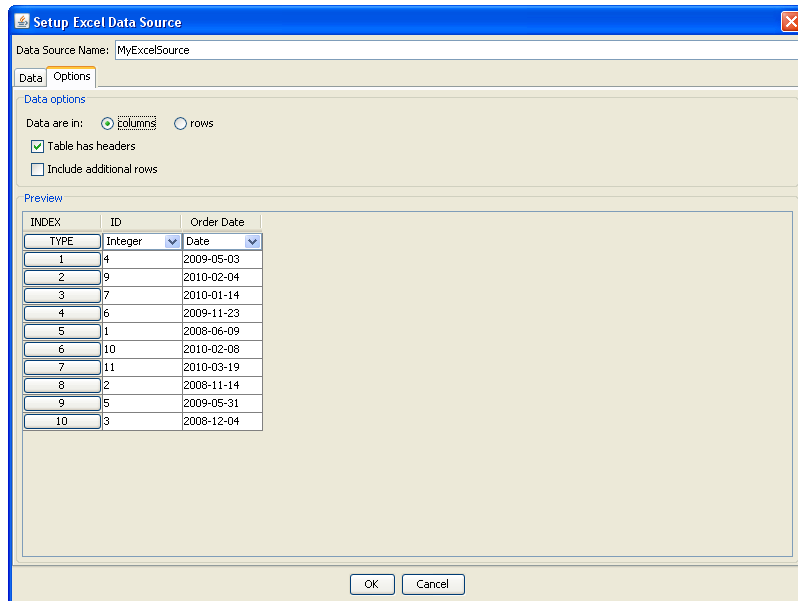
Please note that EDAB can process both *.xls files and *.xlsx files. The *.xlsx files are used in Microsoft Excel 2007 and newer and they are based on Open XML.

EDAB can also process basic Excel formulas. If it is not able to process the entered formula, an error message will be displayed.

Empty rows (in case the data are in columns) or columns (in case the data are in rows) at the end of the sheet are automatically removed from the data source, even in case they have been selected.

Click on the *Options* tab to specify whether the data are in columns or in rows to get the data structure correctly. You can also specify whether table headers are included in your data selection. The option *Include additional rows* or *Include additional columns* (depending on whether the data are in columns or rows) allows you to automatically include data rows or columns into the Excel file after the data source has been created without the need to change the data source in the Data Source Manager manually.

The bottom part of the dialog on the *Options* tab shows you the data source content following the current configuration. You can change the data type for each column of the data source there, if desired. The data types are detected automatically, therefore changing the type should not be necessary in most cases.



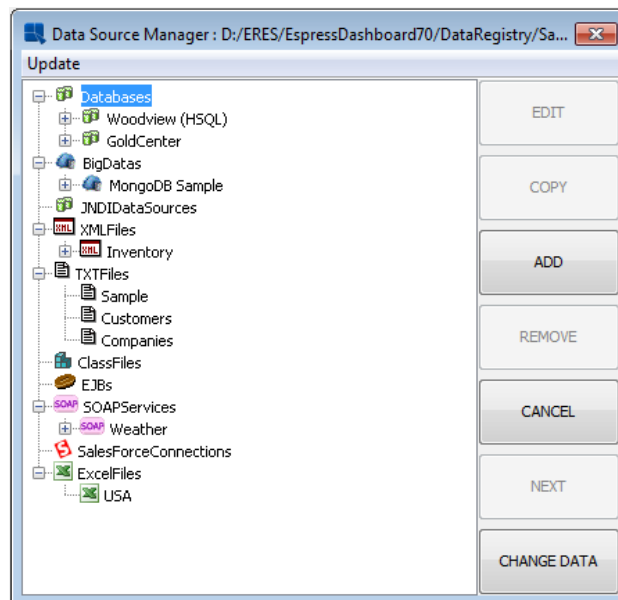
Setup Excel Data Source dialog - Options

Click **OK** to save the data source when the configuration is finished.

3.1.11. Using Data for Charts

Once you set up a data registry, the sources defined in the registry can be used for charts, in the QuickDesigner for reports and charts, and in the Map Builder for maps. The first time you select to start a new ChartDesigner chart, you will be prompted to select a registry you want to use. If there are currently no available registries (meaning that you haven't created any, or you don't have privileges to view any), you will be prompted to go to the data registry manager to create one.

Once you select a registry, a data source manager window will open allowing you to select, add, or modify a data source that you want to use for the chart or report.



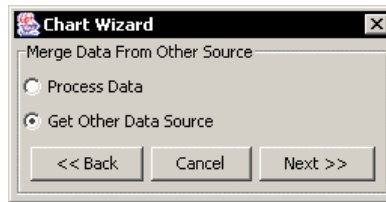
Data Source Manager Window for Chart Designer

After you first select a registry, the next time you design a chart, it will automatically connect to that registry and open the data source manager. You can change registries by clicking the *Change Data* button.

3.1.11.1. Using Multiple Data Sources

Once you select the data source you want to use for a chart and click the *Next* button, the next screen will display first twenty records from the data source (With the exception of data views, which will require you to select fields and set conditions first). You can display all of the records by checking the *Show All Records* box.

EDAB allows you to construct charts from multiple data sources. Once you have selected your first source, and click *Next* from the data table window, you will be presented with a dialog asking if you would like to process the current data or select another data source.



Additional Data Source Dialog

If you select *Process Data* and click the *Next* button, you will continue to the next step in the chart wizard. If you select *Get Other Data Source*, you will return to the Data Source Manager to select another data source for the chart. You can repeat this process to select as many sources as you want.

Multiple data sources are combined sideways in the data table. This means that the columns from the second (or third, fourth, etc.) data sources are placed to the right next to the columns from the first data source. If the columns from one data source have more rows than the other ones, null values will be placed in extra rows.

For example, assuming you want to use two data sources to create a chart. The data table generated by the first source will look like this:

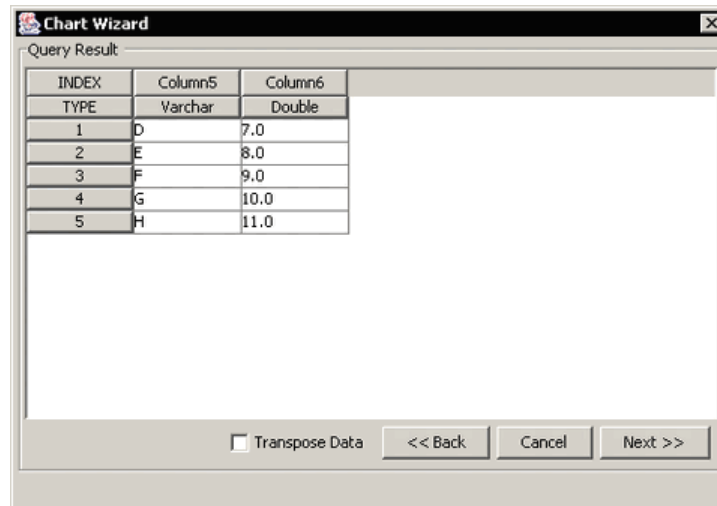
 A screenshot of the 'Chart Wizard' dialog box, specifically the 'Query Result' tab. It displays a table with the following data:

INDEX	Column1	Column2	Column3	Column4
TYPE	Varchar	Double	Double	Double
1	A	0.0	5.0	10.0
2	B	2.0	4.0	8.0
3	C	3.0	9.0	27.0

 Below the table, there is a checkbox labeled 'Transpose Data' which is currently unchecked. At the bottom right, there are three buttons: '<< Back', 'Cancel', and 'Next >>'.

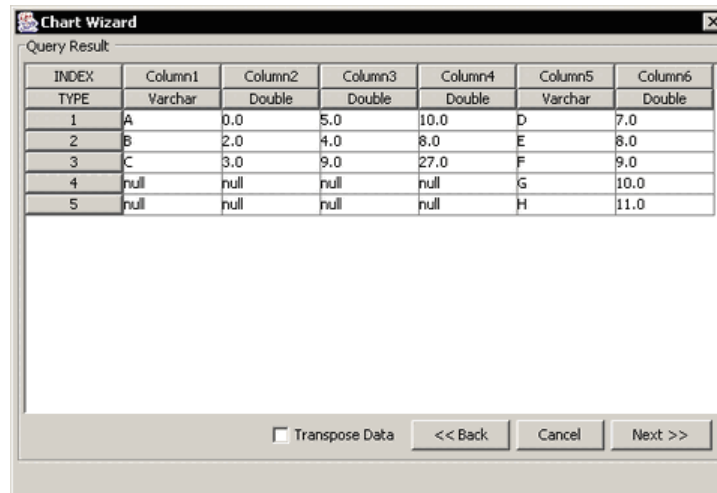
Data From the First Data Source

The data from the second source will look like this:



Data From the Second Data Source

When you combine two data sources, you will get the following data table:



Data From Combined Sources

As you can see, the two data sources have been placed side by side. Since the second source has more rows of data than the first, additional rows of null data were added.



Note

You cannot use parameterized data sources to create a multiple data source.

3.1.11.2. Change Data Source

At any point during chart design, you can select to change the template's data source. Since chart templates are saved with their data source information, you must use the option within Chart Designer to change the template's data source. Simply altering a data source in the registry will not affect the template unless you use the data source updating feature (For more information about this, see Section 3.1.11 - Using Data for Charts). To change a template's data source, select *Modify Data Source* from the Data menu, or click the *Modify Data Source* button on the toolbar. This will bring up the Data Source Manager allowing you to select a new data source or modify an existing one.

If the new data source is significantly different from the previous data source in number of columns or data types, you might need to re-map the data to the chart. For more information about data mapping, please see Section 4.1.3 - Chart Types and Data Mapping for charts.

3.1.12. Data Source Updating

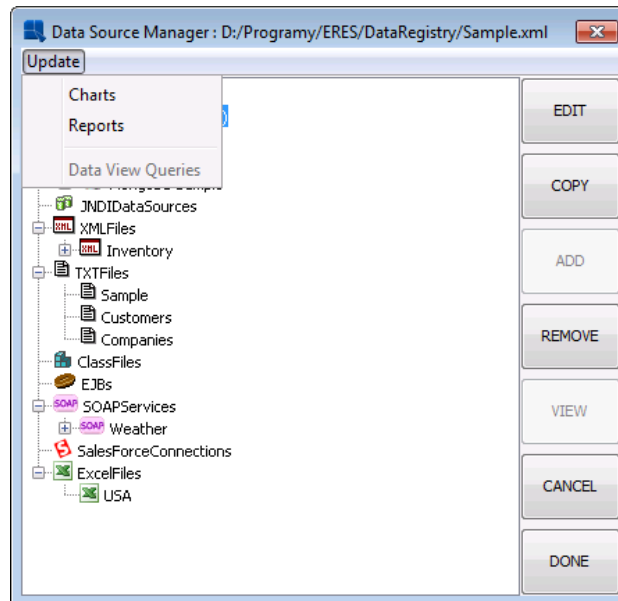
There are many circumstances where you will want to move a group of templates or a complete installation of EDAB from one location to another. For example an application can move from a development to a production environment. In each environment, the location and connection information for data sources may be different. In this scenario, updating report templates one by one as detailed in the previous section isn't a feasible way to change the connection information for a large number of templates. Instead, EDAB allows you to quickly update a group of templates based on information in the data registry.

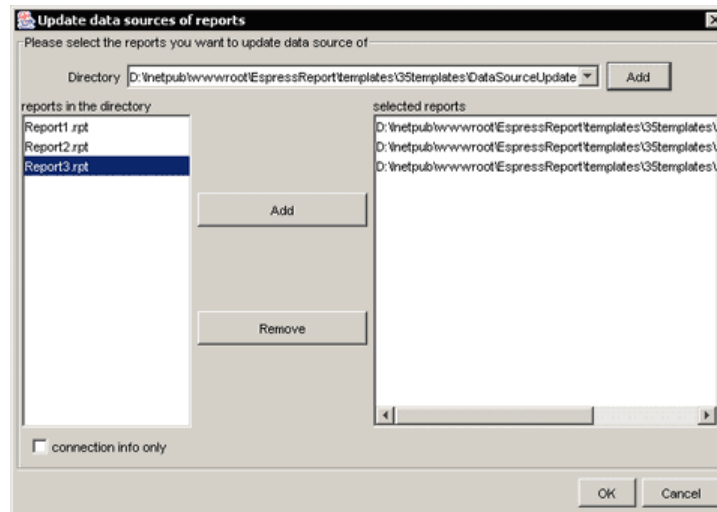
Although chart templates maintain an internal copy of the data source information (allowing them to be deployed independently), they also maintain information (location and source) about the data registry from which they were created. Hence, when you modify a chart's query (as detailed in Section 3.1.3.4 - Editing Queries), you have the option to save the changes back to the data registry. In order to use this feature, you will have to keep your data registry up to date. This means that query changes should be saved back to the registry and changes in data view structure should be propagated to data view queries.

To use this feature, first make any modifications to the data registry that you want to propagate to the templates. These modifications can include database connection information, file locations for text, XML data files, and even changes to data views or queries that you want to pass to the templates.

Note that if you're moving reports or charts between installations, the data registry file will need to be moved to the same relative location in the new installation. In addition, the associated query files for that registry (`.qry` / `.dvw` / `.ddt`) will need to be moved to `/queries/` directory of the new installation (Query file names begin with the name of the registry).

From the data source manager, select *Reports* or *Charts* from the *Update* menu depending on whether you want to update charts or reports. This will bring up a dialog allowing you to select which files you want to update.

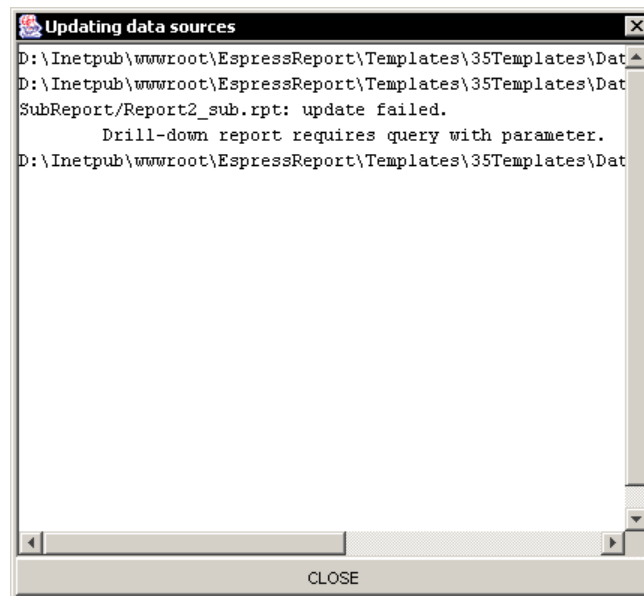




Select Reports for Data Source Updating

To select files to update, first browse to the directory that contains the reports/charts. After you select a directory, any chart templates will appear in the left side of the dialog. You can select any templates you want to update and click the *Add* button. You can navigate to as many different directories as you want to select templates. Selecting the *Connection Info Only* option will only update the connection information (database URL, driver, username, password, and locations for XML files, text files and Java classes). Queries and data view information will not be updated in the templates.

Once you finish selecting the templates you want to update, click *OK* and the updating process will begin. A dialog will display the current progress and any errors.



Updating Data Sources Progress/Log Dialog

A log file named `UpdateDataSources` will also be written in the root directory of the installation with the contents of the progress screen. Reports that fail the updating for various reasons can be updated manually using the option in ChartDesigner.



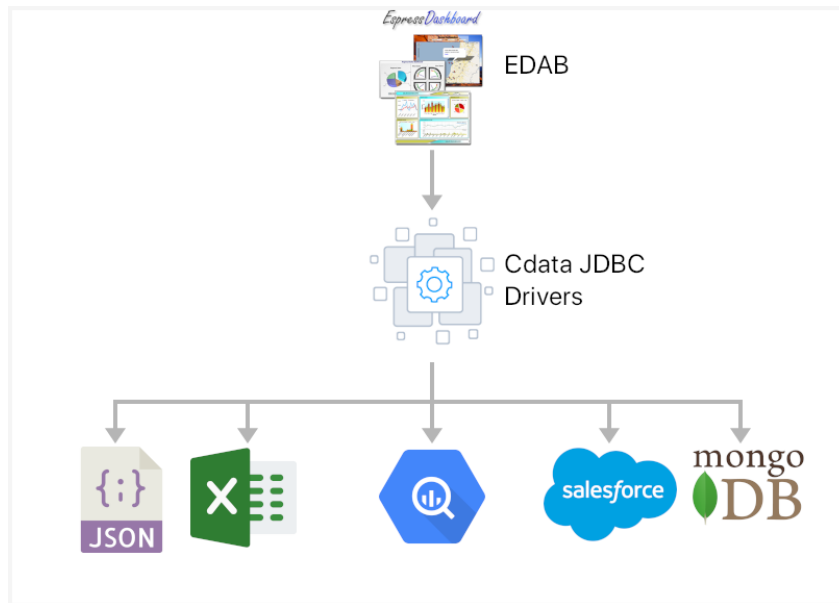
Note

Only reports or charts for the current data registry will be modified. If you select templates that do not retrieve their data from a source in the current registry, they will be ignored.

3.1.13. CData JDBC drivers

If you want to connect to a data source driver that doesn't ship with ExpressDashboard, CData JDBC drivers are an interesting option.

CData JDBC drivers can be also used to increase capabilities of non-database data sources like Excel or JSON.



Tip

Although CData JDBC drivers are a 3rd party commercial product, you can install a free trial version to test the product before purchasing.



Note

CData JDBC drivers are a 3rd party product. If you want to use the drivers, you have to purchase them at <https://www.cdata.com>

3.1.13.1. Supported CData Drivers

Currently, we support the following CData JDBC drivers:

- Salesforce
- BigQuery
- Excel
- JSON
- MongoDB
- Kintone

Other CData JDBC drivers might work too but we can not guarantee full functionality for unsupported drivers. If you require a connection to a driver that is not listed here, please contact us at [<support@quadbase.com>](mailto:support@quadbase.com)

3.1.13.1.1. Excel

Download: <https://www.cdata.com/drivers/excel/jdbc>

Official CData Documentation (for the JDBC driver only): <https://cdn.cdata.com/help/RXF/jdbc>

After downloading the driver, see the following chapters: Section 3.1.13.2 - CData JDBC driver installation, Section 3.1.13.3 - Deploying the CData JDBC Driver in EspressoDashboard and Section 3.1.13.4 - Using the CData JDBC drivers in DataSource Manager

The CData Excel driver allows you to run SQL queries on top of an Excel file just like it was a database. This allows you to use the QueryBuilder, build Data Views in a graphical user interface as well as write SQL queries manually (in Query Builder).



Tip

You can also write queries with parameters and multi-value parameters using Excel files as the data source.

Since Excel does not have data types set for columns as a normal database has, determining the data type for the columns can be a bit tricky at times. By default, we added the following parameters to the CData Excel driver connection URL:

```
TypeDetectionScheme=RowScan;
RowScanDepth=10;
```

This makes the CData Excel driver scan the first ten rows when loading the selected Excel file and detect the data source for each column automatically based on the data in the first ten rows.

However, there are multiple options of how to determine the data type for each column.

For more options, read the CData JDBC Excel driver documentation about the TypeDetectionScheme parameter: https://cdn.cdata.com/help/RXH/jdbc/RSBExcel_p_TypeDetectionScheme.htm

You can change the TypeDetectionScheme value in the Setup Database... dialog in Data Source Manager in the URL field (the field where you entered the Excel file path).

3.1.13.1.2. JSON

Download: <https://www.cdata.com/drivers/json/jdbc>

Official CData Documentation (for the JDBC driver only): <https://cdn.cdata.com/help/DJF/jdbc>

After downloading the driver, see the following chapters: Section 3.1.13.2 - CData JDBC driver installation, Section 3.1.13.3 - Deploying the CData JDBC Driver in EspressoDashboard and Section 3.1.13.4 - Using the CData JDBC drivers in DataSource Manager

The CData json driver allows you to run SQL queries on top of a json file just like it was a database. This allows you to use the QueryBuilder, build Data Views in a graphical user interface as well as write SQL queries manually (in Query Builder).



Tip

You can also write queries with parameters and multi-value parameters using JSON files as the data source.

3.1.13.1.3. Salesforce

To connect to Salesforce:

Download: <https://www.cdata.com/drivers/salesforce/jdbc>

Official CData Documentation (for the JDBC driver only): <https://cdn.cdata.com/help/RFF/jdbc>

After downloading the driver, see the following chapters: Section 3.1.13.2 - CData JDBC driver installation, Section 3.1.13.3 - Deploying the CData JDBC Driver in EspressoDashboard and Section 3.1.13.4 - Using the CData JDBC drivers in DataSource Manager

3.1.13.1.4. MongoDB

To connect to MongoDB:

Download: <https://www.cdata.com/drivers/mongodb/jdbc>

A bug fix was added to newer MongoDB driver from https://cdatabuilds.s3.amazonaws.com/support/DGR-JV_8598.exe

Official CData Documentation (for the JDBC driver only): <https://cdn.cdata.com/help/DGF/jdbc>

After downloading the driver, see the following chapters: Section 3.1.13.2 - CData JDBC driver installation, Section 3.1.13.3 - Deploying the CData JDBC Driver in EspressoDashboard and Section 3.1.13.4 - Using the CData JDBC drivers in DataSource Manager

3.1.13.1.5. BigQuery

To connect to BigQuery:

Download: <https://www.cdata.com/drivers/bigquery/jdbc>

Official CData Documentation (for the JDBC driver only): <https://cdn.cdata.com/help/DBF/jdbc>

After downloading the driver, see the following chapters: Section 3.1.13.2 - CData JDBC driver installation, Section 3.1.13.3 - Deploying the CData JDBC Driver in EspressoDashboard and Section 3.1.13.4 - Using the CData JDBC drivers in DataSource Manager

3.1.13.1.6. Kintone

To connect to Kintone:

Download: <https://www.cdata.com/drivers/kintone/jdbc>

Official CData Documentation (for the JDBC driver only): <https://cdn.cdata.com/help/DBF/jdbc>

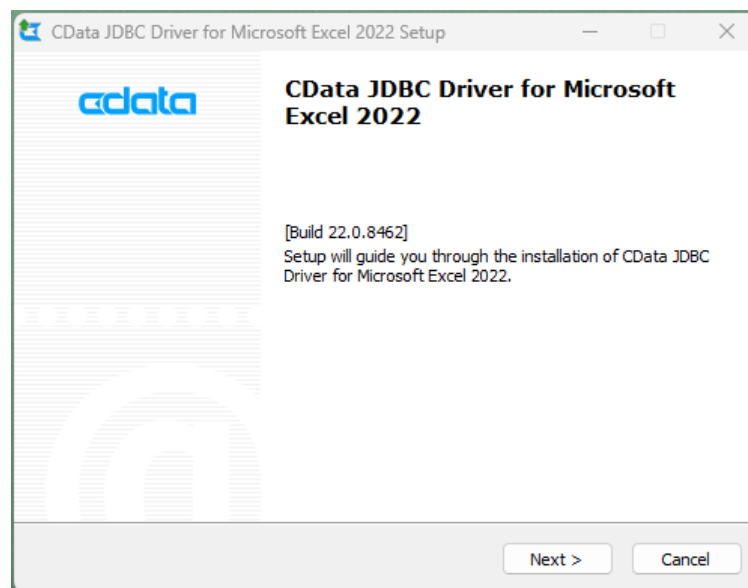
After downloading the driver, see the following chapters: Section 3.1.13.2 - CData JDBC driver installation, Section 3.1.13.3 - Deploying the CData JDBC Driver in EspressoDashboard and Section 3.1.13.4 - Using the CData JDBC drivers in DataSource Manager

3.1.13.2. CData JDBC driver installation

The installation process of all CData JDBC drivers is basically the same for all the supported data sources. We'll guide you through the process of installing the CData Excel JDBC driver for example.

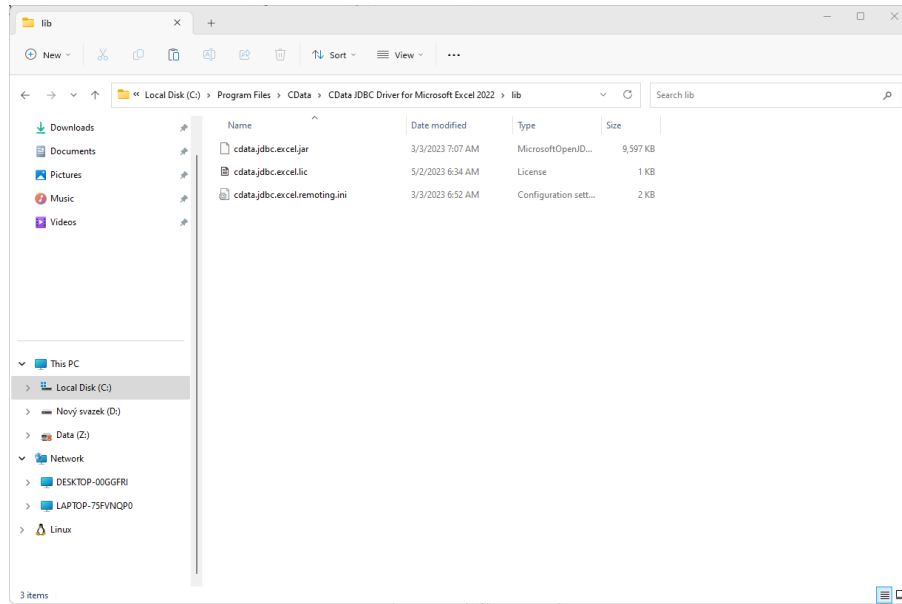
First, install the CData JDBC Driver of your choice. You will find the links to each supported CData JDBC driver in the chapters below.

After downloading the installer, proceed with the setup according to the dialogs.



In the following dialog, you can choose to either install a paid version of the product or a free 30-day trial version. In this example, we'll use the trial version.

When the setup is complete, JDBC Driver will be created in "CData JDBC Driver for Microsoft Excel 2022\lib".



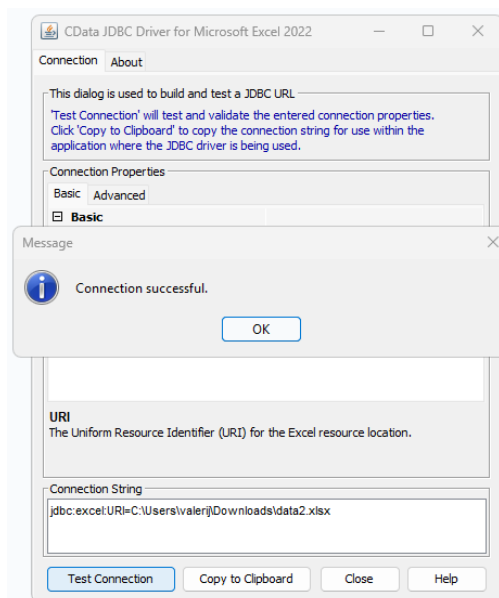
Tip

Executing "cdata.jdbc.excel.jar" will launch the connection test tool. You can use this tool to test or troubleshoot the CData driver.



Tip

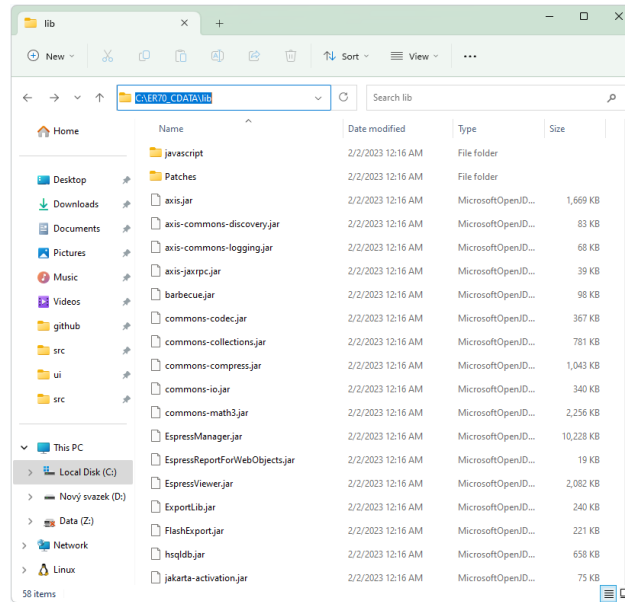
The connection test tool also displays you the connection URL that can be used in EspressDashboard Data Source Manager.



3.1.13.3. Deploying the CData JDBC Driver in EspressDashboard

Locate the "CData Installation Directory"\CData JDBC Driver for Microsoft Excel 2023\lib (for example: C:\Program Files\CData\CData JDBC Driver for Microsoft Excel 2023\lib) on your hard drive. The directory should contain three files: cdata.jdbc.excel.jar, cdata.jdbc.excel.lic, cdata.jdbc.excel.remoting

Copy the three files to EDAB/WEB-INF/

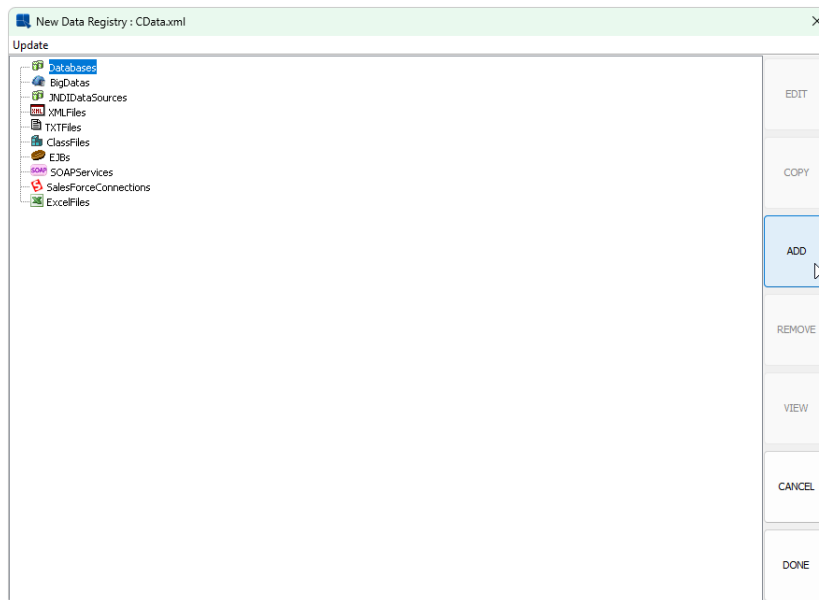


After you've copied the files, restart The EspressoManager (if it is running).

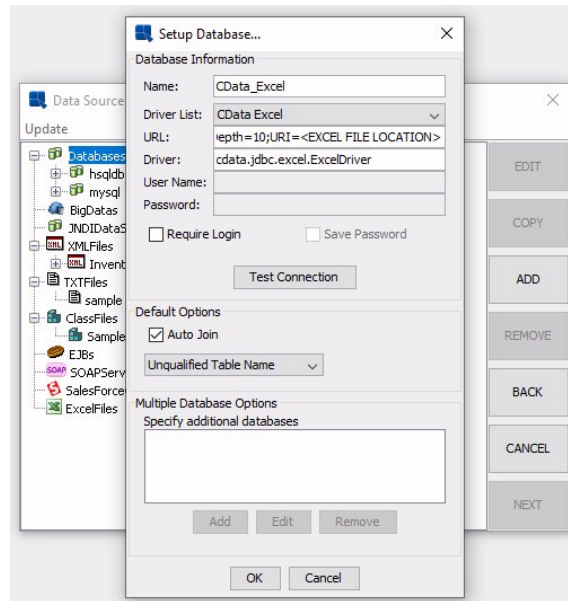
3.1.13.4. Using the CData JDBC drivers in DataSource Manager

Launch the DataSource Manager and open a data registry in it (existing one or a new one).

Select the “Databases” option in the tree-list and press the “ADD” Button.



In the “Driver List:” drop-down menu, select “CData Excel” (or any other CData JDBC driver you might be installing).



In the “URL:” text field, replace the placeholders (like “EXCEL FILE LOCATION”) with real values.



Tip

Alternatively, you can replace the text in the “URL:” text field with the connection string obtained by the CData Test Connection tool described in the previous chapter.

Click OK. You’re done. You can start using Excel files as if they were a database.

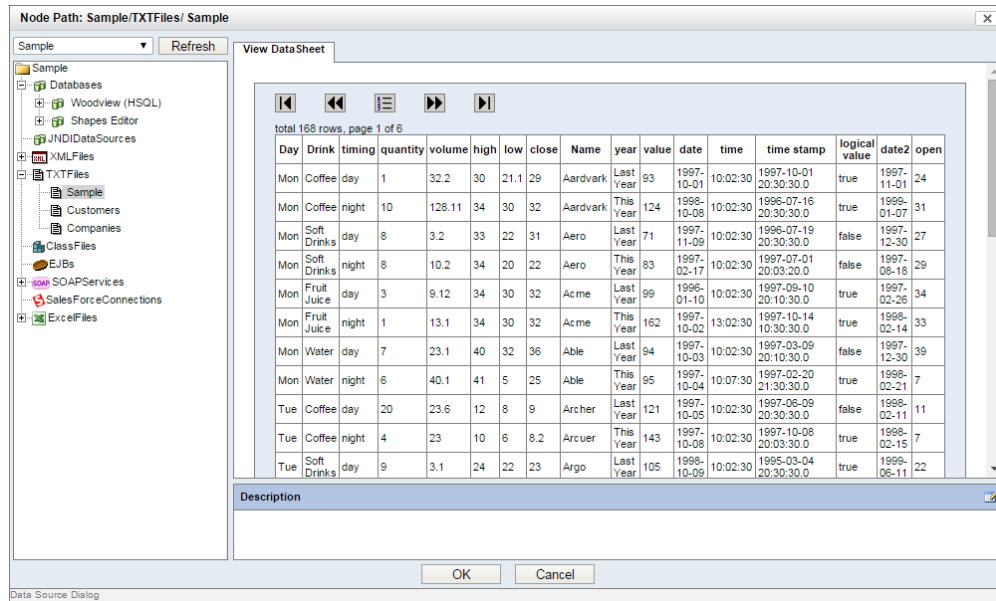
3.2. Data in QuickDesigners and Maps

3.2.1. Select a Data Source

QuickDesigner Reports, QuickDesigner Charts, Online Maps and SVG Maps use the same interface to manage data - *Data Source Dialog* . The *Data Source Dialog* allows end users to select, filter, and present data without mastering database structures, and all with zero client download. For full control of data sources, please see Section 3.1 - Data in Organizer.

The *Data Source Dialog* prompts you to select the data registry and the data source that you would like to use. A user must have read privileges to one of the registries defined in the Organizer. For more about creating and managing data registries, please see Section 3.1.1 - Managing Data Registries.

Select the data registry from the drop-down menu in the upper part on the left. The pane below displays the content of the selected registry. Select a data source you want to use.



Data Source Dialog

Once a data source is selected, you can see records of the source in the right-hand side *View DataSheet* pane. If you select a *DataView* as the data source, you will be able to create a new *DataView* query in the *DataView Builder* in the right pane. You can also select an existing *DataView* query and modify it in the right pane.

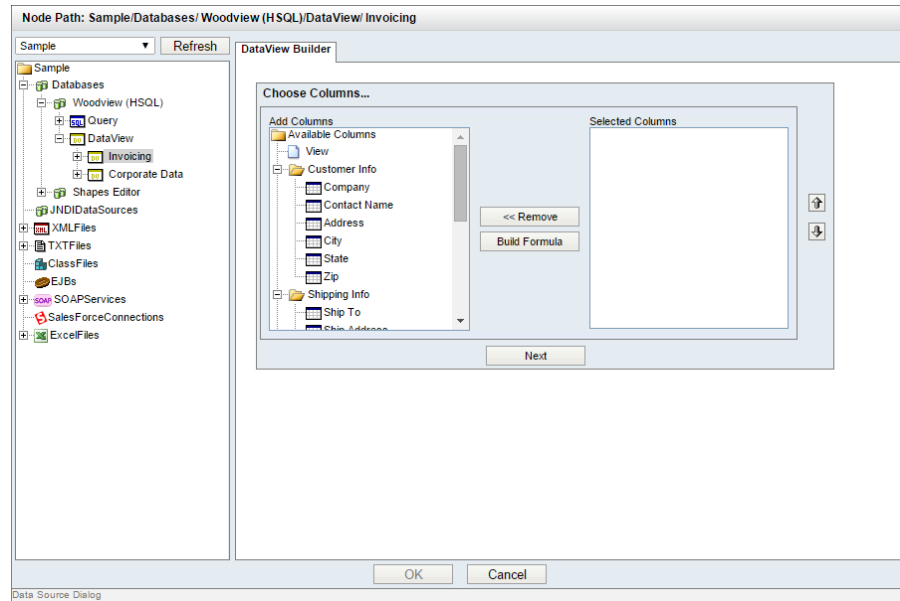
3.2.2. Queries

Queries work in conjunction with Data Views. Data Views allow an administrator to pre-select tables and fields from a database and create a local view from which end users can run queries. For more information about data views and setup, please see Section 3.1.3.3 - Data Views.

If you select either a data view or a data view query as the data source, you will be taken to *Data Source Dialog* query interface in the right-hand pane where you can create or modify a query based on the data view.

3.2.2.1. Select Fields

If you select a data view as the data source, the *DataView Builder* will open in the right pane allowing you to select fields from the view for your query.

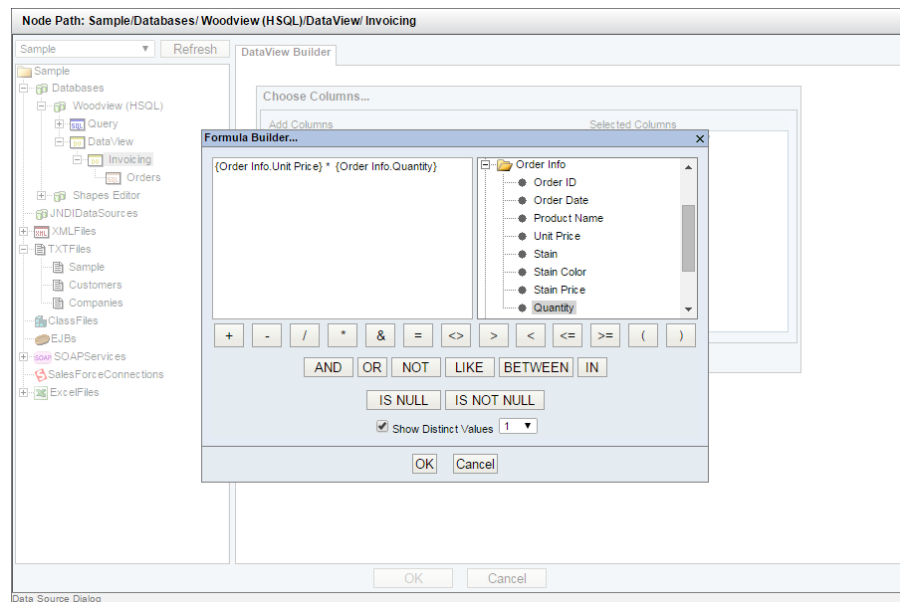


Select Columns Dialog

A select list for each heading in the view is generated on the left-hand side of the *DataView Builder*. To select fields for the query, click on a field on the left-hand side. The selected column will be added to the dialog on the right-hand side. To remove fields, you can select the columns from the right-hand side and click the *Remove* button.

3.2.2.1.1. Build Columns

You can also create computed columns from the *Choose Columns* dialog. To create a computed column, click the *Build Formula* button. This will launch the Formula Builder interface in a new window.

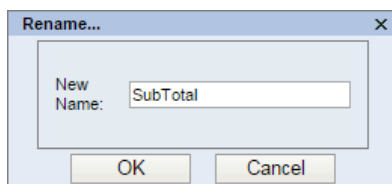


Formula Builder Window

The right-hand side of the Formula Builder contains a list of fields in the view, as well as the database functions. The buttons at the bottom are available operators and conditionals. The Boolean and conditional operators are more commonly used for building conditions which is covered in Section 3.2.2.2 - Set Conditions.

Once you have finished constructing the field, click *OK* to close the Formula Builder and return to the *Choose Columns* window where the computed field is added to the *Selected Columns* list. You can edit the formula by selecting it in the list and clicking the *Modify Formula* button.

You can also create an alias for any expression column by selecting it and clicking the *Rename* button. This will bring up a new window for you to specify the alias.



Rename Dialog

Once you have finished selecting the columns you would like to use, click the *Next* button to set conditions.

3.2.2.1.2. Querying Encrypted Data

Suppose you have a database that has encrypted fields, e.g. social security numbers, tax ID, passport numbers, etc in a tax payer database or immigration database. How do you allow authorized users of your (EDAB) system to make ad hoc reports or view reports online so that they can see the actual field values? The other concern is that you do not want any users of your system to know the encryption/decryption key but you want them to be able to see the decrypted data.

EDAB provides an advanced feature that allows you to run QuickDesigner Reports and Menu Page/report URL with automatic decryption enabled.

To do this, you need to do two things:

1. You need to create an XML file that gives the database URL, database driver, name of column to be decrypted, and the function to be applied when the data is being retrieved. An example file is as follows:

```
<?xml version="1.0"?>
<!DOCTYPE ReplaceColumnInfoList SYSTEM "QBReplaceColumnInfoList.dtd">
<ReplaceColumnInfoList>
  <ReplaceColumnInfo>
    <Driver>com.mysql.jdbc.Driver</Driver>
    <URL><![CDATA[jdbc:mysql://prodigy:3306/CAIT]]></URL>
    <TableName>user</TableName>
    <Pair>
      <ColumnName>user.SSN</ColumnName>
      <ReplaceValue><![CDATA[cast(AES_DECRYPT(user.SSN,'111111111111111') as CHAR)]]></ReplaceValue>
    </Pair>
  </ReplaceColumnInfo>
</ReplaceColumnInfoList>
```

The DTD file used is shown below.

```
<?xml encoding="US-ASCII"?>
<!ELEMENT ReplaceColumnInfoList (ReplaceColumnInfo*)>
<!ELEMENT ReplaceColumnInfo (Driver, URL, TableName, Pair*)>
<!ELEMENT Pair (ColumnName, ReplaceValue)>
<!ELEMENT Driver (#PCDATA)>
<!ELEMENT URL (#PCDATA)>
<!ELEMENT TableName (#PCDATA)>
<!ELEMENT ColumnName (#PCDATA)>
<!ELEMENT ReplaceValue (#PCDATA)>
```

In this example, we are using a MySQL database, the table name is *user*, the encrypted column is *SSN*. When a query is run, the function `AES_DECRYPT(user.SSN, '1111111111111111')` will replace `user.SSN` wherever it may appear in the SQL statement. The function `AES_DECRYPT` is the decryption function in MySQL. The key for decrypting the data is `'1111111111111111'` in this example.

2. You need to set the path to the XML file in the Admin Console (*Server Options* tab, *Security Options* category - see Section 1.4.1.3 - Server Options for more details) and then restart the EDAB server.

Once this is set up, when you run a report in Menu Page, using generated report URL or generating reports in QuickDesigner Reports, you will be able to see the real (decrypted) values of the encrypted columns.

You can find the example XML and DTD files shown above under the `<EDABInstallDir>/help/examples/DataDecryption/` directory. You will also find a script that you can use to generate some sample data in MySQL.

When you run a report (in Menu Page, QuickDesigner Reports or with report URL) with the following query,

```
select user.firstName, user.lastName, user.SSN from user;
```

you will see the following result set:

firstName	lastName	SSN
John1	Smith1	1111
John2	Smith2	2222
John3	Smith3	3333
John11	Smith11	11111

This query below,

```
select user.firstName, user.lastName, user.SSN from user where SSN LIKE '1%';
```

should produce the following result in your report.

firstName	lastName	SSN
John1	Smith1	1111
John11	Smith11	11111

You can specify multiple fields in the xml file. To add another field from the same table, simply add another `<pair>` element. To add a field from another table or datasource, add a full `<ReplaceColumnInfo>` element. This feature also supports sub-reports and drill-downs. Any field in these components matching the information in the xml will also be decrypted.

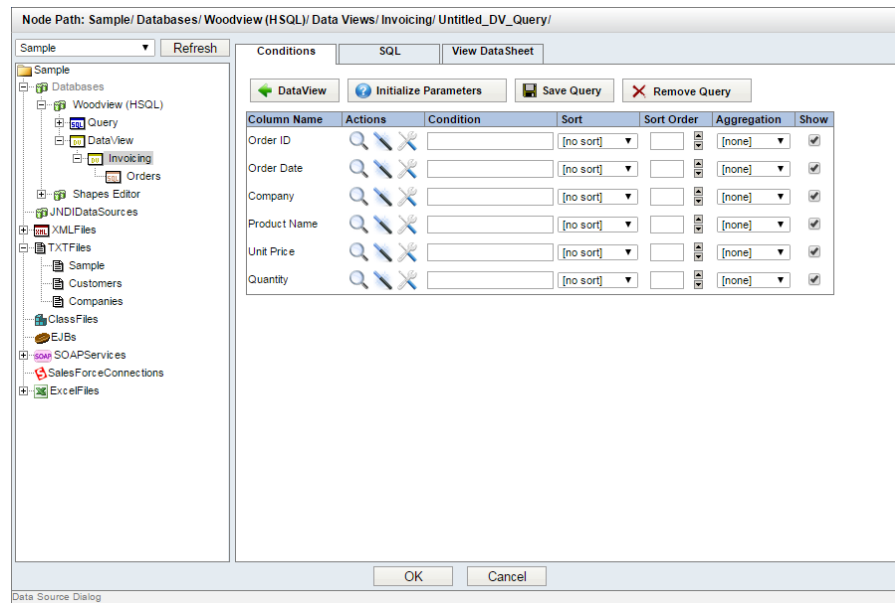


Note

If you are entering SQL for your query, you must use the full name for the field in order for this feature to work. For example, `SELECT user.SSN FROM user` works, but `SELECT SSN FROM user` will not.

3.2.2.2. Set Conditions

After selecting fields, or if you select a data view query as the data source, the conditions window opens allowing you to set aggregation, grouping, and filtering for the selected columns.

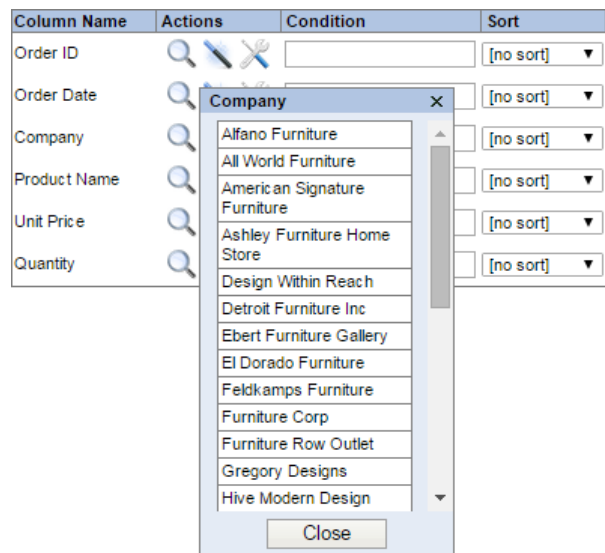
*Conditions Dialog*

Each selected column is listed along with several options:

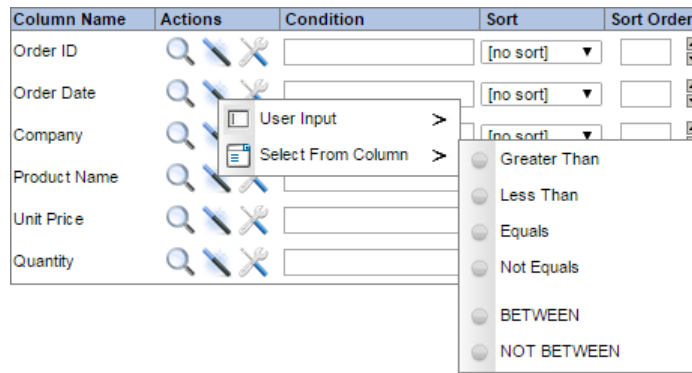
Actions:



Show Column Data: This will bring up a new window with column data.

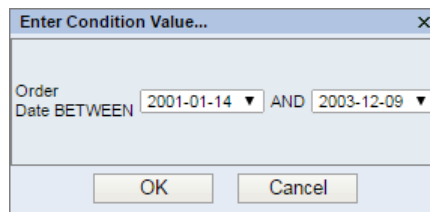
*Show Column Data*

Condition Wizard: This will bring up a dialog allowing you to set basic conditions from a field. The first dialog allows you to select whether you would like to enter values or select them from a list.




Condition Wizard

Once you have selected the condition, a new dialog opens allowing you to set the values for the condition.



Condition Value Dialog

Select the values that you would like and click *OK*. The condition will automatically be added to the *Condition* field.

 **Build Condition:** This will bring back up the Formula Builder, allowing you to construct an expression for the condition.

Condition: This allows you to type a condition for the column.

Sort: This allows you to indicate if the column should be sorted. You can select whether to sort in ascending or descending order.

Sort Order: This allows you to specify the priority (order) for sorting columns.


Aggregation: This allows you to specify any grouping or aggregation in the query.

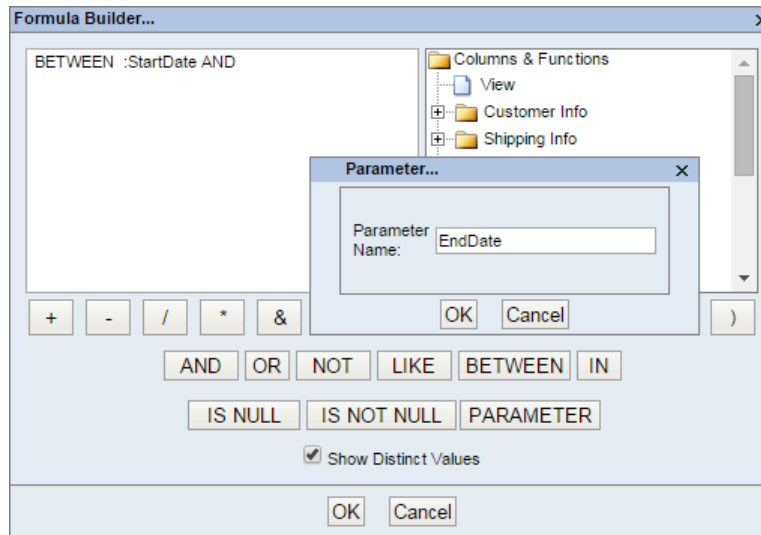
Show: This option indicates whether the column should be visible or not.

3.2.2.2.1. Parameterized Queries

Data Source Dialog also allows you to define query parameters. Parameters are defined in the same way as with other query interfaces - using the ":" character. You can define parameters directly in the conditions field, using the convention of :ParameterName to define a parameter.

Parameters can also be defined in the Formula Builder when building conditions. To define a parameter this way,

click the  *Build Condition* button to open the Formula Builder, click the *Parameter* button to insert a parameter. This will bring up a new dialog allowing you to enter the parameter name.

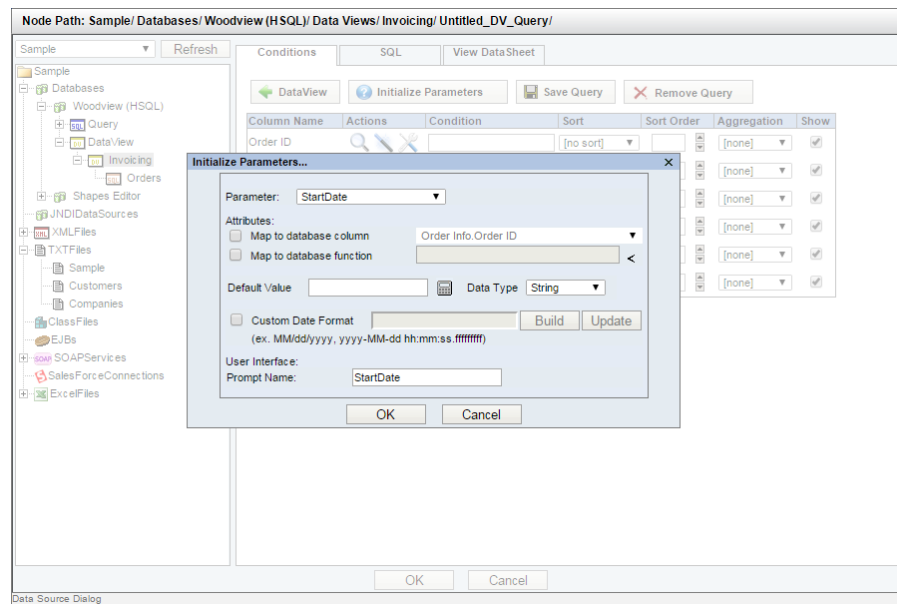


Specifying a Parameter in Formula Builder

For more information about query parameters, please see Section 3.1.3.2.2 - Parameterized Queries.

3.2.2.2.1.1. Initialize Query Parameters

Once you have specified parameters in the query, they will need to be initialized. You can initialize parameters by clicking the *Initialize Parameters* button. The initialization dialog will also open if you preview the query by clicking on the *View DataSheet* tab or if you click on the *OK* button closing *Data Source Dialog*.



Initialize Parameter Dialog

The drop-down list at the top of the dialog indicates which parameter is currently being initialized. You can access and set the options for each parameter by selecting it from this list.

The following options are available for each parameter:

Map to database column:

This allows you to specify a column from the database whose values will be used for the parameter input. Selecting this option modifies the parameter prompt that you will get when running the report or chart. If you map the parameter to a database column, you will be prompted with a drop-down list of distinct values from which to select a parameter value. If you do not map, you will have to type in the specific parameter value.

Initialize Parameters...

Parameter: StartDate

Attributes:

☒ Map to database column Order Info.Order Date

☐ Map to database function

Default Value: 2001-01-14 Data Type: Date

☒ Custom Date Format: yyyy-MM-dd Build Update
(ex. MM/dd/yyyy, yyyy-MM-dd hh:mm:ss.ffffff)

User Interface:
Prompt Name: StartDate

OK Cancel

Initialize Parameter Dialog

Map to database function:

Mapping a parameter to a database function is very similar to mapping to a column. This feature allows you to map to the results of any function from your database. For example, suppose you have the following condition in your query:

```
Year({Orders.OrderDate}) IN (:OrderYear)
```

In the initialize parameter dialog, check the *Map to database function* box, select the function from the list and enter the parameter value for your function. The default value and data type will be automatically updated

Initialize Parameters...

Parameter: Year

Attributes:

☐ Map to database column Order Info.Order ID

☒ Map to database function YEAR(Enter Param Type)

Enter Function Parameter Values:
Order Info.Order Date

Default Value: 2001 Data Type: Integer

☐ Custom Date Format Build Update
(ex. MM/dd/yyyy, yyyy-MM-dd hh:mm:ss.ffffff)

User Interface:
Prompt Name: Enter Starting Date:

OK Cancel

Initialize Parameter Dialog

For certain databases, you may notice that the functions available do not specify the parameter types for functions. In this case, there will only be one function parameter field and it will be longer than normal. Selecting a column from the drop down list will be appended at the end of the field rather than replacing the existing text. You must enter all parameter values for the selected function, separating values with commas. Once you click away from this field, the default value field will be filled with function results and the data type will be automatically determined.

Default Value:

This allows you to specify a default value for the parameter. Although you do not have to specify a default value, it is recommended that you do so.

**Date Variable:**

This option is only available when the parameter is not mapped to a database column or function, and is only intended for parameters with variable type date/time. When you click this button, the following panel will pop up, listing all the supported keywords.

The dialog box titled "Set Date Variable..." contains a text input field with a "Clear" button. Below the input field, an example text reads "ex. FirstOfYear(CurrentDate) - 1 WEEK". The main area of the dialog is populated with buttons for date and time keywords and functions, organized as follows:

CurrentDate			CurrentTime			CurrentDateTime			+	-
YEAR	MONTH	WEEK	DAY	HOUR	MINUTE	SECOND				
FirstOfYear()			FirstOfQuarter()			FirstOfMonth()			FirstOfWeek()	
LastOfYear()			LastOfQuarter()			LastOfMonth()			LastOfWeek()	
StartOfDay()						EndOfDay()				

At the bottom of the dialog are "OK" and "Cancel" buttons.

Enter Date Variable Dialog

This dialog allows you to select one of the three keywords: *CurrentDate*, *CurrentTime*, and *CurrentDateTime*. You may add or subtract units of time from the current date/time, allowing you to have a dynamic date range. For example, a report may have the following default values:

StartDate: CurrentDate - 1 WEEK
 EndDate: CurrentDate

This would indicate that every time the report is run, the default prompt should be one week ago to the current date. Other supported measures are *YEAR*, *MONTH*, *DAY*, *HOUR*, *MINUTE*, and *SECOND*. This feature only supports a single addition or subtraction. This feature does not support multi-value parameters. Note: QuickDesigner Reports date values (in the right-hand panel) correspond to the variable values entered here, not the keywords themselves as in Report Designer.

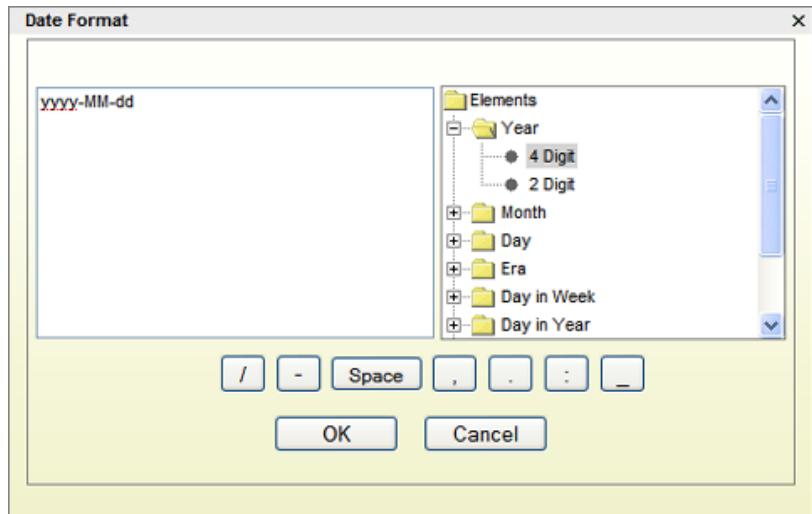
You can also use functions to define the parameter value.

Data Type:

This allows you to specify the data type for the parameter value(s). If you have mapped the parameter to a column, the data type is set automatically.

Custom Date Format:

This allows you to set the format in which the date parameter should be entered. This option is only available when you have mapped the parameter to a date/time column or when you not mapped the parameter at all and the parameter data type is date, time, or timestamp. When you check this option you can specify a date format by either using the builder interface or by entering the date format in a combination of characters that represent time elements.



Date Format Dialog

If you click the *Build* button, the above dialog will appear. The date/time representations are listed on the right and optional spacers and symbols are shown as a collection of buttons on the bottom. Once you have finished creating the format, click *OK* to save.

The date and/or time format is a series of characters and delimiters. Letters are used to represent different elements of date/time data. The characters and what each represent are listed below:

Character	Represents	Out-put (text/number)	Example
G	era	text	AD
y	year	number	1996, 96
M	month in year	text or number (depends on length)	July, Jul, 07
d	day in month	number	10
h	hour am/pm (1-12)	number	1
H	hour 24 hr. (0-23)	number	18
m	minute in hour	number	30
s	second in minute	number	55
S	millisecond	number	978
E	day in week	text	Tuesday, Tue
D	day in year	number	189
F	day of week in month	number	2 (as in 2nd Wed. in July)

Character	Represents	Output (text/number)	Example
w	week in year	number	27
W	week in month	number	2
a	am/pm marker	text	AM, PM
k	hour 24 hr (1-24)	number	24
K	hour am/pm (0-11)	number	0
z	time zone	text	Pacific Standard Time, PST

You can piece together almost any combination of these characters to produce a date expression in the format that you would like. The count of groups of characters determines the form that the element will take. For text elements 4 or more characters in a group will cause the full form of the element to be used. If less than four characters are used the short form will be used if one exists. For example, EEEE would return Monday and EE would return Mon. For month M which can display as either text or a number, four or more in a group will display the full version, three will display the abbreviation, and two or less will display the number form.

For numeric elements, the count of characters is the minimum number of digits that the element will take. Shorter numbers will implement leading zeros. For example if the day of the date is 2, dd would return 02 and d would return 2.

If the parameter is mapped to a column, once you have finished designing the format, click on the *update* button to refresh the parameter prompt with the new format.

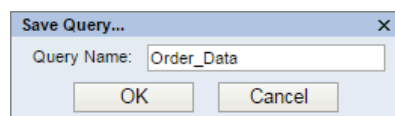
Prompt Name:

This allows you to specify the prompt that is given to the user in the parameter dialog.

Once you have finished setting values for all the parameters, click *Ok* to close the dialog and save the settings.

3.2.2.2.2. Save Queries

You can save the created query, or save the changes to the query if you selected a data view query to begin with by clicking the *Save Query* option in the conditions window. This will bring up a dialog prompting you to specify a name for the query.

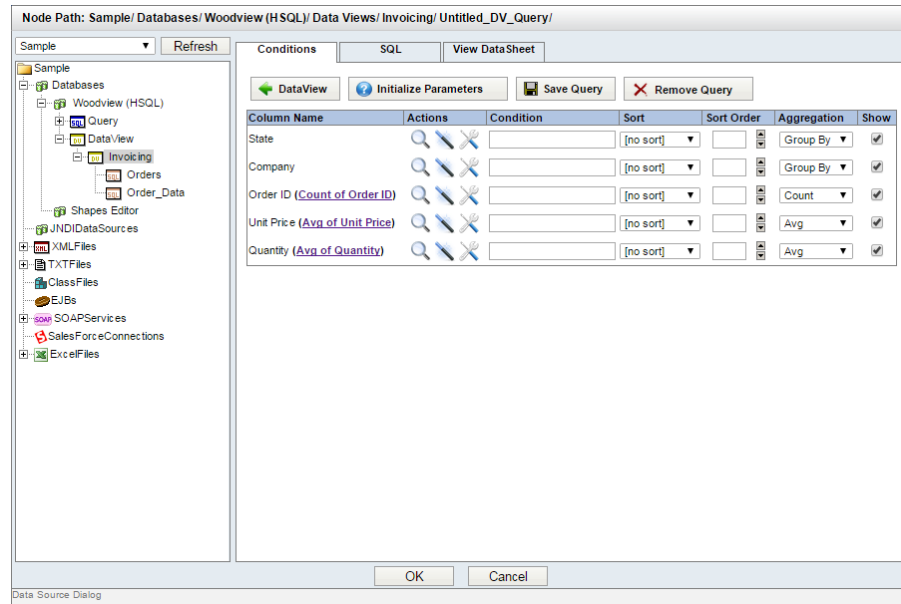


Save Query Dialog

Once you have specified the name, click *OK*. The query will be saved in the data registry as a new node under the data view.

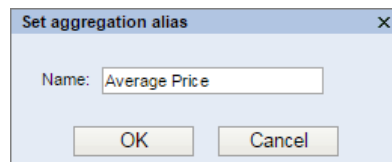
3.2.2.2.3. Aggregation Alias

If you add aggregation and group bys to the query, you are also able to specify an alias for the aggregated field.



Set Alias

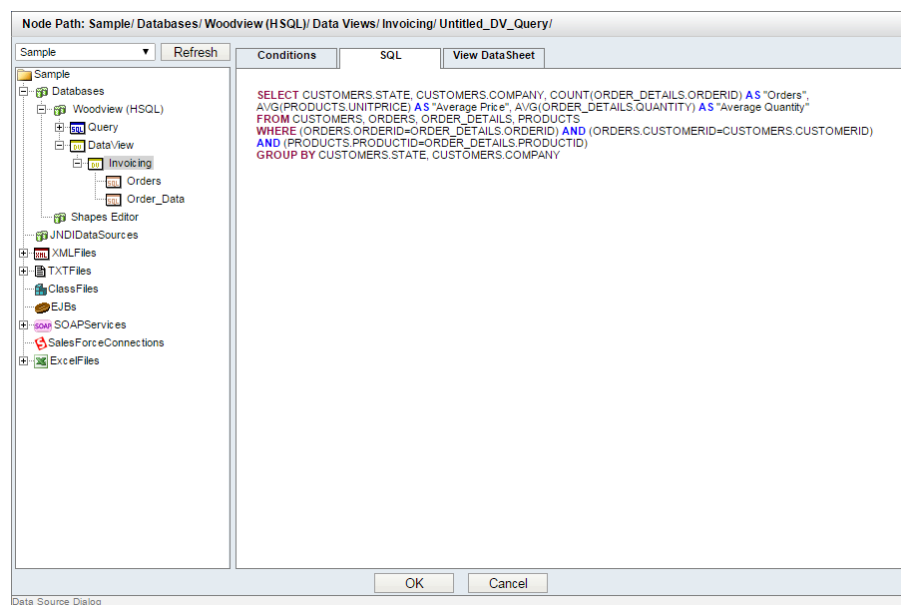
Clicking on the underlined option next to the column name will pop up a dialog allowing you to specify a name for the field. The name you specify here will replace the actual field name.



Alias Dialog

3.2.2.3. View SQL Query

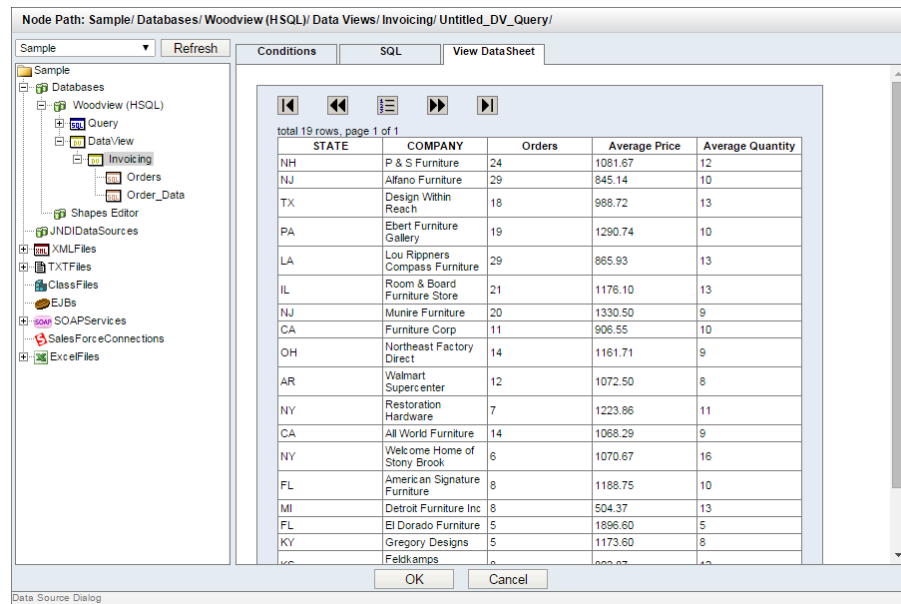
You can view the constructed query in the *SQL* tab of the *Data Source Dialog*.



SQL View


3.2.2.4. Preview Query Results


You can preview the query at anytime by clicking the *View DataSheet* tab in the *Data Source Dialog*.





Query Results


You can navigate through the result set using the icons at the top of the window.

 Go to the first page of the data table.

 Go to the previous page of the data table.

 Set the number of records to display per page. This will open a new dialog prompting you to specify the record number.

 Go to the next page of the data table.

 Go to the last page of the data table.

Chapter 4. Designing Reports & Charts

4.1. Chart Designer

4.1.1. Introduction to Chart Designer

The Chart Designer is a graphical user interface, launched within the Organizer that allows users to create and customize charts. The simple drop and drag style interface and extensive editing/formatting capabilities, makes chart design quick and easy.

4.1.1.1. Working with Charts

It is important to note that in EDAB there are two ways that you can use charts. Charts can either be placed within reports or designed and run as stand-alone entities. A unique feature of charts placed in reports is that they can use the report data as their data source. In this instance, the chart cannot be deployed outside of the report.

Charts that use their own data sources can be deployed independently from reports, either in menu pages or using image URLs to embed them in web pages.

4.1.1.2. Starting Chart Designer

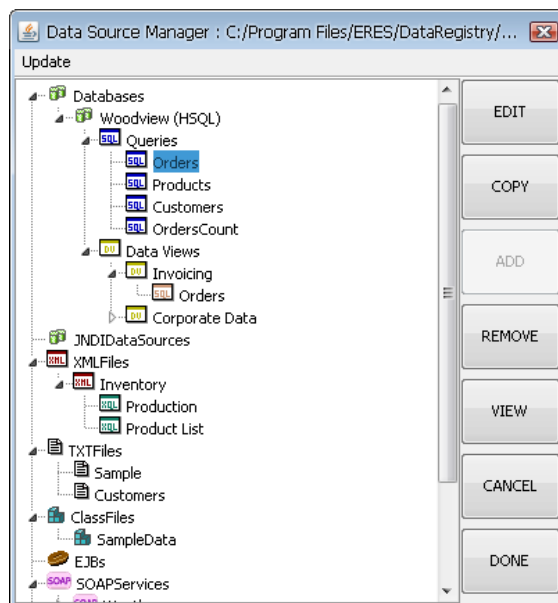
The Chart Designer interface is always loaded from within the Organizer. To start a new chart, you can click the *Chart Designer* button on the toolbar or select View → Chart Designer. You can also start Chart Designer when creating or editing data sources in the registry. The *View* button allows you to preview a data source and to build a chart or report. In addition, you can open the Chart Designer to edit a chart template file in the Organizer. To do so, first, select the file that you would like to open and then you can select File → Open File, right click on the file and select *Open File* from the pop-up menu, or double-click on the file.

If you are creating or adding an embedded chart within a report, the Chart Designer is launched from within Report Designer by either clicking on the *Insert Chart* button on the toolbar or by selecting Insert → Insert Chart.

4.1.1.3. Selecting a Data Source

The first step in designing a chart is to select the data source from which the chart is to be drawn. The first time you select to start a new chart you will be prompted to select the data registry that you would like to use. If there are not currently any available registries (meaning that you have not created any or you do not have privileges to view any), you will be prompted to go the Data Registry Manager to create one. For more on data sources, please see Section 3.1 - Data in Organizer

Once you have selected a registry, a Data Source Manager window will open allowing you to select, add, or modify a data source that you would like to use for the chart or report. Note that the registry will not open if you are building a chart from the registry in the *Modify Data Sources* dialogs.



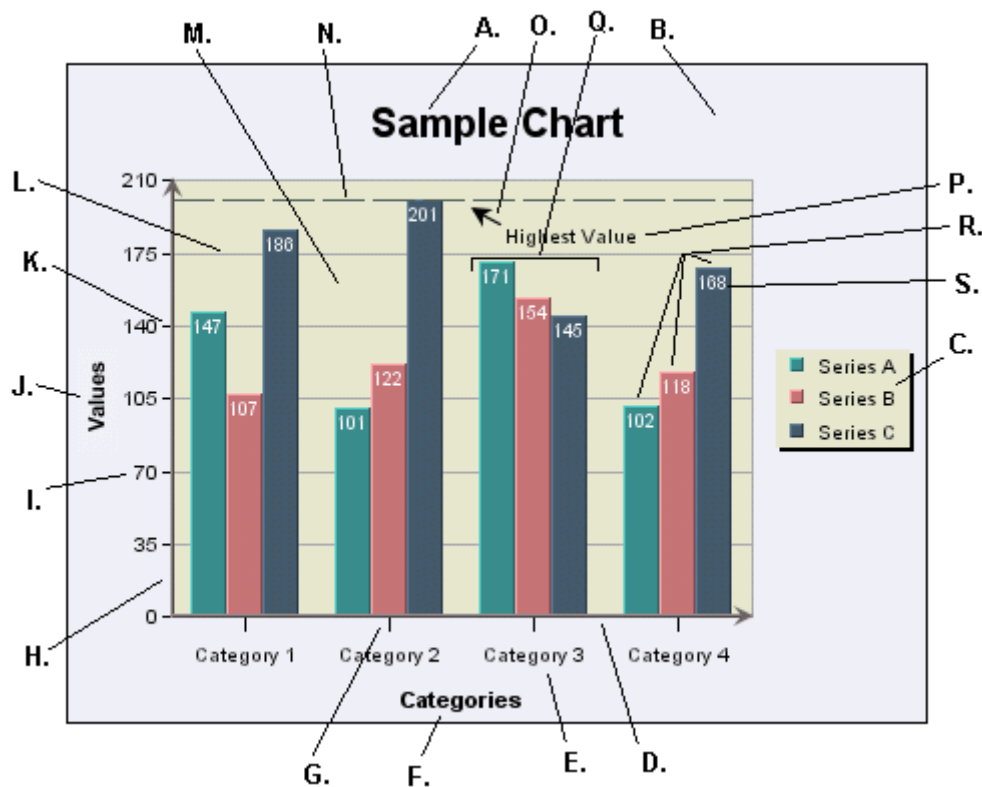
Data Source Manager Window for Chart Designer

4.1.2. Charting Basics

This section covers some of the basics of charts including the different parts of a chart and the way in which tabular data either from the report or another data source is mapped to a chart.

4.1.2.1. What is a Chart

The following diagram illustrates the various components that make up a chart. Also, this diagram refers to various terms that will be used throughout this guide.



A. Main Title	This is the main title for the chart.
B. Chart Canvas	This is the background on which the chart is drawn. The canvas serves as the size/boundary for the chart and it is generally the same size as any exported image. You can modify the canvas color or place/add an image file as a background.
C. Legend	This is the chart legend. The legend shows your category or series names along with a color point. Secondary values, as well as added trend/control lines and control areas, can also be displayed in the legend.
D. Category (X) Axis	This is the X or category axis of the chart. Generally, the category axis plots the distinct entries from the dataset for which you want to plot values in the chart. (The values are generally plotted on the Y-axis.) Certain chart types such as bar and Gantt reverse this by drawing the categories on the Y-axis, and plotting the values on the X-axis. Other chart types like scatter and bubble plots, plot values on each axis to create a point in 2D or 3D space.
E. X-Axis Labels	These are the labels for the X-axis elements or categories.
F. X-Axis Title	This is the X-axis title.
G. X Ticker	These are the X-axis tickers. By default the tickers match each data point in the chart.
H. Value (Y) Axis	This is the Y or value axis of the chart. Generally, the Y-axis plots the values for each of the categories. By default the scale of the Y axis is generated to provide a best fit for the dataset; however, it can be manually adjusted. For combination charts, the second value axis is drawn at the right-hand side of the plot.
I. Y-Axis Labels	These are the labels for the Y-axis values.
J. Y-Axis Title	This is the Y-axis title.
K. Y-Axis Ticker	These are the Y-axis tickers.
L. Y Grid	These are grid lines drawn along each scale step in the Y-axis. Grid lines can also be drawn for the points on the X-axis (and Z-axis for 3D charts).
M. Plot Area	This is the area, bounded by the axes, where all the data points are plotted. You can fill the area with color and/or draw a border around the area, along with other options. The plot area can be moved and resized on the chart canvas.
N. Control Line	This is one of the special types of lines that can be added to a chart. In this instance, it is a control line that follows the highest value in a series. Control lines can also be drawn for averages and multiples of standard deviation. Users can also add a variety of trend lines to charts.
O. Floating Line	A floating line is an arbitrary line added to a chart. In this case it is being used as a pointer with an arrow. Floating lines move in relative position with the chart plot. They can also be used to create filled shapes.
P. Annotation Text	This is a piece of arbitrary text added to a chart (not labels or titles). You can place text anywhere in the chart canvas. Like floating lines, annotation text moves in relative position to the chart plot.
Q. Category Elements	This is the plot for a category element. There are three points plotted for each category because this chart has a data series.

R. Data Series Elements

These are the individual points that make up a category. A series allows groups of data to be plotted on a single chart. For more about information about categories and series, see Section 4.1.2.2 - Basic Data Mapping

S. Data Top Labels

These are labels placed at each data point in the chart that display the exact value for each point.

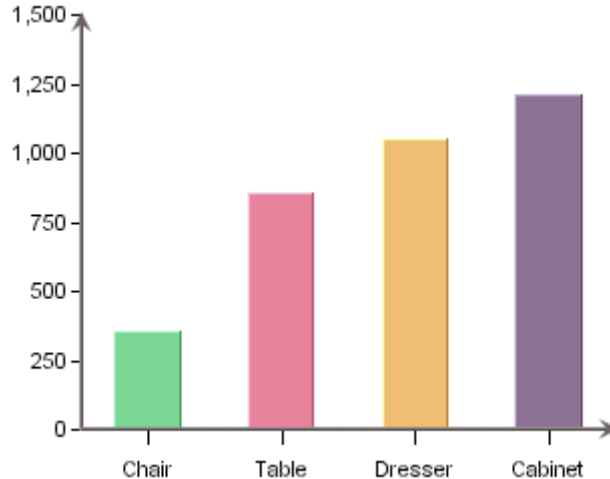
4.1.2.2. Basic Data Mapping

Data mapping is the way that raw data is rendered in the chart. Although data can be drawn from many sources, a chart looks for the basic structure of the data to be in the form of a table. Hence, data passed in as arguments, from the report or from XML files is converted to a table structure before mapping.

A basic set of data might look something like this:

Product	Sales
Chair	362
Table	862
Dresser	1052
Cabinet	1211

To plot this data in a chart, you would want to plot the **Sales** value for each entry in the **Product** column. Hence, the products are your category and the sales numbers are your values. In a chart you would map the **Product** column to the X (category) axis and the **Sales** column to the Y (value) axis. The resulting plot would look like this in a column chart:

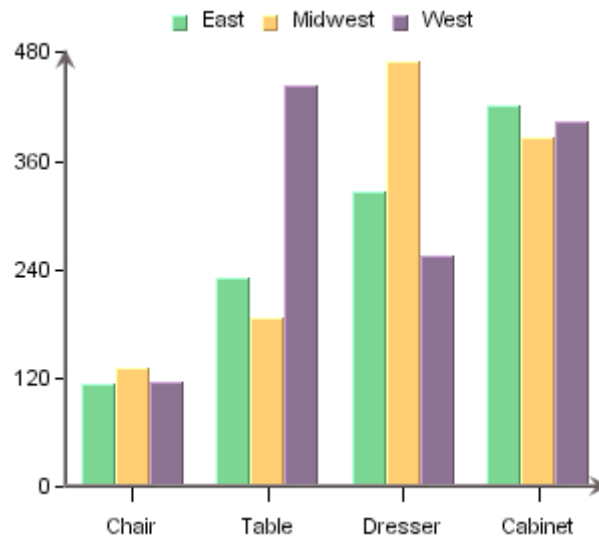


Here, a column is drawn to show the value for each distinct element in the category column. On top of the basic category values, additional information can be displayed in the form of a data series. For example, say that there is another element to our data that shows sales data not only for product, but also over a sales region. Our adjusted table would look like this:

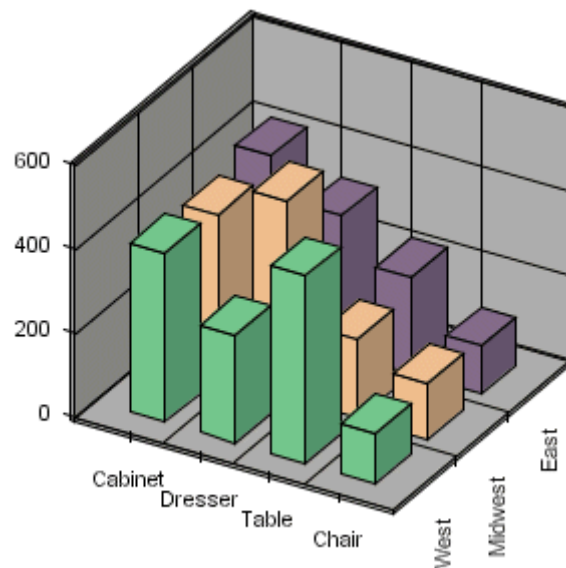
Product	Region	Sales
Chair	East	114
Chair	Midwest	131
Chair	West	117
Table	East	231

Product	Region	Sales
Table	Midwest	187
Table	West	444
Dresser	East	327
Dresser	Midwest	469
Dresser	West	256
Cabinet	East	422
Cabinet	Midwest	386
Cabinet	West	403

In order to show the value for each region per product we could add the **Region** column to the data mapping as a data series. Doing this gives us the following chart:



Now each category has three data points, one for each region. For two-dimensional charts the series is always displayed in-line. In three-dimensional charts, the series is drawn on the Z-axis by default, although it can be drawn in-line as well. Below, is the same chart show in 3D.



In this chart, the data series is drawn along the Z-axis. Note that the order of the categories has been changed to provide a better view of the data. This is the basic concept behind data mapping. Most chart types use this mapping technique or mapping options similar to this. Detailed data mapping instructions for each chart type are available in Section 4.1.3 - Chart Types and Data Mapping.

4.1.2.3. Saving and Exporting Charts

There are several options available both for saving chart definitions and exporting charts as image files. More information on how to save and export charts in the Chart Designer can be found in Section 4.1.6 - Saving & Exporting Charts.

4.1.2.3.1. Saving Chart Definitions

There are two primary methods which you can use to store chart definitions created in ChartDesigner: either as chart or as template files.

Chart files	Chart files save the chart in a binary file called <code>filename.cht</code> . A chart file stores both the definitions of the chart (type, dimension, etc.) as well as the data that was used to create the chart. Hence, chart files are portable. Any time you open a chart file, it will open with the original data that was used to create the chart. After opening the chart, you can refresh the data from the source or change the chart's data source entirely.
Template files	Template files save the chart in a binary file called <code>filename.tpl</code> . A template file stores only the chart definitions and stores only 10 records of data with the chart. Hence, any time a template file is opened it will try to connect to the original data source to retrieve the data. Because of this, template files can be less portable than chart files. Template files can also be used to pass chart attributes from one chart to another. To do this, you can apply a template to a chart. This will carry over many of the attributes of the template to the current chart. For more information about applying templates, please see Section 4.1.6.1.1 - Working with Templates.

In addition to saving chart definitions in binary format, chart files and template files can also be saved in XML format. These XML chart definition files can be modified outside of Chart Designer or Chart API.

4.1.2.3.2. Generating Image Files

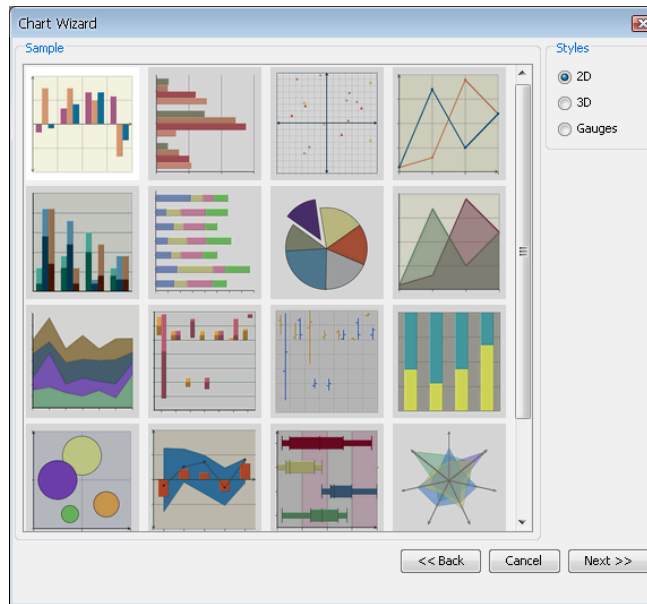
For image files, EspressDashboard can render charts in the following formats:

GIF	EDAB can generate GIF images using one of two compression methods - RLE or LZW. The LZW method is faster and produces smaller files; however, its use is protected by patent. You must obtain a license from Unisys in order to unlock the LZW compression. By default, GIF files are generated using RLE compression.
JPEG	JPEG is another popular image format. It is a higher resolution image format than GIF and it is not patent protected. When generating a JPEG file you can specify the quality and compression of the file. The higher the quality, the larger the file.
PNG	PNG is an image format that is less popular but it can be displayed in most browsers. It is a high-quality image with a smaller file size than JPEG. There are three different compression options available for this format.
SVG	SVG (Scalable Vector Graphics) is a relatively new image format that saves the image as vectors in an XML-based text format. Generally, you will need a browser plug-in to view these images.
SWF	SWF is an Adobe Flash file. The flash format is vector based and it allows the chart to be resized after export. Also, flash allows for high-resolution printing and produces a small file size.
BMP	This is a Windows bitmap format.
WMF	WMF is the Windows Meta File format. This can be used for import/export into MS Office documents.

In addition to static images, ChartDesigner also allows the chart data to be exported as a text file, PDF, or an XML file.

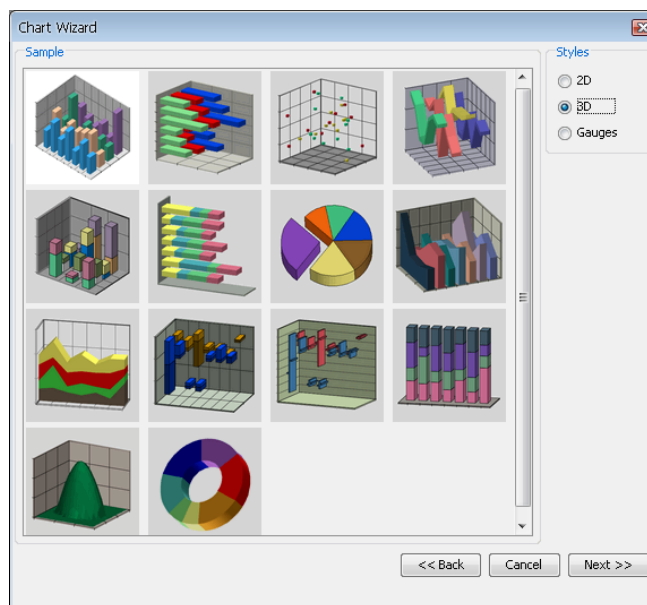
4.1.3. Chart Types and Data Mapping

Once you have selected the data that you would like to use for the chart, the next step in the Chart Wizard is to select the chart type that you would like to use. You will be presented with a dialog that allows you to specify the chart type.



Two-Dimensional Chart Types Selection Dialog

Each chart type represents a different way in which the data points are plotted to give a pertinent representation to all kinds of data. The different types of charts are broken down by dimension. In addition to basic chart types, users can create many different types of composite/combination charts by adding secondary values/series to the chart. You can toggle between the chart categories by selecting either *2D*, *3D* or *Gauges* in the right-hand side of the chart types dialog.



Three-Dimensional Chart Types Selection Dialog

This dialog enables you to select your chart type by either selecting the image and clicking *Next* or by double clicking on the chart image. You can select from one of the following chart types:

- Column

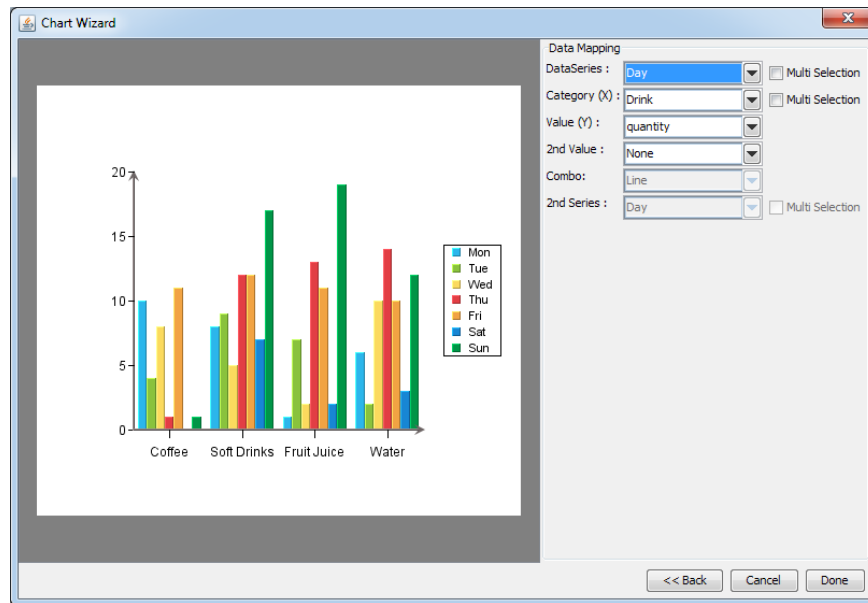
- Bar
- XY(Z) Scatter
- Line
- Stack Column
- Stack Bar
- Pie
- Area
- Stack Area
- High Low
- HLCO
- Percentage Column
- Doughnut
- Surface (Three-Dimensional Only)
- Bubble (Two-Dimensional Only)
- Overlay (Two-Dimensional Only)
- Box (Two-Dimensional Only)
- Radar (Two-Dimensional Only)
- Dial (Two-Dimensional Only)
- Gantt (Two-Dimensional Only)
- Polar (Two-Dimensional Only)
- Circular Gauge
- Square Gauge
- Semi Circular Gauge
- Square Gauge
- Quarter Circular Gauge
- Heatmap

Each of the chart types is described in detail later in this chapter, starting with Section 4.1.3.2 - Column Charts

For information on gauges, see Section 4.1.3.20.2 - Gauges

4.1.3.1. Data Mapping

After you have selected the chart type, the next step is to specify the data mapping for the chart. Data mapping is the way that the selected data source is rendered in the chart. The basics of data mapping are described in Section 4.1.2.2 - Basic Data Mapping. The data mapping screen in the Chart Wizard allows you to set the mapping options and also preview the results.



Data Mapping Dialog

The left-hand side of the dialog shows a preview of your chart. The right-hand side shows which columns from your data source have been selected to plot in the different chart elements (series, category, value, etc). By default, EDAB will select the first available columns, based on the data type, to plot. Changing the data mapping is easy - click on the down-facing arrow next to a data field and select a different field. The chart preview in the left-hand part of the data mapping dialog will be updated immediately.

The specific mapping options vary for each chart type and are discussed later in this chapter starting with Section 4.1.3.2 - Column Charts.

4.1.3.1.1. Data Transposition

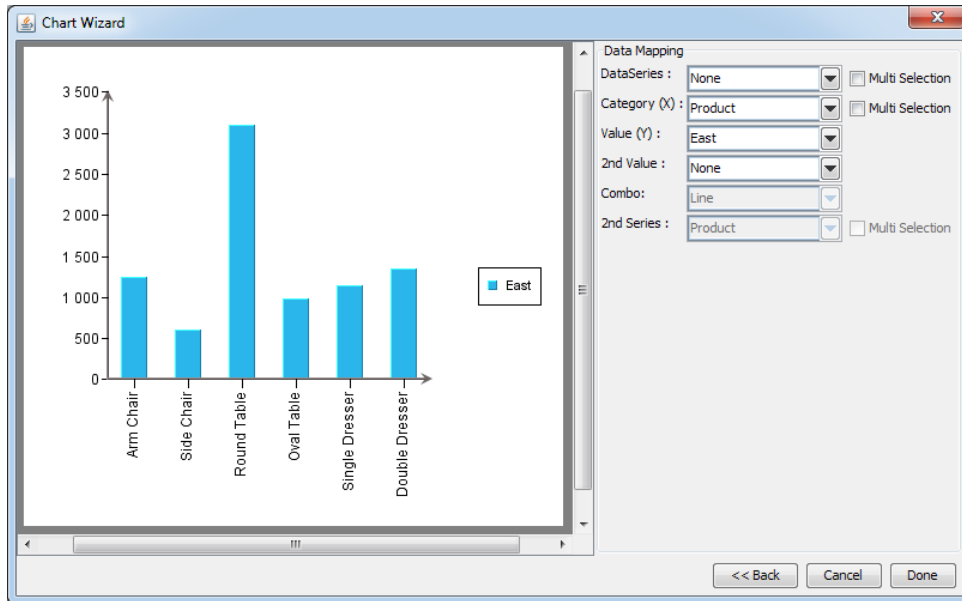
You can also set data transposing from the data mapping dialog. Transposing allows you to plot multiple data source columns in one chart without modifying the data source.

For example, we have a data source that looks like this:

INDEX	Product	East	Midwest	West
TYPE	Varchar	Integer	Integer	Integer
1	Arm Chair	1241	2100	800
2	Side Chair	600	2940	1150
3	Round Table	3100	2500	2630
4	Oval Table	980	1660	1210
5	Single Dresser	1145	2340	1970
6	Double Dresser	1345	3560	1200

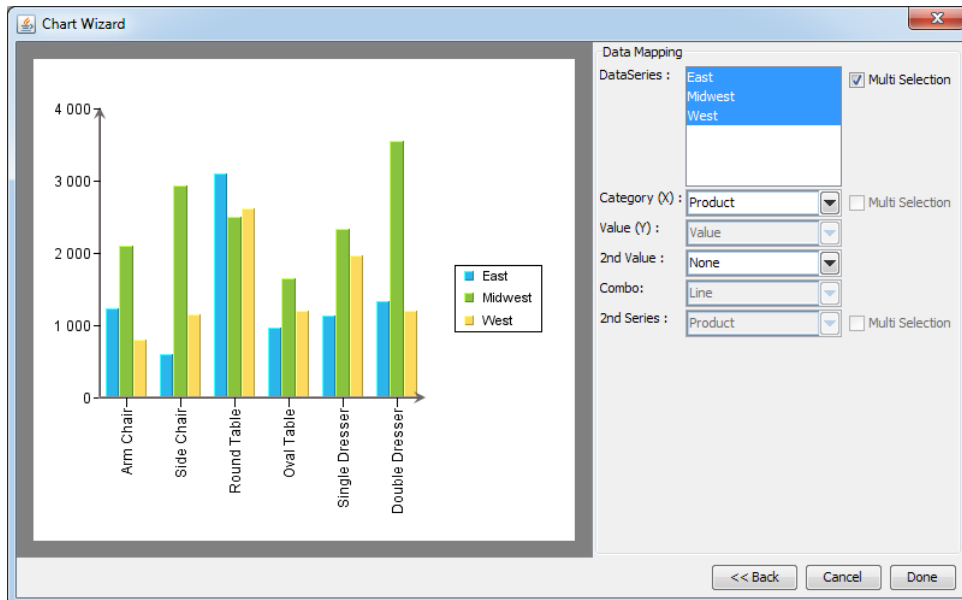
Example Data Source

We would like to plot data from all regions in one chart. The default mapping without data transposing does not allow us to do that. We would have to plot only one region in the chart or modify the data source, so we will have to transpose the data source.



Default Chart Data Mapping

To transpose the data, select the *Multi Selection* option next to the *Data Series* field. The field will change to a list allowing us to select several columns at the same time. We will just select all region columns and the chart will look like this:



Default Chart Data Mapping

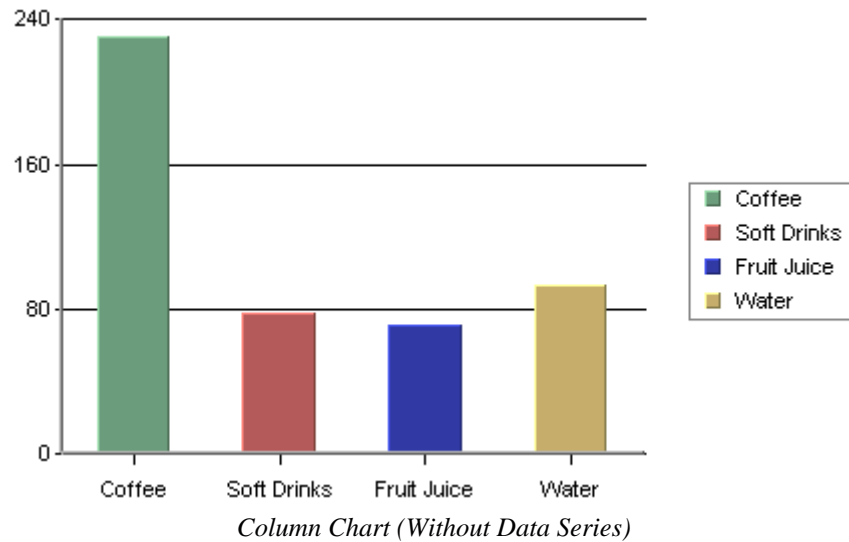


Note

Only one chart element (such as data series, category, value, or second series) can be transposed at a time. If you select the *Multi Selection* checkbox for one chart element, the *Multi Selection* checkboxes will be deactivated for other chart elements.

Transposing is not available for drill-down charts.

4.1.3.2. Column Charts



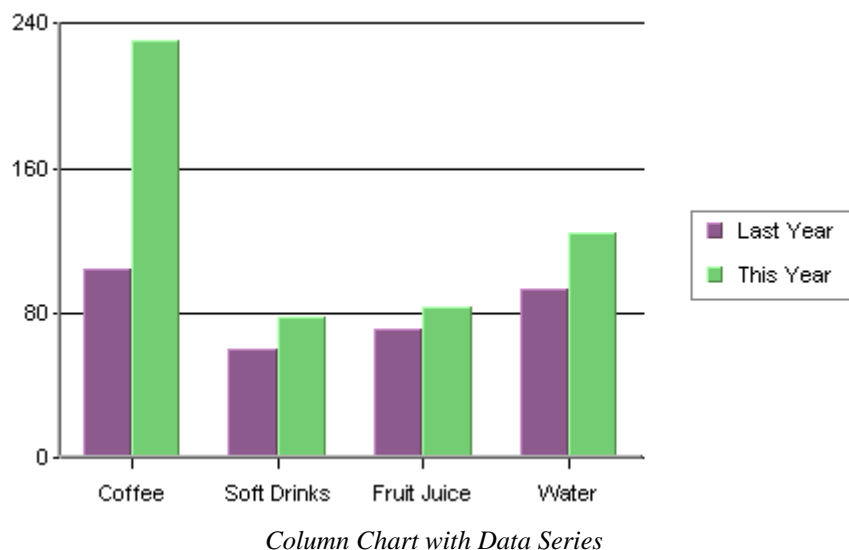
A column chart displays each row in the data table as a vertical bar (or column). The categories are listed along the X-axis and the values plotted along the Y-axis. Column charts are good for comparing discrete values for different groups. Each group is represented by a different color.

In a two-dimensional chart, if a data series has been selected then the entire series for a single category will be displayed in the XY plane. If a three-dimensional column chart is being used then all the vertical bars in a given data series are drawn using the same color along the Z-axis. If a data series is not present in the chart, the categories will be represented by the same color by default. Different colors for the categories are possible to set by clicking the



Change chart options icon on the toolbar (or selecting Format → Chart Options), and unchecking the *Single Color For All Categories* option.

In the examples given, we have chosen Drink as the category variable and Value as the value variable (most examples use the `Sample.dat` file included in the EDAB installation). Year has been selected as the data series variable for the chart below. Therefore, each name has two columns shown: one for *last year* and one for *this year* which are the only two values present in the data series column.



4.1.3.2.1. Data Mapping

Data mapping for column charts is fairly straightforward. It is similar to the examples first presented in Section 4.1.2.2 - Basic Data Mapping. For column charts, the following options are available:

Data Mapping

DataSeries : year ☐ Multi Selection

Category (X) : Drink ☐ Multi Selection

Value (Y) : value ☐ Multi Selection

2nd Value : quantity ☐ Multi Selection

Combo: Line ☐ Multi Selection

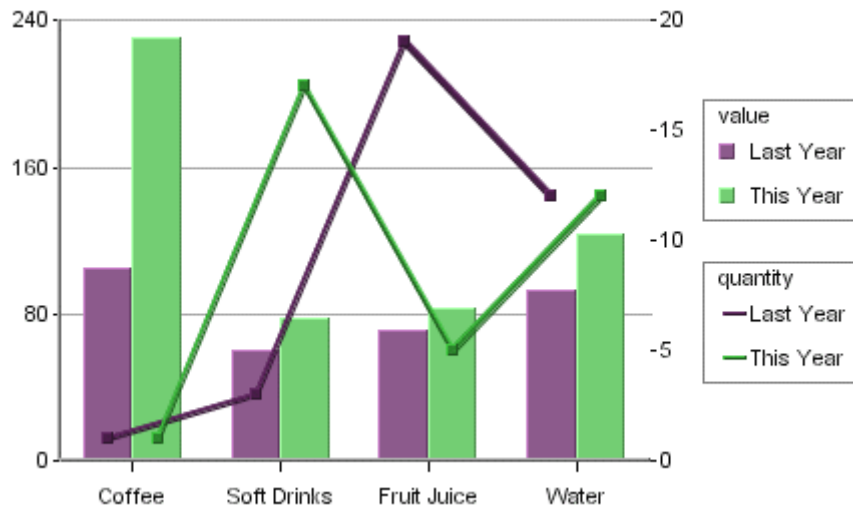
2nd Series : Day ☐ Multi Selection

Mapping Options for Column Charts

- Data Series:** Choose a data column whose distinct values will determine the number of data series in the chart. Each element in a data series is drawn using the same set of colors and other attributes.
- Category (X):** Choose a data column whose distinct values will determine the categories.
- Value (Y):** Choose a data column to provide values for each category.
- 2nd value:** Add a second value to create a combination chart.
- 2nd Series:** Choose another column to be series for the secondary chart. This option is applicable only if the secondary chart is an overlay chart.
- Combo:** Choose the chart type for the secondary chart. For column charts the combo options are *Line* and *Overlay*.

The data mapping also allows you to transpose the data (in other words: to select several columns for a single category). To learn more about data transposition, please see Section 4.1.3.1.1 - Data Transposition.

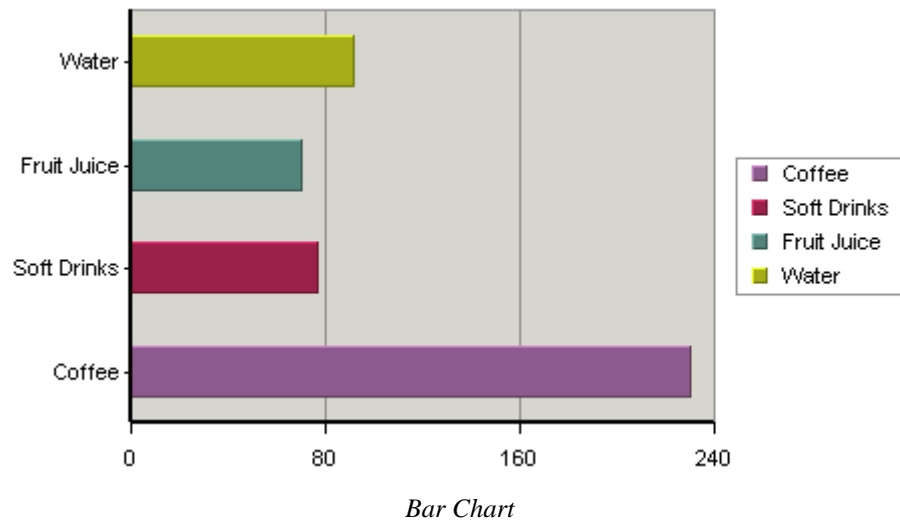
The last three data mapping options allow you to add a second value to the chart. ChartDesigner supports secondary values for all chart types except pie, radar, bubble, dial, surface, and scatter charts. In the example below, a second value of Quantity has been added to our column chart.



Column Chart With 2nd Value


As you can see, the second axis labels are drawn on the right-hand side of the plot area (you can choose to make this axis visible). Usually, the second value will share the same categories and series as the primary value. However, for two-dimensional column, stack column, stack area, high low, HLCO, and percentage column charts you can specify an overlay combination, which allows you to specify a second series. For more on overlay charts, please see Section 4.1.3.17 - Overlay Charts.

4.1.3.3. Bar Charts



A bar chart is essentially the same as a column chart, except that horizontal bars are drawn in the chart as opposed to the vertical bars which are used in a column chart. In a bar chart, the categories are plotted along the Y-axis and the values along the X-axis.

Just as for a column chart, if a data series has been selected then the entire series for a single category is displayed in the XY plane. If a three-dimensional bar chart is being used, all the horizontal bars in a data series are drawn using the same color. Each category is drawn using a different color. If the data series is not present in the chart, the categories will be represented by the same color by default. Different colors for the categories are possible to set by

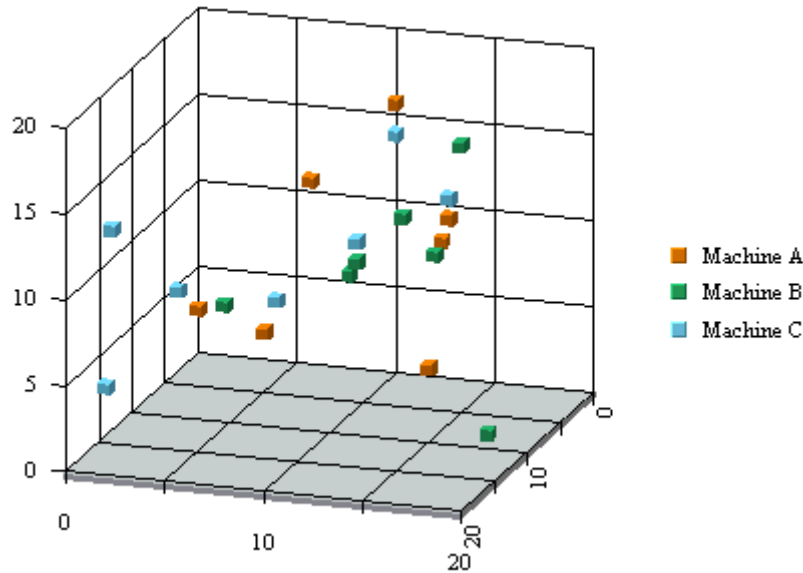
clicking the  *Change chart options* icon on the toolbar (or selecting Format → Chart Options), and unchecking the *Single Color For All Categories* option.

4.1.3.3.1. Data Mapping

Mapping Options for Bar Charts

The mapping for this chart type is similar to that of a column chart, except the category (X) and values (Y) under column chart become category (Y) and values (X) respectively. This is because values are represented vertically in a column chart, but horizontally in a bar chart. Please note that the *2nd Series* and *Combo* options are not available for bar charts. This is because the only combination available with bar charts is a line.

4.1.3.4. XY(Z) Scatter Charts



XY(Z) Scatter Chart

In an XY(Z) scatter chart, each selected row in the data table defines a point in two or three-dimensional space. Thus each column must contain either numeric or date/time values. A marker represents each point. The data columns that are in each row of the data table determine the spatial position of the marker.

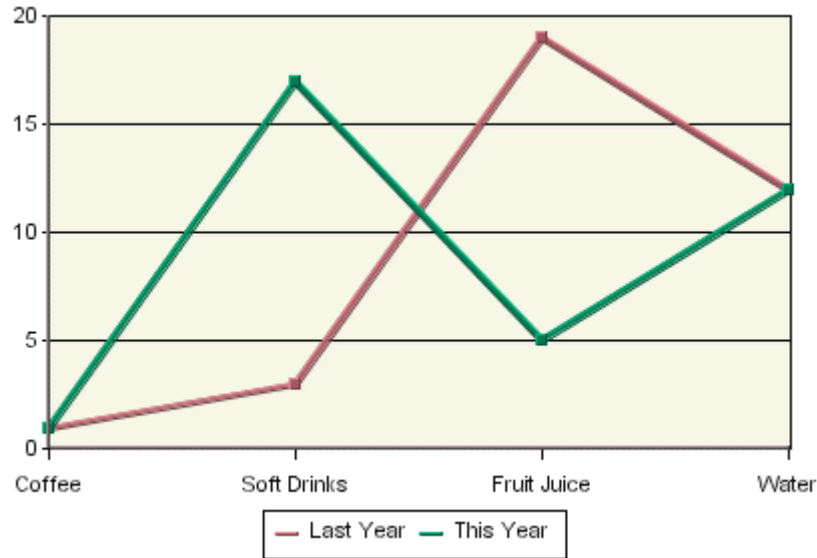
Optionally, another data table column can be chosen to separate the markers into groups. Elements of each group have the same value on this column which is referred to as the data series column. Markers in the same group are drawn using the same drawing attributes; in other words, using the same shapes and colors. The X-axis scale of a scatter chart is linear. This means that unlike other chart types, the data points may or may not be evenly spaced along the X-axis.

4.1.3.4.1. Data Mapping

Mapping Options for Scatter Charts

In a scatter chart, the X-axis, Y-axis, and Z-axis values determine the X, Y, and Z coordinates of a point respectively. The data series box allows you to choose a data column whose distinct values will determine the number of data series in the chart. Each element in a data series is drawn using the same set of drawing attributes(e.g., colors and markers).

4.1.3.5. Line Charts

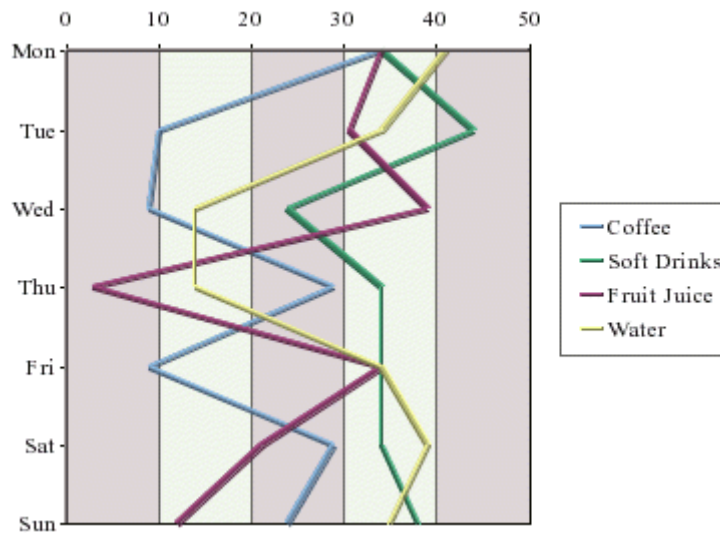


Line Chart

Line charts present data in a manner similar to column charts. For a two-dimensional line chart, a marker denotes a data point for each category. The value column, which must be numeric, determines the height of a marker. Each marker is joined with a line. If the chart has a data series, a separate line is drawn for each element in the series. Please note, the markers (points) are not shown by default. You can enable showing markers (points) in the *Line*

and *Point* dialog. This dialog can be opened from the *Format* menu or by clicking the  *Format line and points* icon on the toolbar.

ChartDesigner allows 2D line charts to be displayed vertically in addition to the default horizontal setting. To use this feature in Chart Designer, create a line chart and then select *Format* → *Chart Options*. Next, select *Vertical*. The chart is rotated clockwise 90 degrees.



Vertical Line Chart

A three-dimensional line chart is an extension of its two-dimensional counterpart. It contains no additional information. The markers disappear (they are not available for 3D line charts) and thicker lines, spaced apart on the Z-axis, replace the thin lines.

4.1.3.5.1. Data Mapping

Data Mapping

DataSeries : year ☐ Multi Selection

Category (X) : Drink ☐ Multi Selection

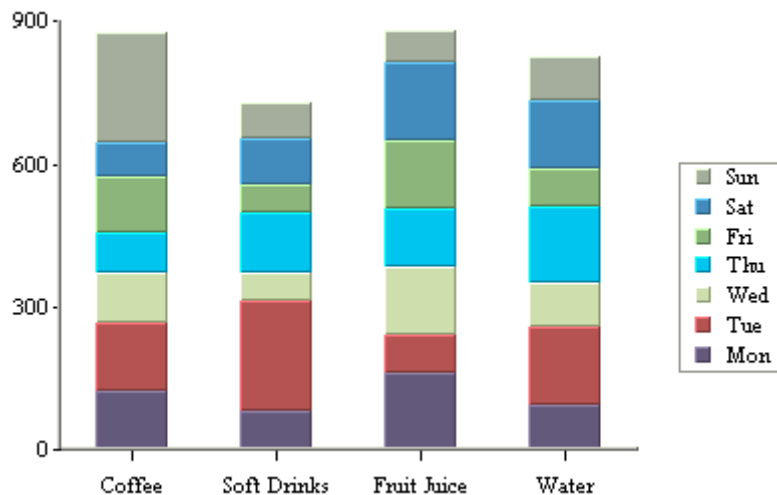
Value (Y) : value ☐ Multi Selection

2nd Value : quantity ☐ Multi Selection

Mapping Options for Line Charts

Data mapping for line charts is almost exactly the same as for column charts (discussed in Section 4.1.3.2.1 - Data Mapping). Line charts, however, do not have the *2nd Series* and *Combo* options. This is because the only combination available with line charts is a line. To create line combinations with other chart types (bar, column, stack column, etc) select the other chart type as the primary chart, and select *Line* as the *Combo* option.

4.1.3.6. Stack Column Charts



Stack Column Chart

A stack column chart is a derivative of a column chart in which each vertical column comprises a stack of bars. Each selected row in the data table is represented by a component of a chart column. For a two-dimensional stack column chart, the categories are plotted on the X-axis and the values on the Y-axis. The value column, which must be numeric, determines the length of each bar component. A third column, known as the sum-by column, represents a group of values for each category. A stack for a given category value is built up stacking the sum-by values for that category value. Each stack component of a chart column has a distinct sum-by value denoted by a distinct color. Each row in the data table must have a unique pair of values on both the category and sum-by columns. Components with negative values are separated from components with positive values and they are stacked up in the “negative” direction below the X-axis.

In our example above, we have chosen types of drink as category value and day representing the sum-by values. Another independent category (to be displayed on the Z-axis) may be optionally specified based on a fourth column as a data series. In such a case, each row in the data table must have a unique triplet of values on the category, sum-by and data series columns. In a two-dimensional chart, the entire data series for a single category is displayed side-by-side in the XY plane. In a three-dimensional stack column chart, the data series is displayed along the Z-axis.

4.1.3.6.1. Data Mapping

In stack column charts, each category is further subdivided into components. Hence, there is an additional sum-by option used to determine the sum-by values for the chart.

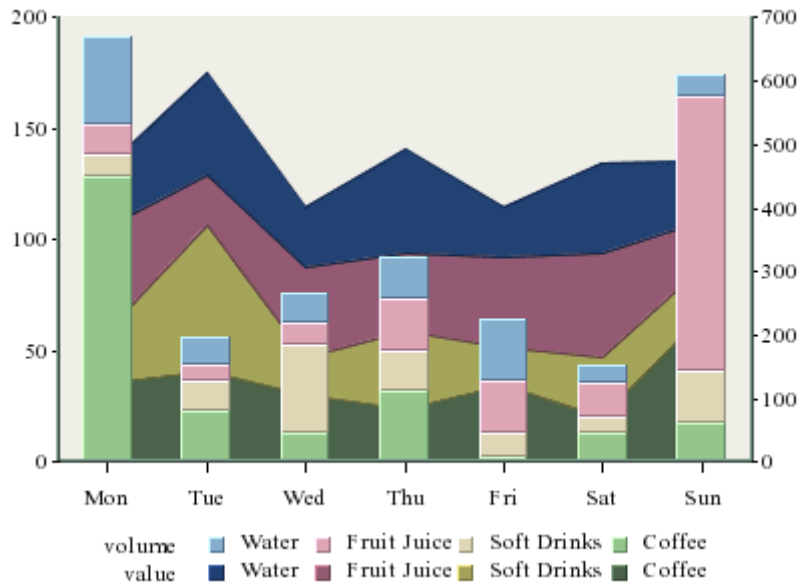
Mapping Options for Stack Column Charts

The data mapping options are as follows:

- Data Series:** Allows you to choose a data column whose distinct values will determine the number of data series in the chart. Each element in a data series is drawn using the same color.
- Category (X):** Allows you to choose a data column whose distinct values determine the categories. Values from this data column are used to calibrate the X-axis. Each category in a data series is drawn as a distinct column in the chart.
- Sum-by:** Allows you to choose a data column whose distinct values determine the components in each category. Each distinct value in this column determines a distinct stack component of a column.
- Value (Y):** Choose a data column to provide values for each category.
- 2nd value:** Add a second value to create a combination chart.
- 2nd Series:** Choose another column to be series for the secondary chart. This option is applicable only if the secondary chart is an overlay chart.
- Combo:** Choose the chart type for the secondary chart. For stack column charts the combo options are *Line*, *Stack Area*, and *Overlay*.

The data mapping also allows you to transpose the data (in other words: to select several columns for a single category). To learn more about data transposition, please see Section 4.1.3.1.1 - Data Transposition.

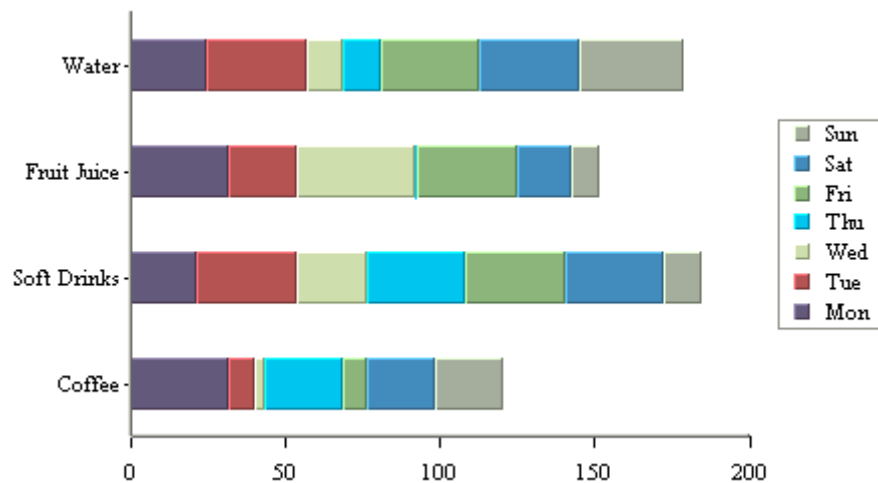
Stack column charts have a unique combination option which allows users to plot a second value as a stack area chart. Both values share the same category and sum-by values. This chart can compare component based categories for different values in the data.



Stack Column-Stack Area Combination Chart

With this combination chart you can specify to hide particular sum-by component in the chart to achieve a strip area effect.

4.1.3.7. Stack Bar Charts



Stack Bar Chart

Like a stack column chart, the stack bar chart displays the chart categories as a sum of different stacked values - the sum-by column in the data source. The difference for a stack bar chart is that the categories are plotted on the Y-axis and the values are plotted along the X-axis.

4.1.3.7.1. Data Mapping

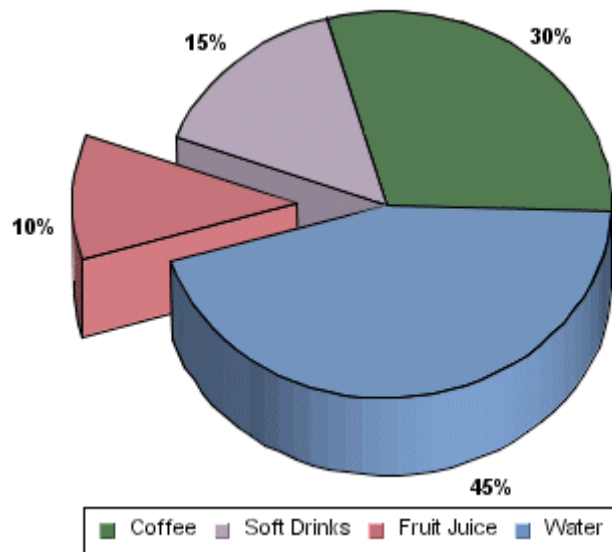
Data Mapping

DataSeries :	None	<input type="checkbox"/> Multi Selection
Category (Y) :	Drink	<input type="checkbox"/> Multi Selection
Sum by :	Day	<input type="checkbox"/> Multi Selection
Value (X) :	volume	
2nd Value :	None	

Data Mapping Options for Stack Bar Charts

The mapping for this chart type is similar to that of a stack column chart, except the category (X) and values (Y) under column chart become category (Y) and values (X), respectively. This is because values are represented vertically in a column chart, but horizontally in a bar chart. Please note that the *2nd Series* and *Combo* options are not available for stack bar charts. This is because the only combination available with stack bar charts is a line.

4.1.3.8. Pie Charts

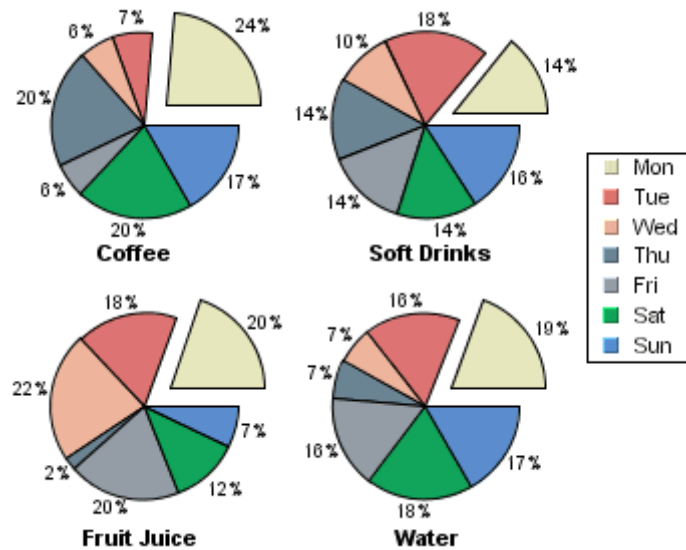


Pie Chart

In a pie chart, each row in the data is represented as a pie sector. The pie chart requires a category column and a value column. The value column must be numeric. The number of distinct values in the category column determines the number of pie sectors in the chart. All the values are summed up and each slice is assigned an angle according to its share in the total. Thus, the value column for each category determines the size of the slice representing that category.

A three-dimensional pie chart is simply an extension of its two-dimensional counterpart and contains no extra information.

Pie charts can also have a data series. If you specify a series in the data mapping, a separate pie will be drawn for each category element and it will be comprised of the series elements. A pie chart with a data series is displayed below:



Pie Chart With Data Series

When a series is present, all the separate pies are drawn on a single plot area and resized in proportion with the plot. You can choose to either stack the different pies or draw them in a line.

4.1.3.8.1. Data Mapping

Data Mapping

DataSeries : ☐ Multi Selection

Category (X) : ☐ Multi Selection

Value (Y) :

Mapping Options for Pie Charts

For pie charts, the mapping is as follows:

Data Series: Allows you to choose a data column whose distinct values will determine the number of data series in the chart. Each element in a data series is drawn using the same color.

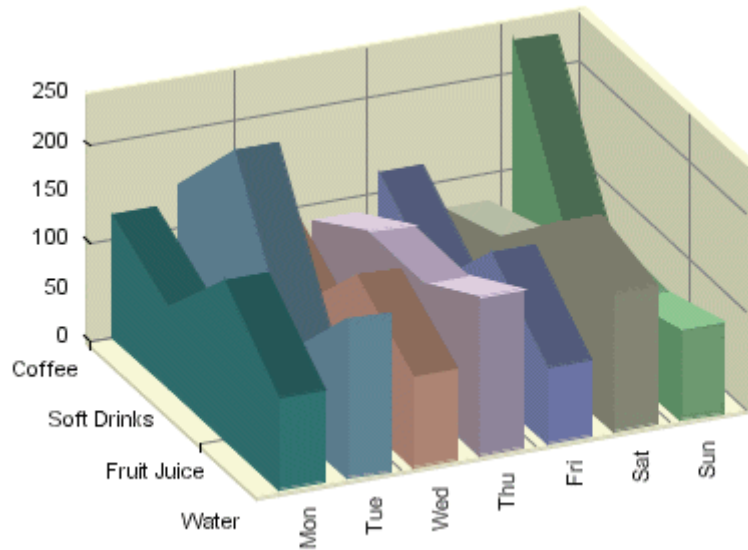
Category (X): Allows you to choose a data column whose distinct values determine the various categories.

Value (Y): Allows you to choose a data column to provide values for each category.

There is no *2nd Value* option, as you cannot make any combination charts with a pie chart.

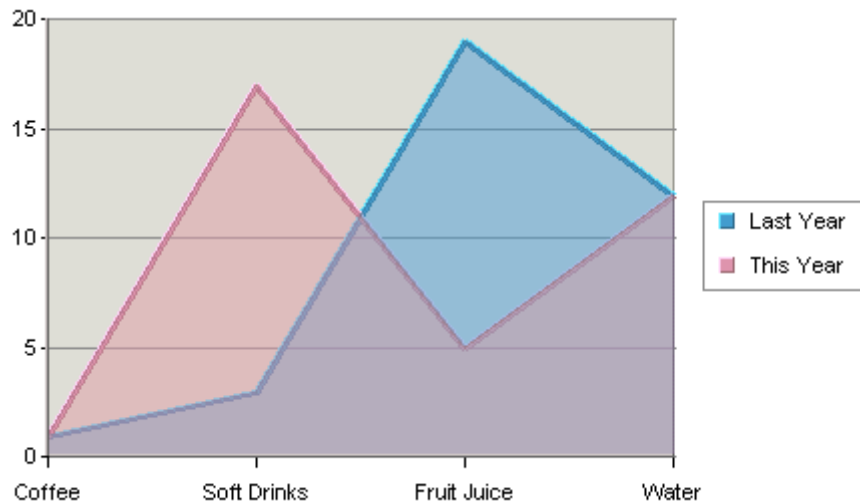
The data mapping also allows you to transpose the data (in other words: to select several columns for a single category). To learn more about data transposition, please see Section 4.1.3.1.1 - Data Transposition.

4.1.3.9. Area Charts



Three-Dimensional Area Chart

A three-dimensional area chart may be viewed as a derivative of a column chart. A three-dimensional area chart may be constructed from a three-dimensional column chart in the following manner. For each data series (Z-axis) value, the tops of all the columns are joined together by a thick line. The columns are then removed and the area (in the XY plane) under each line is filled with a distinct color.



Two-Dimensional Area Chart

A two-dimensional area chart is essentially a projection of its three-dimensional counterpart viewed along the Z-axis. As a result, a two-dimensional area chart may sometimes hide a data series value altogether. Therefore, it must be used with caution. The chart above illustrates this point: in a two-dimensional display some parts of each series are concealed by a series in front. To ameliorate this problem, you can change the ordering of the data series (see Section 4.1.4.8.3 - Data Ordering) or set the area translucent (see Section 4.1.4.1.3 - Format Menu) as in the example above.

4.1.3.9.1. Data Mapping

Data Mapping

DataSeries : ☐ Multi Selection

Category (X) : ☐ Multi Selection

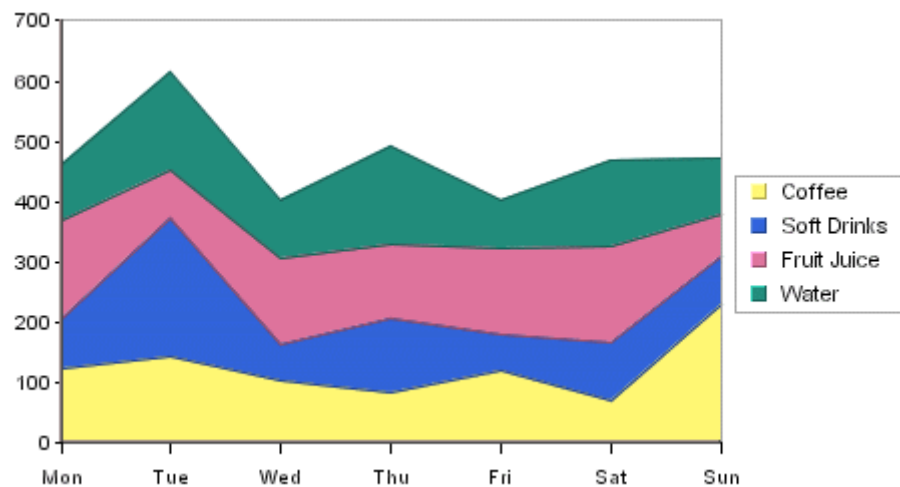
Value (Y) :

2nd Value :

Mapping Options for Area Charts

Data mapping for area charts is almost exactly the same as for column charts (discussed in Section 4.1.3.2.1 - Data Mapping). However, area charts do not have the *2nd Series* and *Combo* options. This is because the only combination available with area charts is a line.

4.1.3.10. Stack Area Charts



Stack Area Chart

A two-dimensional stack area chart may be viewed as a derivative of a two-dimensional stack column chart but without a data series. This chart may be constructed from a stack column chart as follows: The tops of each stack component that have the same color are joined together by lines. The stack columns are removed and the area between each set of lines is filled with a distinct color.

A three-dimensional stack area chart can have a data series and can also be derived from a three-dimensional stack column chart using the process stated above. In this case, the stack columns for each distinct data series value (a fixed Z-axis value) are joined separately, giving rise to multiple stacks.

Both two-dimensional and three-dimensional stack area charts may be constructed using a similar set of data as needed for their respective stack column chart counterparts. Note, as stated earlier, that a two-dimensional stack area chart cannot have a data series.

4.1.3.10.1. Data Mapping

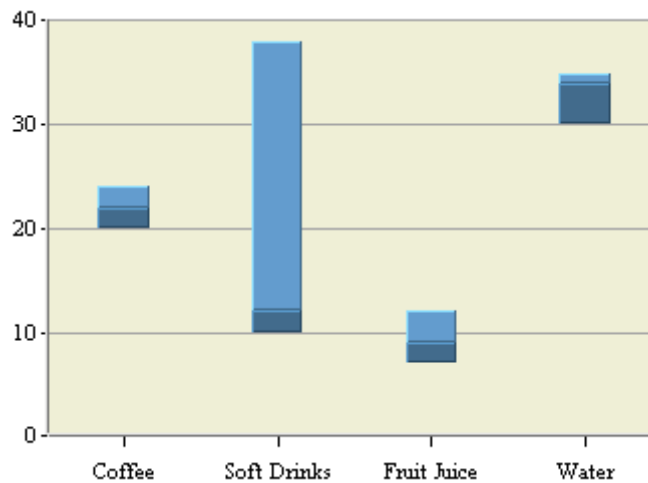
Data Mapping

DataSeries :	None	<input type="checkbox"/> Multi Selection
Category (X) :	Drink	<input type="checkbox"/> Multi Selection
Sum by :	Day	<input type="checkbox"/> Multi Selection
Value (Y) :	volume	
2nd Value :	None	
Combo:	Line	
2nd Series :	drink	<input type="checkbox"/> Multi Selection

Mapping Options for Stack Area Charts

Data Mapping for stack area charts is almost exactly the same as for stack column charts (covered in Section 4.1.3.6.1 - Data Mapping). However, two-dimensional stack charts cannot have a data series, and the combo options are *Line* and *Overlay*.

4.1.3.11. High-Low Charts



High-Low Chart

A high-low chart is also a derivative of a column chart, only that instead of one value column it uses two columns for high and low bounds of values. The data for the chart must contain at least three columns: category, high, and low. The category column can contain values of any type, while the high and low columns must be numeric.

Another option for a high-low chart is a data column called the *close* column. If a close column is also used, then the close value will lie somewhere between the high and low values. The portion of the bar between the low point and the close point is rendered in a darker form of the same color as the portion between the close point and the high point. As for many other types of charts, a high-low chart may include a data series column.

4.1.3.11.1. Data Mapping

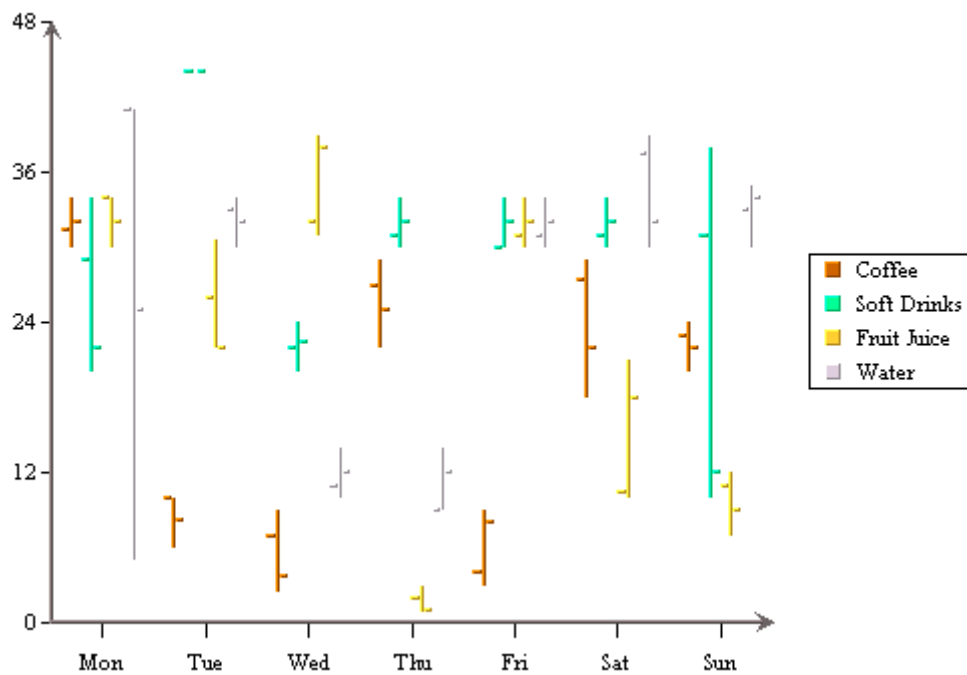
Data Mapping

DataSeries :	None	<input type="checkbox"/> Multi Selection
Category (X) :	Drink	<input type="checkbox"/> Multi Selection
High :	high	
Low :	low	
Close :	close	
2nd Value :	None	
Combo:	Line	
2nd Series :	Day	<input type="checkbox"/> Multi Selection

Mapping Options for High-Low Charts

Data mapping for high-low charts is similar to column charts. You can select series and category columns in the same manner. The difference for high-low charts is that you need to specify at least two value columns - a high and a low value. These are the two points that are plotted for each category. A third value called *Close* can also be specified. Combo options for high-low charts are *Line*, *Column*, and *Overlay*.

4.1.3.12. HLCO Charts



HLCO Chart

A HLCO (High Low Close Open) chart is similar to the high-low chart described above, except that it also contains an *open* column. It is useful in presenting data that fluctuates over discrete periods of time, such as stock prices or inter-day temperatures. The HLCO chart can also be displayed in a Candlestick format for both 2D and 3D charts. This feature is described in Section 4.1.4 - The Chart Designer Interface.

4.1.3.12.1. Data Mapping

Data Mapping

DataSeries :	Drink	<input type="checkbox"/> Multi Selection
Category (X) :	Day	<input type="checkbox"/> Multi Selection
High :	high	
Low :	low	
Open :	open	
Close :	close	
2nd Value :	None	
Combo:	Line	
2nd Series :	Day	<input type="checkbox"/> Multi Selection

Mapping Options for HLCO Charts

Data mapping for HLCO charts is similar to that for High-Low charts. The only difference is that instead of specifying two different value columns, you must specify four different columns - *High*, *Low*, *Open* and *Close*. Combo options are *Line*, *Column* and *Overlay*.

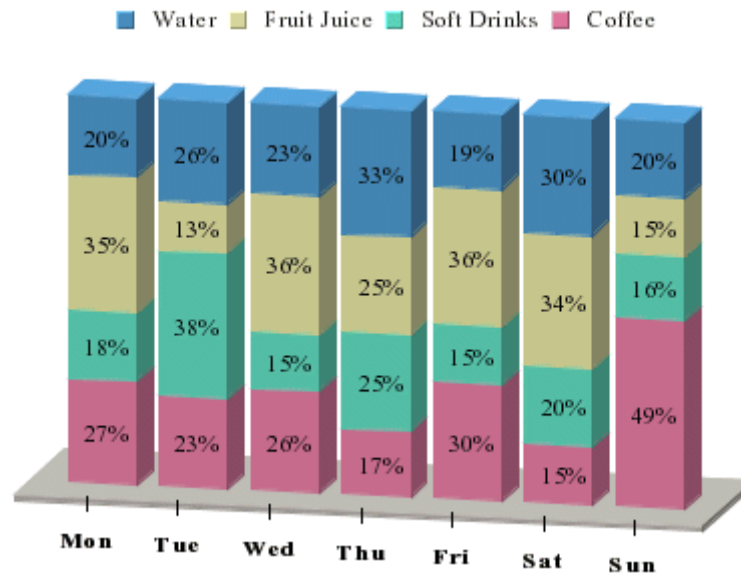
A common kind of chart is a one that plots both stock price and trading volume in the same chart. This can be accomplished using an HLCO-column combination in ChartDesigner.



HLCO-Column Combination Chart

This example plots stock price and volume data over a three month period.

4.1.3.13. Percentage Column Charts



Percentage Column Chart

A percentage column chart may be viewed as a derivative of pie and column charts together. Each column in the chart corresponds to one pie. A percentage column chart has a category column corresponding to the X-axis value. A sum-by column represents the category in this case and the value column is the same as that of a pie chart.

4.1.3.13.1. Data Mapping

Data Mapping

DataSeries : None ☐ Multi Selection

Category (X) : Day ☐ Multi Selection

Sum by : Drink ☐ Multi Selection

Value (Y) : value

2nd Value : None

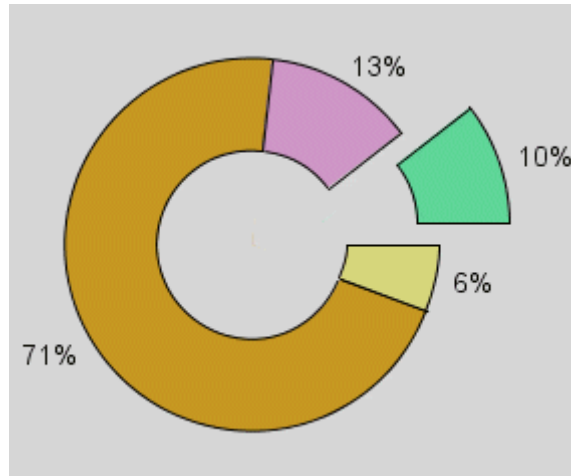
Combo: Line

2nd Series : Day ☐ Multi Selection

Mapping Options for Percentage Column Charts

Data mapping for percentage column charts is the same as for stack column charts (covered in Section 4.1.3.6.1 - Data Mapping). The only difference is that the sum-by components are plotted as a percentage of the Value column instead of discrete values. Combo options for percentage column charts are *Line* and *Overlay*.

4.1.3.14. Doughnut Charts



Doughnut Chart

A doughnut chart is the same as a pie chart, except for the “hole” in the center.

Just as for a pie chart, if a data series is selected, a separate doughnut will be drawn for each category element and each doughnut will be comprised of the series elements. When a series is present, all the separate doughnuts are drawn on a single plot area and resized in proportion with the plot. You have an option to either stack the different doughnuts or draw them in a line.

4.1.3.14.1. Data Mapping

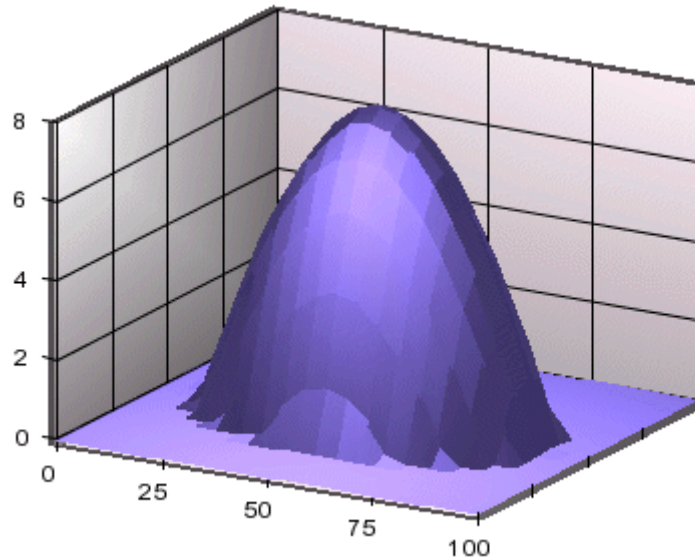
Data Mapping

DataSeries :	Day	<input type="checkbox"/> Multi Selection
Category (X) :	Drink	<input type="checkbox"/> Multi Selection
Value (Y) :	volume	

Mapping Options for Doughnut Charts

Data mapping for doughnut charts is exactly the same as for pie charts (discussed in section Section 4.1.3.8.1 - Data Mapping).

4.1.3.15. Surface Charts



Surface Chart

A 3D surface chart is a scientific/business chart, which uses three sets of numeric data values to form a smooth surface in a three dimensional space. For each data point, two of the three values represent the coordinates on the horizontal plane and the third value is used to determine the vertical position of a data point. Input data can be in a form of a rectangular matrix (as shown below) or arranged in three columns where each column represents an axis. In the following sample data, the top row (column header) and the left column (row header) represent the coordinate values of the axes that form the plane on which the surface will be drawn. When the chart is viewed from above the plane, you can see a grid formed by joining every four adjacent points together. A given set of data cannot plot a surface chart if this grid cannot be drawn.

Sample surface chart data:

	0	20	40	60	70	80	100
20	0	0	0	0	0	0	0
30	0	10	10	10	10	10	10
40	0	10	25	25	25	2	10
80	0	10	25	30	30	30	25
90	0	10	25	30	30	30	25
100	0	10	10	25	25	25	10

Note: Sample data for creating a surface chart can be found in `surface.dat` file, which is located in `<EDAB install dir>/help/examples/DataSources/text/` directory.

The values of the column header and the row header are not required to be equally separated because a surface chart can properly scale the horizontal distances. This dataset can create a distorted inverted cone. A surface chart can be very impressive when animation is turned on.

A surface chart uses a set of X, Y, and Z coordinates to plot a 3D chart. The vertical axis is called Y-axis and each Y value is represented by a function of X and Z, $f(X, Z)$. Data on XY plane (horizontal plane) simply looks like a square matrix. Surface charts do not support data series and cannot be converted to a 2D chart.

4.1.3.15.1. Data Mapping

Data Mapping

X axis : RowLabel

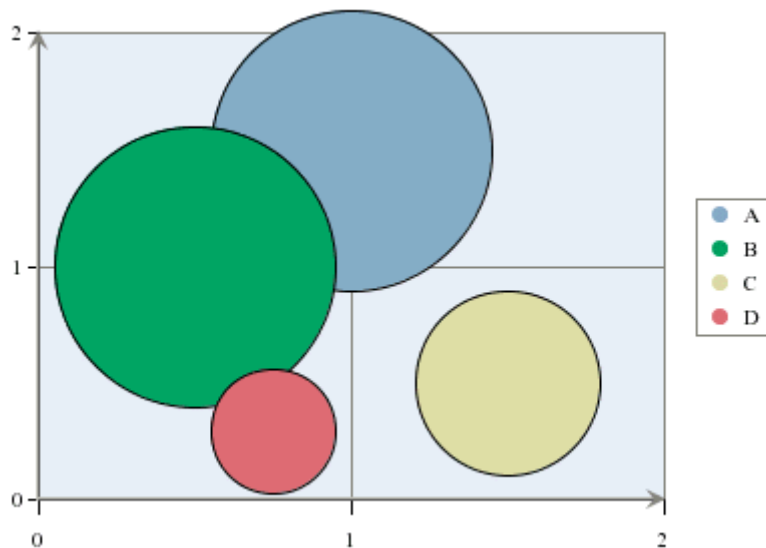
Y axis : Value

Z axis : ColumnLabel

Mapping Options for Surface Charts

The data mapping for a surface chart is similar to a three-dimensional scatter chart; the X-axis, Y-axis, and Z-axis values determine the X, Y, and Z coordinates of a point respectively. However, unlike a scatter chart, surface charts do not support data series and cannot be converted to a two-dimensional chart.

4.1.3.16. Bubble Charts



Bubble Chart

A bubble chart is used to represent data wherein the size of the bubble also provides information. A data point is represented by an XY coordinate and a third point, which is the radius of a circle or bubble.

This chart is available in a two-dimensional form only. For more information about the bubble charts and their display options, please see Section 4.1.4.9.1 - Bubble Charts.

4.1.3.16.1. Data Mapping

The data mapping for bubble charts is similar to three-dimensional scatter charts; however, instead of plotting a Z-axis position, the third value determines the size of the bubble (specifically the radius).

Data Mapping

DataSeries : Time

X axis : X

Y axis : Y

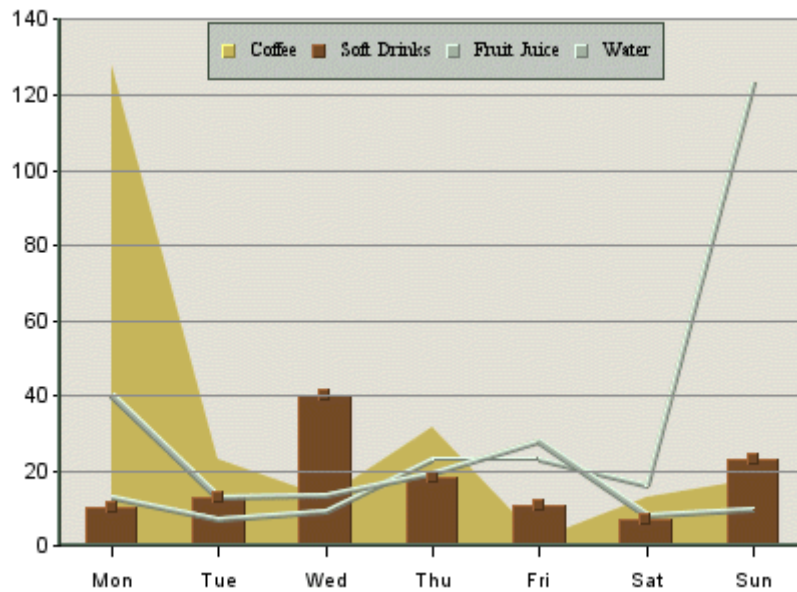
bubble size: BubbleSize

☐ Multi Selection

Mapping Options for Bubble Charts

Note that similarly to a scatter chart, the columns have to be numeric in order to be mapped successfully.

4.1.3.17. Overlay Charts



Overlay Chart

ChartDesigner supports a special chart type called overlay chart, which allows you to super-impose more than two charts with a common category axis. A different chart can be used to represent each element of a data series. This allows for more freedom while creating the chart and also allows more information to be represented. The chart types supported in an overlay chart are column, area, and line charts. Only two-dimensional charts can be included in an overlay chart.

4.1.3.17.1. Data Mapping

Data Mapping

DataSeries : Drink ☐ Multi Selection

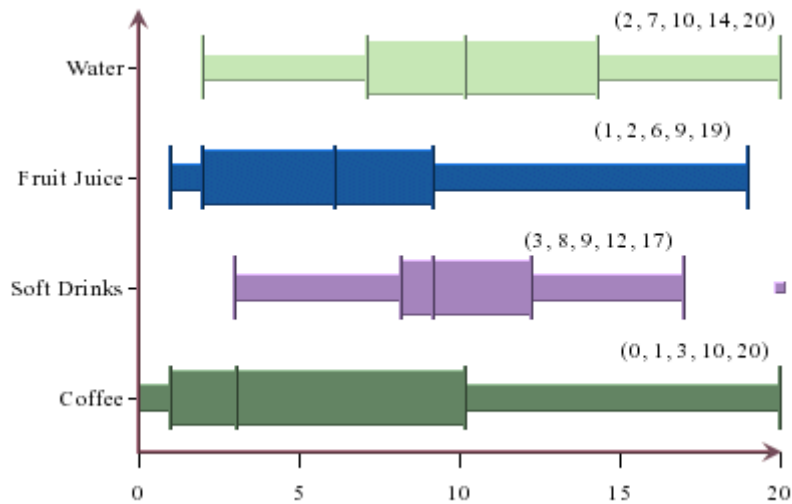
Category (X) : Day ☐ Multi Selection

Value (Y) : volume

Mapping Options for Overlay Charts

The data mapping for an overlay chart is similar to a column chart (covered in Section 4.1.3.2.1 - Data Mapping), except that the overlay chart plots each element in the data series as a separate chart. Please note that the *2nd Value*, *2nd Series* and *Combo* options do not apply to overlay charts. Overlay charts do not support secondary values. However, the different series elements can be plotted on different axes. For more about plotting options for overlay charts, see Section 4.1.4.9.3 - Overlay Charts.

4.1.3.18. Box Charts

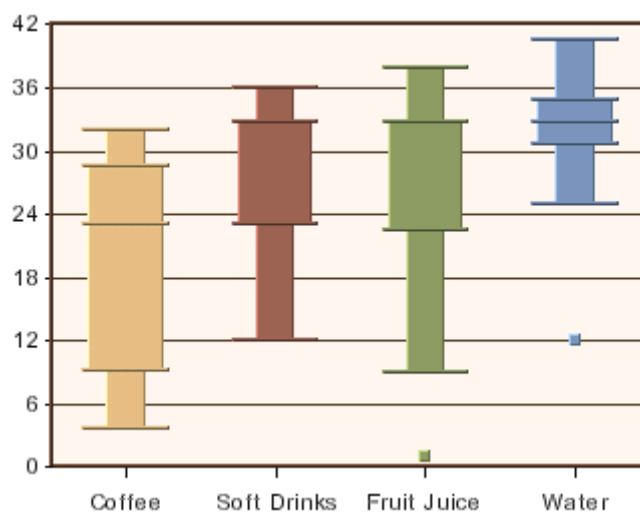


Box Chart

ChartDesigner provides a statistical chart type, the Box Chart, also known as Box and Whiskers Chart. Basically, the box chart provides a quick visual realization of summary statistics. A histogram shows the distributions of observed values so you can identify the peak(s), minimum, maximum values and clustering of data. A box chart, on the other hand, displays a summary of data distribution in quarterly percentiles (Minimum data point, 25th percentile, median, 75th percentile, and Maximum data point).

The middle line in the box represents the median. The other lines in each direction represent the 25th percentile increment (decrement). The minimum and maximum points are the observed minimum and maximum which are not outliers. The lines that extend from the edge of the box to the minimum/maximum are sometimes called whiskers. Hence, the name box and whisker chart. Outliers are any values that are 1.5 times (or more) the length of the box, that being the difference between the 75th and 25th percentiles. Outliers are calculated and shown as dots away from the box. The other nice feature about box plot is that you can stack up the boxes in one chart to compare summaries of different data sets.

ChartDesigner allows box charts to be displayed vertically in addition to the default horizontal setting. To use this feature in Chart Designer, create a box chart and then select Format → Chart Options, and select *Vertical*.



Vertical Box Chart

This chart is available in two-dimensional form only.

4.1.3.18.1. Data Mapping

Data Mapping

Category (X) : Drink ☐ Multi Selection

Value (Y) : quantity

2nd Value : None

Mapping Options for Box Charts

For box charts the mapping is as follows:

Category (X): Allows you to choose a data column whose distinct values determine the various categories.

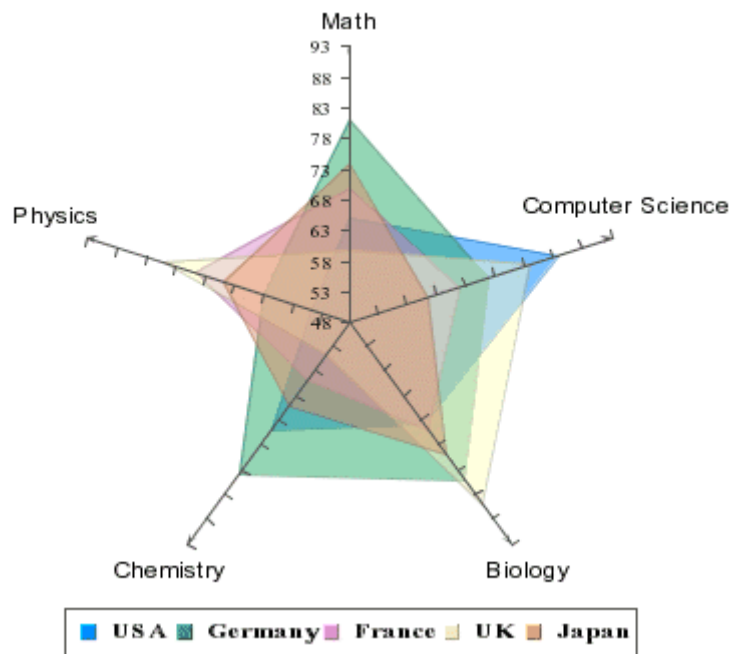
Value (Y): Allows you to choose a data column to provide values for each category. These values are used to calculate the minimum data point, the 25th percentile, the median, the 75th percentile, and the maximum data point.

2nd value: Add a second value to create a combination chart.

Please note that the *2nd Series* and *Combo* options are not available for box charts. This is because the only combination available with box charts is a line.

The data mapping also allows you to transpose the data (in other words: to select several columns for a single category). To learn more about data transposition, please see Section 4.1.3.1.1 - Data Transposition.

4.1.3.19. Radar Charts



Radar Chart

Radar charts are useful when you need to compare performance/measurement results, statistics, etc from different sources. For example, we can compare median test scores of children in the industrial nations on subjects such as math, computer science, and biology. A radar chart has multiple axes along which data can be plotted. Each axis is a category. The data is shown as points on the axis. The points belonging to one data series can be either joined or the enclosed area filled. A point close to the center on any axis indicates a low value while a point near the end represents a high value. In some cases, low values may be more desirable than high values, e.g. price/earning, price/sales, price/book of stocks.

This chart is available in a two-dimensional form only.

4.1.3.19.1. Data Mapping

Mapping Options for Radar Charts

For radar charts the mapping is as follows:

Data Series: Allows you to choose a data column whose distinct values will determine the number of data series in the chart. Each element in a data series is drawn using the same color along each axis.

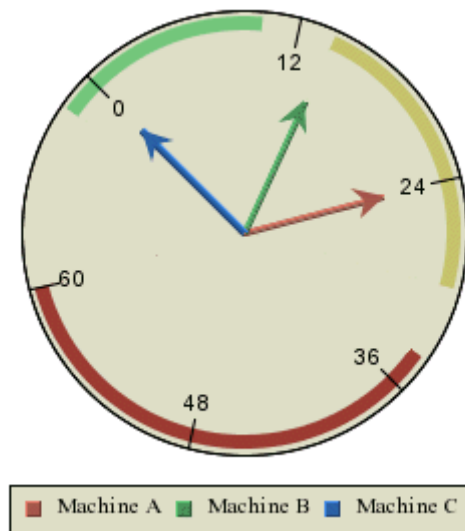
Category (X): Allows you to choose a data column whose distinct values determine the categories. The number of unique elements in this column determines the number of axes in the radar chart.

Value (Y): Allows you to choose a data column to provide the numeric value to plot against the category.

Please note that the *2nd Value*, *2nd Series*, and *Combo* options do not apply for radar charts. Radar charts do not support secondary values.

The data mapping also allows you to transpose the data (in other words: to select several columns for a single category). To learn more about data transposition, please see Section 4.1.3.1.1 - Data Transposition.

4.1.3.20. Dial Charts

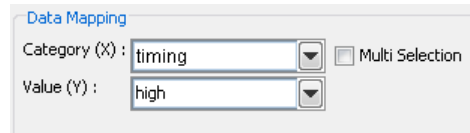


Dial Chart

Dial Charts are used to build circular charts (such as temperature gauges, speedometers, and clocks), create dashboards, or balanced scorecard applications. Here the data is represented as a “hand” on a “dial”. You can either use the whole dial (for instance for a clock) or just part of it (a semi- circular chart such as a gasoline gauge). Dial Charts support pop-up labels, drill-down, and user interactions such as rotating the hands or sectors of a dial.

This chart type is available in a two-dimensional form only.

4.1.3.20.1. Data Mapping

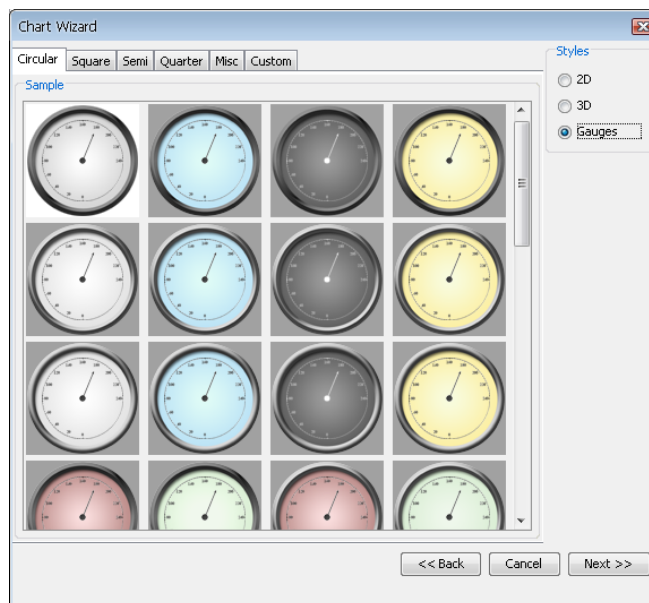


Mapping Options for Dial Charts

The data mapping of a dial chart is similar to a pie chart (detailed in Section 4.1.3.8.1 - Data Mapping). However, instead of pie wedges, the categories become dial hands. Also, dial charts do not support data series or secondary values.

4.1.3.20.2. Gauges

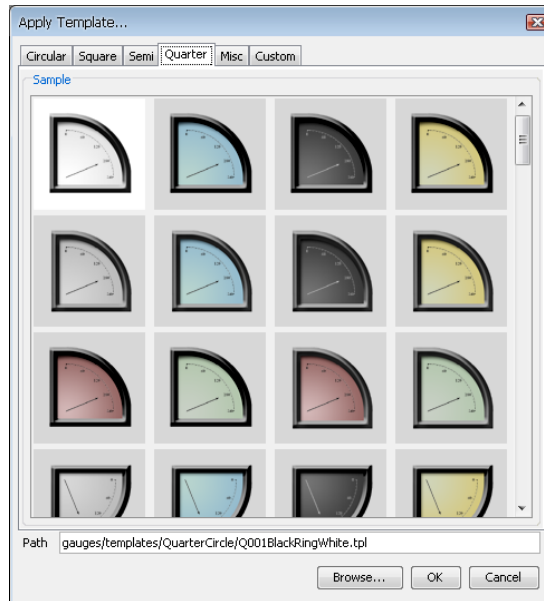
Gauges are special types of dial charts containing a plot background image and/or plot foreground image. To create a new gauge, select the *Gauges* radio button on the *Chart Type Selection* dialog.



Selecting a Gauge Template

You can also create your own gauge by creating a template, saving the tpl file in the <EDAB Install>/gauges/templates/Custom/ folder (please note that the Custom folder is not created automatically by the EDAB installer so you will have to create it manually in your favorite file manager). To add this template to the Custom tab, you will also need to provide two images, a screenshot of the template placed in <EDAB Install>/gauges/screenshots/selected/Custom/, and a dimmer version of the same template placed in <EDAB Install>/gauges/screenshots/unselected/Custom/. An easy way to create the screenshot is to export the template to gif for the regular screenshot and resize it to 100px by 100px. Then change the background to a darker color, export and resize again for the dimmer version.

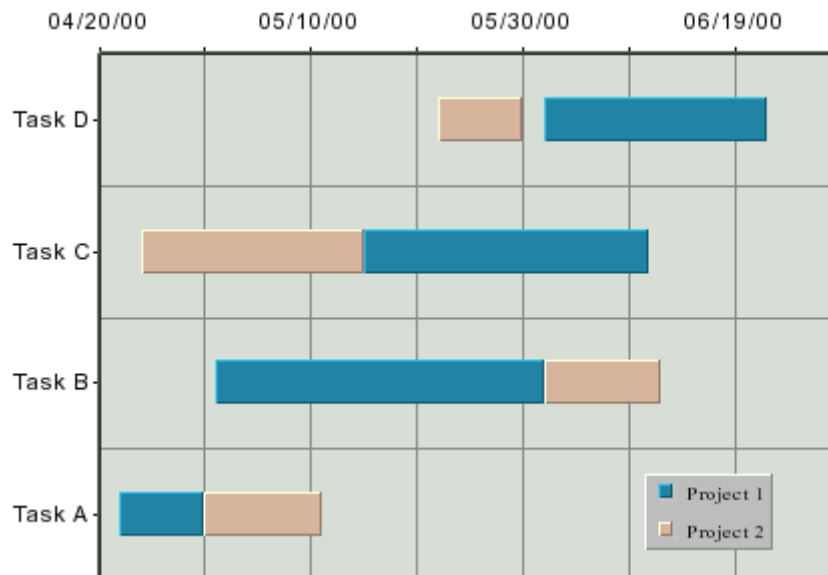
It is also possible to apply a gauge template onto an existing dial chart. Selecting apply template when the current chart is a dial chart will display the gauge tabs just like when creating a new chart.



Applying a Gauge Template

For more information regarding the dial background and foreground images, see Section 4.1.4.9.2 - Dial Charts.

4.1.3.21. Gantt Charts



Gantt Chart

A Gantt or time chart is used to represent data where a time factor is being measured. This is often used in project or time management situations. A Gantt chart resembles a bar chart with the category axis on the Y-axis and the value axis on the X-axis. The value axis uses date, time, or timestamp data (numeric values can also be used). Each data point has a start and end time associated with it.

This chart type is available in a two-dimensional form only.

4.1.3.21.1. Data Mapping

The data mapping for Gantt charts is similar to high-low charts. However, instead of selecting a column for the high and low values, you select the start and end values.

Data Mapping

DataSeries : ☐ Multi Selection

Category (X) : ☐ Multi Selection

Start :

End :

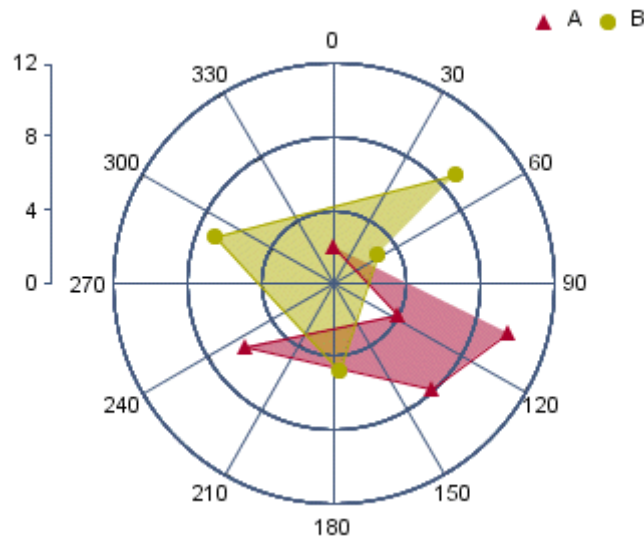
Mapping Options for Gantt Charts

The mapping is as follows:

- Data Series:** Allows you to choose a data column whose distinct values will determine the number of data series in the chart.
- Category (X):** Allows you to choose a data column whose distinct values will determine the categories.
- Start:** Allows you to choose a data column whose distinct values represent the starting time interval for the category.
- End:** Allows you to choose a data column whose distinct values represent the ending time interval for the category.

The data mapping also allows you to transpose the data (in other words: to select several columns for a single category). To learn more about data transposition, please see Section 4.1.3.1.1 - Data Transposition.

4.1.3.22. Polar Charts



Polar Chart

Like a two-dimensional scatter chart, a polar chart plots points on a plane. However, instead of rectangular coordinates, a polar chart plots points using polar coordinates (r, θ) , where r represents the distance from the center of the circular plot and θ represents the angle of a ray emanating from the center of the plot and passing through the point. Like scatter charts, the data for polar chart coordinates must be numeric. The θ values can be supplied in degrees or radians. A third data series column can be used to separate the data points into groups.

4.1.3.22.1. Data Mapping

Data Mapping

DataSeries : Wavelength ☐ Multi Selection

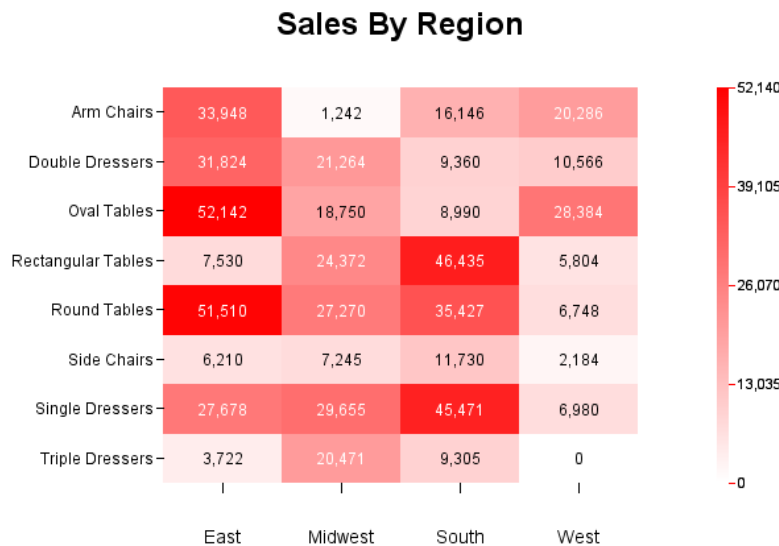
Angle : Angle

Radius : Distance

Mapping Options for Polar Charts

The data mapping for polar charts is similar to scatter charts. The angle and radius options allow you to select the columns whose values will make up the polar coordinates. These must both be numeric. The data series box allows you to choose a data column whose distinct values will determine the number of data series in the chart. Each element in a data series is drawn using the same set of drawing attributes, e.g., colors and markers.

4.1.3.23. Heatmap Charts

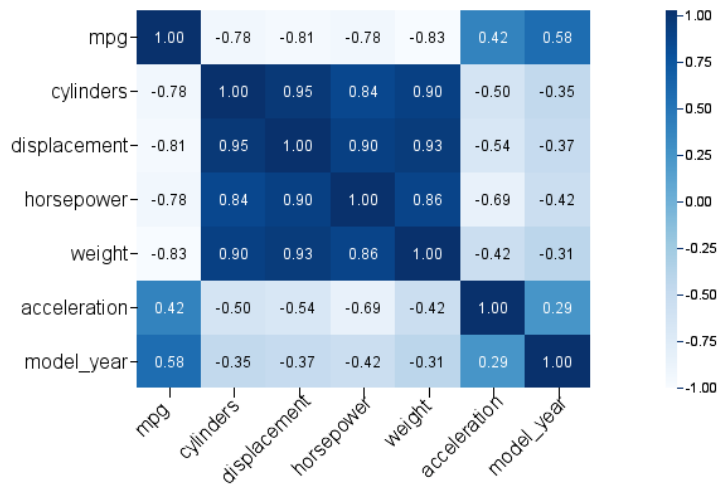


Heatmap Chart

A heatmap is a data visualization tool that uses color to represent the values of a matrix. This method is particularly effective for displaying large amounts of data in a way that is easy to understand at a glance. The colors in a heatmap typically range from light (white) to dark (e.g., red) along the color gradient, or from cool (e.g., blue) to warm (e.g., red), with each color representing a different value. This visual representation allows for quick identification of patterns, trends, and outliers within the data. A simple example that shows the revenue of certain product category in certain region is presented above. In this case, region is the X axis, product category is the Y axis and the plot is (X,Y, value).

In addition to just plotting (X,Y, value) straight from the input data, there is a correlation option that you can apply to the data. As such, you can easily visualize the correlation coefficients between pairs of columns from the data source, i.e. each entry in the matrix is the correlation coefficient between the corresponding columns on the X axis and the Y axis. Below is an example. In this example, the columns along the X axis are the same as those along the Y axis. But they do not have to be the same.

Correlation heatmap for automobile data



Correlation Heatmap Chart



Note

Heatmap chart is available in a two-dimensional form only.

4.1.3.23.1. Data Mapping

Data mapping of heatmap without applying the correlation function is straightforward.

Data Mapping

X axis : REGION

Y axis : CATEGORYNAME

value: EXPRESSION

☐ Correlation Coefficient

Heatmap data mapping

Data mapping of heatmap with correlation function applied.

Data Mapping

X axis : mpg
cylinders
displacement
horsepower
weight
acceleration

Y axis : mpg
cylinders
displacement
horsepower
weight
acceleration

☒ Correlation Coefficient


Correlation heatmap data mapping


Note that the “Correlation Coefficient” checkbox is checked, and the X-axis and Y-axis show all the available numeric columns in the data source. You can select multiple columns in each of the axes.

In our earlier example, we selected all the columns for the X axis and all the columns for the Y axis. But you can choose different columns for each axis.

4.1.3.24. Changing Data Mapping or Data Source

Once you have finished setting the mapping options you will be taken to the main Chart Designer interface. Once you are in the Chart Designer you can return to the Data Mapping window by selecting Data → Modify Data

Mapping, or by clicking the  *Change data mapping* icon. This will return you to the data mapping screen, allowing you to adjust the mapping for the chart.

You can also change a data source directly from the Chart Designer by selecting Data → Modify Data Source, or by clicking the  *Modify data source* icon on the toolbar. This will open the Data Source Manager.

4.1.4. The Chart Designer Interface

Once you have completed all the steps in the Wizard, the main Chart Designer interface will appear. Here you can customize and modify the appearance of the chart by changing properties and adding new elements.

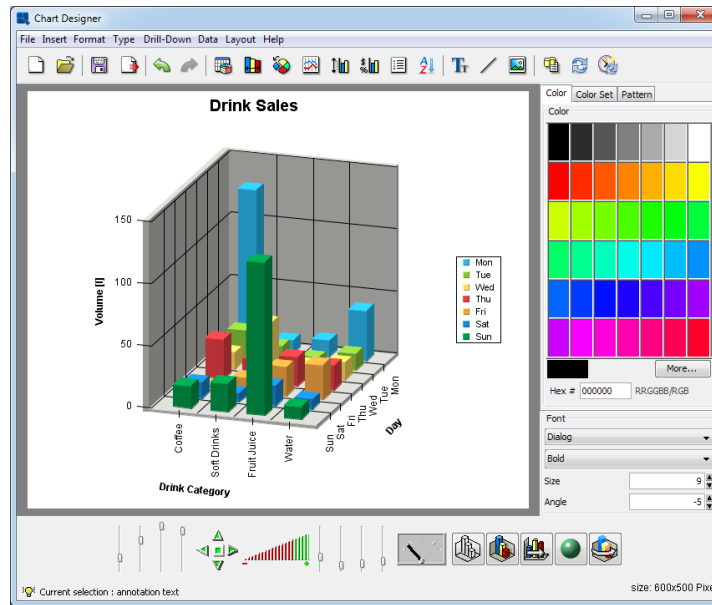


Chart Designer Interface

4.1.4.1. The Designer Menus

Most of Chart Designer's functions can be controlled from the menu bar at the top of the designer window. This section provides a brief overview of the available options. All of the features here are discussed later in this chapter.

4.1.4.1.1. File Menu

This menu performs basic file operations such as opening, closing, and saving files. Note that saving and opening options are not available when you use the report data as the data source for an embedded chart. For more information about file options, please see Section 4.1.6 - Saving & Exporting Charts.

- New:** This will return you to the Data Source Manager to begin creating a new chart. If you have not saved your current chart, you will be prompted to do so.
- Open:** This allows you to open a saved chart definition. Files that can be loaded by Chart Designer include .cht, .tpl, and .xml chart definition files.
- Close:** This closes the current chart.
- Apply Template:** This option allows you to apply a template to the current chart. You can apply any chart template in either .tpl or .xml format. For more information about working with chart templates, please see Section 4.1.6.1.1 - Working with Templates.
- Save:** This saves the current chart.

Save As:	This allows you to save the current chart. You can save the chart as a .cht , .pac or .tpl (a chart, a complete packed chart or a template file). The Create XML check-box allows you to create an XML chart definition file.
Export:	This allows you to export the chart to a number of static image formats. The chart data can also be exported as XML file.
Exit:	This closes the Chart Designer with the possibility to save the current file.

4.1.4.1.2. Insert Menu:

This menu allows you to add various elements to a chart.

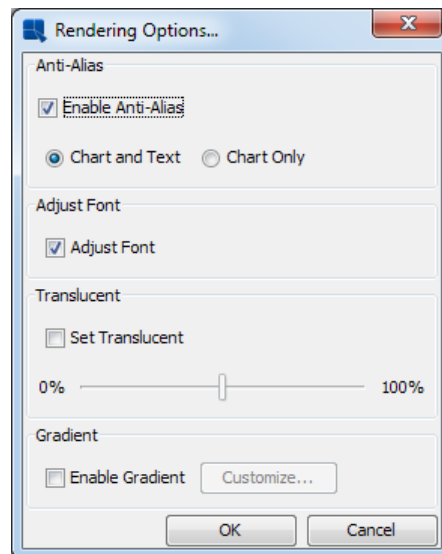
Titles:	This option allows you to automatically add titles to the chart. A main title can be specified as well as titles for each of the axes. Unlike annotation text, titles will size and position automatically with the chart.
Text:	This allows you to add text or annotation to a chart. Text can be added anywhere on the chart and can have a number of different formatting properties. Variables can be added to the text for run-time substitution. For more on adding text to charts, see Section 4.1.4.6.1 - Adding Text
Background:	This allows you to select an image to use as the chart background. Background images can be tiled, centered, or stretched to fit the entire canvas.
Dial Foreground:	This option is only available for dial charts. This allows you to select an image to use as the dial chart foreground. The image can be stretched.
Dial Background:	This option is only available for dial charts. This allows you to select an image to use as the dial chart background. The image can be stretched.
Link:	This allows you to add a hyperlink to a data point or a collection of data points in a chart. To use hyperlinks in a chart, you will need to export a map file along with the image.
Line:	This allows you to add an arbitrary floating line to a chart. These lines can be placed anywhere and can be used to draw enclosed shapes as well. Floating lines are often used as pointers and can be generated with arrowheads.
TrendLine:	This allows you to add a trend line to the chart. ChartDesigner allows you to draw many different types of trend lines including: linear, a polynomial of any degree, a power, exponential, logarithmic, a moving average, exponential moving average, triangular moving average, cubic B-spline, and a normal distribution curve.
Horz/Vert Line:	This allows you to add a fixed horizontal or vertical line to a chart. You have the choice of either adding a constant line (one that draws at a fixed value on the X or Y-axis) or a control line (draws lines based on a certain value range either average, minimum, maximum, or multiples of standard deviation).
Control Area:	This allows you to draw a fixed area (either on the plot area of a 2D chart or on the face of a dial chart) to compare against the data values of the chart.

4.1.4.1.3. Format Menu

This menu allows you to edit and modify the properties of many different chart components.

Undo:	Cancels the last operation performed in the designer and reverts back to the previous state. The designer will remember the last 10 actions made.
Redo:	Reverses the action of the undo command. For example, if you change the font color from black to red, you can undo this command to change it back to black, and then redo this command to have it change to red again.

Data Properties:	This option allows you to control several options for how the data is displayed. You can control the thickness of columns/bars, how null data is displayed, whether to draw data top labels or not, as well as enable and select a color for negative top labels.
Histogram Options:	This allows you to specify if you want to draw the chart as a histogram and display additional options to specify the frequency count.
Aggregation Options:	This allows you to specify whether to aggregate the data before drawing the chart and display additional options to specify type of aggregation.
Zoom Options	This allows you to enable/disable and set options for time based zooming. This option only applies if you have date/time data mapped to the category axis.
3D Display Options:	This allows you to set several options that control the display of 3D charts. You can specify an inline series for 3D column (or similar) charts, as well as specify rendering approximation for 3D scatter and surface charts. (This improves performance for charts with a lot of data points.)
Rendering Options:	This allows you to specify various rendering options for the chart for a better presentation.



Rendering Options Dialog

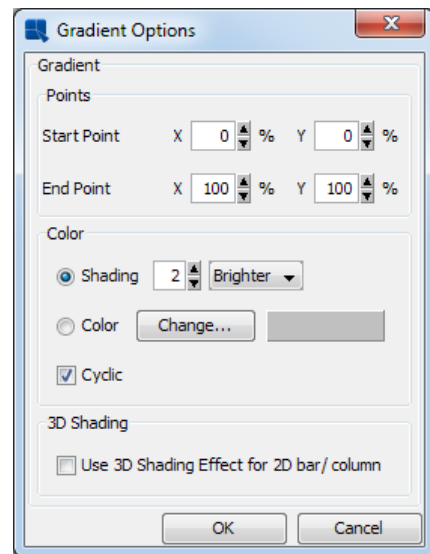
Among the options available are:

Enable Anti-Alias:	This allows you to specify anti-aliasing for the chart. Anti-aliasing will smooth text and lines in the chart, creating a smoother appearance. This feature can be applied to either the entire file or just to the chart, which will leave the text unaffected.
Adjust Font:	This allows you to specify the font size for text in charts to be relative to the screen resolution. This feature allows for more precise conversions of charts to various export formats and between installations.
Translucent	This allows you to specify whether the columns/bars/areas should be translucent. This

allows any columns/bars/areas “hidden” behind to show through. You can also specify the opacity by adjusting the slider, where 0% is completely opaque and 100% is completely transparent. This option is available for all 3D charts and for 2D Area, Radar and Gantt charts with data series.

Gradient:

This allows you to enable gradients. The gradient can be applied either across the entire canvas or to chart data points only. You can change the gradient options by clicking on Customize to show the following dialog:



Gradient Options Dialog

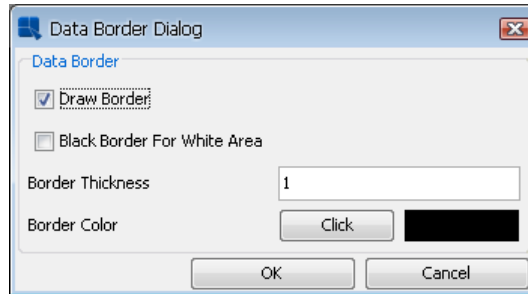
This dialog allows you to specify the start and end points for the gradient (based on the x-y plane), the gradient color scheme, and whether the gradient should be cyclic. The start and end points are represented as percentages with (0,0) being the top left corner of the canvas and (100, 100) being the bottom right corner of the canvas. You can also specify whether the gradient change is shading (becoming darker or brighter) or a different color. Lastly, you can specify the gradient to be cyclic i.e., toggle between the two opposites in the gradient. Note that the start and end points represent the first segment if the gradient is cyclic.

You can also enable 3D shading for certain 2D charts (Column, Bar, Stack Column, Stack Bar, 100% Column and Overlay) by selecting the *Use 3D Shading Effect for 2D bar/column* option (please note that this option is available for 2D bar/column charts only).

Font Mapping:

This allows you to map system (true type) font files for the PDF export. For more information about this feature, please see Section 4.1.6.2.1 - PDF Font Mapping

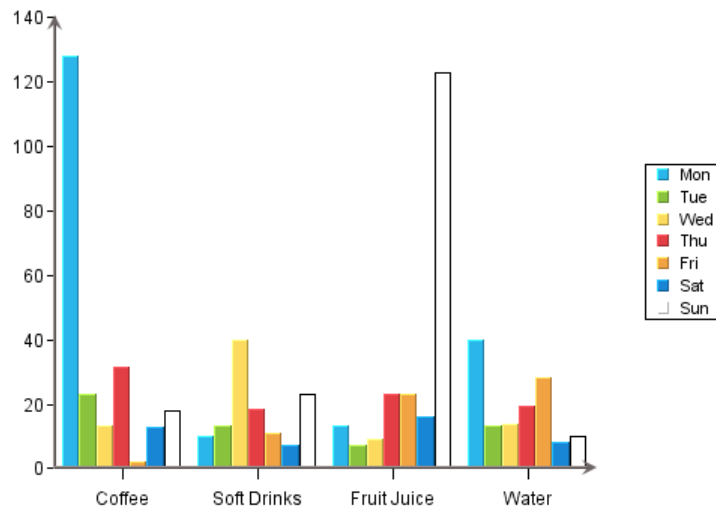
Chart Options:	This brings up some specific options for the chart type that you are using. Options vary depending on chart type.
Axis Scale:	This allows you to adjust the scale for any value axes. Automatic (best fit) scaling is used by default.
Axis Elements:	This allows you to modify the appearance of the axes and axis labels. Options here include axis thickness, grid lines, label steps, and data formatting.
Canvas:	This allows you to adjust the background canvas size. You can also specify whether to use scroll bars when the canvas size exceeds the view port.
Data Border:	Add a border to data elements (columns, bars, etc).



Data Border Dialog

If you select the *Draw border* option, the *Border thickness*, and the *Border color* fields will be enabled allowing you to configure the border properties.

The *Black border for white area* option can be selected even when the *Draw border* is disabled. If you do so, a simple black border will be added only to white data elements.



Column chart with the “Black border for white area” enabled



Note

This option is not available for Surface, Scatter, Line, Bubble, Box, Radar, Dial, and Polar charts.

Legend:	This allows you to modify the display properties of the chart legend.
----------------	---

Lighting Model:	This allows you to modify some of the lighting options for three-dimensional charts. You can modify both the light ambient color and the intensity.
Line and Point:	This allows you to modify the display properties of any lines in a chart. For two-dimensional charts you can choose to display lines and points for any dataset as well as customize their appearance. You can also use this menu item to modify the display properties for any trend, floating, or horizontal/vertical lines.
Plot Area:	This option allows you to customize appearance of the area bounded by the X and Y axes for two-dimensional charts.
No Data To Plot Message:	This option allows you to set a message that appears if there is insufficient data to plot the chart. By default, the “No Data To Plot” message appears.
Flash Hintbox Customization:	This option allows you to specify the font properties as well as the border and background color for the hint box in the flash export.
NULL Data Properties:	This option allows you to show any category axis point that contains Null data and replace it with a different string. By default, any Null data category point is skipped.
Table:	This allows you to add and configure a table displaying the chart data.
Text Properties:	This option allows you to set a resize ratio for any text in the chart. From this option you can also specify to use Java 2D rotate text. This option gives rotated text a cleaner appearance. You can also specify any text replacement options.
Viewer Options:	This menu item allows you to specify some configuration options for the Chart Viewer JNLP (More about Applets in JNLP: Section 1.3.4 - Run Applets in WebStart with JNLP file) when the chart is viewed. You can control what options are available in the Viewer pop-up menu. These options can also be controlled through HTML parameters.

4.1.4.1.4. Type Menu

This menu allows you to change the current chart and its dimension. You can switch between 2D and 3D for chart types that support representations in each dimension. You can also change chart types. Note that as you switch between chart types, some formatting information will be lost. Also, you cannot switch between Gantt charts and other chart types.

4.1.4.1.5. Drill-Down Menu

This menu contains options allowing you to add and navigate drill-down layers in a chart. The drill-down features are explained in Section 4.1.5 - Chart Drill-Down. Note that drill-down is only available (functional) when charts are deployed independently from reports.

Add:	This allows you to add a layer of data drill-down.
Remove This:	This removes the current level of data drill-down.
Remove All:	This removes all levels of data drill-down.
Previous	This navigates to the previous layer of data drill-down.
Next:	This navigates to the next layer of data drill-down.
Go To Top Level:	This navigates to the top-level chart for data drill-down.
Dynamic:	This allows you to enable dynamic data for drill-down.
Parameter Drill-Down:	This brings up the parameter drill-down navigation window allowing you to edit, add, and remove layers of parameter drill-down.

4.1.4.1.6. Data Menu

This menu contains options that allows you to refresh, re-order or completely change the chart data.

Modify Data Mapping:	This will bring back the Data Mapping Window, allowing you to change the data mapping for the chart.
Modify Data Source:	This will take you back to the Data Source Manager, allowing you to select a new data source for the chart.
Modify Database:	If the chart uses an independent data source that is from a database, this feature, like in Report Designer, will allow you to modify the database connection that the chart use. For more information about this feature, please see Section 3.1.3.5 - Editing Database Connections.
Modify Query:	If the chart uses an independent data source that is from a database, this feature, like in Report Designer, will allow you to modify the query that retrieves the chart data. For more information about this feature, please see Section 3.1.3.4 - Editing Queries.
Query Parameters:	This will allow you to re-initialize query parameters and change parameter values for the current chart. This option is only available if the chart uses a parameterized query as the data source.
Ordering:	This option allows you to re-order the data points in a chart. You can arbitrarily change the order of any of the category elements or you can sort the categories by value.
Refresh:	This will update the current chart with the latest data. The original data source must be available for this option to work.
Schedule Refresh:	This allows you to set a schedule to refresh the data. This option is for deploying charts in the Chart Viewer JNLP (More about Applets in JNLP: Section 1.3.4 - Run Applets in WebStart with JNLP file). You can set a periodic refresh interval so that the chart will update itself with the latest data.
View Table:	This option will bring up a window containing the data table from which the current chart is generated. The table will initially display only the first 20 records. Clicking on the <i>Show All Records</i> checkbox will display all of the records.
View Chart Data:	Rather than viewing the entire data table you can just view the data points plotted in the current chart.
View Data Source Info:	If the chart contains an independent data source, this option will bring up a dialog containing information about the data source that was used to create the chart. The data source type and location are displayed.
Go Back:	This will go back to the original chart if you have traversed a link.

4.1.4.1.7. Help Menu

This menu allows you to view a version of the program and to open a documentation.





















About:	This shows you information about the version of the program.
Contents:	This opens the EspressoDashboard User's Guide.

4.1.4.1.8. Layout Menu

The options in this menu allow you to toggle various elements in the Chart Designer interface. From this menu you can turn on and off the font and color panel, the Chart Designer toolbar, and the navigation panel for 3D charts.

4.1.4.2. The Designer Toolbar

The toolbar at the top of the Chart Designer window offers easy access to the Chart Designer's most commonly used features and functions. The buttons perform the following functions:

-  Start a new chart
-  Open an existing chart
-  Save the current chart
-  Export the current chart
-  Undo the last change
-  Redo the last undone change
-  Change data mapping for the current chart
-  Modify data display properties
-  Modify chart-specific options
-  Modify line and point attributes
-  Change axis scale
-  Modify axis elements/display properties
-  Modify legend display
-  Change data ordering
-  Insert annotation text
-  Insert floating line
-  Add/change background image
-  Modify/change chart data source
-  Refresh chart data
-  Schedule periodic data refresh

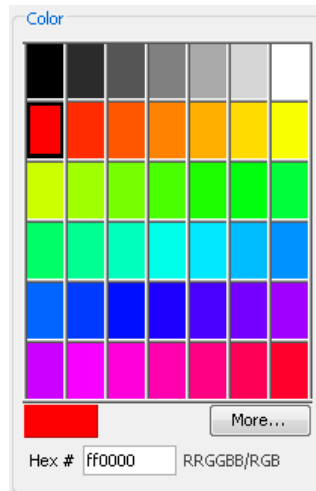
4.1.4.3. Color, Color set, Pattern, and Font Panels

The color, color set, and font panels on the right-hand side of the Chart Designer window allow you to modify the color of any chart object, as well as modify the font size and style for any text/labels in the chart. You can choose not to display these panels by toggling the Layout → Show font/color panel option.

For heatmap charts, you can use the *Color Panel* or *Color Set Panel* to set the colormap. The *Select Axis* option in the *Color Set Panel* is not applicable to heatmap charts. Additionally, the *Pattern Panel* does not respond to heatmap charts.

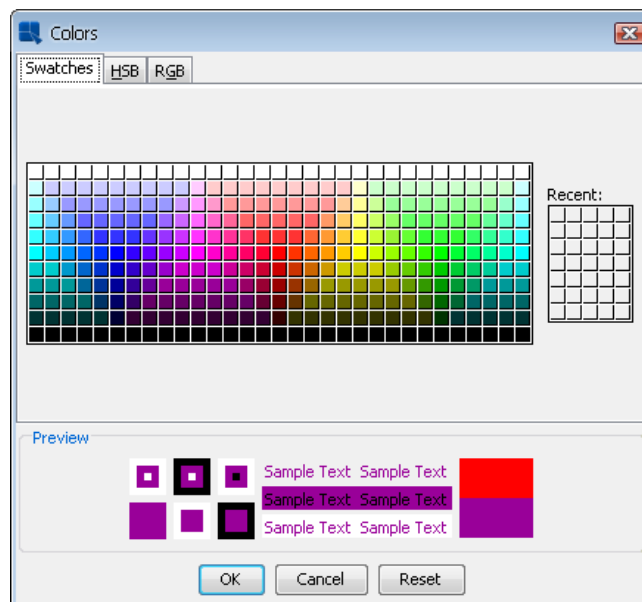
4.1.4.3.1. Color Panel

You can use the color panel to change the color of any element in the chart. To modify an element's color, first, click on it. The status bar at the bottom of the Chart Designer window will indicate which element has been currently selected. After you have selected the object, click on one of the panels in the color panel and the color of the object will change to reflect the color you selected.



Color Panel

To create a custom color for the object, first select it, then click the *More* button. This will bring up a new dialog allowing you to pick a color from a larger palette or to create a new color.

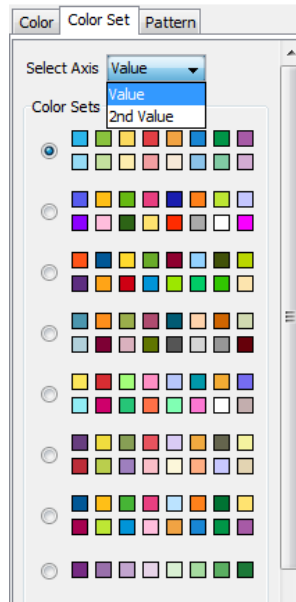


Additional Colors Dialog

From this dialog you can pick a new color from swatches or configure a custom color using HSB values or RGB values.

4.1.4.3.2. Color Set Panel

The color set panel allows you to choose a color scheme for your chart. It contains predefined color sets that can be applied to chart data points on the value or the second value axis.




Color Set Panel

In order to change a color set for the value or the second value axis, first select the *Color Set* tab. Then select the axis (*Value* or *2nd Value*) from the *Select Axis* select box and click the appropriate color set radio button. After that chart data points will get colors from the selected color set. Note that the *Select Axis* select box is only visible when the chart has the second value axis. By default, the value axis uses the first color set while the second value axis uses the seventh.

Please note that if you change a color of a data point manually, it will automatically unselect the selected color set. This is because of the color set that no longer corresponds to colors of data points in the chart. It is also important to note that if chart data points have more different colors than there are colors in the color set, it will automatically use colors from the beginning of the next color set, and so on. If there is no next color set available, it will use the first one instead.

4.1.4.3.2.1. Save Colors for Categories feature

Data points colors are closely related to the *Save Colors for Categories* feature that is available in the *Data Properties* dialog. The dialog can be opened by clicking the  *Change data properties* icon on the toolbar, or by using *Format* → *Data Properties*. (Please note that this feature is not available for Stack charts and it is disabled if the chart has *Single Color for All Categories*.) If the feature is turned on, colors of chart data points are assigned to names of categories (or series for charts with series). If such categories (or series) then appear in the chart, it will automatically use their assigned colors. If the feature is turned off (default), colors are assigned by data points order (i.e. the first data point will get the first available color from the color set, the second will get the second color, etc.). *Save Colors for Categories* setting is automatically saved in the .pac file. The following example shows scenarios with the feature on and off:

Assume that a chart has three categories (“A”, “B”, and “C”) with colors taken from the following color set (blue, green, yellow, red).

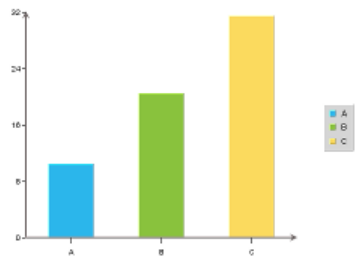


Image 1 - Example Chart

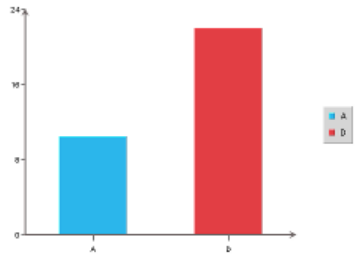


Image 2 - “Save Colors for Categories” Feature On

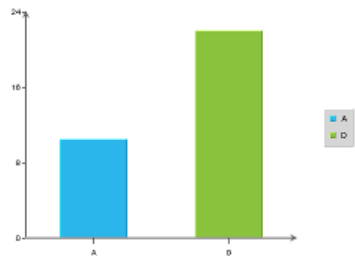


Image 3 - “Save Colors for Categories” Feature Off

Save Colors for Categories feature on

We saved the chart with the “Save Colors for Categories” feature enabled (see Image 1), so the following categories and colors were saved to the list of saved categories:

category A ... blue color

category B ... green color

category C ... yellow color

Now if you open the chart and data changes (e.g. there will be only categories A and D in the data - see Image 2), categories will have the following colors:

category A ... blue color (category A has blue color, because the category name A is present in the list of saved categories)

category D ... red color (category D has the next available color in the color set, because the category name D isn't present in the list of saved categories. In this case, the category will have red color, because blue, green, and yellow colors are already assigned to categories A, B, and C.

For this scenario, the color set will not be selected under the *Color Set* tab because colors of data points do not correspond to the color set.

Save Colors for Categories feature off

Here is the same situation, but assume that the chart has been saved with the “Save Colors for Categories” feature off (see Image 3). The list of saved categories will be empty this time.

After opening the chart, the categories will have the following colors:

category A ... blue color (category A has blue color, as it is the first available color from the color set)

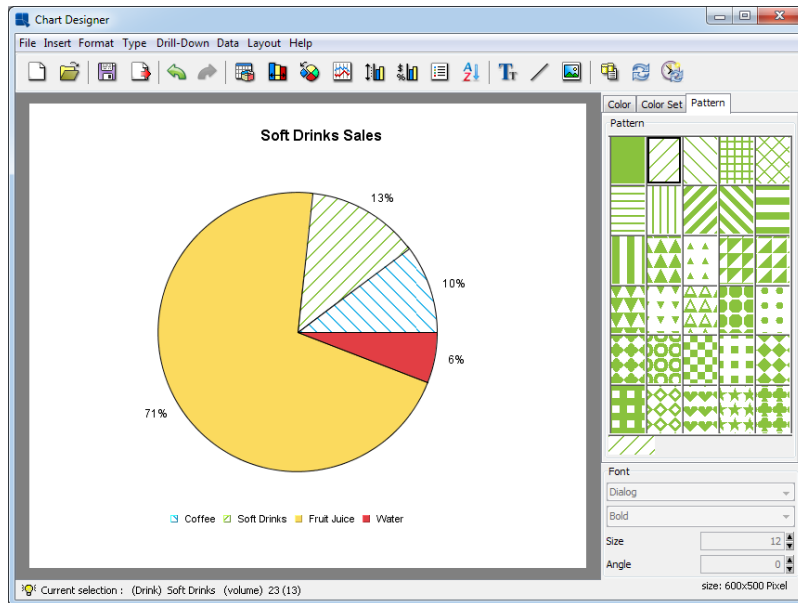
category D ... green color (category D has the second color from the color set, because the first is already assigned to the category A)

For this scenario, the color set will be selected under the *Color Set* tab because colors of data points correspond to the color set.

4.1.4.3.3. Pattern Panel

Unlike color and font panels, the pattern panel can only be applied to data points. There is a predefined pattern palette available for you to use. Similarly to how the color panel is used, you will need to first pick the data point you would like to change and then pick any pattern from the pattern palette. The pattern will be applied to the data point directly.

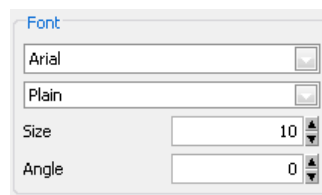
If a pattern has already been defined for a data point, the user can still change the color by selecting the color panel tab and picking a different color from the color palette. The color on the pattern will be changed to the new color immediately. The patterns shown on the pattern palette will change to the new color as well.



Pattern Example

4.1.4.3.4. Font Panel

You can use the font panel to modify the font, the font style, font size, and the font angle for labels, titles, or other text in a chart. To modify the font, you must first select the text object whose font you would like to change by clicking on it. The status bar at the bottom of the Chart Designer window will indicate which is the currently selected object.



Font Panel

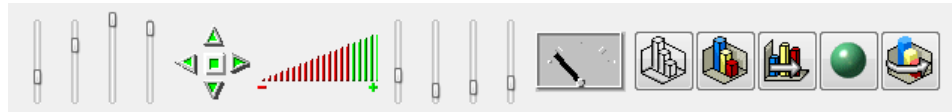
The first drop-down box allows you to select the font that you would like to use. The second drop-down box allows you to select the font style either plain, bold, italic, or bold and italic. The third box allows you to select the font size. The last box allows you to specify the angle of the text.

Certain groups of text (i.e. axis labels or data top labels) will all have the same properties. Hence, if you select one of text and modify the font properties it will apply to all of them.

ChartDesigner allows you to use the Java graphics libraries to give your text a cleaner appearance. For regular text you can use the chart anti-alias feature by selecting Format → Rendering Options. For rotated text (i.e. text not at 0 degrees), you can use the Java 2D rotate text feature by selecting Format → Text Properties. Note that these methods require Java 1.2 or higher.

4.1.4.4. The Navigation Panel

The navigation panel provides options for controlling a number of the properties specific to three-dimensional charts. It does not appear for two-dimensional charts and it can be hidden for three-dimensional charts by toggling the Layout → Show Navigation Panel option.



Navigation Panel

There are six controls and five buttons in the Navigation panel. Starting from the left, the six controls are step size, light position for X, Y and Z axes, navigation, zoom, scale and navigation speed. The buttons on the right control wire frame/solid mode, on/off border drawing, on/off inline series (for columnar and bar charts with series), on/off Gouraud shading, and on/off animation.

Step size:

This slide bar allows you to set the incremental amount used by the navigation control to rotate or move the 3D chart. The higher the position of the horizontal bar, the larger is the increment for each step.

Light position for X, Y, and Z:

User-defined light position is useful to control the position of the light and therefore the direction from which it will shine on each axis.

Navigation:

This control allows you to rotate or translate the chart. The center button is a toggle switch. When it is in a depressed state (denoted by the red color of the button), clicking on any of the four triangular buttons moves the chart in the direction indicated by that button. When the center button is in an elevated state, clicking on any of the four triangular buttons rotates the chart in the direction indicated by that button. The speed of navigation may be controlled by the Navigation Speed control.

Zoom:

This control allows you to effectively move the chart closer to or farther away from the viewer. To operate, you point the mouse to the control and click on the left mouse button. Then, while holding down the button, you drag the mouse to the left or right. When you drag the mouse to the right, the red area is extended to the right, and the chart will appear to be closer to you (zoom in to blow up the chart). When you drag the mouse to the left, the red area will move to the left, and the chart will appear to move farther away from you (zoom out to shrink the chart).

Scale:

This is a set of four slide bars. The first three bars allow you to change the scale factors for the X, Y, and Z axes respectively. The fourth bar determines the thickness of a three-dimensional column, bar, line, or pie (depending upon the chart type). The higher the position of the horizontal bar, the larger the value. For two-dimensional charts, you can adjust the bar width for bar, column, stack bar, stack column, high-low, and HLCO charts by selecting Format → Data Properties.

Animation speed:

This allows you to set the speed for chart navigation. This control is useful with the Navigation control and the Animation On/Off switch. It determines

how fast the chart moves or rotates. To operate, click and drag the indicator needle to the desired position. Moving the indicator needle to the right will speed up the translation or rotation and moving the needle to the left will slow it down.

In addition, the five buttons do the following:



Wire frame/ solid mode On/Off: This toggle switch allows you to view the three-dimensional chart as a wire frame when the switch is on or as a solid object when the switch is off.



Border Drawing On/Off: This toggle switch will draw a black outline on each edge of the chart when the switch is on.



Inline Series On/Off: This toggle switch will draw the series column in the same XY plane. This option is only available for columnar and bar charts with a data series and it does not appear on the navigation panel if the chart type is not applicable.



Gouraud Shading On/Off: Gouraud shading is a sophisticated and very realistic shading feature for three-dimensional charts. When the switch is on it will begin rendering the chart to shade each individual surface.



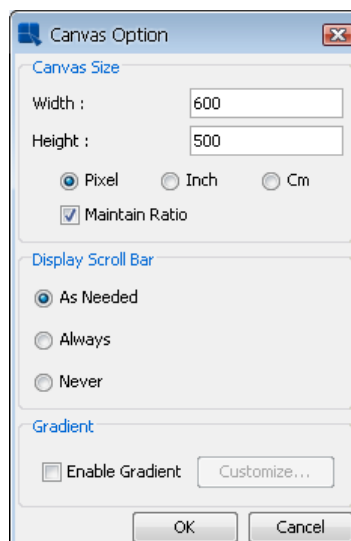
Animation On/Off: This toggle switch allows you to start/stop the animation of a three-dimensional chart. The speed of animation may be set using the Navigation Speed control. During animation, all panel controls except the animation speed control are disabled.

4.1.4.5. The Viewport

The viewport compromises the central portion of the Chart Designer window. Within the viewport you can select, move, and size all of the various chart elements on the canvas.

4.1.4.5.1. The Chart Canvas

The chart canvas is the background on which all of the chart elements are drawn. Its dimensions are the size of the finished chart. You can modify the size of the chart canvas by selection Format → Canvas. This will bring up a dialog prompting you to specify the new canvas dimensions.



Canvas Formatting Dialog

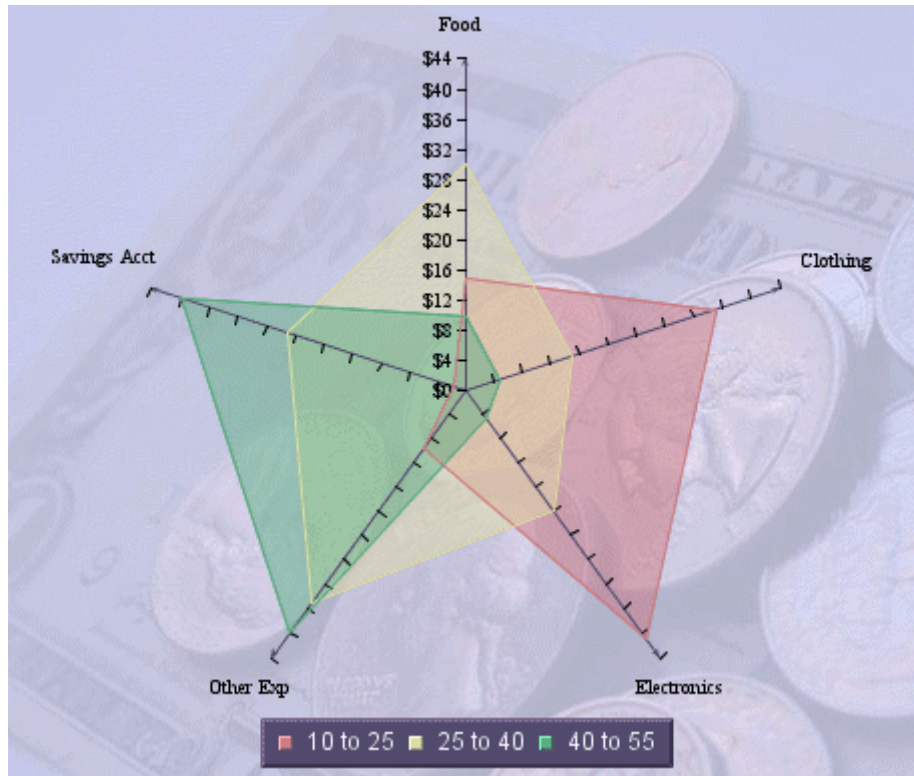
You can specify the canvas size in pixels, inches, or centimeters. From this dialog you can also specify when to use scroll bars in the viewport. By default the viewport will display scroll bars when the canvas is larger than the window. When the canvas is smaller than the viewport window, a dark gray area will appear around it.

If you're creating an embedded chart in a report, the canvas will resize to match the space that you have defined for the chart in the Report Designer. For example, if you set the space in the report to be 3 inches by 4 inches, the next time you edit the chart, the canvas will automatically size to 3 inches by 4 inches.


On this dialog, you can also set up gradient background for the canvas. The gradient settings are the same as in the *Rendering options* described in the Section 4.1.4.1.3 - Format Menu.

4.1.4.5.1.1. Background Images

You can add an image as the background of the chart instead of having a plain or a colored canvas.



Radar Chart with Background Image

You can add a background image by selecting Insert → Background or by clicking the  *Background* button on the toolbar. This will bring up a dialog allowing you to specify the background image for the chart.



Add Background Image Dialog

To insert a new image, select the *Enable Background Image* option. If you want to remove an existing background image and use simple background color instead, unselect this option.

There are two ways of inserting background images: either locate the image on the hard drive or retrieve it from an URL.

To locate an image on the hard drive, click on the *Browse* button.

To retrieve image from an URL, enter the URL in the *Image URL* text field. After that, click on the *Refresh Preview* button to verify the URL. If the image from the URL appears in the *Preview* section, the URL is correct.

If you add a background image and save the chart as TPL or CHT, the image itself is not stored with the chart. Only the path or URL is saved. If you move a TPL or CHT chart, you need to be sure that it can still access the image along the path specified. If you save the chart as PAC, the background image will be stored in the PAC file along with the chart.

4.1.4.5.2. Moving and Sizing Chart Elements

You can select any element in the chart by clicking on it. The status bar at the bottom of the Chart Designer window will indicate which object has been currently selected. Clicking and dragging on an object will move it around the chart canvas. Note that some objects like axis or data top labels move in tandem, while other objects like legends move independently.

You can move the entire chart plot by clicking in the plot area and dragging the mouse. This will move the entire chart around the canvas. To resize a chart click the plot area and 8 points will appear in the corners and on the sides of the plot area. Drag any of those points using the mouse to enlarge or shrink the plot area. You can also resize three-dimensional charts by using the zoom control in the navigation panel.

4.1.4.6. Adding Chart Elements

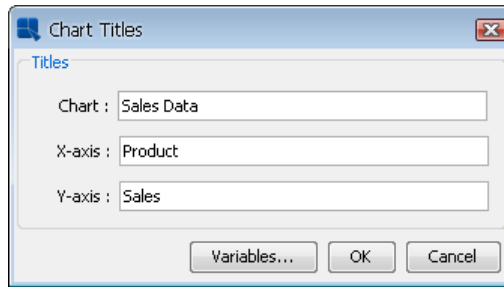
In addition to the default chart elements, ChartDesigner provides a number of additional elements that you can add to a chart.

4.1.4.6.1. Adding Text

There are two ways to add text to a chart: as titles or as plain text elements.


Adding Titles:

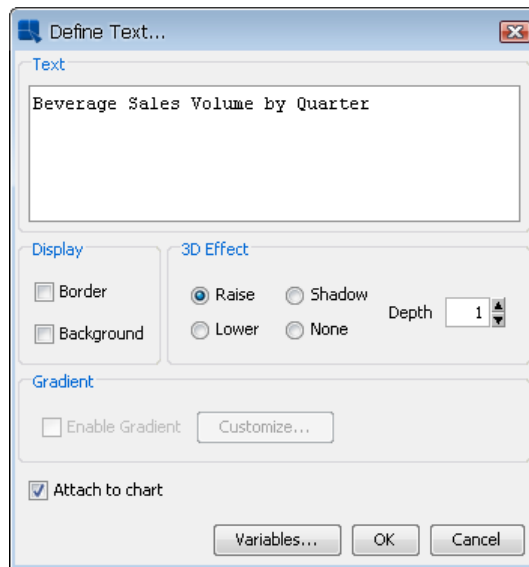
To add titles to a chart, select Insert → Titles. This will bring up a dialog prompting you to enter titles for the chart.

*Add Titles Dialog*

The dialog allows you to specify a main title for the chart and a title for each of the axes. Pie and dial charts, which do not have axes, prompt you for the chart title only. After you have finished specifying the titles click *OK* and they will be added to the chart. Titles are placed and sized automatically. However, they can be moved and the fonts can be changed.

Adding Text:

To add individual text fields to the chart, select *Insert* → *Text* or click the  *Add Text* button on the toolbar. This will bring up a dialog prompting you to add text to the chart.

*Add Text Dialog*

From this dialog you can specify the text and configure some display options. You can specify whether to draw a border around the text or around a background. You can also specify what effect you want to apply to the background.

On this dialog, you can also set up gradient background for the text label. The gradient settings are the same as in the *Rendering options* described in the Section 4.1.4.1.3 - *Format Menu*.


Once you have finished specifying the text, click *OK*. You will then be able to place the text in the chart. A small rectangle will follow your pointer around the chart canvas. Click the mouse where you would like to place the text.

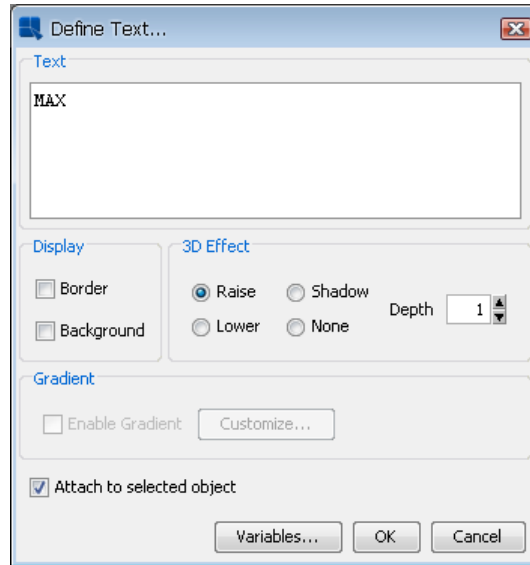
Annotation Text:

ChartDesigner also supports annotation. Annotation allows you to attach labels or text fields to a particular element, such as a line or the chart plot. For example, you can insert a control line showing the maximum value to a chart and attach a text label called **MAX** to this control line. Each time the maximum value changes, the label will adjust its

position along with the control line. For more information about the control lines, please see Section 4.1.4.6.2.3 - Fixed Horizontal/Vertical Lines.

You can specify text to be annotation in two ways. To attach the text to the chart plot, select the *Attach to Chart* option when adding text. To attach the text to a specific object

like a control line, first select the object, and then select Insert → Text or click the  *Add Text* button on the toolbar. The option for *Attach to selected object* will be automatically checked. Leave it checked and any text you add will be automatically attached to the object.

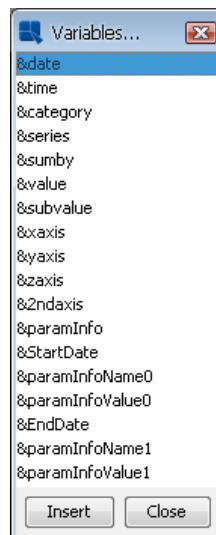


Add Text (Annotation) Dialog

4.1.4.6.1.1. TextVariables

ChartDesigner allows you to specify certain variables within text that allow for run time substitution based on certain values/objects in the chart. For example, if your chart uses a parameterized query as the data source, you could use the **¶mInfo** variable to display which parameter value(s) were selected at runtime.

Both the insert titles dialog and the add text dialog have a button marked *Variables* at the bottom. This will bring up a dialog with a list of variables you can use and it will allow you to select one to add to the title or to the text.



Variables Dialog

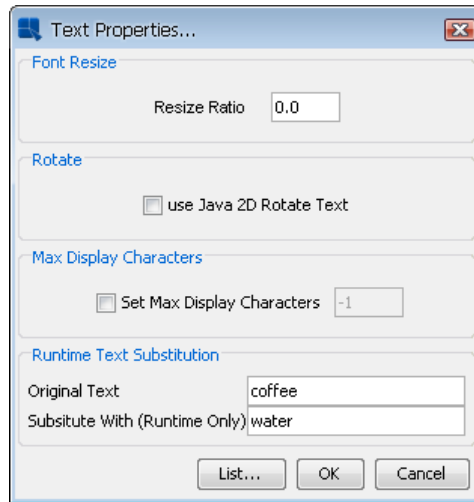
The following text variables are supported:

&drillInfo:	This displays which data point is being drilled on for drill-down charts. This variable does not work for parameter drill-down.
&paramInfo:	This displays the parameter value(s) that were selected. You can use this variable instead of &drillInfo for parameter drill-down charts.
&date:	This displays the date when the chart was last drawn/redrawn.
&time:	This displays the time when the charts were last drawn/redrawn.
&category:	This displays the name of the category column.
&series:	This displays the name of the data series column.
&sumby:	This displays the name of the sum-by column.
&value:	This displays the name of the value column
&subvalue:	This displays the name of the secondary value column
&xaxis:	This displays the name of the column that is mapped to the X-axis. This is for charts that map a value instead of a category to the X-axis like scatter or bubble charts.
&yaxis:	This displays the name of the column that is mapped to the Y-axis. This is for scatter, bubble, and surface charts.
&zaxis:	This displays the name of the column that is mapped to the Z-axis. This is for scatter, bubble, and surface charts.
&2ndaxis:	This displays the name of the column that is mapped to the 2nd-axis.
&paramInfoName<index>:	If the chart contains parameters, this displays the name of the parameter for the selected index.
&paramInfoValue<index>:	If the chart contains parameters, this displays the supplied parameter value for the selected index.
&<paramName>>:	If the chart contains a parameter with this name, this will display the value selected for that parameter.

4.1.4.6.1.2. Text Replacement

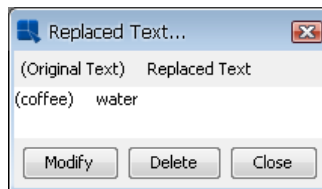
ChartDesigner allows you to overwrite a particular piece of text in a chart. This can be useful if the data source does not use particularly intuitive column names. Note that this feature will replace all instances of the text. For example, if you have a column chart without a series that displays a column name for both the X-axis label and the legend item, you cannot use text replacement to change only the label and not the legend item. The text replacement feature will also change only whole strings and not instances where there is a partial match.

To use text replacement, select Format → Text Properties. This will bring up a dialog allowing you to specify replaced text.

*Text Properties Dialog*

Please note that when making successive changes to the same text, the original text must be used. For example, assume you replaced the word “coffee” with “water”. Now if you want to change “water” to “soft drink” the text replacement should have the original text, which is “coffee” and then “soft drink” as the replacement. To remove any text replacement, simply replace the original string with itself. Hence, using the same example, you would replace “coffee” with “coffee”.

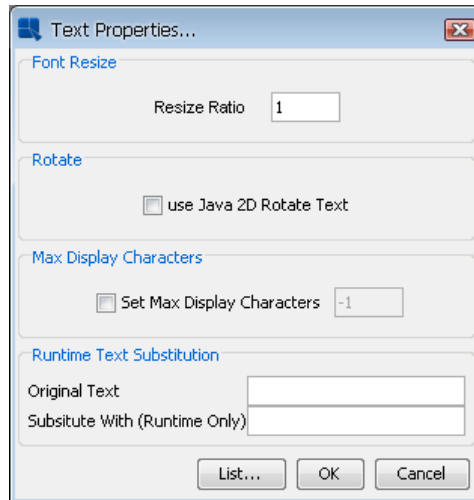
You can see a list of all the original text and the replacements by clicking on the *List* button. This will bring up a dialog listing all of the text replacement in the chart.

*Replaced Text List*

From this dialog you can select any of the replaced text and modify or undo the replacement by clicking the buttons at the bottom of the dialog.

4.1.4.6.1.3. Automatic Text Resizing

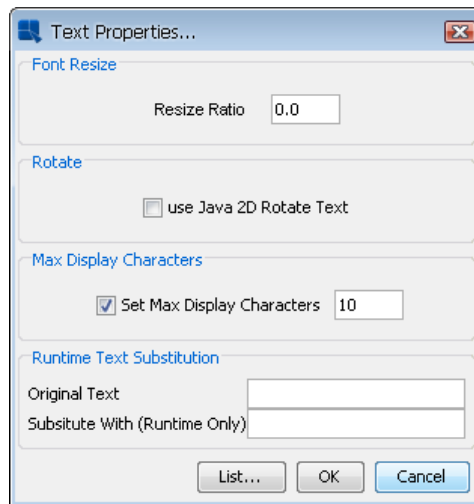
ChartDesigner has the ability to automatically adjust the font size of the text in a chart as it is resized. This is useful if you're using the same chart template to produce a number of charts in different sizes. You can specify a ratio for the font size to adjust based on changes in the chart canvas. To specify a resize ratio, select Format → Text Properties.

*Text Properties Dialog*

The ratio dictates the relative percent that the font should resize in regards to the canvas. For example, say you resize a chart from 500 x 500 pixels to 250 x 250. With a resize ratio of 1 then text with 12 point font would decrease to 6 point, decreasing by the same percent as the canvas. However, with a resize ratio of 0.5 the font would decrease half as much as the canvas so our 12 point font would only decrease to 9 point.

4.1.4.6.1.4. Text Cropping

Long labels or text in a chart can sometimes take up too much space in the chart plot, leaving little room for the actual chart. For situations like this, ChartDesigner offers a text cropping feature for chart text. Text that is longer than a user-supplied threshold will be truncated with "...". The hint box for the chart will show the whole label. To specify text cropping, select Format → Text Properties.

*Text Properties Dialog*

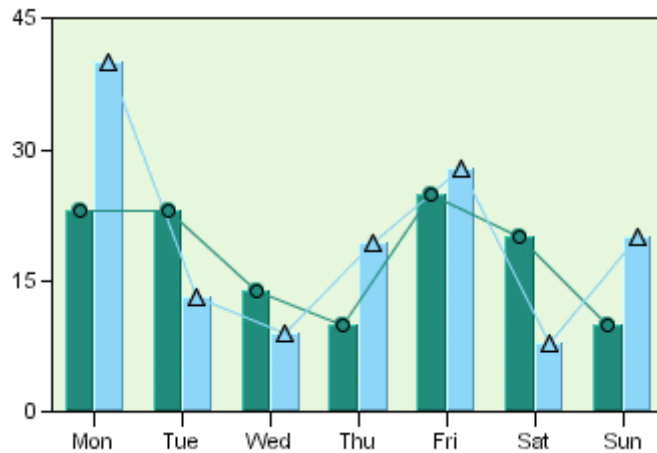
To enable text cropping, check the *Set Max Display Characters* option and specify the maximum character length in the dialog. Any text longer than the specified number of characters will be truncated.

4.1.4.6.2. Adding Lines


ChartDesigner allows you to add and format a number of different types of lines for charts.

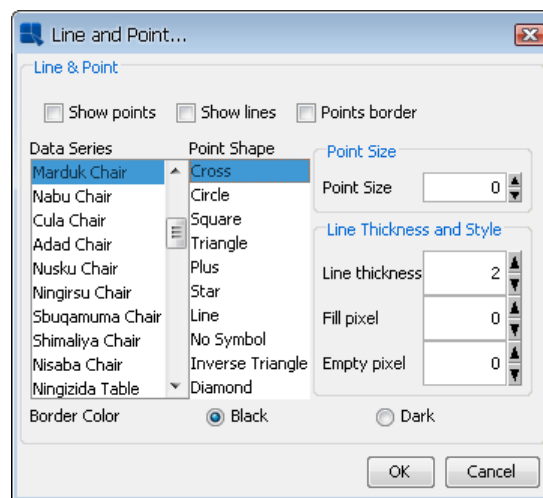
4.1.4.6.2.1. Line and Point Formatting

You can choose to display lines and points for all the data points in the chart for any two dimensional chart type. Note that some chart types already use this representation (i.e. line or scatter charts).



Column Chart with Lines and Points Defined

Line and point display is controlled by selecting Format → Line and Point, or click the  *Line and Point* button on the toolbar. This will bring up a dialog presenting several options.



Line and Point Dialog


The first three options allow you to specify whether you would like to show lines, points, and a points border for the chart. For radar charts you also have the option of showing areas. The remaining options allow you to customize the line and point displays for each element in the data series.

For each data series element, you can specify the point shape that you would like to use. You can also control the size of the points. The default point size is 0. You can specify point sizes of -1, -2, & -3 which represent sizes of 0.75, 0.5, and 0.25 respectively. At -3 (0.25), the point will be drawn as a dot regardless of the selected point shape.

For lines you can specify the line thickness, as well as customize a dash pattern. The dash pattern is created by specifying the number of filled pixels and the number of empty pixels (between 0 and 255). The line is then drawn by dividing into segments - the number of filled pixels followed by the number of empty pixels. Setting 0 for both will result in a solid line. Setting 255 for both will result in no line being drawn.

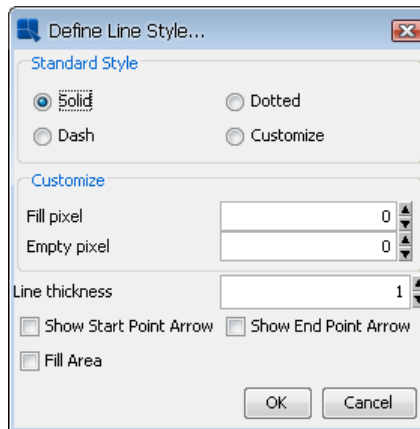
The last option allows you to change data point symbol border color to black or darker shade of symbol color.

4.1.4.6.2.2. Floating Lines

Floating lines are free-form lines that can be arbitrarily added to any place on the chart canvas. Often floating lines are used to point to a specific element in a chart. To add a floating line select Insert → Line, or click the  *Insert Line* button on the toolbar.

When you select this option, your mouse pointer will change to a cross. Click within the chart canvas where you would like the line to begin. Each additional click will add a point to the line, allowing you to add another segment. This way you can use floating lines to draw shapes as well. Once you have finished, right-click to stop drawing. The line will then be added.

Once a line has been placed on the canvas, it cannot be moved individually. It will move with the chart plot, like annotation text. To specify options for a floating line, first select it, and then select Format → Line and Point or click the *Line and Point* button on the toolbar. This will bring up a dialog allowing you to format various properties for the floating line.



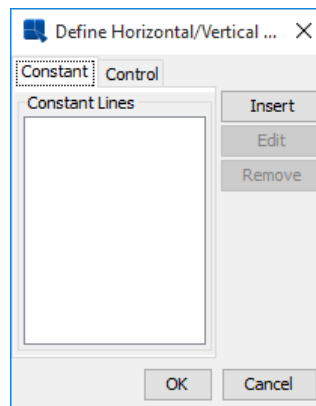
Line and Point Dialog for Floating Lines

The dialog allows you to specify a standard line style or to create a custom dash pattern in the same manner as line and point formatting. You can also specify the line thickness in pixels. The checkboxes at the bottom of the dialog allow you to place an arrowhead at the start and/or end of the line, as well as fill the area enclosed by the line to create a solid shape.

Floating lines can be deleted by selecting them and hitting the delete key.

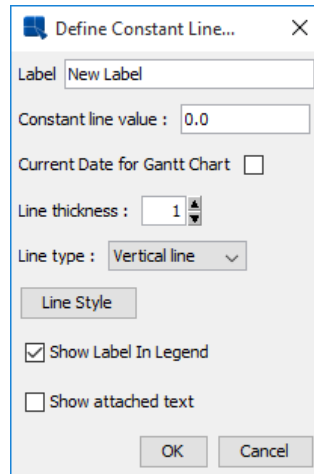
4.1.4.6.2.3. Fixed Horizontal/Vertical Lines

Fixed horizontal or vertical lines are lines that are drawn on one of the chart axes. These lines can also be drawn on three-dimensional charts where they appear as planes. There are two types of fixed lines: constant lines and control lines. Constant lines are lines that are fixed to a certain value in an axis. Control lines are drawn based on computed values that allow you to spot data points that are outside of certain value ranges. To add either type of line to a chart, select *Horz\Vert Line* from the *Insert* menu. This will bring up a dialog showing the list of existing horizontal/vertical lines, allowing you to edit the selected line, remove the selected line, and/or create a new line.



Define Horizontal/Vertical Lines Dialog

Clicking on *Insert* or clicking on *Edit* when an existing line is selected, respectively, will bring up a dialog allowing you to configure the selected line.

*Constant Line Dialog*

For constant lines you need to specify a label for the line, as well as the numeric value to use for the line. Note that for the category axis, the data points start with 0.5. You can also specify the line thickness and whether the line is horizontal or vertical. The last two options allow you to add an item to the chart legend for the line and whether to display any annotation for the line.

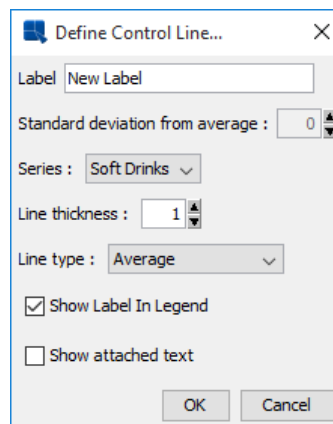
For Gantt charts, there is one extra option called *Current Date for Gantt Chart*. If you choose this option, the *Constant line value* field will deactivate and the current date will be used as the line value (the line position will be updated every time you run the chart).

For radar charts, the horizontal/vertical option is disabled. Radar charts draw constant/control lines at the same point around all the chart axes (in a similar manner to the radar grid). In addition, for radar charts, an additional option named *Circular Style* is present. By default, lines in radar charts are drawn in a segmented fashion - straight lines connect the points on each axis. Selecting this option will draw the constant/control as a circle.

**Note**

If you have not specified any annotation for the line then none will appear if you select the last option. For more on adding annotation to a chart, please see Section 4.1.4.6.1 - Adding Text.

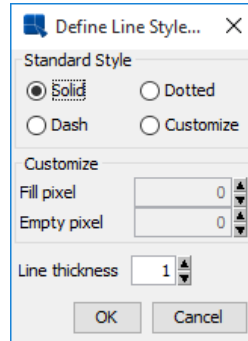
To add a control line, click on the *Control Line* tab in the dialog.

*Control Line Dialog*

For control lines you need to specify which series element you want to compute the value for (this option does not appear if no series is present) and how to compute the value. Options for control lines are average, minimum, maximum, and multiples of standard deviation.

After you have specified all of the options, the line will be added to the chart or the selected line will be changed, respectively. To edit any of the properties specified in the previous dialogs, you can select Insert → Horz/Vert Line again and select the line from the list. You can also double-click on the line that you want to modify.

You can change the appearance of fixed lines by first selecting the line and then selecting Format → Line and Point, or clicking the *Line and Point* icon on the toolbar. This will bring up a dialog allowing you to customize the line.



Line and Point Dialog for Fixed Lines

This dialog allows you to specify a standard line style or to create a custom dash pattern in the same manner as line and point formatting.

4.1.4.6.2.4. Trend Lines

A powerful feature of ChartDesigner is the ability to add trend lines to charts. Trend lines can help to show more details of a chart's data by exposing and highlighting certain trends.

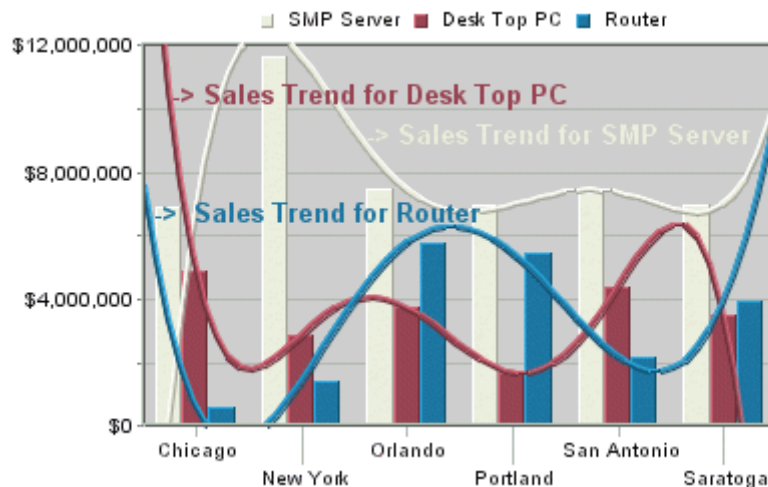
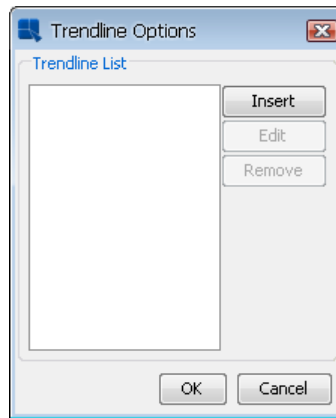


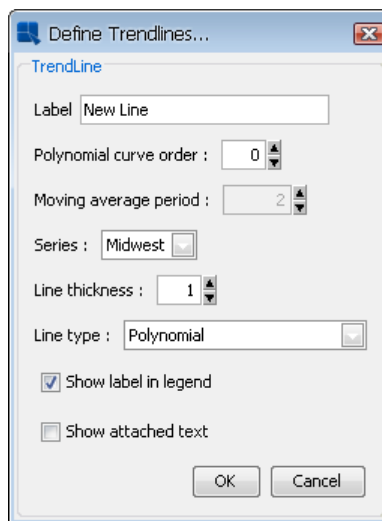
Chart with Trend Lines

To add a trend line to a chart, select Insert → Trendline. This will bring up a dialog showing the list of existing trend lines, allowing you to edit the selected trend line, remove the selected trend line, and/or create a new trend line.



Trend Line Options Dialog

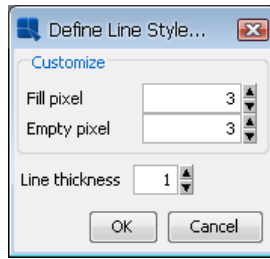
Clicking on *Insert* or clicking on *Edit* when an existing trend line is selected, respectively, will bring up a dialog allowing you to configure the selected trend line.



Define Trend Lines Dialog

In this dialog you can specify a label for the line, as well as what element of the data series to base the calculation on. The following types of trend lines are supported: a polynomial of any degree (please note that a linear trend line is a polynomial trend line of the 1st degree, i.e. the *Polynomial curve order* option has to be set to 1), a power, exponential, logarithmic, a moving average, an exponential moving average, an triangular moving average, cubic B-spline, and a normal distribution curve. For moving averages you will need to specify the average period and for a polynomial you will need to specify the curve order. You can also specify the thickness of the line and configure whether a label in legend and the attached text should be shown. In case the chart has data series, you can configure the trendline for a specific series.

After you have specified all of the options, the trend line will be added to the chart or the selected trend line will be changed, respectively. To edit any of the properties specified in the previous dialog, you can select 'TrendLine' from the Insert menu again, and select the line from the list. You can change the appearance of the trend line by first selecting it, and then selecting Format → Line and Point, or clicking the *Line and Point* button on the toolbar. This will bring up a dialog allowing you to customize the lines.



Line and Point Dialog for Trend Lines

This dialog allows you to create a custom dash pattern in the same manner as line and point formatting.

4.1.4.6.2.4.1. Normal Distribution Curve

A special type of trend line that allows you to draw a normal distribution curve for the data in the chart. In order to plot a normal distribution curve, the chart must either be a two-dimensional column or bar chart, it cannot have a data series, and the category should be numeric. Assuming these conditions are met, you can specify a normal distribution curve as one of the trend line options.

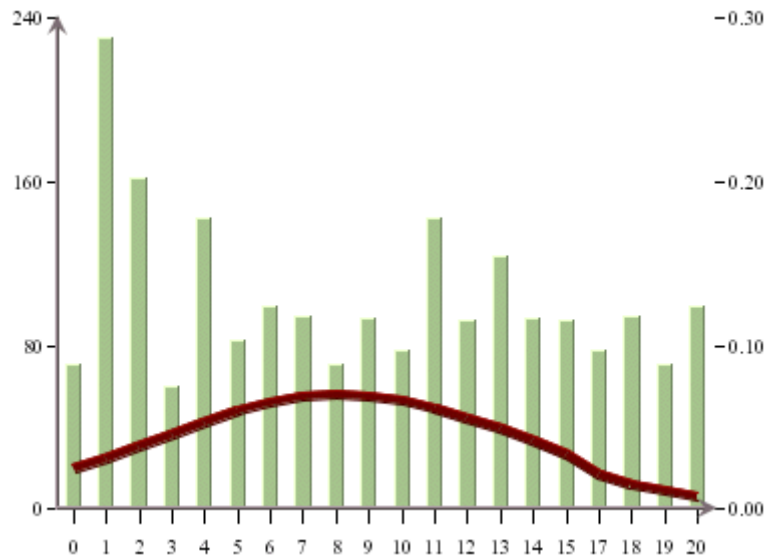
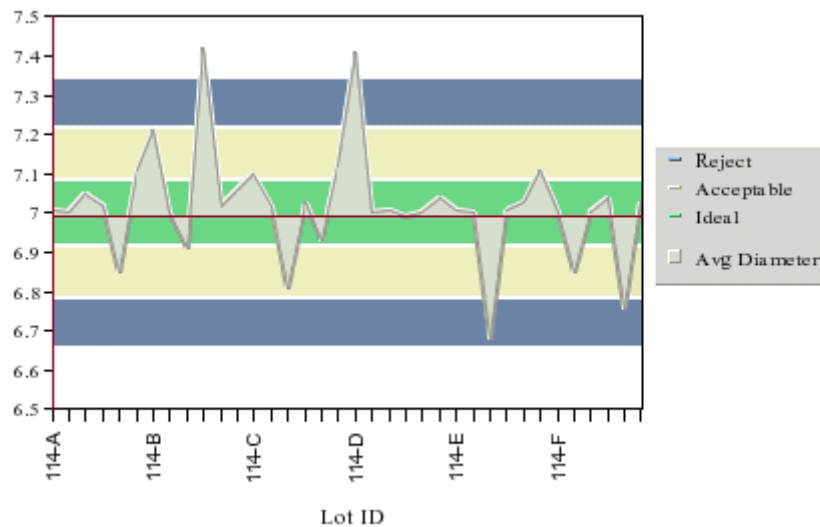


Chart with Normal Distribution Curve

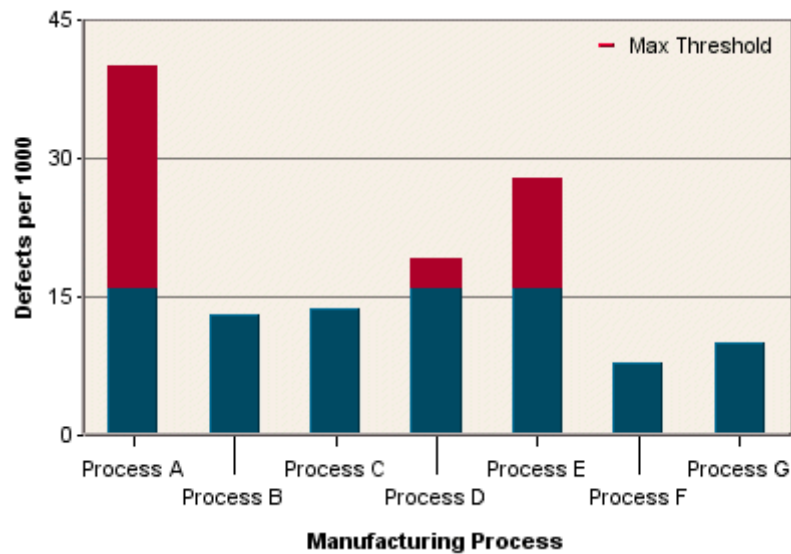
Since the scale for the curve is usually different than the scale for the value axis, the curve is shown on a secondary axis. You can modify the scale by changing the scale for the secondary axis.

4.1.4.6.3. Adding Control Areas

Control areas are useful for comparing the chart data against a certain range of data. For most two-dimensional charts, control areas are drawn as filled areas on the chart plot between a range of values on the chart's value axis and/or category axis. The data points are then drawn over top of the control areas giving you a quick visual reference to see which data points fall within the designated range. Instead of drawing a background area on the plot, the control areas can also be shown only where the data points intersect the control area. This feature gives users a clear visual reference when specific threshold values are reached.

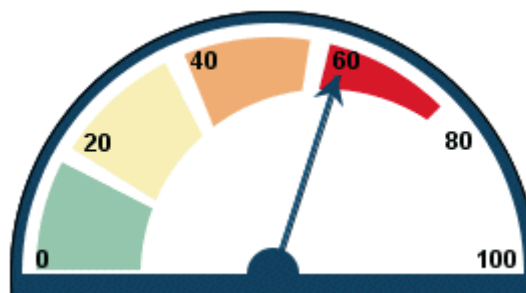


Two-Dimensional Area Chart with Control Areas



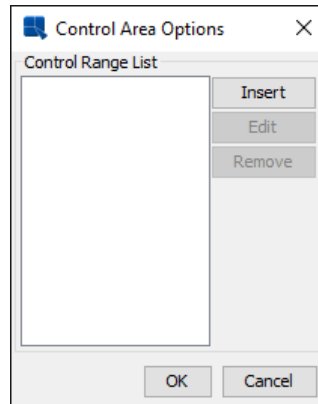
Column Chart with Control Area Drawn for Data Points

A special instance of control areas can be used for dial charts. For dial charts control areas are drawn as arcs on the face of the dial, allowing you to see if the dial hands (data points) fall within the range. Note that control areas are not available for radar and pie charts.



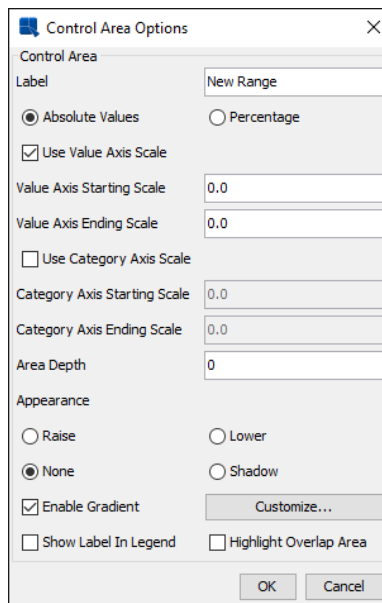
Dial Chart with Control Areas

To add a control area to a chart, select **Insert** → **Control Area**. The following dialog will appear showing the list of existing control areas, allowing you to edit the selected control area, remove the selected control area, and/or create a new control area.



Control Area List

Clicking on *Insert* or clicking on *Edit* when an existing control area is selected, respectively, will bring up a dialog allowing you to configure the selected control area. If your chart is not a dial chart, the following dialog will open.



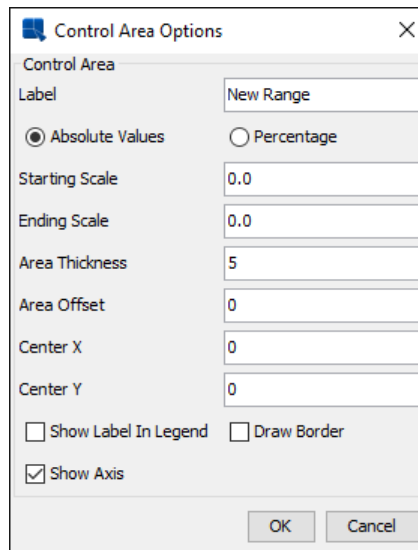
Control Area Configuration Dialog

The following options are provided for control areas:

- | | |
|-----------------------------------|--|
| Label: | This option allows you to specify a label for the control area. |
| Absolute Values: | This option allows you to specify the scale in absolute values. |
| Percentage: | This option allows you to specify the scale in percentage. |
| Use Value Axis Scale: | This option allows you to specify whether the control area should be bounded by values on the value axis of the chart. |
| Value Axis Starting Scale: | This is the lower bound for the control area on the value axis. |
| Value Axis Ending Scale: | This is the upper bound for the control area on the value axis. |

Use Category Axis:	This option allows you to specify whether the control area should be bounded by values on the category axis of the chart.
Category Axis Starting Scale:	This is the lower bound for the control area on the category axis.
CategoryAxis Ending Scale:	This is the upper bound for the control area on the category axis.
Area Depth:	This option specifies the depth for any of the appearance styles. If no style is selected, the depth will have no effect.
Appearance:	This option allows you to specify a 3D or shadow effect for the control area. If the area depth is specified as zero, the appearance will not take effect.
Enable Gradient:	Enable color gradient for the control. Gradient settings are described in Section 4.1.4.1.3 - Format Menu
Show Label In Legend:	This option specifies whether or not to show the control area label in the chart legend.
Highlight Overlap Area:	This option will only show the control area where the data points overlap the control area boundaries.

If your chart is a dial chart, then a different dialog will appear when you select Insert → Control Area and then click on the *Insert* or *Edit* button.



Control Area Dialog for Dial Charts:

The following options are provided for dial chart control areas:

Label:	This option allows you to specify a label for the control area.
Absolute Values:	This option allows you to specify the scale in absolute values.
Percentage:	This option allows you to specify the scale in percentage.
Starting Scale:	This is the value where the control area begins.
Ending Scale:	This is the value where the control area ends.
Area Thickness:	This option allows you to set the thickness for the control area
Area Offset:	This option allows you to specify the offset in pixels from the edge of the dial chart

Center X:	This sets the X coordinate for the center of the control area. 0 shares the same center point as the dial face. You can specify a new number (either negative or positive) pixels to specify an offset position from the center of the dial.
Center Y:	This sets the Y coordinate for the center of the control area. It works in the same way as the previous option.
Show Label in Legend:	Specifies whether to show the control area label in the chart legend.
Draw Border:	This option allows you to draw a border around the control area.
Show Axis:	This option allows you to show or hide the axis for the remaining area not covered by the control range.

After you have specified all of the options, the control area will be added to the chart or the selected control area will be changed, respectively. To edit any of the properties specified in the previous dialog, you can select Insert → Control Area again, and select the control area from the list. You can also double-click on the control area that you want to modify.

4.1.4.6.4. Adding Tables

In addition to displaying charts, you can also display a table showing the data points displayed in the chart. The table can be placed below or to the right of the chart plot.

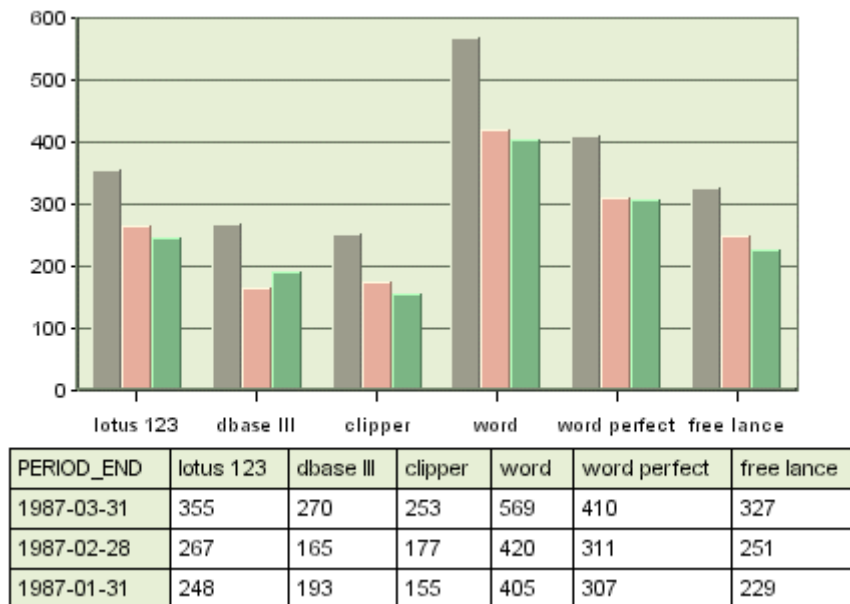
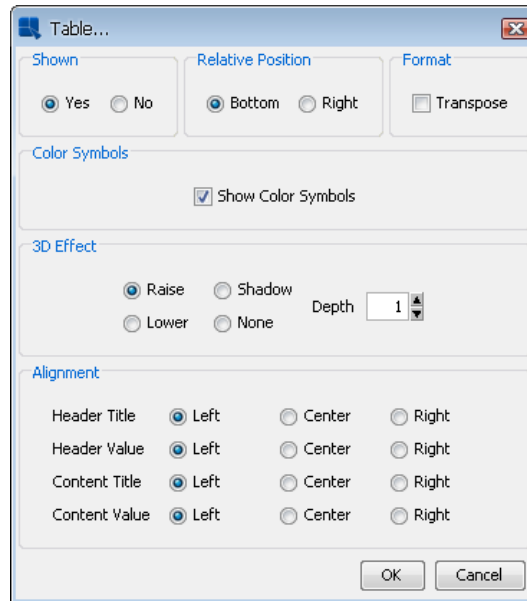


Chart with Table

To add a table to a chart, or to modify the various display options for a table, select Format → Table. This will bring up a dialog allowing you to customize various options for the table display.



Format Table Dialog

From this dialog you can specify whether or not to display the table, as well as what relative position to give the table either to the bottom or right-hand side of the chart plot. You can also specify a 3D effect for the table and its depth.

The *Transpose* checkbox allows you to swap the columns and rows of the table. By default, the category elements are drawn as columns and the data series elements as rows.

The *Show Color Symbols* option allows you to show/hide color boxes for data series in the table.

quarter/drink	Coffee	Fruit Juice	Soft Drinks	Tea	Water
Q1	181	89	264	114	303
Q2	149	144	212	144	156
Q3	169	70	498	162	275
Q4	195	171	230	144	186

Chart table with color boxes

The *Alignment* options allows you to specify horizontal alignment of the text in the table cells, either left, center, or right. The alignment can be set for row headers, column headers, and inner table cells.

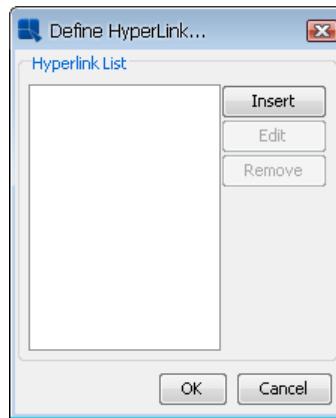


Note

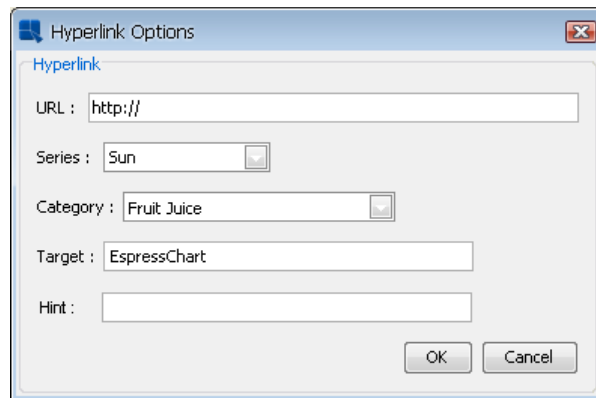
If there isn't enough room in the chart canvas, not all data points will be displayed in the table. The table size adjusts with the canvas size and also with the font size in the table cells.

4.1.4.6.5. Adding Hyperlinks

ChartDesigner has an capability to add hyperlinks to any data point in a chart. Links can be specified for either single data points or multiple elements. Any added hyperlinks will be applied to both the data points on the chart and to their respective fields on the legend. To add a hyperlink to a chart, select *Insert* → *Link* or right-click on a data point and select *Insert Link* from the pop-up menu. This will bring up a dialog showing a list of existing hyperlinks which you can configure, remove, and/or create new ones.

*Insert Link Dialog*

Clicking on *Insert* or clicking on *Edit* when an existing hyperlink is selected will bring up a dialog allowing you to configure the selected hyperlink.

*Define Link Dialog*

The URL field allows you to specify a Web page that you want to link.

The *Series* and *Category* drop down menus allows you to select an element in the data series and category elements for the hyperlink. You can also link to all data series elements or all category elements.

You can specify the *Target* parameter recognized by HTML when specifying a hyperlink to be attached to a data point or data series. This can be used to determine whether the new HTML page should open in a new browser window, in the same browser window, or whether the new page should occupy the same portion of the page as the current page.

The *Hint* field allows you to enter text that will pop-up when you move your mouse cursor over a data point. If you want to create pop-up labels without hyperlinks, you can leave the URL field blank and only specify the hint.

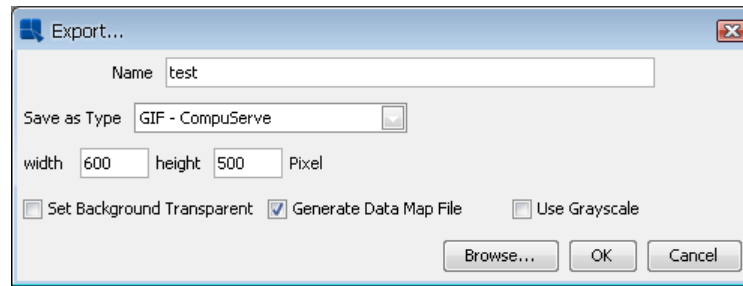
4.1.4.6.5.1. Viewing Hyperlinks

If you specify hyperlinks for charts, they will be active only when the chart is exported to Flash format*. For most image formats such as PNG, JPG, GIF, etc, an image map file containing information for the link will be automatically generated when you export the chart. You can insert the image and image map into an HTML file to view the image with clickable links.



Note

* For Flash export, when the user clicks on the hyperlink, there will likely be a warning message prompted by Flash saying that there was a potentially unsafe operation. You can turn this warning off by clicking on settings and adding the chart into the list of trusted locations.


*Export Dialog*

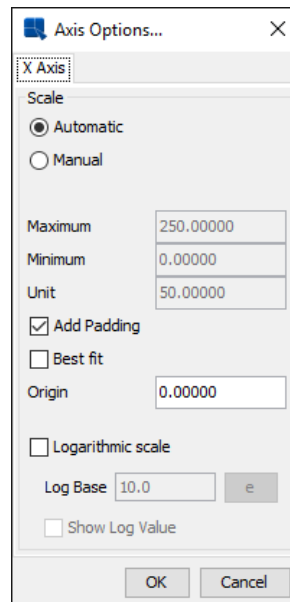
4.1.4.7. Formatting Chart Axes

ChartDesigner provides a number of extensive formatting capabilities for the chart axes. Users can customize everything from the axis scale to the way how axis labels should be displayed.

4.1.4.7.1. Axis Scale

By default, the scale of any value axes in the chart is calculated to provide a 'best fit' for the data being plotted. This is often a useful feature if the data being displayed can change radically. However, you may often want to set

the scale of the axes manually. To modify the axis scale, select Format → Axis Scale, or click the  *Axis Scale* button on the toolbar. This will bring up a dialog allowing you to format the scale for any value axes in the chart.

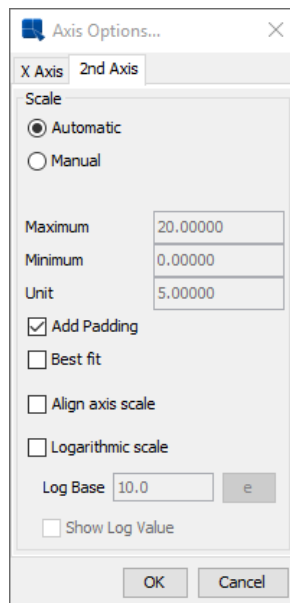
*Axis Scale Dialog*

The following options are provided:

Automatic:	This turns on automatic scaling for the axis. This is the default option.
Manual:	This turns on manual scaling, allowing you to set the axis scale to your preference.
Maximum:	This is the highest value on the axis scale.
Minimum:	This is the lowest value on the axis scale.
Unit:	This is the step interval between successive labels.
Add Padding:	This will raise the highest value of the axis to create a cushion between the max value of the data and the top of the chart.

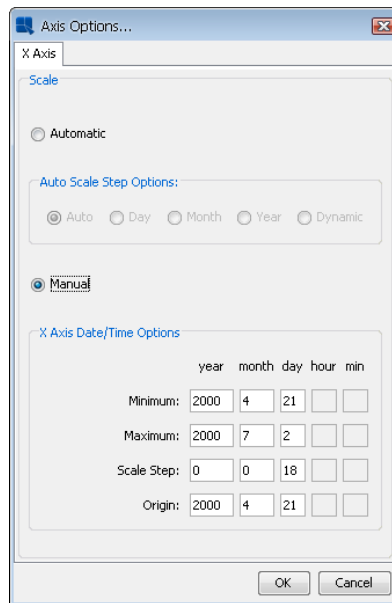
Best fit:	This will automatically place the origin of the chart based on the minimum and maximum values of the data.
Origin:	This allows you to specify where the X and Y axes should intersect. This is usually set to zero.
Logarithmic Scale:	This option creates a logarithmic scale for the given axis. It's valid only if the axis in question contains positive values.
Log Base:	This allows you to specify the base for the log value.
Show Log Value:	This specifies whether to show log values in the axis labels or not.

The axis scale dialog will have a tab for each axis in the chart. There's a unique option available for secondary axes which allows you to align the axis scale. It will apply all options from the primary axis to the secondary axis, giving both axes the exact same scale.




Axis Scale Dialog for Secondary Axis

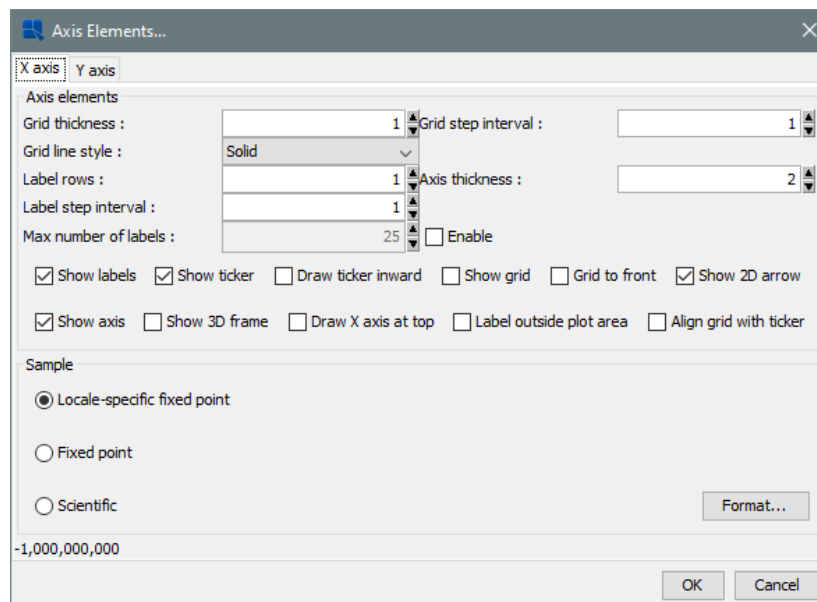
The axis scale dialog is different for a Gantt chart. You can still select the *Automatic* option to have the scale configured automatically or the *Manual* option to set the scale manually. The *Automatic* axis scale has a few Auto Scale Step Options: Auto, Day, Month, Year, and Dynamic. The *auto* option always finds a best fit. The *dynamic* option also finds a best fit, but unlike the *auto* option, it uses standard step intervals only (for example: 1 month, 1 year etc...). When the *Manual* axis scale is chosen, the *Maximum*, *Minimum*, *Unit* and *Origin* are replaced with *Maximum Date*, *Minimum Date*, *Scale Step* and *Origin Date* respectively and these new settings take in a Date/Time (represented by yyyy, MM, dd, hh, mm).



Axis Scale Dialog for Gantt Chart

4.1.4.7.2. Axis Elements

The appearance properties of the axes and the axis labels are controlled through the axis elements dialog. To invoke the axis elements dialog, select Format → Axis Elements or click the  *Format Value Elements* button on the toolbar. This will bring up the following dialog, allowing you to customize elements in all chart axes. You can also customize the appearance of dial and pie chart labels from this dialog.



Axis Elements Dialog

A tab will appear in this dialog for each axis in the chart. The dialog allows you to perform the following options. Note that some options are only available for certain chart types, certain data types, and on certain axes.

Grid thickness:

This allows you to specify the thickness of any grid lines along the axis.

Grid step interval:

This option allows you to set the grid step interval for any grid lines along the axis.

Grid line style:	This option allows you to select the grid line style (solid, dotted, dash).
Label rows:	This option allows labels to be displayed in alternating rows. This can prevent overlapping. This option is only available for X-axis.
Axis thickness:	This option allows you to set the thickness of the axis in pixels. Note that this setting is applied to all axes in the chart.
Label step interval:	This option allows you to set the label step interval for the data. For example, setting this to 2 will draw the label for every other data point in the chart.
Label interval unit:	This option allows you to select the unit to be used when sorting and representing time-based data (date, time, or timestamp). Selecting tickers will use the data as it is read by Chart Designer. You can also select Dynamic for time-based data and that will choose an appropriate scale (depending on number of data points and range of data).
Max number of labels:	This option allows you to select the maximum number of labels and tickers displayed on the axis. If the number of labels exceeds the max count, the label step will be re-calculated.
Ascending/Descending:	This option allows you to order and filter time-based data. You can sort the data in ascending or descending order, as well as specify the starting (or ending) point for the data.
Show labels:	This option allows you to remove or display the labels for each axis.
Show ticker:	This option allows you to remove or display the axis tickers.
Draw ticker inward:	This option will draw the axis tickers inside the plot area instead of outside (default).
Show sub-tickers:	This feature is only available for the value axis when the axis scale is set to logarithmic with a log base of 10. This feature will draw interval tickers (non-uniform) between the points on the value axis.
Show grid:	This option allows you to remove or display the grid for each axis.
Grid to front:	This option allows you to draw the grid lines on top of the data elements in the chart. By default the data points are drawn on top of the grid.
Show 2D arrow:	This option allows you to remove or display the arrowhead at the end of the axis. Note that this option applies to all chart axes and it is only available for two-dimensional charts.
Show axis:	This option allows you to remove or display the axis (for two-dimensional charts) or the wall (for three-dimensional charts).
Show 3D frame:	This option allows you to remove or display a frame around the chart. Note that this option applies to all chart axes and it is only available for three-dimensional charts.
Label outside plot area:	This option sets the labels to be placed outside of the plot area, irrespective of where the axis is. This feature can be useful for category axis labels if you're plotting data with both positive and negative values.
Align grid with ticker:	This option aligns the grid line with the ticker instead of placing it between tickers. This places the ticker and the corresponding grid line along the same line. This option only applies to the category axis of column-type charts.
Swap Y-axis position:	This option will swap the primary and secondary value axes. This option can only be found under the 2nd Axis tab.

Draw X-axis at top:

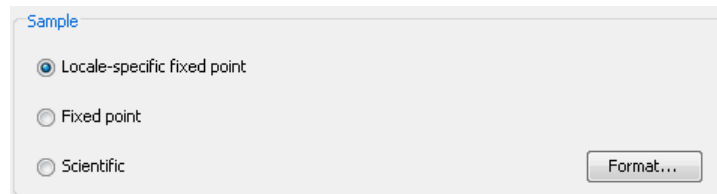
This option allows the X-axis to be positioned at the top of the chart instead of the default bottom position. This option is available for two-dimensional column, bar, scatter, high-low, HLCO, bubble, and Gantt charts.

4.1.4.7.2.1. Axis Label Formatting

The axis elements dialog also allows you to format the appearance of the axis labels, depending on what type of data is plotted on the axis. The *Format Options* portion of the dialog contains the label formatting dialog.

Formatting Numeric Data

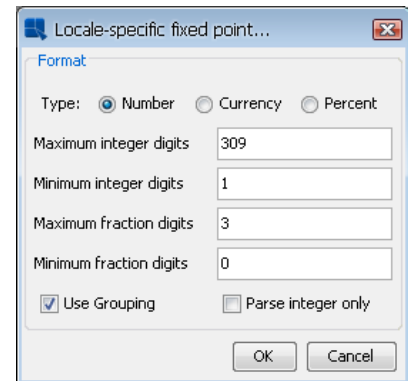
For numeric data there are three primary options for display formatting: locale-specific fixed point, fixed point, and scientific. Additional options will be displayed if you click on the *Format* button.



Numeric Data Format Options

Locale-Specific Fixed Point

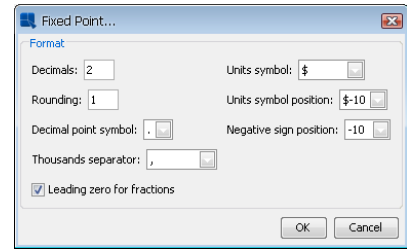
This will change the format of the data depending on the locale in which it is being viewed. Additional formatting for this option allows you to specify whether the data should be displayed as a number, currency, or percentage. In addition, you can set the maximum and minimum number of integer digits and fraction digits. Other display attributes will vary depending on locale.



Locale-Specific Formatting Options

Fixed Point:

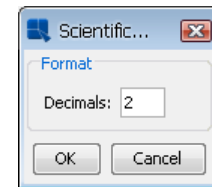
This will keep the data format consistent, regardless of locale. Additional formatting for this option allows you to set the number of decimals, rounding for digit number, unit symbols, negative sign position, decimal and thousands separator, and enable leading zeros for fractions



Fixed Point Formatting Options

Scientific:

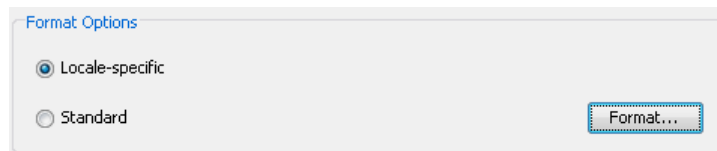
This will display the data in scientific notation. Additional formatting for this option allows you to set the number of decimals.



Scientific Formatting Options

Formatting Date/Time Data

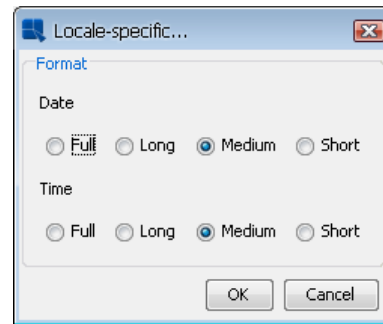
For date/time data there are two primary options for display formatting: locale specific and standard. Additional options will be displayed if you click on the *Format* button. The available options will vary depending on the nature of your data. Date, time, and timestamp data will bring up date, time, and date & time options respectively.



Date/Time Data Format Options

Locale-Specific:

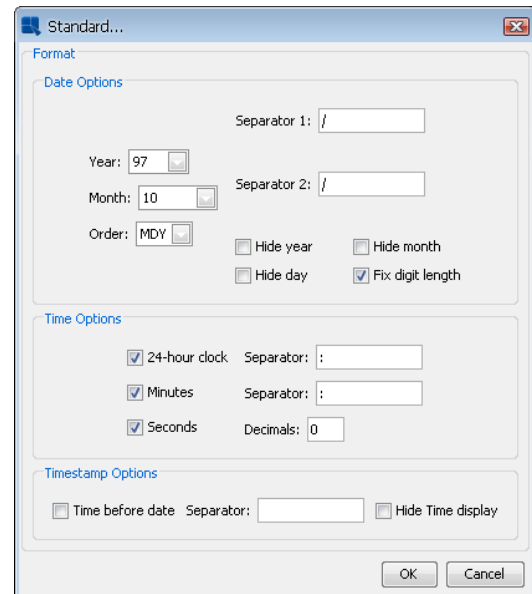
This will change the format of the data depending on the locale in which it is being viewed. Additional formatting for this option allows you to select full, long, medium, or short notations for date and time information. Other display attributes will vary depending on locale.



Locale-Specific Formatting Options

Standard:

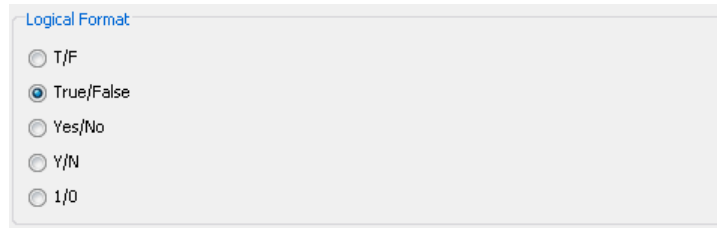
This will keep the data format consistent, regardless of locale. Additional formatting for this option allows you to select year and month displays, as well as the order in which month, day, and year information is presented. You can also select the characters that you want to be used as separators. Time options allow you to display hours, minutes, and/or seconds and also to select the separators between them. For timestamp data, you can select to display the time before or after the date, as well as the separator to be used between them.



Standard Formatting Options

Formatting Logical Data:

There are five options available for displaying logical or Boolean data: T/F, True/False, Yes/No, Y/N, and 1/0.

*Logical Data Formatting Options*


Any changes you make to the data formatting will take effect after you click on the *OK* button in the axis elements dialog. Note that there are no additional formatting options for string data.

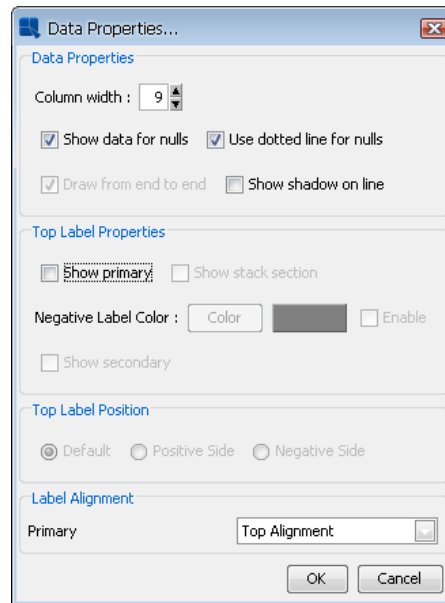
4.1.4.8. Formatting Plot/Data Elements

ChartDesigner provides a number of ways to customize and configure the way data points are drawn and annotated on the chart, as well as the chart plot itself.

4.1.4.8.1. Data Properties

Many of the data display options are controlled through the data properties dialog. From this dialog you can control the size of bars/columns, set display options for null values, and specify options for data labels. To invoke the data

properties dialog, select *Format* → *Data Properties* or click the  *Data Properties* button on the toolbar. This will bring up the following dialog:

*Data Properties Dialog*

This data properties dialog contains the following options:

Column width:

This specifies the ratio of the bar/column width with respect to the gap between successive bars in the chart. Each unit represents 1/10th of the space between data points. Therefore, entering **9** would leave 10% of the space between data points blank, while **10** would eliminate all space between bars/columns. This option only pertains to two-dimensional bar, column, stack bar, stack column, high-low, HLCO, and Gantt charts. To control the column thickness in three-dimensional charts, you can use the thickness of shape slider in the navigation panel.

Show data for nulls:

This option will connect lines when null data is present. For example, if you have three points and the value of point 1 is 4, point 2 is null, and point 3

is 6, then a line will be drawn from 4 to 6. This option is only available for line charts and other two-dimensional charts with lines. All other chart types will not plot null data.

Use dotted lines for nulls:

You can use this option to replace the full line with a dotted line. Like the show data for nulls option, this property is only available for lines.

Draw from end to end:

This option allows you to draw two-dimensional line and area charts across the entire plot area, rather than offsetting to the first and last data points on the chart.

Show shadow on line:

This option allows you to use shading on two-dimensional lines. However, the line must be thicker more than one pixel.

Show primary:

This option will display data top labels for the primary values in the chart.

Show stack section:

This option will display individual labels for each stack section for stack bar, stack column, and stack area charts.

Negative Label Color:

This option will display the top labels with a value smaller than that of the origin in a different color that can be selected using the *Color* button after enabling the feature.

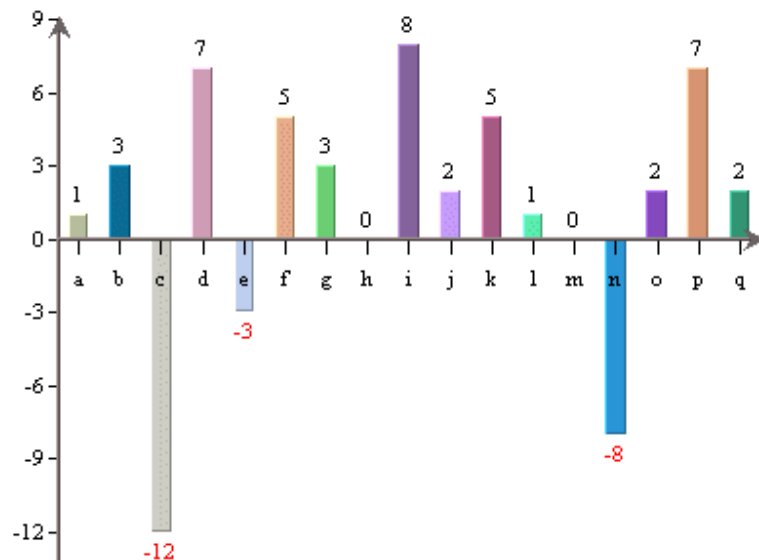


Chart with Colored Negative Top Labels

Show secondary:

This option will display data top labels for the secondary values in the chart.

Top label position:

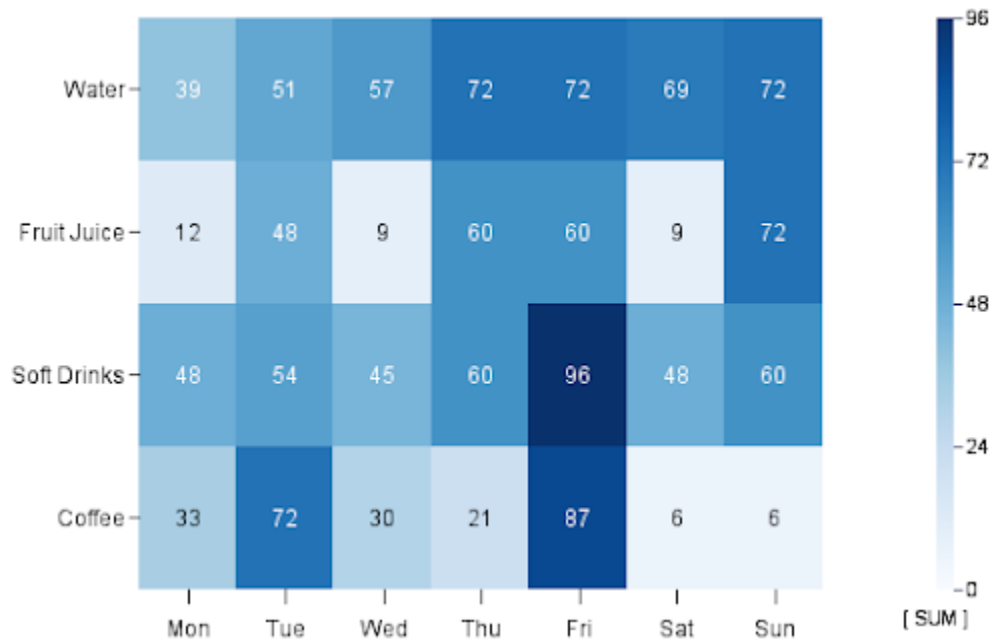
This option allows you to specify where the data top labels should be drawn. By default, they are drawn above data points if they are positive and below data points if they are negative. Other options allows you to draw the labels to either positive or negative side.

Label Alignment:

This option allows you to set the alignment for the data top labels. You can draw them at the top, bottom, or middle of the data points. In addition, you can select to draw the label inside the data point at the top or bottom. An additional option stack charts offers you to set the alignment for stack section labels.

4.1.4.8.1.1. Data Properties for Heatmap Chart

For heatmap charts, the Data Properties dialog allows users to configure the values displayed in the cells in the matrix.



Data Properties for Heatmap Chart

4.1.4.8.2. Date/Time Based Zooming

For charts displaying date or time data on the category axis, ChartDesigner provides a unique feature allowing users to perform date/time based zooming. Using this feature, you can group the category elements into user-defined intervals and aggregate the points in each group. You can also filter the data by specifying upper and lower bounds for the results.

For example, suppose your data contains daily sales volume for the past two years. Using zooming, you could aggregate the data to look at average volume per month, quarter, or year. Using the upper and lower bounds, you could narrow the range to look at weekly sales volume within a specific quarter.

Zooming is available for all chart types except scatter, surface, box, dial, polar, radar, bubble, and Gantt.

4.1.4.8.2.1. Adding Zooming

When you create a new chart with date, time, or timestamp data in the category axis, you can specify zooming options by selecting Format → Time Zooming Options.

Time Zooming Options...

Enter data range

	year	month	day	hour	minute	second
Lower Bound:	1995	3	4	20	30	30
Upper Bound:	1998	10	1	20	30	30

Scale: 1 months

Linear: false

☐ Enable Zooming ☒ Disable Lower Bound ☒ Disable Upper Bound

Advanced OK Cancel

Zoom Options Dialog

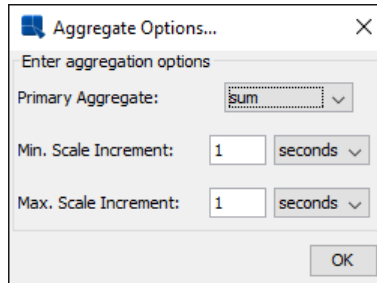
This dialog allows you to specify a lower and upper bound for the data, as well as the interval by which you would like to group the data. The scale specified here must be within the maximum and minimum scale specified in the aggregation options dialog.

This dialog also allows you to preserve a linear scale for the chart. By setting the *Linear* option to true, the chart will always display points for the grouped intervals, even if there is no data associated with a particular group. For

example, say again that you are measuring sales volume over a three month period - April, May, and June. If the input data has no records for May and you set the *Linear* option to true, a point will be drawn for May with a value of zero. If you set the *Linear* option to false, the data point for April will be immediately followed by June.

You can disable/enable zooming, as well as the lower and upper bound restrictions by using the checkboxes at the bottom of the dialog.

If you enable zooming (if you check *Enable Zooming* option), the dialog *Aggregate Options* will appear, prompting you to specify aggregation options for the grouped data points.



Aggregation Options Dialog

In this dialog, you can specify the Primary Aggregation, as well as the maximum and minimum scale increments that can be used when zooming the data. After you have specified your desired options, click on the *OK* button to return to zooming options.

Once you have finished specifying all the options, click on the *OK* button and the zooming will be applied to the chart.


4.1.4.8.2.2. Zooming In Chart Viewer

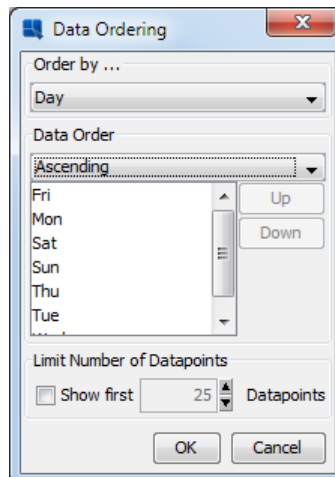
When deploying charts using Chart Viewer, end users can perform dynamic zooming. To perform a time-series zoom in the Chart Viewer, **Ctrl+Click** on a point on the chart and drag it to another point in the chart. This will automatically zoom in based on the lower and upper bounds selected using the mouse. The aggregation is performed according to the options that were set during design time. You can undo the zoom by **Ctrl+Right-Click**.

The scale interval is chosen automatically, depending on the data and chosen bounds (as long as minimum 2 data points can be shown). The scale interval can also be changed in the Chart Viewer by pressing **Alt+Z**. This will bring up a dialog allowing the user to change the zoom settings.

4.1.4.8.3. Data Ordering

ChartDesigner allows you to change the order of the category and series elements. To modify the ordering, select

Data → Ordering or click the  *Change Data Ordering* button on the toolbar. This will bring up the following dialog:

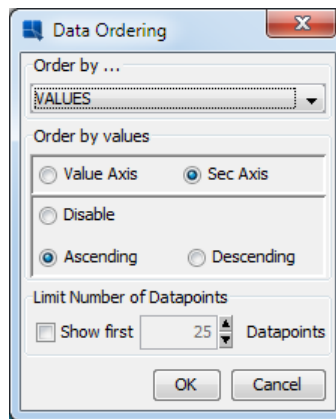


Data Ordering Dialog

There is an *Order By* list which contains category element, series element, and an option marked *VALUES*. You have the following options for the category and series elements:

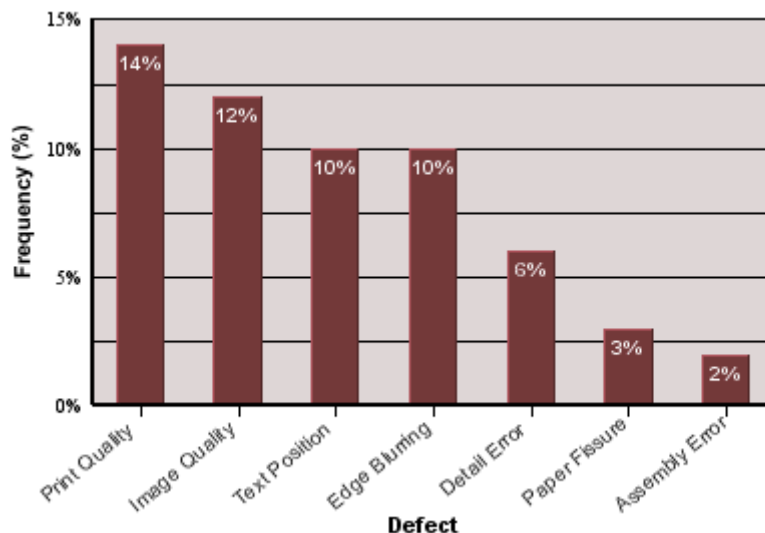
- DataSource Order:** This option turns the ordering off. The categories/series order will depend on the data source only and will not be altered by the EDAB at all.
- Ascending:** This option will arrange the categories/series elements in ascending order. For example, if the category elements are strings, they will be arranged alphabetically.
- Descending:** This option will arrange the categories/series elements in descending order.
- Customize:** This option allows you to customize the categories/series order. To customize the order, select an item from the list of Categories/Series items and then move it upwards or downwards in the list by clicking on the *Up* or *Down* button (the buttons are inactive until you select the *Customize* option).

You can also sort the category elements based on their corresponding values. To do this, select the *VALUES* option in the data ordering dialog.



Value Data Ordering Dialog

If you choose the *VALUES* option, the entire dialog changes. From this dialog, you can specify to sort the category elements based on their corresponding values in the value, or secondary value axis. You can also specify whether to sort them in ascending or descending order. This type of ordering is called a *Pareto* chart and is often used in process control applications.




Pareto Chart

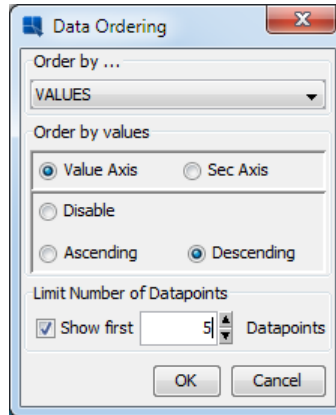
Please note that any sorting set will be re-applied if the chart is refreshed and/or if the data changes.

4.1.4.8.3.1. Top/Bottom N Charts

Sometimes you want to plot only a few highest or lowest values. To do that, you can use the *Top/Bottom N* function.

To enable this feature, choose Data → Ordering, or click the  *Change Data Ordering* button on the toolbar.

If the chart doesn't have any data series, the *Ordering* dialog will show the *Limit Number Of DataPoints* option.



This option can only be enabled if you sort the chart by categories or values, or by the values in ascending or descending order. If you have such chart, you can enable this function by selecting the *Show first* option. Then you can specify the maximum number of items that will be shown in the chart. If the data source returns more items than you specify in this option, excessive items will not be shown in the chart as if they didn't exist.

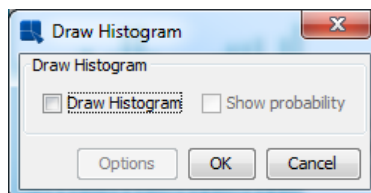
4.1.4.8.4. Histograms

Histogram is a useful analysis tool that allows you to track how often events occur and when and how a set of data falls into specific ranges. ChartDesigner allows you to plot histograms based on the category elements in a chart. You can plot histograms for all category data types except time-based data (date, time, or timestamp).

Histograms are calculated by counting data points or instances of each category element. For numeric categories, you can further specify upper and lower bounds, as well as the number of bins or bin width to create ranges for the frequency counts.

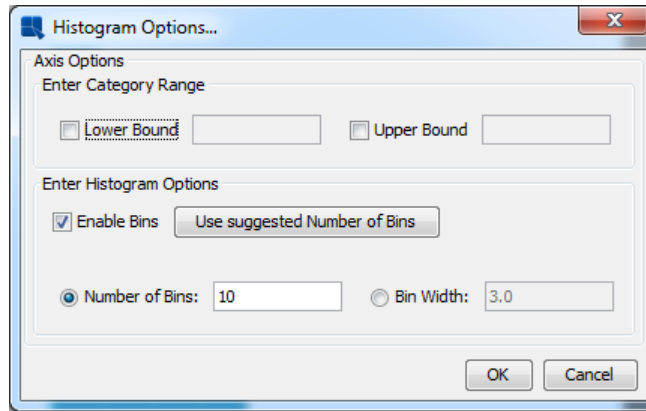
To create a histogram, you must start with a 2D column chart, bar chart, line chart or area chart. In the Data Mapping dialog, make sure *DataSet* is set to *None* and the field you want to plot in the histogram is set in the Category axis.

Once you click *Done* in the Data Mapping dialog and the chart is shown on the canvas, select Format → Histogram Options. A dialog will appear allowing you to select a histogram plot. By default, the histogram is displayed as frequency count. You can choose to change it to display probability by checking the *Show probability* check box.



Select Histogram Dialog

If the values in the Category axis is numeric, you will see that the *Options* button is enabled. When you click Options button, another dialog will appear, allowing you to specify options for the histogram plot.

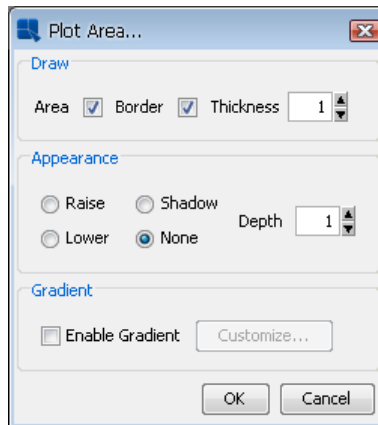
*Histogram Options Dialog*

From this dialog, you can set a lower or upper bound for that data being plotted. When you place bound restrictions, the histogram will not count data that falls outside of the range specified by the upper and lower bounds. If you select *Enable Bins*, you can specify the number of bins or set width of each bin. The default number of bins is 10. You can also click *Use suggested Number of Bins* if you wish to use a value calculated by the system. If *Enable Bins* is deselected, the frequency count will be performed on each Category value instead of a range of values for a bin.

If you enable scale, you can specify the number of bins or set width of each bin.

4.1.4.8.5. Formatting Plot Area

The plot area is the plane on which the data points are drawn for two-dimensional charts. You can customize the appearance of the plot area by selecting Format → Plot Area. Assuming the current chart is a two-dimensional chart, the following dialog will appear.


*Plot Area Dialog*

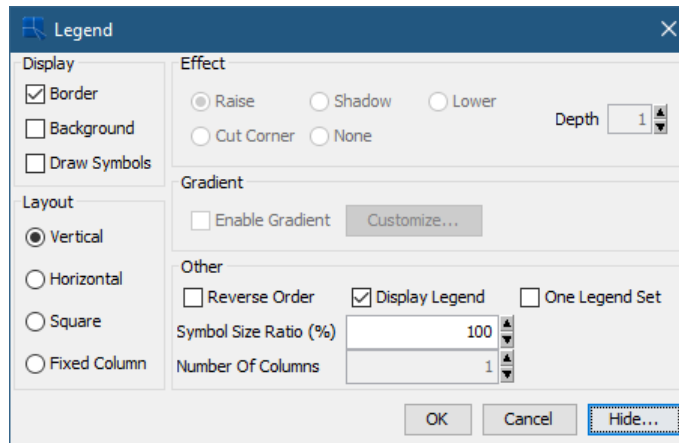
This dialog allows you to draw a border around the plot area, or fill it with a background color. If you fill the area, you can also specify certain 3D effects like raising, lowering or shadow.

On this dialog, you can also set up gradient background for the plot area. The gradient settings are the same as in the *Rendering options* described in the Section 4.1.4.1.3 - Format Menu.

4.1.4.8.6. Formatting Chart Legend

You can control and modify the display of the chart legend either by selecting Format → Legend, or by clicking on

the  *Format Legend* button on the toolbar, or by selecting *Legend properties* from legend pop-up menu (which pops up when you right-click on the legend). This will bring up the following dialog, allowing you to customize the legend properties.



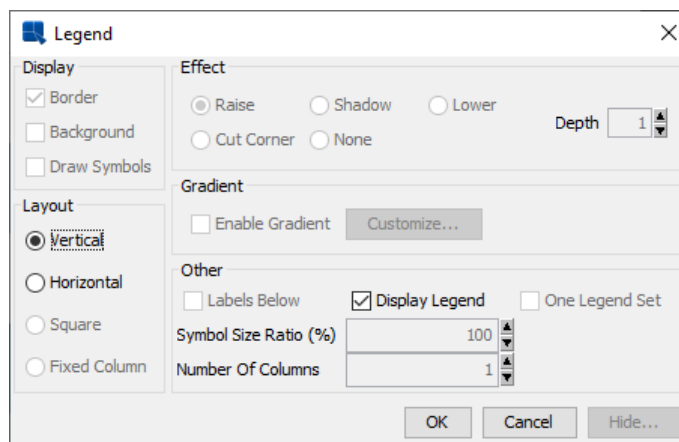
Format Legend Dialog

The dialog contains the following options:

- Display:** These options allow you to turn on or off the legend border and background. This also allows you to display the point symbols instead of lines or blocks in the legend.
- Effect:** This allows you to add a 3D effect to the legend. You can raise it, lower it, or draw a shadow. In addition to the 3D effects, you can also display the legend with cut corners.
- Layout:** This allows you to change the legend from vertical, horizontal, square, or fixed column layout.
- Gradient:** Allows you to configure gradient for the legend background. The gradient settings are the same as in the *Rendering options* described in the Section 4.1.4.1.3 - Format Menu.
- Other:** This allows you to choose whether or not to display the legend, or to draw the legend in reverse order. You can set the fixed number of columns in legend in the *Number of columns* field. This field will be active only if you choose the *Fixed columns* layout option in the *Layout* section. You can set trend/control lines and chart data legend drawn as *One Legend Set*. You can also change the size of the symbols in the legend.

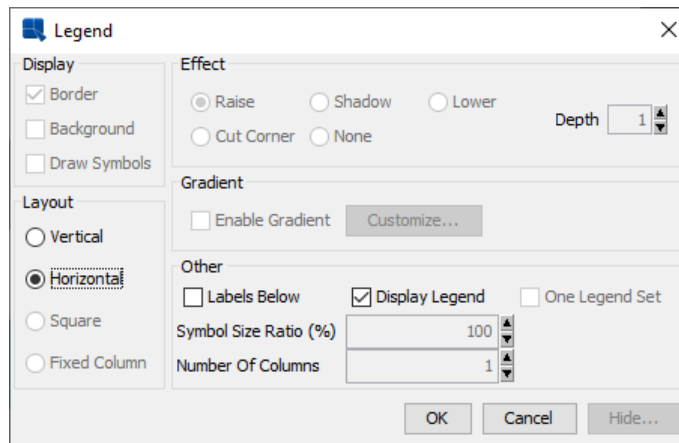
Additionally, you can remove specific category/series elements from the legend by clicking on the *Hide* button. This will bring up a list of the legend items, where you can select which elements you would like to hide.

The legend for heatmap charts differs from those of other charts, and some legend options in the *Legend* dialog are not applicable to heatmap charts. By default, the legend for heatmap charts is displayed vertically.



Heatmap Vertical Legend

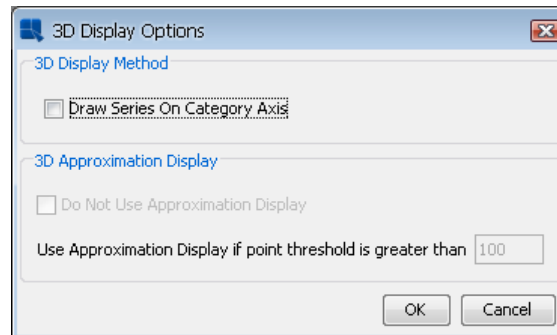
In the horizontal layout, you can choose to display tickers and labels below the legend by selecting the *Labels Below* option.

*Heatmap Horizontal Legend*

4.1.4.8.7. 3D Display Options

ChartDesigner renders three-dimensional charts in true 3D, allowing light source modification, panning, zooming, and rotation. However, 3D rendering can be very memory and CPU intensive. When charts have a lot of data points (like 3D scatter and surface charts), it's possible to run out of memory when generating the chart. To solve this problem, a rendering approximation feature is provided. Using this algorithm the chart is not rendered perfectly, but it's usually acceptable when a lot of points have to be shown.

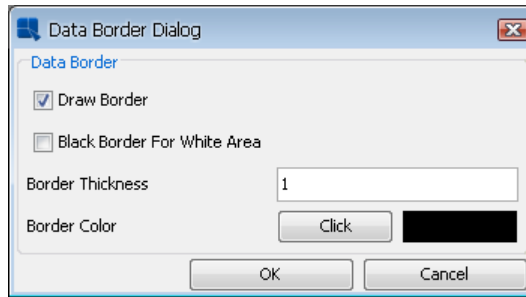
By default, approximation is turned on at a threshold value of 100 points. This means that if a 3D chart has more than 100 data points, the approximation will be used. You can turn this feature off, or change the threshold value by selecting Format → 3D Display Options. This will bring up the following dialog.

*3D Display Options Dialog*

The two options for 3D approximation allows you to turn on/off the approximation and set the threshold value. The other option in this dialog allows you to draw the series in-line (the same option is in navigation panel).

4.1.4.8.8. Data Border

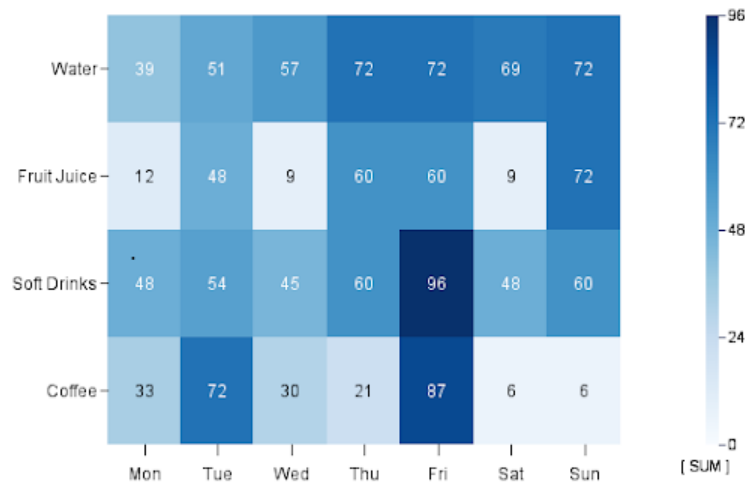
For column, bar, stack column, stack bar and HLCO charts, ChartDesigner allows you to configure a border around the columns. To set the border option, select Format → Data Border. This will bring up a dialog allowing you to set border options.



IData Border Dialog

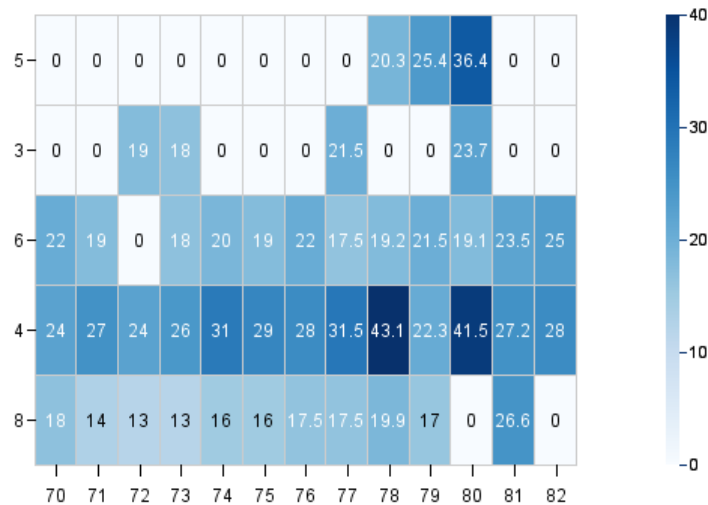
The first option allows you to turn on/off the data border. The second option allows you to set a black border for any white areas in the chart. Please note that the border is black only if the first option is unchecked and will only appear around white areas in the chart. The third and fourth options allow you to set border thickness and border color. If you click on the *Click* button, a new dialog will appear allowing you to select or enter a new color.

For heatmap charts, you can configure a border around the data rectangles. This is particularly useful when there are many light-colored cells.



Heatmap Data Border

The following is an example heatmap chart with border thickness set to one.

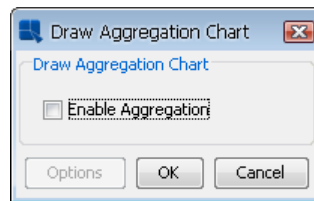


Heatmap chart with light gray colored border

4.1.4.8.9. Aggregation

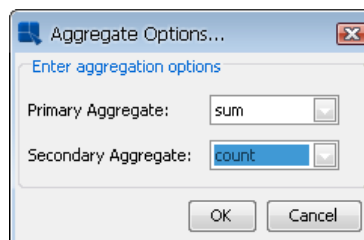
ChartDesigner allows you to aggregate data if there is more than one data point associated to a given category (and its series and/or stack, if a series and/or stack is present). This allows for a broader look at the data rather than just a single data point (out of many).

To aggregate the data, select Format → Aggregation Options. A dialog will appear allowing you to enable aggregation.



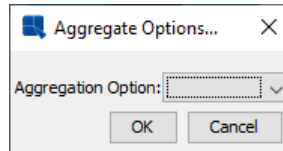
Select Aggregation Dialog

When you select *Enable Aggregation*, a second dialog box will appear, asking you which type of aggregation should be applied. You can choose from minimum, maximum, average, sum, count, first, last, sumsquare, variance, stddev, and countdistinct for the aggregates. You can specify a primary aggregate (aggregate applied to the column mapped to the primary axis) as well a secondary aggregate (aggregate applied to the column mapped to the secondary axis), if a secondary axis exists.



Aggregate Options Dialog


For heatmap charts, the aggregation option can be used when there are multiple data values at a single (x, y) point.



Aggregate Options

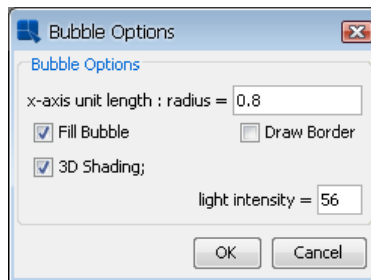
By default, no aggregation is applied to heatmap charts, and the first data value is always selected if there are multiple values at the same (x, y) point.

4.1.4.9. Chart-Specific Options

There are a number of formatting options that are unique to certain chart types. These options can be modified by selecting Format → Chart Options, or clicking the  *Chart Options* button on the toolbar. This will bring up a dialog that varies depending on the type of the current chart. Some chart types have no additional options.

4.1.4.9.1. Bubble Charts

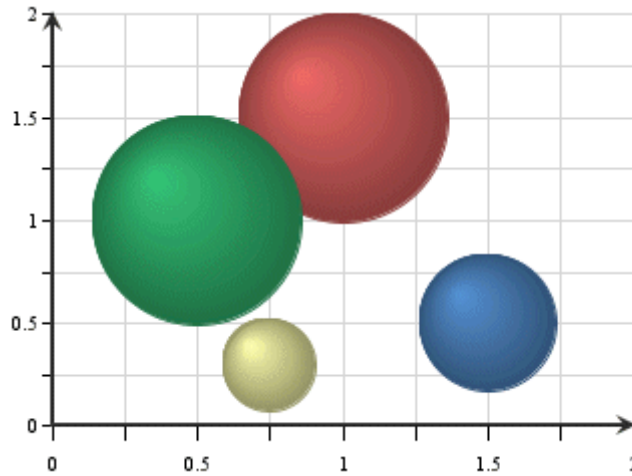
For bubble charts, the following dialog is displayed:



Bubble Options Dialog

The following options are available for bubble charts:

x-axis unit length: radius	This option specifies the ratio of X-axis unit length to the radius of the bubble.
Fill Bubble:	This option allows you to specify whether or not to fill the bubble area.
Draw Border:	This option allows you to specify whether or not to draw a border around the bubbles.
3D Shading:	This option allows you to add 3D shading to the bubbles. You can also specify the light intensity for the shading.



Bubble Chart with 3D Shading

4.1.4.9.2. Dial Charts

For dial charts, the following dialog is displayed.

Dial Chart Options

Draw Options

Starting Angle: 0.0

Ending Angle: 360.0

Center Point Radius: 0.05

Needle Style

☒ Pointed Blade

☐ Round Headed Blade

☐ Triangular Pointer

☐ Pointer

☐ Rectangular Pointer

Other Options

☒ Draw Border

☐ Draw Arrow

☒ Draw Full Circle

Needle Length and Thickness

	Length	Thickness
<input type="checkbox"/> Select All		
<input type="checkbox"/> Mon	0.5	4
<input type="checkbox"/> Tue	0.77	5
<input type="checkbox"/> Wed	0.64	2
<input type="checkbox"/> Thu	0.51	2
<input type="checkbox"/> Fri	0.39	2
<input type="checkbox"/> Sat	0.26	2
<input type="checkbox"/> Sun	0.13	2

Apply To Selected Item(s)

Length: 0.5

Thickness: 4

Dial Options Dialog

The following options are available for dial charts:

Starting Angle:

This option specifies the angle where the first axis label is to be set. This property also determines where the border and dial area will start if the *Draw Full Circle* option is unchecked. The angle is represented in degrees and is 0 by default. Assuming the dial chart is a clock face, 0 degrees is 12 o'clock.

Ending Angle:

This option specifies the angle where the last axis label should be set. This property also determines where the border and dial area will end if the *Draw Full Circle* option is unchecked. The angle is represented in degrees and is 360 by default. Hence by default the labels (and data points) encompass the entire circumference of the dial.

Center Point Radius:

This option specifies the radius for an inner circle, which starts from the center of the dial chart. The radius is specified as a ratio to the radius of the

dial plot. Hence, a value of 1 will make the inner circle encompass the entire dial. If the *Draw Full Circle* option is unchecked, only the portion of the center point that is within the starting and ending angles will be shown.



Dial Chart with Center Point Radius

Needle Style:

This option specifies the type of needle to draw.



The Pointed Blade is a smooth line with a thick base. It becomes slightly thinner as it extends outwards, but finishes with a very sharp pointed tip.



The Round Headed Blade is similar to the Pointed Blade except for a round tip.



The Triangular Pointer is sharp and resembles a very thin triangle.



The Pointer is a step ladder pointer with three segments.



The Rectangular Pointer (as shown above on the picture) is a simple straight needle.

Default needle is Pointed Blade.

Draw Border:

This option specifies whether or not to draw a border around the dial.

Draw Arrow:

This option specifies whether or not to draw arrowheads at the end of the dial hands.

Draw Full Circle:

This option specifies whether to draw the dial as a complete circle (360 degrees) or only draw the portion of the circle determined by the starting and ending angles.

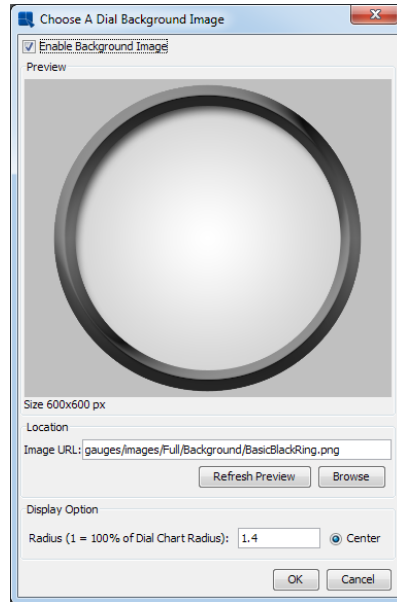
Needle Length and Thickness:

This option specifies the length and thickness of the needle. The needle length is measured from the center of the dial. The range is from 0 (center of the dial) to 1 (the end of the dial). The thickness determines the width of the needle, larger values results in a wider pointer. When creating a chart, the needle length is randomly generated. Default thickness is 2. Each category element is represented by a different needle. You can either change properties individually or change multiple categories at once. To change the property of each needle individually, directly change the values to the right of the category. To make changes to multiple needles, check each category or check the *Select All* option, set the properties in the lower right corner

and then click on the *Apply Now* button for each property changed. This will modify all checked categories to new values.

4.1.4.9.2.1. Gauge Images

Dial charts have an additional option to display a foreground or background image for the dial plot area. To add a foreground or background image, select **Insert → Dial Foreground...** or **Insert → Dial Background...**. These options are only enabled for dial charts.

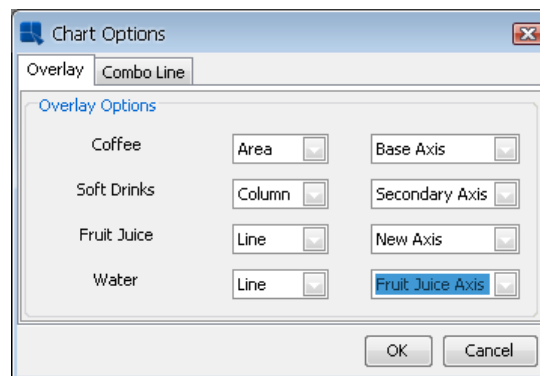


Dial Chart Background Image Dialog

Selecting an image works in the same way as the background image dialog, see Section 4.1.4.5.1.1 - Background Images. In the dial chart image dialog, there is also an option to specify the radius of the image. Specifying 1 for the radius will make the image the same width and height as the plot area. Increasing or decreasing this value will enlarge or shrink the images.

4.1.4.9.3. Overlay Charts

For overlay charts, the following dialog is displayed:

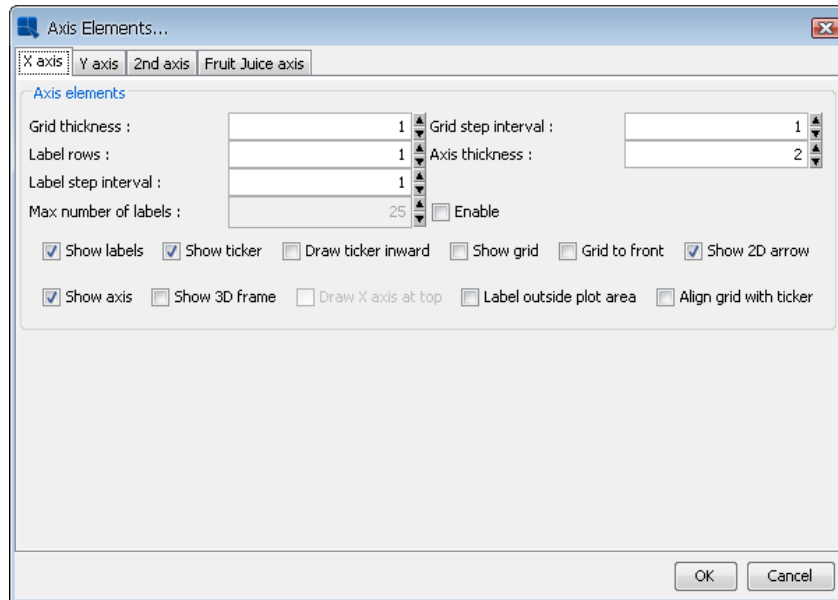


Overlay Options Dialog

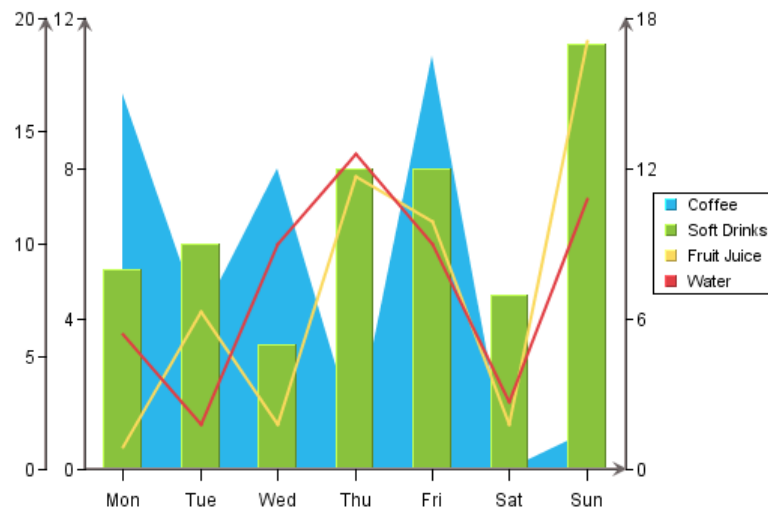
From this dialog, you can specify which chart type you would like to use for each element of the data series. Available chart types for the series elements in an overlay chart are column, area, and line. You can also choose not to display certain series elements.

From this dialog, you can also specify which axis you would like to use to plot a series element. You can place elements on the primary or secondary axes, or you can create new value axes for the series elements. To create a new value axis, select *New Axis* from the drop-down menu. Once you specify to use a new axis, a new option will be added to the drop-down menus for the other data series elements, allowing them to be drawn on the same axis

that you previously specified. Using a variety of axes allows you to precisely tune the scale that the different series elements use. Each of these axes will have its own tab in the Axis Elements window, where you can change the label step interval for each axis independently of the others.

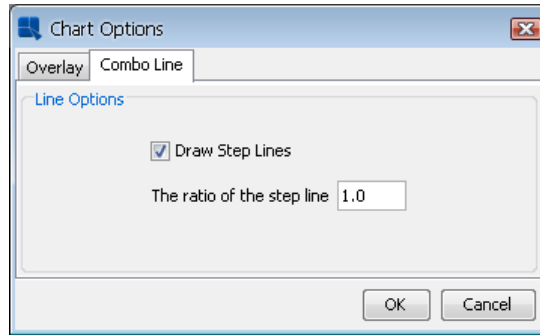


Axis Options Dialog for Multiple Value Axes



Overlay Chart with Multiple Value Axes

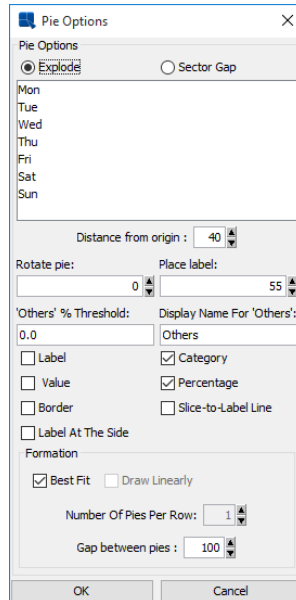
If one of the chart types used by the overlay chart is a line, you can specify whether or not to draw the line as a step line using the *Combo Line* tab. For more about step lines see Section 4.1.4.9.5 - Line Charts:



Combo Line Options for Overlay Charts

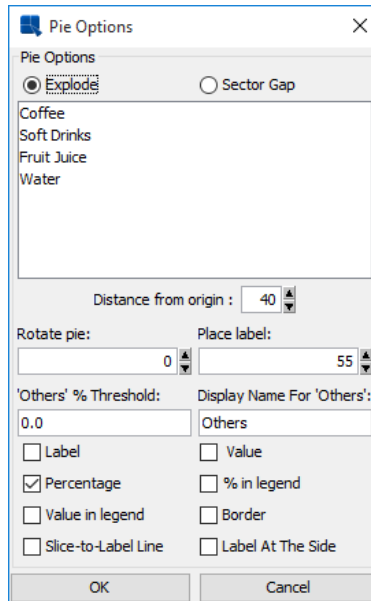
4.1.4.9.4. Pie Charts

For pie charts, the following dialog is displayed. (Note that different options will appear/disappear depending on whether the chart has a series, and if it's a 2D or 3D chart.) Here are the options available for a 2D chart with series:



Pie Options Dialog (With Series)

Here are the options available for a 2D chart without series. Notice that the formation options are removed and the % in legend and Value in legend options are added:

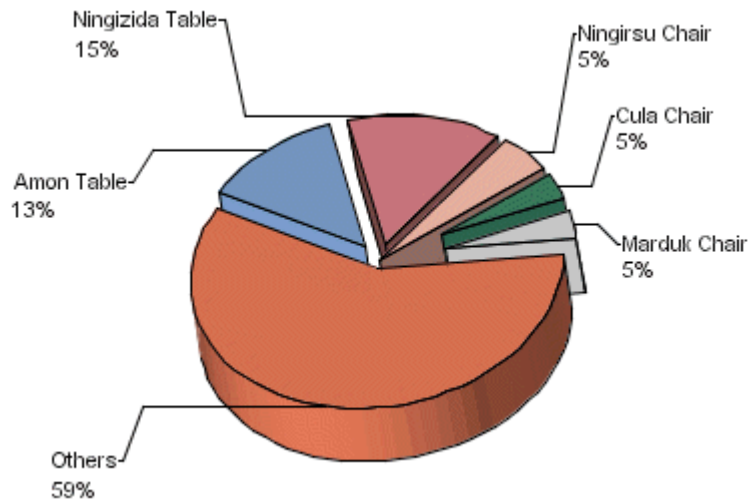


Pie Options Dialog (Without Series)

The following options are available for pie charts:

- | | |
|-----------------------------------|---|
| Explode: | This option allows you to pick one or more category/series elements whose sections are to be drawn at a certain distance away from the center of the pie. |
| Sector Gap: | This option allows you to pick one or more category/series elements whose sections are drawn at a certain distance away from the center of the pie and still maintain the same distance between pie slices and a circular boundary. |
| Distance from origin: | This option allows you to specify how far the exploded/sector gap sections are to be drawn away from the center. This number, represented as a percentage of the radius, indicates the distance between the center and the tip of the pie slice to be exported. |
| Rotate pie: | This option allows you to specify the number of degrees that the chart should be rotated in a clockwise direction. Available values are between 0 and 360 |
| Place label: | This option indicates the distance of the labels from the center of the pie. The position of an individual label can also be adjusted by dragging the text. |
| “Others” % Threshold: | This feature is useful for pie charts that have a large number of small categories. Rather than draw a slice for each category, users can select a threshold value. Any category whose percentage of the value column is less than the threshold value will be lumped into an “Others” slice. |
| Display Name for “Others”: | This option allows you to set the display name for the “Others” slice that is created for categories that fall below the supplied threshold value. This label will appear in the legend, and/or for the slice label. |
| Label: | This option determines whether a category/series label should be drawn for each pie slice. By default, these only appear as legend items. Note that the label will not appear if the data for the slice is 0 or null. |
| Value: | This option allows you to specify whether to display the actual value of each pie slice. Note that the value will not appear if the data for the slice is 0 or null. |
| Category: | This option allows you to specify whether to display the category of each pie slice. |

Percentage:	This option allows you to display the percentage for each pie slice. The percentages are calculated by dividing the value of each section by the sum of all the values. Note that the percentage will not appear if the data for the slice is 0 or null.
% in legend:	This option allows you to display the percentage represented by each slice in the pie in the legend. This can be a preferable presentation if the pie slices become too thin. This option is only available if the pie chart does not have a data series.
Value in legend:	This option allows you to display the value represented by each slice in the pie in the legend. This can be a preferable presentation if the pie slices become too thin. This option is only available if the pie chart does not have a data series.
Border:	This option specifies whether to draw a border around each pie slice. This option is only available for two-dimensional pie charts. For three-dimensional pies, you can use the border drawing option on the navigation panel. Note that the border will not appear if the data for the slice is 0 or null.
Slice-to-Label Line:	This option will draw a line from any label(s) to it's corresponding pie slice. Note that the slice-to-label line will not appear if the data for the slice is 0 or null.
Label at the Side:	This option will place labels for the pie chart away from the plot around the outside of the chart. When used with the <i>Slick-to-Label Line</i> option, it gives users a way to display the pie labels for charts with many small categories without any text overlapping. Note that the label will not appear if the data for the slice is 0 or null.



Pie Chart with Side Labels and Lines

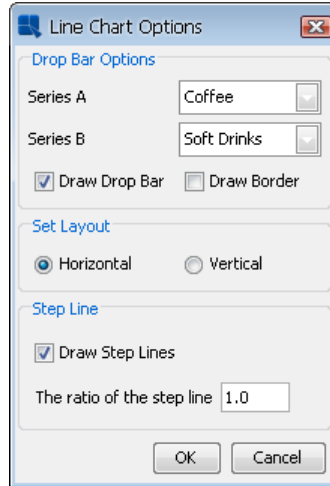
Best Fit:	This option will arrange multiple pies in best configuration to fit the chart canvas. It's only available for pies with data series.
Draw Linearly:	This option will arrange multiple pies in a straight horizontal line. It's only available for pies with data series.
Number of Pies Per Row:	This option allows you to create a custom arrangement of multiple pies, by specifying the number of pies to draw in each row of the arrangement.

Gap between pies:

This option allows you to specify the gap between the multiple pies. The number is a multiple of the pie radius, so the gap will adjust with the size of the chart plot.

4.1.4.9.5. Line Charts:

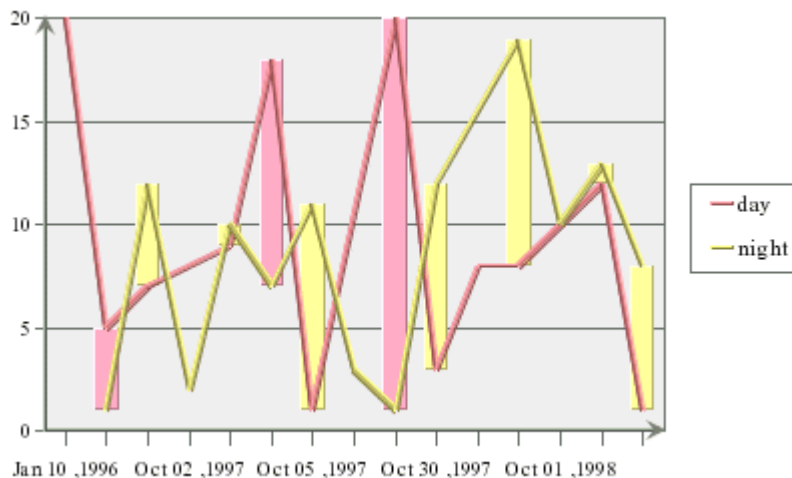
For two-dimensional line charts, one of two different dialogs will be displayed depending on whether the chart has a data series or not. If the chart has a data series then the following dialog is displayed.



Line Options Dialog (with series)

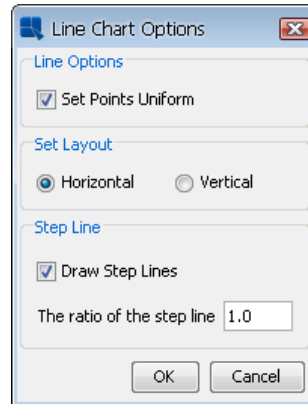
Line charts with data series have a specific option that allows you to draw drop bars between two series elements. The dialog options are as follows:

- Series A:** This option specifies the first series element for the drop bar.
- Series B:** This option specifies the second series element for the drop bar.
- Draw Drop Bar:** This option specifies whether or not to draw drop bars.
- Draw Border:** This option specifies whether or not to draw a border around drop bars.
- Set Layout:** This option specifies whether to draw a line chart in vertical or horizontal orientation.
- Step Line:** This option allows you to draw line chart as a step line. You can also specify the step line ratio to use.



Line Chart with Drop Bars

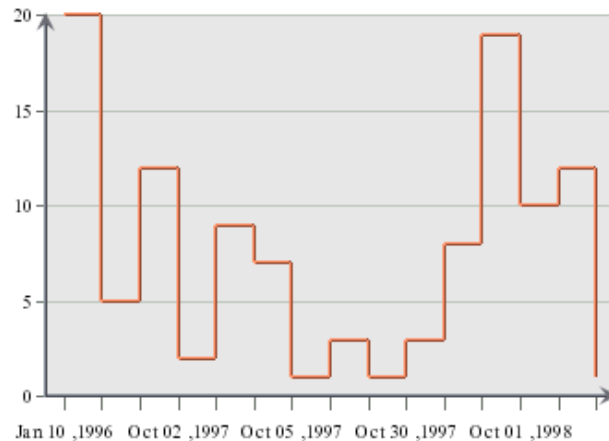
Note that the color of the drop bar will vary depending on which series has the higher value for a given point. If the line chart does not have a series, then the following dialog will appear.



Line Options Dialog (without series)

The dialog options are as follows:

- Set Points Uniform:** This option specifies whether the point shapes and colors are uniform or not. Un-checking this option allows you to set multiple colors and point shapes for the data points. The points can be customized in the line and point dialog.
- Set Layout:** This option specifies whether to draw the line chart in vertical or horizontal orientation.
- Step Line:** This option allows you to draw the line chart as a step line. You can also specify the step line ratio to use.



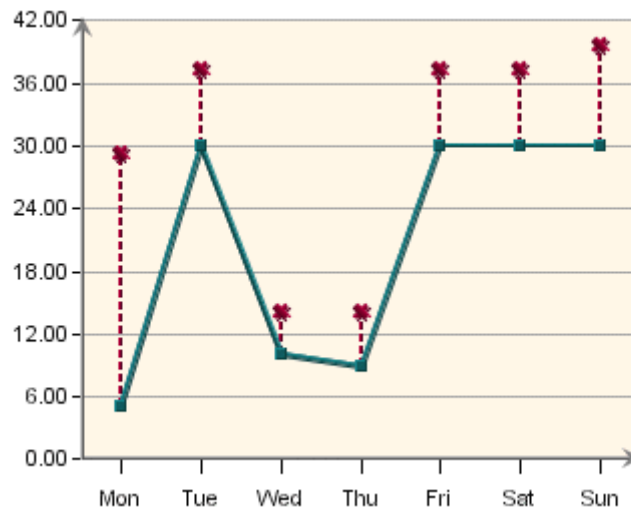
Line Chart with Step Lines

The step line ratio allows you to specify how far between points the horizontal portion of the step line should be drawn. A ratio of 1 draws the line horizontally to the next point on the chart and then draws vertically to that point, while a ratio of 0.5 draws the horizontal portion halfway between the two points. A ratio of 0 will result in the vertical portion of the line drawn first and then connect to the next point horizontally. Values for the ratio are between 0 and 1.

There are no additional options for three-dimensional line charts.

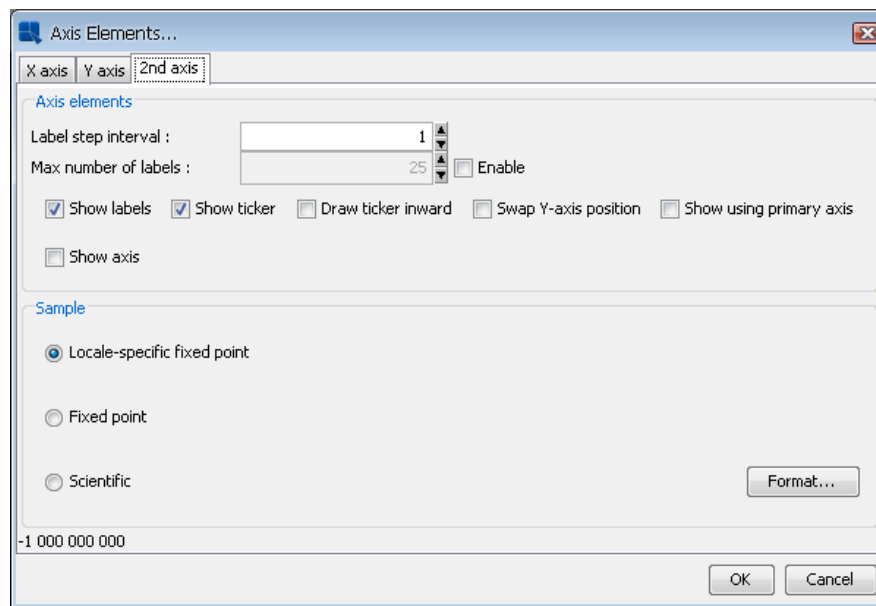
4.1.4.9.5.1. Double Value Line Charts

ChartDesigner contains a special option for line charts that allows you to have two values shown for the same line. Here the secondary axis is used to plot the second value (as in a line-line combination) and then combined with the line on the primary axis.



Double Value Line Chart

To create a double value line chart, design a line-line combination chart (a line chart with primary and secondary values). Then select Format → Axis Elements. This will bring up the axis elements dialog.

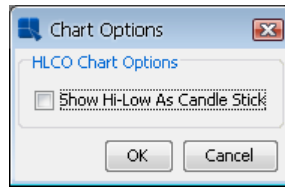


Axis Elements Dialog for Line-Line Combination Charts

Under the 2nd Axis tab, there is a checkbox marked *Show using primary axis*. Check this box and click on the OK button. Your chart will now be drawn as a double value line chart.

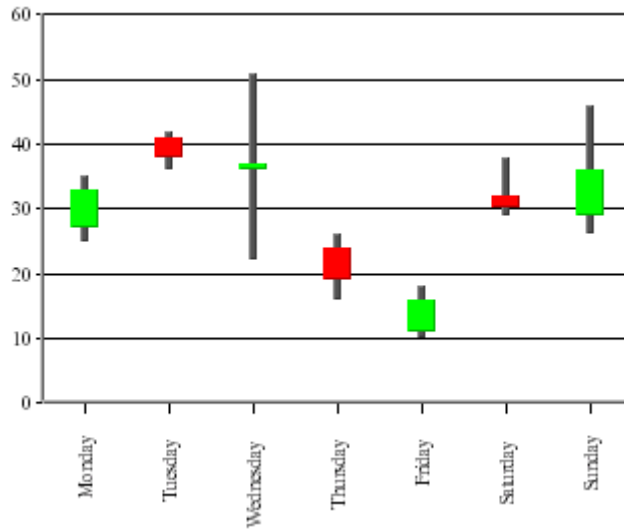
4.1.4.9.6. HLCO Charts

For HLCO charts, the following dialog is displayed:



HLCO Options Dialog

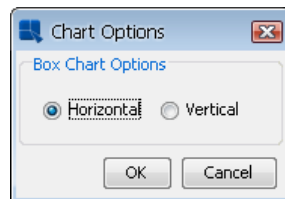
The *Show Hi-Low As Candle Stick* option will turn the HLCO chart into a candle representation. A candle HLCO chart blends high, low, close, and open data into a single object that resembles a candlestick.



HLCO Candlestick Chart

4.1.4.9.7. Box Charts

For box charts, the following dialog is displayed:

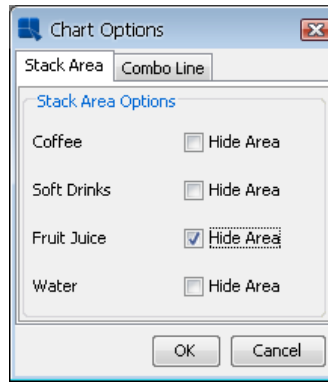


Box Options Dialog

This dialog allows you to specify whether to display the box chart in a horizontal or vertical orientation.

4.1.4.9.8. Stack Area Charts

For stack area charts, the following dialog is displayed:

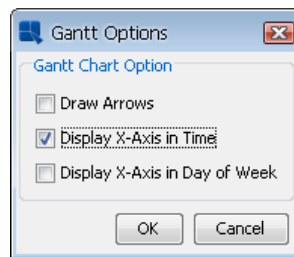


Stack Area Options Dialog

You can choose to hide any of the stacks in the chart by clicking on the corresponding check box. The *Combo Line* tab allows you to specify step lines if the chart is a line stack area combination. There are no additional options for three-dimensional stack area charts.

4.1.4.9.9. Gantt Charts

For Gantt charts, the following dialog is displayed:

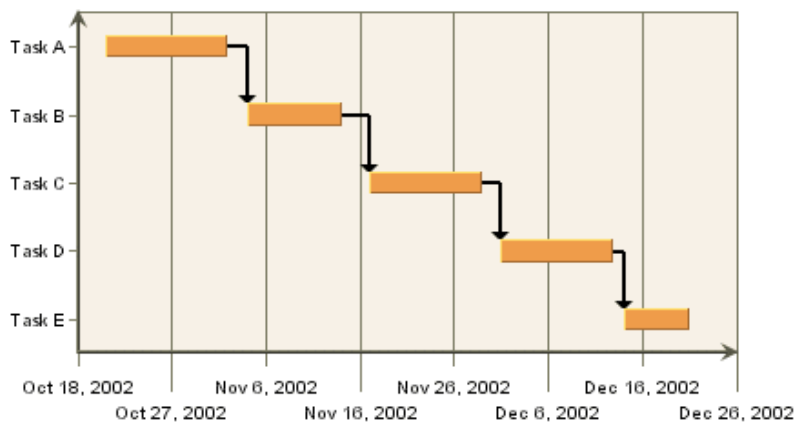


Gantt Options Dialog

The following options are available for Gantt charts:

Draw Arrows:

This option will draw connecting arrows between category elements of the Gantt chart. This allows you to illustrate a sequence between scheduled events. The arrows are drawn in the order the category elements appear in the data source.



Gantt Chart with Arrows

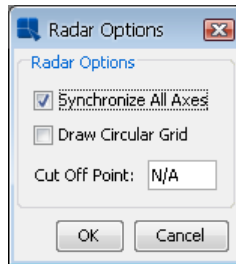
Display X-Axis in Time:

This option shows the ticker labels as time values instead of numeric values for the X-axis.

Display X-Axis in Day of Week: This option shows the ticker labels as days of the week with the date for each Sunday shown as well.

4.1.4.9.10. Radar Charts

For radar charts, the following dialog is displayed:



Radar Options Dialog

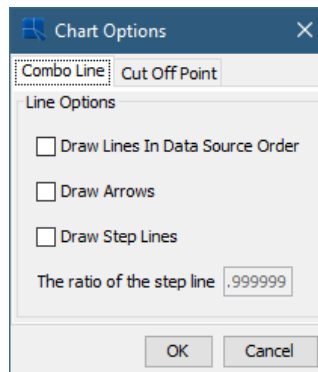
By default, the scale is same for all axes in the radar chart. Unchecking the *Synchronize All Axes* option will allow each axis in the radar chart to be scaled independently. You can select to use auto-scaling for each axis, or you can set the scales manually by invoking the axis scale dialog.

The second option allows you to set how the grid is drawn for the radar chart. By default, if the grid is enabled, it is drawn in straight lines that connect the tickers on each axis. Enabling the *Draw Circular Grid* option will draw the grid in a circle, similar to the polar chart grid.

The third option allows you to specify a cut-off point for the data points (areas) in the radar chart. You can enter a maximum value that should be shown in the chart. The areas bounded by the data points will not be drawn beyond the specified cut-off point.

4.1.4.9.11. Scatter Charts

For scatter charts, the following dialog is displayed:



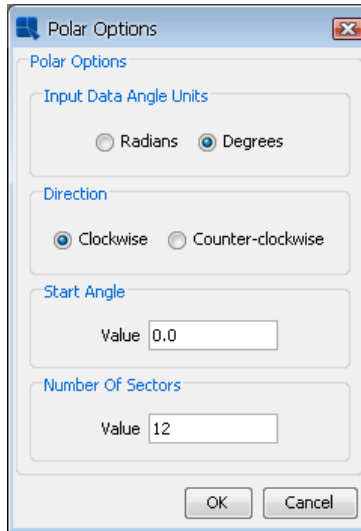
Scatter Options Dialog

The *Combo Line* tab allows you to set Draw Lines In Data Source Order, Draw Arrows on lines or Draw the connecting lines as a Step Lines, as well as specify the step lines ratio.

The *Cut Off Point* tab allows you to specify a maximum value for the Y point of the scatter coordinates. Any coordinates that fall beyond this threshold will not be plotted. Connecting lines will draw up to the edge of the threshold and continue to the next data point.

4.1.4.9.12. Polar Charts

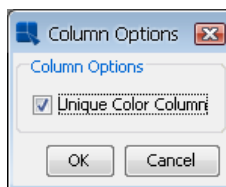
For polar charts, the following dialog is displayed:

*Polar Options Dialog*

- Scale:** This option allows you to specify whether the input data for the angle (θ) portion of the data points is in radians or degrees. The chart will always display angles from 0 to 360. If the input data is in radians, it will be displayed as degrees.
- Direction** This option allows you to specify whether the circular plot should be drawn clockwise or counter-clockwise.
- Start Angle:** By default, the top of the polar chart plot is 0 degrees. This option allows you to specify a different angle for the top of the plot. The argument for this angle is supplied in degrees or radians, depending on the scale you have chosen.
- Number of Sectors:** This option allows you to select number of sectors you'd like to show in the chart. Sectors are created by drawing additional polar axis lines at specified angle intervals. By default, four sectors are shown.

4.1.4.9.13. Column Charts with Series

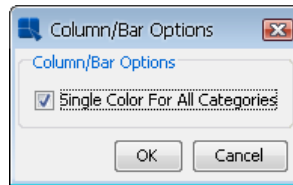
For column charts with data series, the following dialog is displayed:

*Column Options Dialog*

Normally, when column charts have data series, each series has its own color that is applied for every category in the chart. If you want to assign different colors to the columns in the chart regardless of the series, you can enable the *Unique Color Column* option in this dialog. When it's turned on, you can set color for each column in the chart individually.

4.1.4.9.14. Column/Bar Charts without Series

For column/bar charts without data series, the following dialog is displayed:

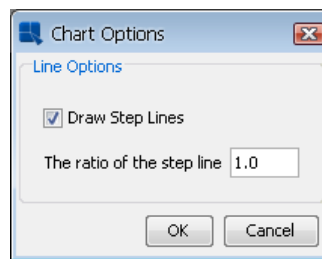


Column/Bar Options Dialog

Normally, when column/bar charts don't have data series, all categories in the chart have single color. If you want to assign different colors to the categories in the chart, uncheck the *Single Color For All Categories* option in this dialog.

4.1.4.9.15. Two-Dimensional Line Combination Charts

For any other two-dimensional line combination chart, the following dialog is displayed:

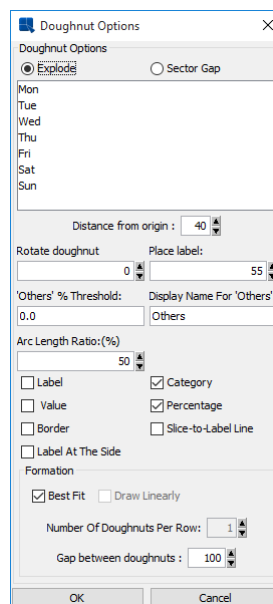


Line Combination Chart Options

This dialog allows you to specify whether to draw the combo line as a step line, as well as specify the step line ratio.

4.1.4.9.16. Doughnut Charts

The chart options for a doughnut chart are almost exactly the same as the options for a pie chart. You can refer to the options under Section 4.1.4.9.4 - Pie Charts for more details.



Doughnut Chart Options

The only option unique to doughnut charts is the Arc Length Ratio (%) which specifies the size of the hole at the center of the chart. A higher number results in a smaller hole.

4.1.5. Chart Drill-Down

Like with reports, it is also possible to add layers of drill-downs to charts. All of the drill-downs added will be applied to both data points on the plot area and their respective fields in the legend. Using chart drill-down, you can create a top-level chart that displays summarized data and allows users to click through specific data points to see underlying details.

The charting engine in EDAB contains several different built-in drill-down mechanisms that allows you to create only one chart template for each level of drill-down. All of the drill-down options are available from the Drill-Down menu in Chart Designer.



Warning

It is important to note that you can only use drill-downs in charts when the chart is deployed independently from the report (in the Chart Viewer, its own servlet, etc). If a chart template that includes drill-down is placed in a report, only the top-level chart will display. Users will not be able to click through to the lower-level charts.

4.1.5.1. Data Drill-Down

Data drill-down allows you to group and display information that is based on a single data source. The advantage to this form of drill-down is that it works with any data source. However, it does not allow you to display loosely related information because all levels of drill-down will share the same value column from the input data.

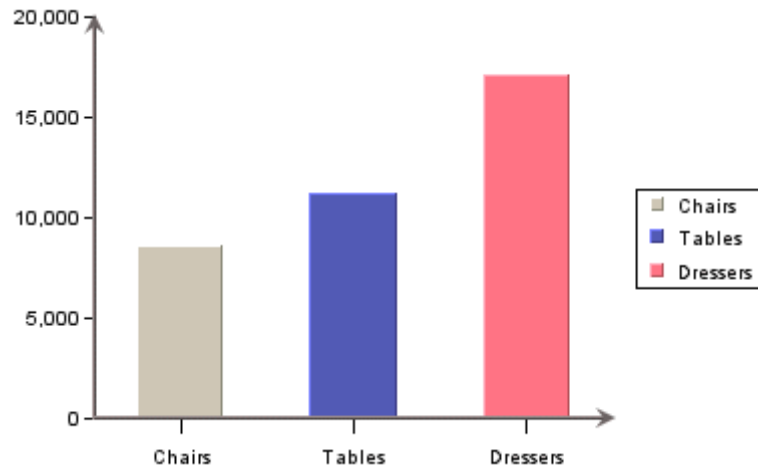
For example, assume you have the following table as the chart data:

Category	Product	Sales
Chairs	Elm Arm Chair	\$8,216
Chairs	Pine Side Chair	\$7,611
Chairs	Redwood Arm Chair	\$8,625
Tables	Elm Round Table	\$10,241
Tables	Pine Oval Table	\$9,663
Tables	Oak Oval Table	\$11,261
Dressers	Oak Single Dresser	\$16,442
Dressers	Elm Double Dresser	\$17,148

Using this data, you could create a top-level chart that displays total sales for each distinct product category and then create a lower-level chart that shows individual sales for each product in a category. The number of drill-down levels available depends on the number of groupings that are present in the input data.

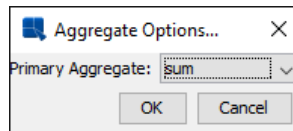
4.1.5.1.1. Adding Data Drill-Down

To add layers of data drill-down to a chart, you must first create a top-level chart. In the above example, we would create a chart with “Category” mapped to the category axis and “Sales” mapped to the value axis. In a column chart, it would look like this:



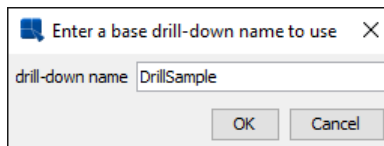
Top-Level Chart

To add a layer of drill-down, select Drill-Down → Add. This will bring up a dialog prompting you to specify the aggregation you want to use for the value axis. For example, selecting sum will display the total for each category in the top-level chart. Available aggregate functions are minimum, maximum, average, sum, count, first, last, sumsquare, variance, stddev, and countdistinct.



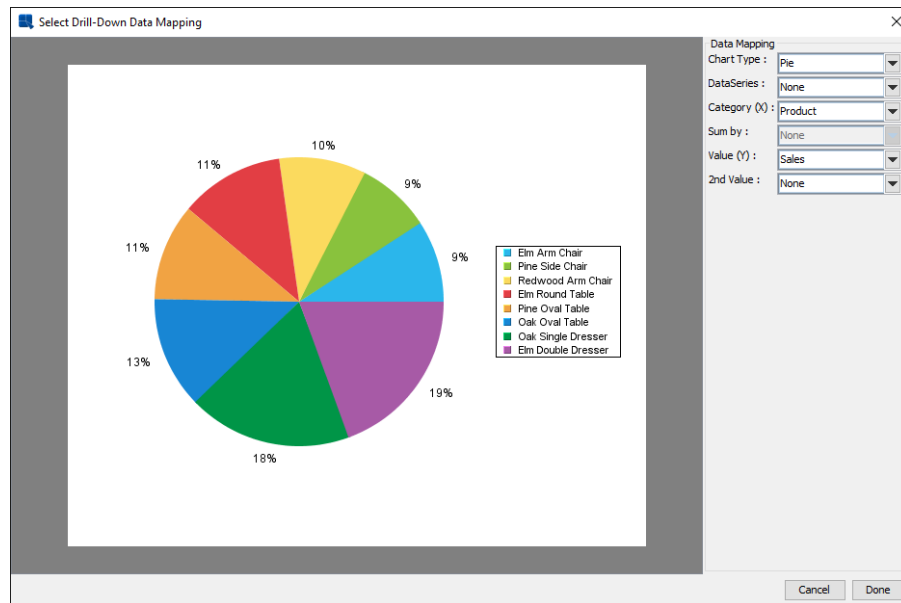
Aggregation Dialog

After you select the aggregation you want to use, click on the *OK* button. A new dialog will appear prompting you to specify a name for the drill-down chart. This name will be assigned to the chart templates that are created by the drill-down process. Note that all sub-level drill-down templates are saved in the `/drilltemplates/` directory.



Drill-Down Name Dialog

Once you have specified a name you want to use, click on the *OK* button. You will then be prompted to specify the chart type and data mapping for the sub-level chart.



Data Drill-Down Mapping Dialog

The options in the dialog are similar to those for normal data mapping. The major difference is that the first option allows you to select a chart type. Available chart types for data drill-down are column, bar, line, stack column, stack bar, pie, area, doughnut, overlay, radar, and dial. The data mapping options will change depending on the type of chart that you select.

Once you have finished specifying the mapping options, click on the *Done* button and you will go back to the Chart Designer where you can customize and modify your sub-level chart. You can add additional layers of drill-down by selecting Drill-Down → Add again.

You can navigate to a particular drill-down chart by selecting Drill-Down → Previous or Drill-Down → Next or by double clicking a data point. The *Previous* selection takes you up a level, while the *Next* selection goes down a level using the left-most category as the data to be drilled on. You can also go from any drill-down chart to the top-level chart by selecting Drill-Down → Go To Top Level. You can customize (i.e. assign colors, add title, axis labels, background image, ...etc.) the drill-down chart in the same way as the top-level chart. Everytime you change and leave a level, you will be prompted to save the chart. Make sure that you answer *yes* if you want the attributes of the drill-down level to be saved, otherwise you will lose all the changes.

You can insert a drill-down chart between two other drill-down charts by going to the higher level drill-down chart and selecting Drill-Down → Add. The drill-down chart that you create will be inserted between the two previous drill-down charts.

You can remove a level of the drill-down by navigating to that level and selecting Drill-Down → Remove This.

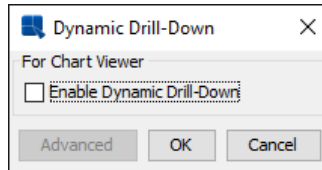
The entire drill-down chart can also be deleted by selecting Drill-Down → Remove All. Selecting Drill-Down → Previous brings the drill-down chart to a higher level, which is the same as right clicking and selecting *Back* from the pop-up menu. The *Next* selection brings the drill-down chart to a lower level, which has the same function as double clicking on an item in the chart.

Once you have finished creating and editing all the levels of the drill-down chart, you can save it by navigating to the top-level chart and selecting File → Save or File → Save as.

4.1.5.2. Dynamic Data Drill-Down

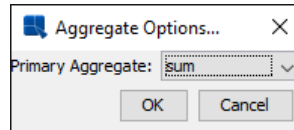
Usually, mapping and drill options for data drill-down are fixed during design time. Dynamic drill-down is an additional option that allows you to select the mapping. The only thing specified during design time is the top-level chart and aggregation.

To create a chart with dynamic drill-down, first create a chart you want to use as the top-level chart (for example, you can create the same top-level chart as before) and then select Drill-Down → Dynamic. A dialog will appear allowing you to enable dynamic drill-down.



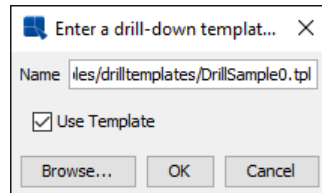
Enable Dynamic Drill-Down Dialog

Check the *Enable Dynamic Drill-Down* box and a new dialog will pop-up prompting you to select the aggregation.



Aggregation Dialog

Options for this dialog are the same as for regular data drill-down. Once you specify the aggregate you want to use, click on the *OK* button. Another dialog will pop-up, allowing you to specify a template you want to use for the sub-level charts.



Select Dynamic Drill-Down Template Dialog

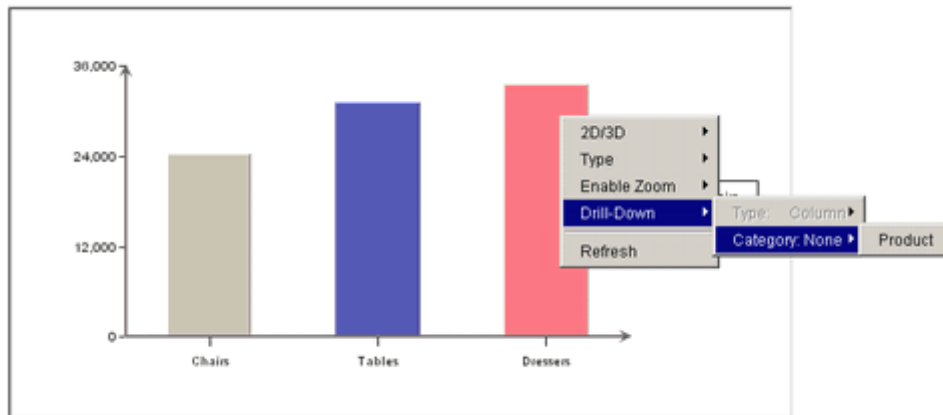
If you do not select to use a template, the sub-level charts will be generated using default appearance properties. After you finish selecting these options, your chart will change to display aggregated data on the value axis. You can change the option by selecting *Drill-Down* → *Dynamic* again and then clicking on the *Advanced* button.

Dynamic drill-down charts can only be viewed in the Chart Viewer JNLP (More about Applets in JNLP: Section 1.3.4 - Run Applets in WebStart with JNLP file). There you can configure setting of the next drill-down chart by right clicking on the desktop area which will bring up a pop-up menu. If the dynamic drill-down is enabled and the chart still has some unused fields (columns) to plot, the item *Drill-Down* will be displayed in the pop-up menu. Upon selecting *Drill-Down* in the pop-up menu, you can view the current setting for next drill-down. If *Category* is **None**, it means you have not configured the next drill-down chart yet. To configure the drill-down chart for next level, you must select *Category* (*Type*, *Series* and *SumBy* can be selected once *Category* has been set). After you create a drill-down chart, you can navigate to different levels in the Chart Viewer by either left clicking on a data point if you want to go down one level, or by right clicking and selecting *Back* from the pop-up menu if you want to go up one level, or by repeating the previous step to create another drill-down chart for next level.



Tip

After traversing to a lower-level of drill-down, right clicking on a data point will navigate you back to the top-level chart. To bring up the pop-up menu, right click on the chart canvas away from the chart data points.



Dynamic Data Drill-Down in Chart Viewer

4.1.5.3. Parameter Drill-Down

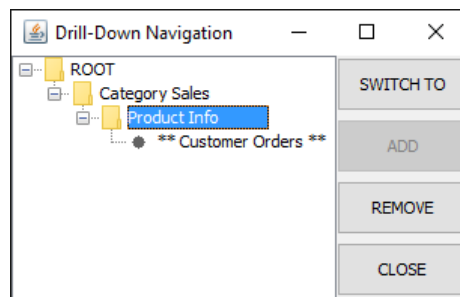
The third type of a drill-down you can use for charts is a parameter drill-down. Parameter drill-down is the most flexible implementation because you can relate the data between drill-down levels in any way you like. Specifically, you do not need to use the same value element for each level. Instead, parameter drill-down uses parameterized query feature to relate various chart levels and since it uses queries, the data source for the sub-level charts must be a database or parameterized class file.

For example, using our previous scenario, rather than always looking at sales, you want your top level chart to look at aggregated sales by category (same as before), then on the next level, you want to look at sales volume for each product, and from there, you want to look at inventory levels for each product by region. This can be accomplished with parameter drill-down.

With parameter drill-down, the category value of an element you click to drill on will be passed to the sub-level chart as the parameter value. Hence, if you click on the “Chairs” column, the value of “Chairs” would be passed to the query. Therefore, anything in the database that could be filtered by category name could be retrieved.

4.1.5.3.1. Adding Parameter Drill-Down

To add and edit various layers of a parameter drill-down, select Drill-Down → Parameter Drill-Down. This will bring up a navigation window that shows various levels of drill-down.

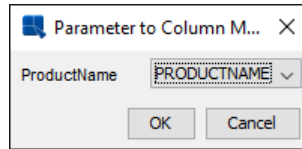


Parameter Drill-Down Navigation Window

Left-hand side of the navigation window displays the hierarchy of drill-down levels. The “ROOT” node is the top-level chart. The level you are currently editing is marked with **. To edit a different level, select it and click the *SWITCH TO* button on the right-hand side. The chart will then open in the Designer. Levels of drill-down can also be removed by selecting the node in the Navigation window and then clicking on the *REMOVE* button.

To add a new level of drill-down, select the chart under which you would like to add the layer (if you have only the top-level chart, then only the “ROOT” node will be visible), and click *ADD*. You will then be prompted to create a new chart or to use an existing chart for the drill-down layer. You can use any existing chart; however, any chart for a drill-down layer must have a parameterized query as the data source. If you select to create a new chart, the Data Source Manager will reopen, allowing you to select a data source for the chart and to continue through the Chart Wizard steps.

The next step is to map the category and/or series columns from the top-level chart to the query parameter(s) in the sub-level chart. You will be prompted to do this when you select an existing chart for the drill-down layer or when you select the data source for a new chart for the drill-down layer. In either case, a dialog will appear prompting you to map the fields.



Parameter Mapping Window

Available options in the drop-down menus are based on data type. For example, if your parameterized query has string as the parameter, only fields containing string data can be mapped. Hence, it is important to consider what type of data will be passed from the top-level chart to the lower-level charts because if your category or series are not of the correct data type or if you do not have enough fields to pass to the lower-levels, the drill-down will not work correctly.


Once you correctly specify the parameter mapping, you will be prompted to specify a display name for the drill-down level. After you select a name, the sub-level chart will appear in the Designer, allowing you to customize it. You can navigate through the layers of drill-down using the Navigation window, or like data drill-down, you can double click on a data point to go down a level and right click and select *Back* from the pop-up menu to go up a level.

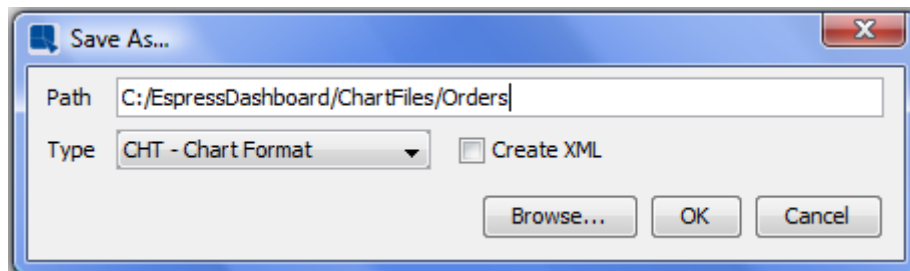
4.1.6. Saving & Exporting Charts

After you finish designing a chart, you can save the definition as a chart or as a chart template, and provide XML definitions for both. You can also generate a number of static image exports of the chart or create a JSP/JNLP page with the Chart Viewer JNLP (More about Applets in JNLP: Section 1.3.4 - Run Applets in WebStart with JNLP file).

4.1.6.1. Saving Charts

If your chart uses an independent data source from the report, it can be saved in any place with any filename. To

save the current chart, select File → Save or File → Save As, or click on the  *Save* button on the toolbar. Assuming you have not saved the chart before, or selected File → Save As, the following dialog will appear:



Save As Dialog

The first option allows you to specify a name and file path for the saved chart. You can browse to the appropriate file path by clicking on the *Browse* button at the bottom of the dialog. The second option allows you to specify which format you would like to use when saving the chart. As detailed in Section 4.1.2 - Charting Basics, there are two principle ways in which chart definitions can be saved.

Chart format: Chart files save the chart in a binary file called `filename.cht`. A chart file stores both the definitions of the chart (type, dimension, etc), as well as the data that was used to create the chart.

Template format: Template files save the chart in a binary file called `filename.tpl`. A template file only stores chart definitions. It doesn't store any chart data. Hence, any time a template file is opened, it will try to connect to the original data source to retrieve the data.

PAC format: PAC files save the chart in such a way that makes them ready for deployment. A .PAC file takes the chart and all the supplementary files associated with it, such as background images, dynamic drill-downs, or parametric drill-downs, and places them in a single binary file.

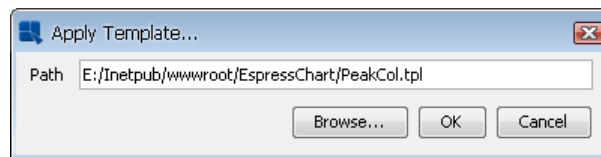
Once you select a format you want to use, click on the *OK* button and the chart will be saved.

4.1.6.1.1. Working with Templates

Chart templates are a special format in which chart definitions can be saved. Unlike chart files (.cht format) which saves both the chart definitions as well as the data for the chart, templates only save the definitions (i.e. the chart attributes and the layout of the individual components), and the data source information. This means that templates store the location/connection information for the data source that was used to create the chart, but they store none of the actual data in the file. (Templates do keep 10 records of back-up data, allowing them to be opened when the data source is not present).

Template files can be used in two ways. First, it can be opened, viewed, or exported, allowing you to see a chart with fresh data. This way data for the chart are retrieved based on the data source specified in the file. If the original data source is not accessible, this method cannot be used. The second way in which templates can be used is to apply its attributes to other charts. In this scenario, the chart to which the template is applied will inherit appearance properties of the template (including colors, fonts, size of chart components, position of legends, etc). This feature allows you to produce a consistent look and feel among charts.

You can apply a template in Chart Designer by selecting File → Apply Template. This will bring up a dialog prompting you to specify the template file you want to use.



Apply Template Dialog

The dialog allows you to specify the template file and its location. You can browse to a file by clicking on the *Browse* button.

Note that when the chart and the template being applied are different sizes (i.e. chart canvas size), the resulting chart may not display correctly. This is because the text size will not change between template and chart to which it is applied. While the other components will adjust to the size of the new canvas, the font will not. To keep a consistent look, the size of your template should be close to the size of the chart to which it will be applied.

Hyperlinks, floating lines, floating text, and axis scales defined in the template files carry over. You may have to redefine them in the chart, if necessary. Chart type and dimension are not modified by the template. For example, a three-dimensional chart will not be changed to a two-dimensional chart if the template is a two-dimensional chart. Likewise, a pie chart remains a pie chart when a template of a bar chart is applied. Although the chart type does not change when you apply a template, some appearance properties will not translate very well from a template with another chart type. For best results, try to apply templates of the same type and dimension to a chart.


4.1.6.1.2. Saving XML Templates

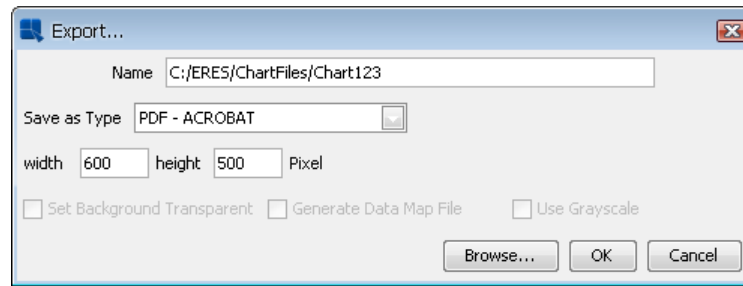
In addition to the two binary chart representations, you can also save the chart definitions in XML format. This option allows you to have a text-based chart template that can be used to modify chart properties outside of Chart Designer.

To create an XML file when saving the chart, check the *Create XML File* box in the save as dialog when saving the chart. This will create an XML file for the chart. Note that the XML file is a representation of the .tpl format and not the .cht format.

4.1.6.2. Exporting Charts

From the Chart Designer, you can generate a number of static image exports for your chart. To export the current

chart, select File → Export, or click the  *Export* button on the toolbar. This will bring up a dialog allowing you to specify options for the generated file.

*Export Chart Dialog*

The first option allows you to specify a name and the file path for the generated file. You can browse to the appropriate file path by clicking the *Browse* button at the bottom of the dialog. The second option allows you to select what type of export you would like. The following options are available:

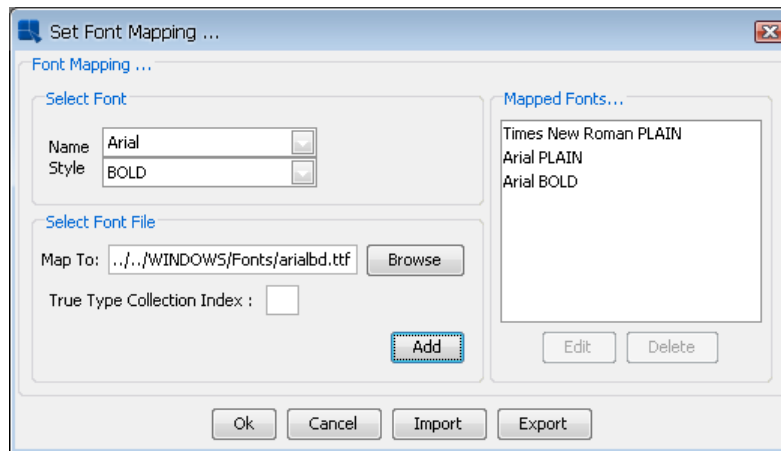
- GIF** EDAB can generate GIF images. GIF has 256 color limit which keeps image file sizes small.
- JPEG** JPEG is another popular image format. It is a higher resolution image format than GIF and it is not patent protected. When generating a JPEG file, you can specify quality/compression of the file. The higher the quality, the larger the file.
If you select JPEG as the export format, after clicking *OK* you will be prompted to specify the quality. The higher quality image you select, the larger the generated file size. It is recommended that with the JPEG export, you use specify a high-quality image, as the low-quality results will most likely be undesirable.
- PNG** PNG is an image format that is less popular, but can be displayed in most browsers. It is a high-quality image with a smaller file size than JPEG. It also provides transparent image background option.
- SVG** SVG (Scalable Vector Graphics) is a relatively new image format, which saves the image as vectors in an XML-based text format. Generally, you will need a browser plug-in to view these images.
- SWF** SWF is an Adobe Flash file. The flash format is vector based and allows the chart to be resized after export. Also, flash allows for high-resolution printing and produces a small file size. When selecting this export type, you can also specify the frame count (number of frames in the animation) and the frame rate (number of frames shown per second) or choose to disable the animation.
- BMP** This is a Windows bitmap format.
- WMF** WMF is the Windows Meta File format. This can be used for import/export into MS Office documents.
- PDF** This will generate the chart in Adobe Portable Document Format. The chart will be generated as a one-page PDF document.
- XML** This will generate an XML data file containing the chart's data. This will not generate an XML chart attribute file, only the data will be written into XML format.
- XLS** Will generate an XLS (MS Excel) file and insert the chart as an image to the first XLS sheet.
- TXT** This will generate a text data file containing the chart's data.

In addition, if your chart contains hyperlinks, you can choose to export a map file along with any of the generated images. The map file contains an HTML image map with the links for the generate chart. Map files are generated in the same location as the image file, and have the same file name. To generate a map file, check the *Generate Data Map File* box prior to exporting.

4.1.6.2.1. PDF Font Mapping

As with reports, you can use any font in the system for the labels, text, and titles in a chart. For most image exports, the text is written in the image and no additional configuration is necessary. For PDF export, however, you will need to specify a .*ttf* (true type font), .*ttc* (true type collection), .*pfb*, or .*afm* file for any system font you want to use in the chart.

To set font mapping for PDF export, select Format → Font Mapping. This will bring up a dialog allowing you to specify font files.



Font Mapping Dialog

For each font and style combination, you can select a specific .ttf, .ttc, .pfb, or .afm file for that font. You can either type the full path or browse to the font file. If you are using a .ttc file you will need to specify the font index in the box provided (.ttc files contain more than one font). Once you have specified the correct file, click the 'Add' button to save the mapping to the list. You can edit or delete existing mappings by selecting them in the list and then clicking the appropriate button.



Note

You only need to apply font mapping when exporting the chart by itself. If the chart is in a report, it will inherit the font mapping defined in the report.

4.1.6.2.1.1. PDF Font Mapping Import/Export

You can pass the font mapping from one chart to another using the Import/Export feature. You can export the font mapping by clicking on the *Export* button on the font mapping dialog. This will bring up a dialog box prompting you to specify a file name. The font mapping will then be saved as an XML file. You can load a font mapping XML file by selecting the *Import* option from the dialog. This will bring up a dialog box prompting you to specify the XML file that you would like to import. Click on *OK* and the mapping stored in the XML file will be applied to the current chart.

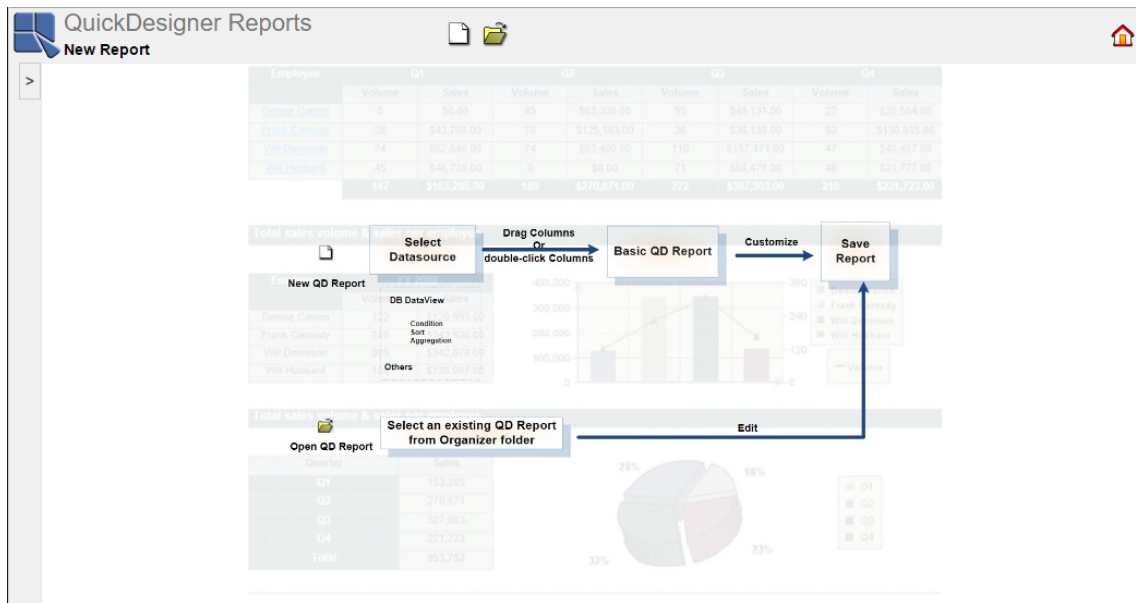
4.2. QuickDesigner Reports

QuickDesigner Reports is a thin-client ad-hoc reporting interface. It provides users a scaled-down design tool to create reports. With QuickDesigner Reports, end users can easily select, filter, and present data without mastering database structures, all with zero client download.

4.2.1. Start


QuickDesigner Reports can be started from the EDAB Start page. If you have logged in as a user with design privileges, then you can follow QuickDesigner Reports link to begin using QuickDesigner Reports.

When the QuickDesigner Reports launches, the first page that appears prompts you to select between opening an existing report, or creating a new one.

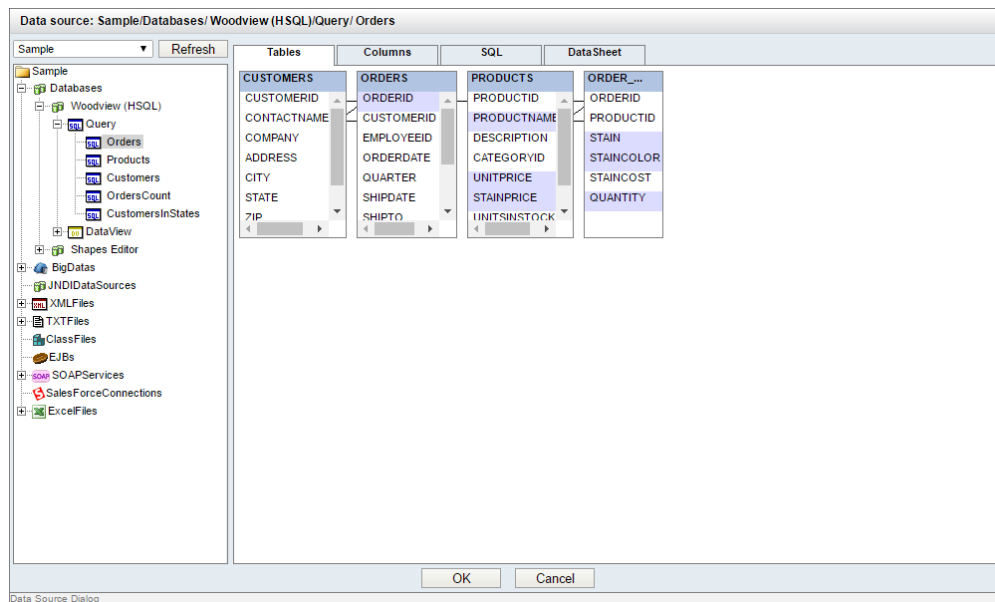


QuickDesigner Reports Start Options

4.2.2. Select a Data Source

If you want to create a new report, click on the  *Create New Report* icon on the toolbar. The *Data Source Dialog* will appear and prompt you to select data registry and data source you want to use with QuickDesigner Reports. In order to use QuickDesigner Reports, a user must have read privileges to one of the registries defined in the Organizer. For more about creating and managing data registries, please see Section 3.1.1 - Managing Data Registries.

Select data registry from the drop-down menu in the upper left corner. The tree-list bellow displays the content of the selected registry. There are all of the data sources that have been defined in the registry that the user has access to. Select a data source you want to use for the report.

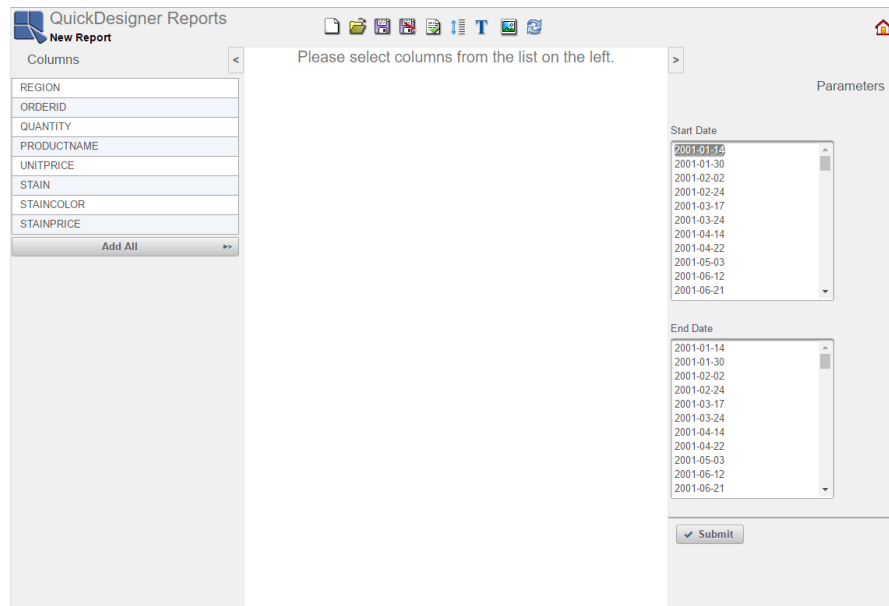


Data Source Dialog

Unlike Report Designer or Chart Designer where the user can select to create new data sources or modify the existing ones, the only option is to select a data source for the report. The only exception to this is for Data Views or

Data View Queries. If data sources of this type are selected, you will see the *DataView Builder* on the right side of the Data Source Dialog. The *DataView Builder* allows you to build or modify queries against the View. For more about managing data sources in QuickDesigner Reports, please see Section 3.2 - Data in QuickDesigners and Maps.

Once you finish selecting a data source, click on the *OK* button to close the *Data Source Dialog*.



QuickDesigner Reports

You can see columns of the data source on the left side, the central part for creating reports, and parameters pane on the right side (this pane is visible only if the data source has some parameter(s)).

4.2.3. Toolbar

A lot of formatting actions for reports in QuickDesigner Reports are accessed through the toolbar. The icons perform the following actions:



Start a new report



Open an existing report



Save the current report



Save report as



Export the current report



Set style for the report



Pagination/Scrolling options



Insert/Edit report title



Insert report logo



Modify DataView Query - This option allows user to modify the DataView and DataView query used for the report. The icon is only visible in toolbar when a DV query is used as data source for the report (otherwise it's hidden).

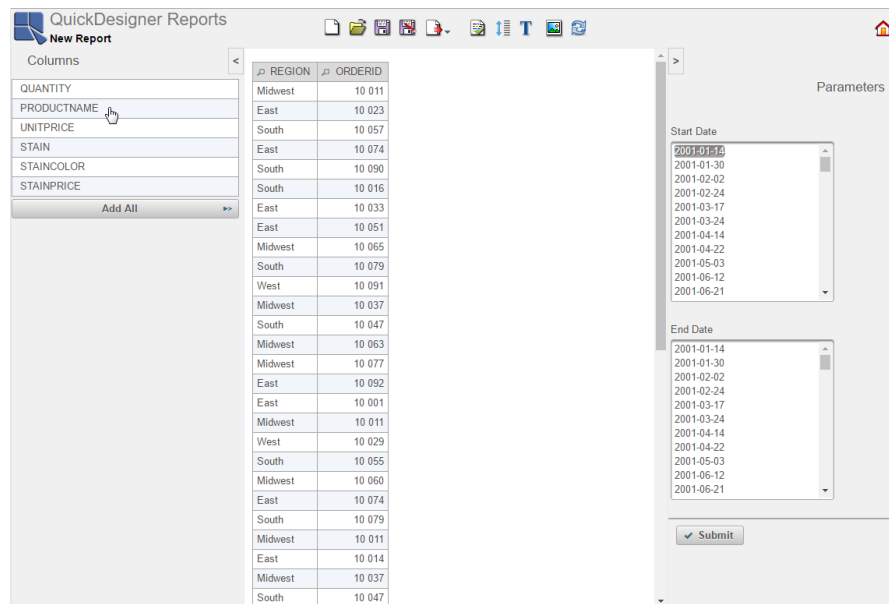


Refresh the report

4.2.4. Format Report Elements

4.2.4.1. Add a Column

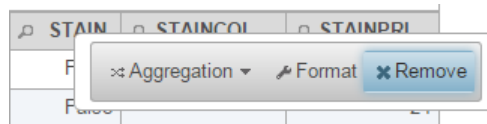
To insert a column to the report, double-click on the column in the left *Columns* pane, or click on the column and drag it to the report. You can also add all columns to the report in one step by clicking the *Add All* button.



Add a Column

4.2.4.2. Remove a Column


To remove a column from the report, right click on the column header and select *Remove* or drag the column back to the *Columns* pane. The column is removed from the report.

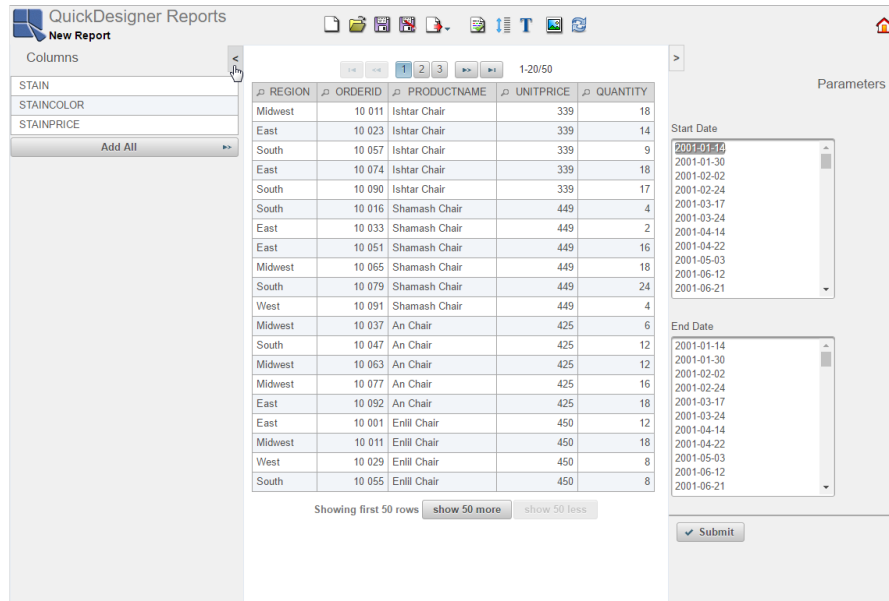


Remove a Column

4.2.4.3. Collapse Sidebars

When you add columns to the report, you can collapse the *Columns* pane to have more space for designing the report.

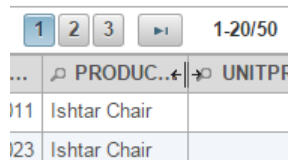
To do this, click on the  *Collapse* button at the top of the *Columns* pane. You can also collapse the *Parameters* pane on the right side (if the report has some parameters).



Collapse Sidebars

4.2.4.4. Column Width

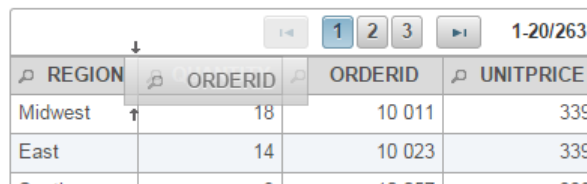
You can simply resize a column width to fit the content or to see the whole column header. Move the mouse over the right side of the column header to see double arrow, left click and drag to set the column width.



Column Width

4.2.4.5. Column Order

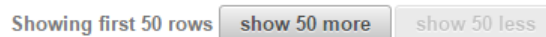
You can also reorder the columns. Left click on the column header and drag it. When you see the arrows, release the mouse button.



Column Order

4.2.4.6. Show More Rows

By default, only the first 50 rows will be displayed in the report. You can display more or less rows by clicking on the *show 50 more* or *show 50 less* buttons under the report.




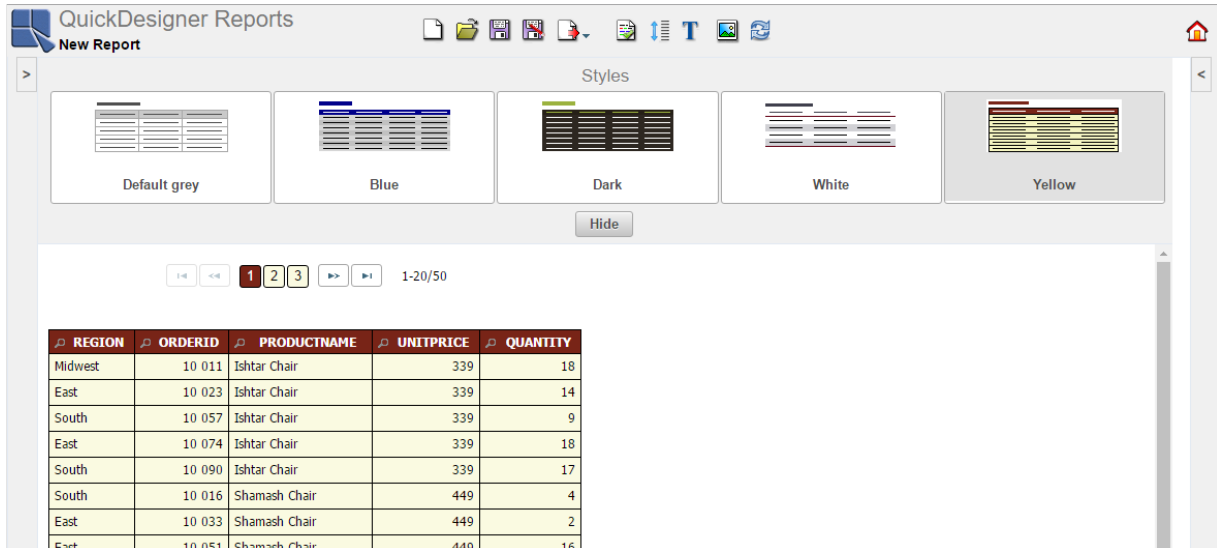
Show More Rows

When all rows are displayed, you will also see the total number of rows.


Showing all 263 rows [show 50 more](#) [show 50 less](#)*Show Less Rows*

4.2.4.7. Report Style

You can choose from five pre-defined report styles. Click on the  *Styles* icon on the toolbar to open the *Styles* dialog. Select the style you would like to use and close the *Styles* dialog by clicking on the *Hide* button.

*Styles Dialog*

4.2.4.8. Pagination/Scrolling

Click the *Pagination/Scrolling* icon on the main toolbar  to open the *Pagination/Scrolling* dialog. You can choose fixed pagination, dynamic pagination that automatically keeps the height of your screen, scrolling, or disable this feature.

Fixed Pagination

For fixed pagination, you can select a number of rows per page and a paginator position on the page.

Pagination Dialog

Paginator divides the report data into separate pages. Once you set pagination to be active, you can switch to any page of the report using buttons with page numbers.

You can also switch to the first page by clicking on the *First Page* button, to the last page by clicking on the *Last Page* button, or to the previous and next page by clicking on the *Previous Page* and the *Next Page* buttons. Next to the buttons you can see which rows are displayed/total number of rows.

1 2 3 21-40/50

REGION	ORDERID	PRODUCTNAME	UNITPRICE	QUANTITY
Midwest	10 060	Enlil Chair	450	22
East	10 074	Enlil Chair	450	18
South	10 079	Enlil Chair	450	24

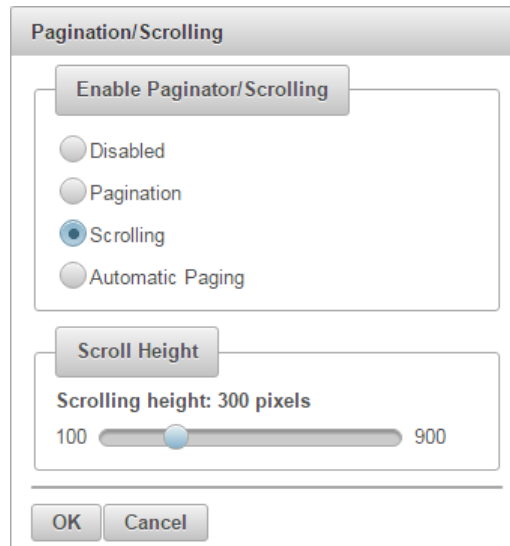
Report with Pagination Buttons

Dynamic Pagination

Dynamic pagination does not require any further setting. It will automatically adjust number of rows per page to the height of your screen. As a result, no scroll bar will appear in QuickDesigner Reports, Dashboard Builder, and Published Files. The number of rows per page may be different for different machines.

Scrolling

Scrolling allows you to display all the report data on a single page while not taking up too much space. All the displayed data is placed into a scrollable table with scrolling height of your choice. The scrolling height in pixels can be set in the *Scroll Height* section of the *Pagination/Scrolling* dialog.





The Scrolling Dialog box has a title bar 'Pagination/Scrolling'. Inside, there is a section 'Enable Paginator/Scrolling' with four radio buttons: 'Disabled', 'Pagination', 'Scrolling' (which is selected), and 'Automatic Paging'. Below this is a 'Scroll Height' section with a slider. The slider is labeled 'Scrolling height: 300 pixels' and has values '100' and '900' at its ends. At the bottom are 'OK' and 'Cancel' buttons.

Scrolling Dialog

REGION	ORDERID	PRODUCTNAME	UNITPRICE	QUANTITY
Midwest	10 011	Ishtar Chair	339	18
East	10 023	Ishtar Chair	339	14
South	10 057	Ishtar Chair	339	9
East	10 074	Ishtar Chair	339	18
South	10 090	Ishtar Chair	339	17
South	10 016	Shamash Chair	449	4
East	10 033	Shamash Chair	449	2
East	10 051	Shamash Chair	449	16
Midwest	10 065	Shamash Chair	449	18

Report with Scrolling

4.2.4.9. Report Title

You can add a report title. Click the  *Report Title* icon on the toolbar, and the *Report Title* dialog appears. Write a title of the report and click on the  *Apply* button or press *Enter* to insert the title.







The Report Title Dialog box has a title bar 'Report Title'. It contains a text input field with the word 'Orders' inside. To the right of the input field is a 'Variables' button with a small icon. At the bottom right is a checkmark button.

Report Title Dialog

The report title is on the left by default. You can move it to the center or right side of the report. Click on the title, hold down the mouse button and move the mouse slightly. You can see three dashed rectangles that represent three positions of the title - left, center, right. Move the title to the position you want to use and release the mouse button.

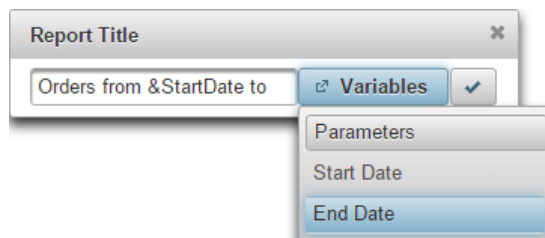


Report Title Moving

You can edit or remove the report title via the pop up  *Edit* button when the cursor is pointed over the title, or via the  *Report Title* icon on the toolbar. Both will open the *Report Title* dialog. There you can rewrite the title and apply it with the  *Apply* button, or remove it by clicking on the  *Remove* button. Then close the *Report Title* dialog by clicking on *X* in the upper right corner. You can also remove the report title directly via the pop up *X* button when the cursor is pointed over the title.

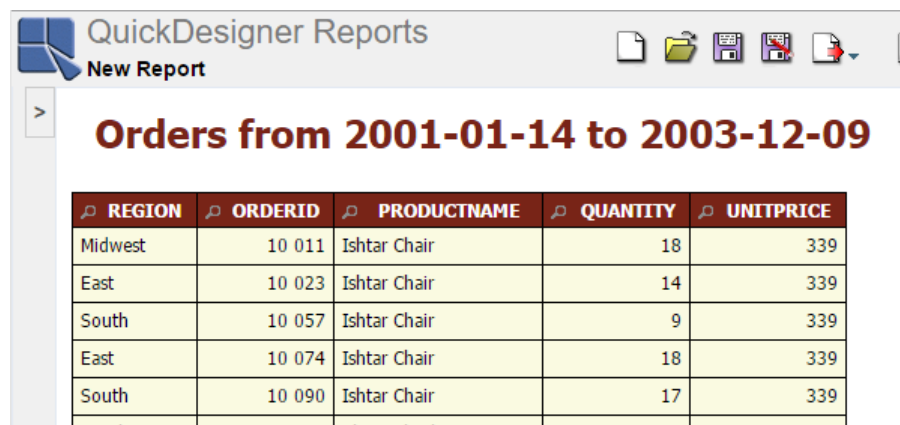
4.2.4.9.1. Report Title with Variables

If the data source is parameterized, you can add variables to the report title. Variables display the current parameter value in the report title and change dynamically according to which parameter value is selected.




Variables for Report Title

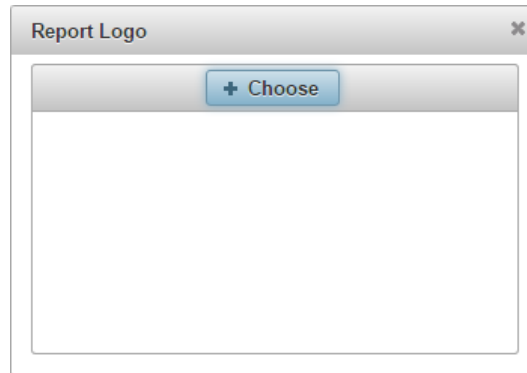
To add a variable, click on the *Variables* button in the *Report Title* dialog, select a parameter from the list, then it will be displayed in this format *&<parameterName>* in the textfield.



Report Title with Variables

4.2.4.10. Report Logo

You can add a logo to your report. To do this, click on the  *Report Logo* icon on the toolbar to open the *Report Logo* dialog. This dialog allows you to choose a logo from your local directory. Once a logo is selected, it is inserted to the top left.





Report Logo Choose

You can move the logo to the center or right side of the report. Click on the logo, hold down the mouse button and move the mouse slightly. You can see three dashed rectangles that represent three positions of the logo - left, center, right. Move the logo to the position you would like to use and release the mouse button.

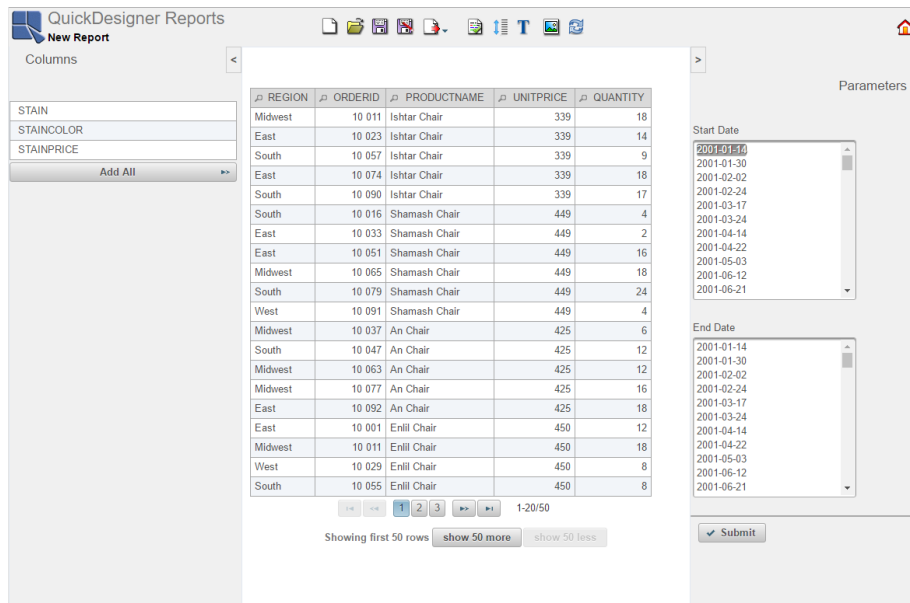


Report Logo Placement



You can remove the report logo via the pop up  button when the cursor is pointed over the logo. If you want to change the logo, just click on the  *Report Logo* icon to choose a new image. The new image will replace the old one.

4.2.5. Parameter Setting

If the report uses a parameterized data source, you can see the *Parameters* pane on the right side of the designer window. You can select parameter(s) value(s) and apply it (them) by clicking on the *Submit* button.

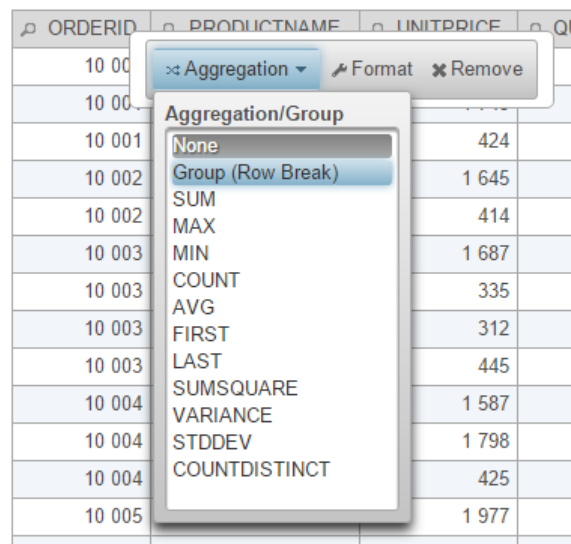


Parameters Pane

You can collapse or open this pane by clicking on the  or  *Collapse* buttons.

4.2.6. Aggregation/Group

You can add an aggregation or group to each column of the report. To do this, right click on the column header and move the mouse cursor over *Aggregation* button. The *Aggregation/Group* list will appear. Select an aggregation by clicking on it in the list. The aggregation will be assigned to the appropriate column. After you finish this setting, an extra line of the aggregation value will appear after each group (row break) and after the last aggregation value line, an extra line with a grand total value will appear.



Aggregation/Group Setting

Once the aggregations are set, you can set label for the row break and grand total. The grand total label is called Report Footer label. To set a label, right click on an empty cell next to the value you want to describe and then click on the pop up *Label* button.

	QUANTITY	SALES
0	12	5 400
5	12	20 940
4	14	5 936
		32 276
5	16	
4	21	8 694
		35 014

Aggregation Label Button

The *Footer Labels* dialog will appear. You can insert a text and decide how to align it in a cell - left or right. Then click on the *OK* button to apply the label.



Footer Labels Dialog

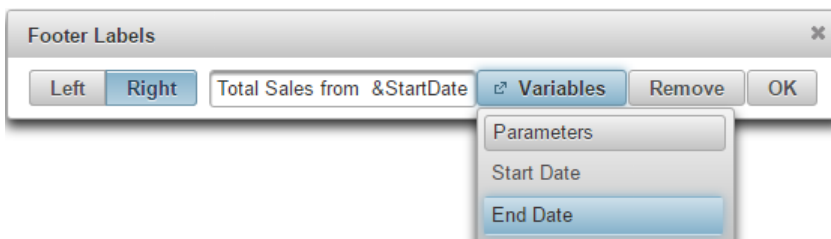
To edit or remove a label, right click on it and then click on the pop up *Label* button and edit the text or click on the *Remove* button in the *Footer Labels* dialog.

The next image shows a sample of the report with *Row Break*, *SUM* aggregation, group footer label, and report footer label.

<div> ◀ << 34 35 36 >> ▶ 351-360/360 </div>				
ORDE..	PRODUCTNAME	UNITPRICE	QUANTITY	SALES
	Apsu Dresser	1 992	8	15 936
			Order Total	30 210
10 095	Shimaliya Chair	424	16	6 784
	Bes Table	1 141	16	18 256
			Order Total	25 040
10 096	Zabada Chair	312	6	1 872
	Ninurta Chair	345	6	2 070
	Horus Table	1 354	12	16 248
			Order Total	20 190
			Total Sales	2 850 764

Aggregation and Label

If the data source is parameterized, you can add variables to the footer labels. Variables display the current parameter value in the selected footer label(s) and change dynamically according to the selected parameter value.



Footer Labels Variables

To add a variable, click on the *Variables* button in the *Footer Labels* dialog and select a parameter from the list. It will be displayed as `&<parameterName>` in the textfield.

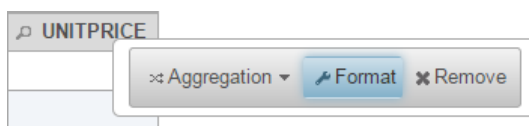
rus Table	1 354	12	16 248
		Order Total	20 190
Total Sales from 2001-01-14 to 2003-12-09			2 850 764

Report Footer Label with Variables

If you add a variable to the label, the text may not fit the cell. In this case it is advisable to use text alignment.

4.2.7. Data Formatting

Right click on the column header and select *Format* from the pop-up menu.



Format Button

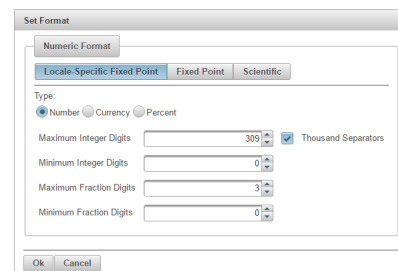
When you select the *Format* option, the *Set Format* dialog will appear. The *Set Format* dialog is dependent on what type of data is present in the selected column: numeric, string, date/time, or logical/boolean.

Formatting Numeric Data:

The dialog for numeric data contains three primary options for the data: *Locale-Specific Fixed Point*, *Fixed Point*, and *Scientific*. Each option opens a different tab.

Locale-Specific Fixed Point:

This will change the format of the data depending on the locale in which it is being viewed. Additional formatting for this option allows you to specify whether the data should be displayed as a number, currency, or percentage. In addition, you can set the maximum and minimum number of integer digits and fraction digits. Other display attributes will vary depending on locale.

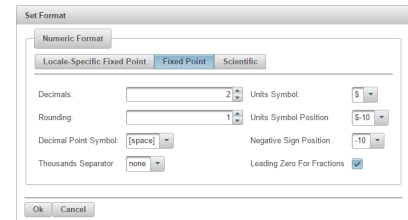


Locale-Specific Formatting

Fixed Point:

This will keep the data format consistent, regardless of locale. Additional formatting for this option allows you to set the number of decimals, rounding for digit number, unit symbols, negative sign position, decimal

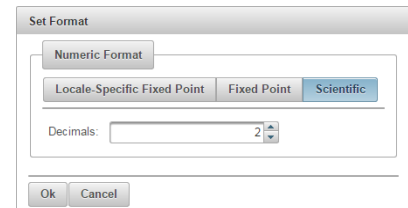
and thousands separator, and specify leading zeroes for fractions.



Fixed Point Formatting

Scientific:

This will display the data in scientific notation. Additional formatting for this option allows you to set the number of decimals.

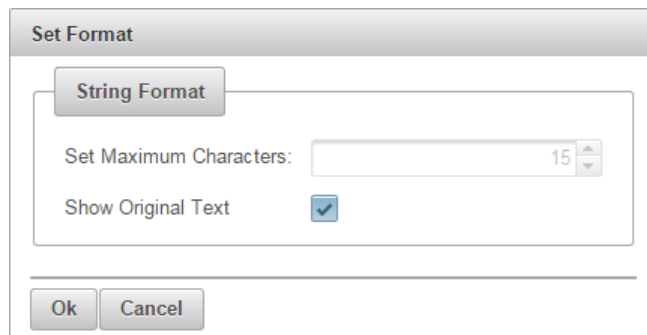


Scientific Formatting

After you finish selecting options, click on the *Ok* button to apply formatting and close the *Set Format* dialog.

Formatting String Data:

In this dialog, you can change maximum characters for the string format. To do this, uncheck the *Show Original Text* option and then set maximum characters.



String Data Format

Click on the *Ok* button to apply changes and close the *Set Format* dialog.

Formatting Date/Time Data:

The dialog for date/time data contains two tabs: *Locale-Specific* and *Standard*.

Locale Specific:

This will change the format of the data depending on the locale in which it is being viewed. Additional formatting for this option allows you to select full, long, medium, or short notations for date and time information. Other display attributes will vary depending on locale.

*Locale-Specific Formatting***Standard:**

This will keep the data format consistent, regardless of locale. Additional formatting for this option allows you to select year and month displays, as well as the order in which month day and year information is presented. You can also select which characters should be used as separators. Time options allow you to display hours, minutes, and/or seconds, and select the separators between them. For timestamp data, you can select to display the time before or after the date and the separator to be used between them.

Standard Formatting

After you finish selecting options, click on the *Ok* button to apply formatting and close the *Set Format* dialog.

Formatting Logical Data:

The dialog for Logical or Boolean data contains five options for displaying the data: T/F, True/False, Yes/No, Y/N, and 1/0.

The 'Set Format' dialog box contains a 'Logic Format' section with five radio button options: 'T/F', 'True/False' (which is selected), 'Yes/No', 'Y/N', and '1/0'. At the bottom of the dialog are 'Ok' and 'Cancel' buttons.

Logic Format

Select a format you want to use and then click on the *Ok* button to change the data format and close the *Set Format* dialog.



4.2.8. Sorting

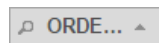
When you hover your mouse cursor over the column header, you will see a label *Sorting: NONE*. Left click on the column header to apply the sorting. First click causes ascending sorting, second click causes descending sorting and third click cancels the sorting.

A table with columns: REGION, ORDERID, and PRODUCTNAME. A mouse cursor is hovering over the 'ORDERID' header, which has a tooltip that says 'Sorting: NONE'. Above the table are navigation buttons (first, previous, next, last) and a page indicator '1-20/263'.


REGION	ORDERID	PRODUCTNAME
Midwest	10014	Lehter Chair
		Enlii Chair
		Enki Chair

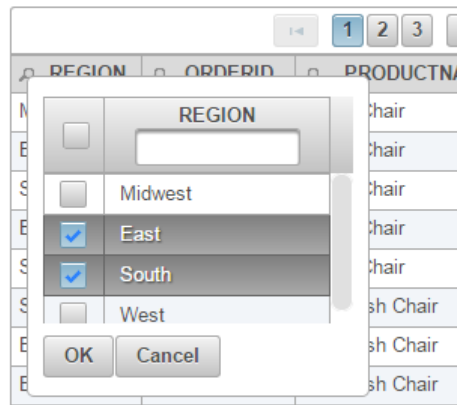
Sorting

Sorted columns have arrow icons on the column header. The  *Up Arrow* icon indicates ascending order and the  *Down Arrow* icon indicates descending order.

*Sorting Arrow*


4.2.9. Data Filtering

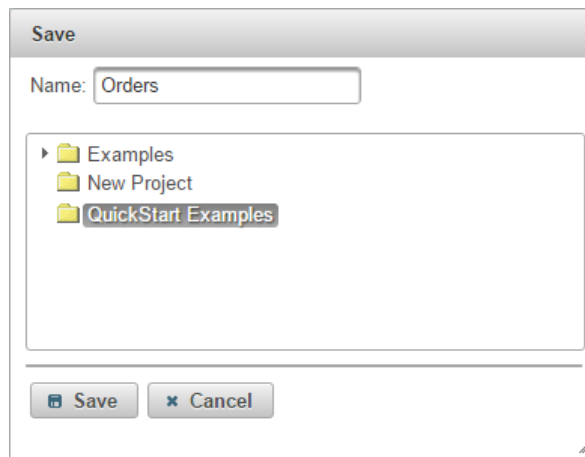
You can filter data by clicking on the  *Filter Column* icon on the column header. A pop-up dialog will appear, which will allow you to select data for filtering. To select values which you want to be displayed in the report, check off the box next to the desired value(s). You can narrow down the values shown in the *Filter* dialog or search for particular values by typing them into the text field at the top of the dialog. To select all available values within the column, check the box in the upper left corner (next to the search field). Click on the *OK* button to apply the filtering and close the dialog.


*Filter Column*

If there are several columns applying data filters, the report will display all filter conditions, i.e. different column filters has “AND” relationship.


4.2.10. Save the Report

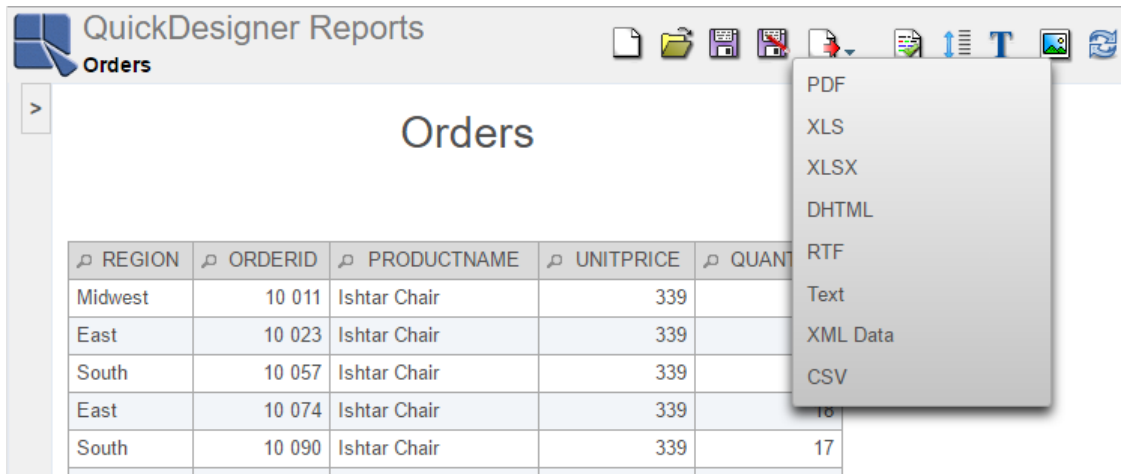
You can save the report by clicking on the *Save* button on the toolbar . This will bring up a dialog allowing you to specify a name for the report. Enter a name for the report, select the project where you want to save it and click on the *Save* button.

*Save Dialog*

After saving or opening existing report, the  *Save As* button allows you to save the existing or modified report under a different name or into a different location.

4.2.11. Export the Report

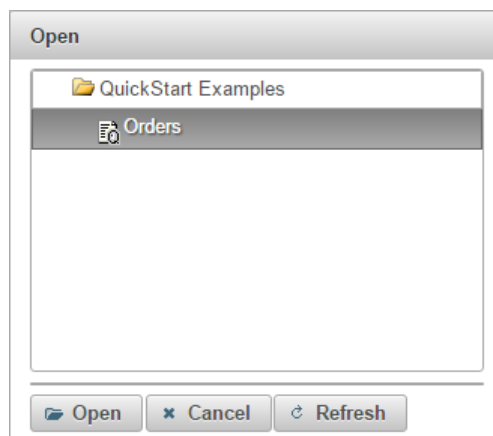
You can also export the report in several file formats. To do so, click on the  *Export* icon on the toolbar. A pop-up dialog will appear, prompting you to specify the export format. Available options are PDF, XLS, XLSX, DHTML, RTF, Text, XML Data, and CSV.

*Export Options*

Select the format and then you can save the generated file to your local system.



4.2.12. Open the Saved Report

You can open the saved report by clicking on the  *Open* icon on the main toolbar. The *Open* dialog will appear.

*Open the Report*

Here you can see all reports from Organizer created in QuickDesigner Reports. Select a report and click on the *Open* button to open it.

4.2.13. Exit

You can exit QuickDesigner Reports either by clicking on the  *Home* icon in the upper right corner, or by clicking on the *QuickDesigner Reports* title, or by clicking on the  *Logo* icon in the upper left corner. Before closing, you will be prompted if you want to save an unsaved report.

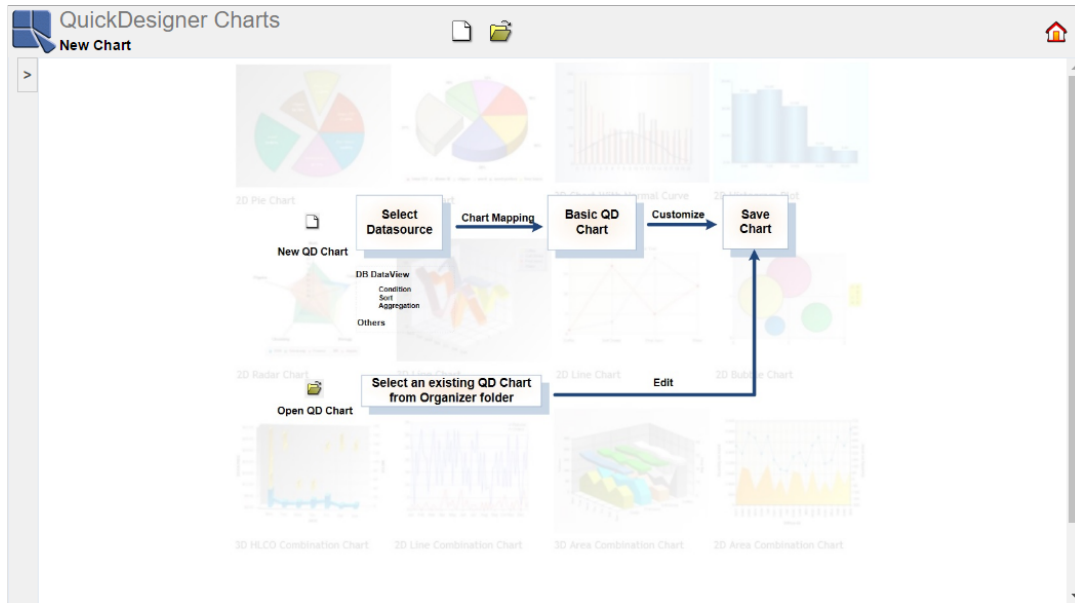
4.3. QuickDesigner Charts

QuickDesigner Charts is a thin-client ad-hoc charting interface. It provides users a scaled-down design tool to create charts. With QuickDesigner Charts, end users can easily select, filter, and present data without mastering database structures, all with zero client download.

4.3.1. Start


QuickDesigner Charts can be started from the EDAB Start page. If you have logged in as a user with design privileges, you can follow QuickDesigner Charts link to start using QuickDesigner Charts.

When QuickDesigner Charts launches, the first page that appears prompts you to select between opening an existing chart or creating a new one.

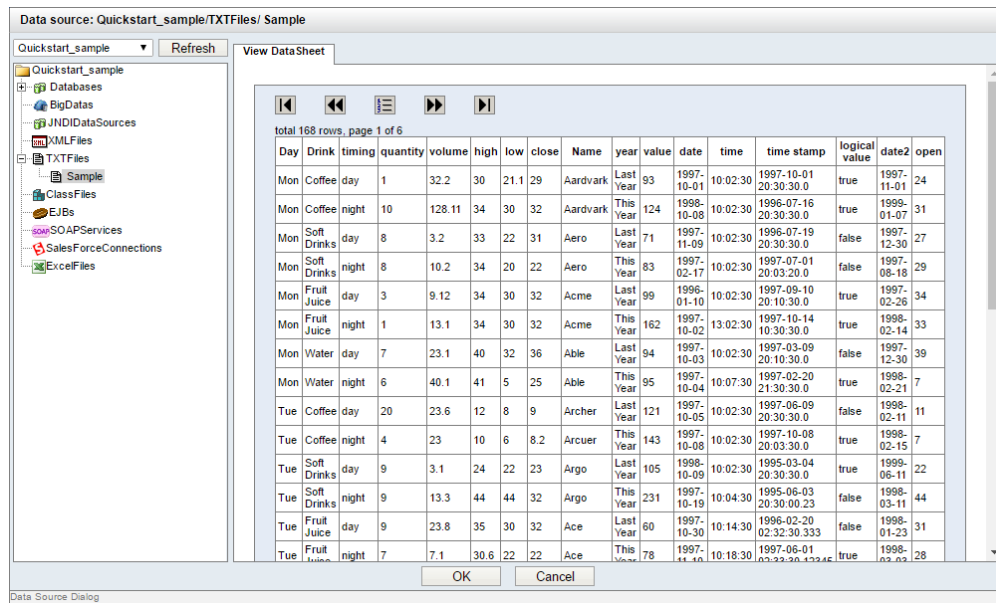


QuickDesigner Charts Start Options

4.3.2. Select a Data Source

If you want to create a new chart, click on the  *Create New Chart* icon. The *Data Source Dialog* appears and prompts you to select data registry and data source you want to use with QuickDesigner Charts. In order to use QuickDesigner Charts, a user must have read privileges to one of the registries defined in the Organizer. For more about creating and managing data registries, please see Section 3.1.1 - Managing Data Registries.

Select data registry from the drop-down menu in the upper left corner. The pane below displays the content of the selected registry. Select a data source you want to use for the chart.




Data Source Dialog

You can see the records of the data source in *View DataSheet* pane on the right. If you select *DataView* as the data source, you will be able to create a new *DataView* query in the *DataView Builder* in the right pane. You can also select an existing *DataView* query and modify it in the right pane. For more about managing data sources in QuickDesigner Charts, please see Section 3.2 - Data in QuickDesigners and Maps.

Once you have finish selecting a data source, click on the *OK* button to close the *Data Source Dialog*.

4.3.2.1. Change a Data Source

After you return to the main QuickDesigner Charts window (after you created a new chart or opened an existing one), you can change the chart's data source by clicking on the  *Change Data Source* button on the toolbar. This will take you back to the *Data Source Dialog* and allow you to pick a new data source.

4.3.3. Data Mapping and Ordering

Once you select a data source or finish building a query and close the *Data Source Dialog* by clicking on the *OK* button, you will see *Chart Mapping* dialog in the left pane of QuickDesigner Charts.

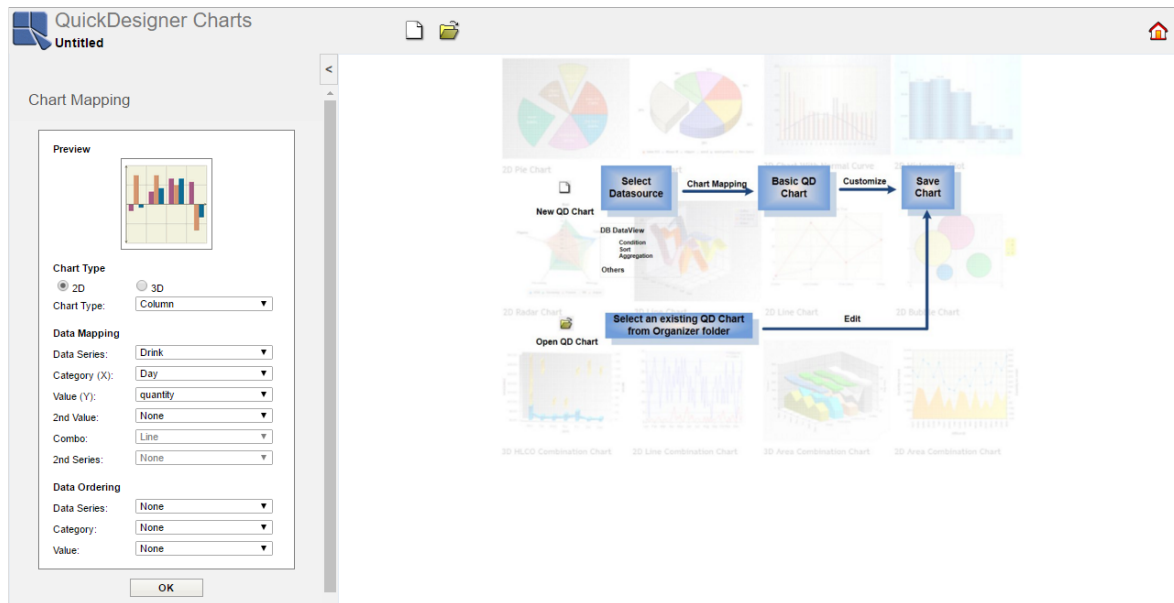


Chart Mapping Dialog

4.3.3.1. Mapping and Ordering Options

The *Chart Mapping* dialog allows you to select which chart type you want to create and which fields from the data source you want to map to the chart elements.

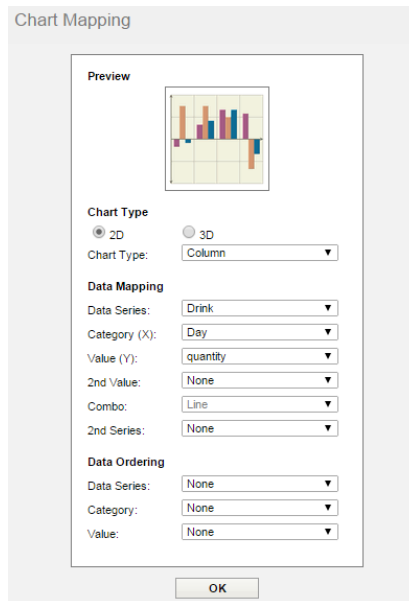
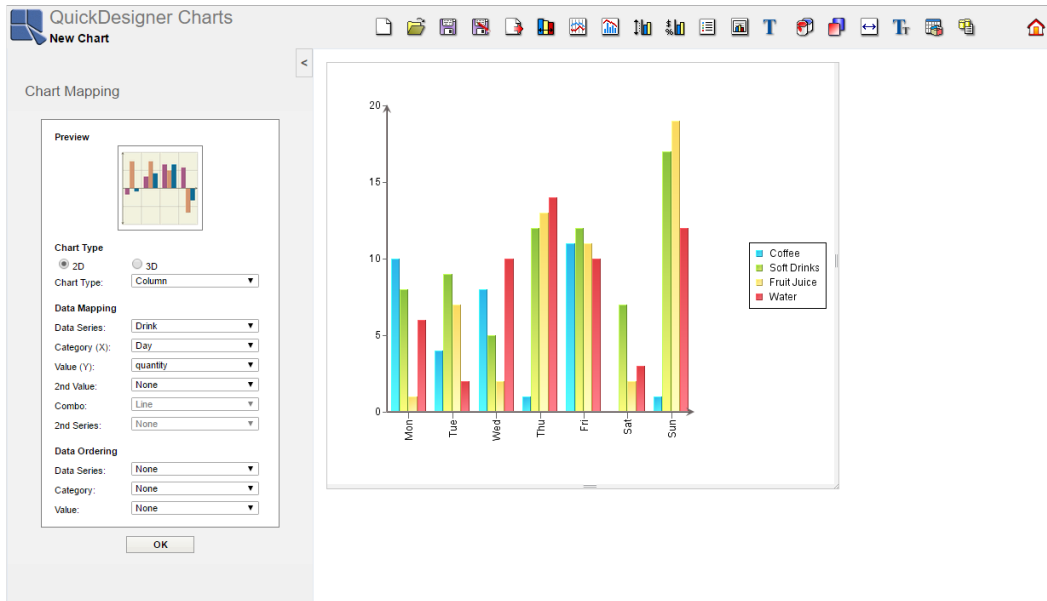


Chart Mapping Options


The *Chart Type* options allows you to select between two-dimensional and three-dimensional chart types, and the chart type you want to use. The *Data Mapping* options allows you to select data mapping for the chart. The *Data Ordering* options allows you to sort data in ascending or descending order. For a detailed description of all available charts and mapping options, see Section 4.1.3 - Chart Types and Data Mapping.

Once you select the type and mapping options, click on the *OK* button and the chart will appear in the right pane, as well as all icons.



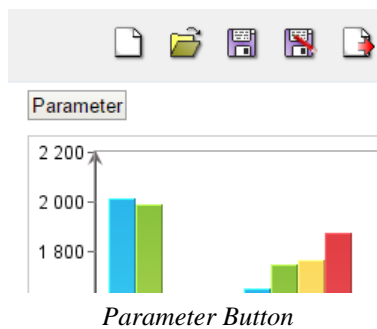
QuickDesigner Charts Main Page

4.3.3.2. Change Data Mapping

You can change the mapping options and the chart type by clicking on the  *Change Mapping/Chart Type* icon on the toolbar. This will open the *Chart Mapping* dialog in the left pane.

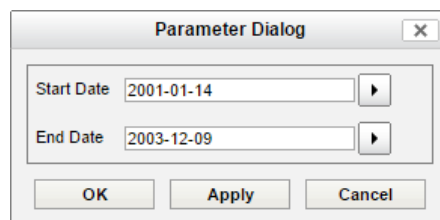
4.3.3.3. Parameter Setting

If the data source is parameterized, you can see *Parameter* button above the chart.



Parameter Button

After clicking on it, the *Parameter Dialog* will open allowing you to set parameter values. To apply a parameter value, click on the *Apply* button (the dialog stays open) or *OK* button (the dialog will be closed).



Parameter Dialog

4.3.4. Toolbar

All of the formatting actions for charts in QuickDesigner Charts are accessed through the toolbar. The icons perform the following actions:

-  Create a new chart
-  Open an existing chart
-  Save the chart
-  Save chart as
-  Export the chart
-  Set data properties for the chart
-  Chart-specific options (This option is only available for Pie, HLCO, Dial, Gantt, Polar, and Doughnut chart.)
-  Line and point settings
-  Trend line settings
-  Format axis scale
-  Format axis elements
-  Format chart legend
-  Format plot area
-  Add/Edit chart titles
-  Set chart colors
-  Set color gradient
-  Set chart canvas size
-  Set label/legend font
-  Change chart type/mapping
-  Change chart data source


4.3.5. Data Formatting

Once you finish specifying all the mapping options, you can format chart elements.

4.3.5.1. Position and Size

There are several options available in QuickDesigner Charts that allows you to control the size and position of the chart.

4.3.5.1.1. Canvas Size

The chart canvas size can be adjusted by clicking on the  *Canvas Size* button on the toolbar. This will bring up a dialog in the left pane allowing you to specify the size of the chart.

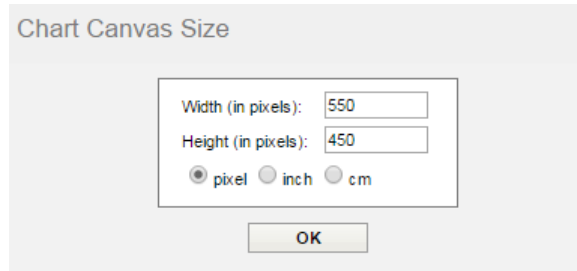
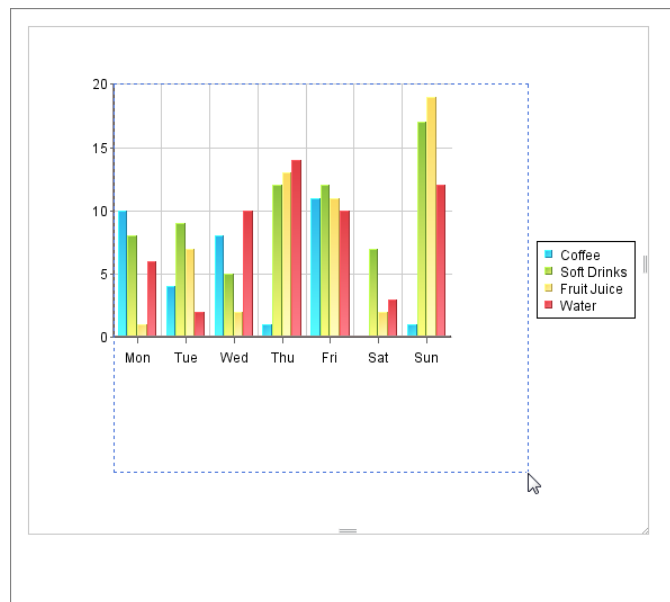


Chart Canvas Dialog

The dialog allows you to specify the width and height of the canvas in pixels, inches, or centimeters. Once you specify the new dimensions, click on the *OK* button and all changes will be applied. You can also change the canvas size freely using drag & drop. In this case, you can see the exact size in the *Chart Canvas Size* dialog.

4.3.5.1.2. Chart Position and Size

You can set the position of the chart within the canvas freely using drag & drop. You can set the canvas size or plot area size by dragging the sides or dragging the corner.



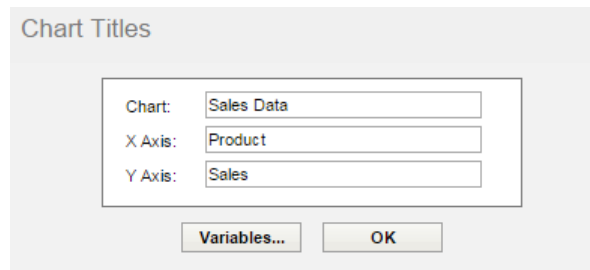
Setting Size

4.3.5.2. Font and Color Options

QuickDesigner Charts interface allows you to add/edit chart titles, set title/label font, and chart colors.

4.3.5.2.1. Chart Titles

To add or edit the chart titles, click on the  *Add/Edit Chart Titles* button on the toolbar. This will bring up a dialog in the left pane.

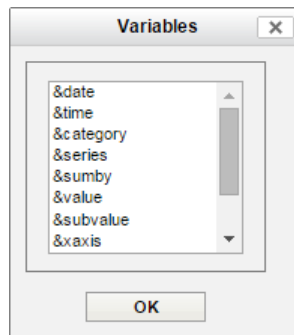


Insert/Edit Titles Dialog

You can add a main title to the chart, as well as one for each of the chart axes. Once you finish adding titles, click on the *Ok* button to apply the changes.

4.3.5.2.1.1. Text Variables

As with the Chart Designer, you can also add text variables that will provide run-time substitution based on certain values in the chart. To bring up a list of variables, click on the *Variables* button in the *Chart Titles* dialog. The variables list will open in a new window.



Text Variables Dialog

The following text variables are supported:

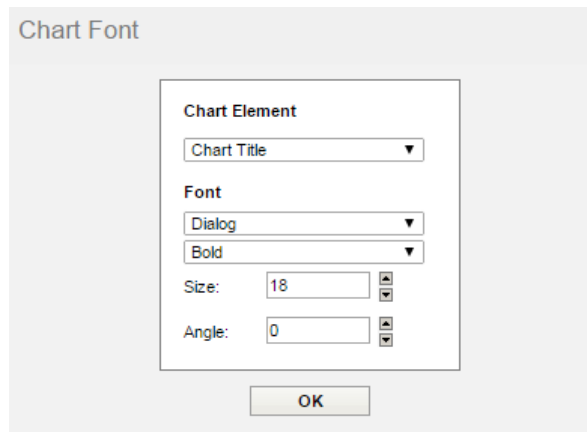
<i>&date</i>	This variable displays the date when the chart was last drawn/redrawn.
<i>&time</i>	This variable displays the time when the chart was last drawn/redrawn.
<i>&category</i>	This variable displays the name of the category column.
<i>&series</i>	This variable displays the name of the data series column.
<i>&sumby</i>	This variable displays the name of the sum-by column.
<i>&value</i>	This variable displays the name of the value column.
<i>&subvalue</i>	This variable displays the name of the secondary value column.
<i>&xaxis</i>	This variable displays the name of the column that is mapped to X-axis. This is for charts that map a value instead of a category to the X-axis like scatter or bubble charts.
<i>&yaxis</i>	This variable displays the name of the column that is mapped to Y-axis. This is for scatter, bubble, and surface charts.
<i>&zaxis</i>	This variable displays the name of the column that is mapped to Z-axis. This is for scatter, bubble, and surface charts.
<i>&2ndaxis</i>	This variable displays the name of the column that is mapped to 2nd-axis.

<i>&paramInfoValue</i>	If the chart contains a parameter, this displays the parameter value(s) that were selected.
<i>&paramInfoName</i>	If the chart contains a parameter with this name, this will display the name of that parameter.

To add a text variable to a title, select it from the list and click on the *OK* button.

4.3.5.2.2. Chart Fonts


To set the font for chart labels and titles, click on the  *Font Settings* button on the toolbar. This will open a dialog in the left pane.



Font Settings Dialog

To set the font for a title or label, select it in the drop-down list at the top of the dialog. Then select font face, style, size, and angle using the options in the lower dialog. Once you finish setting up fonts, click on the *OK* button to apply the changes.

4.3.5.2.3. Chart Colors

You can set color for all chart elements by clicking on the  *Set Chart Colors* button on the toolbar. The *Chart Colors* dialog will open in the left pane.

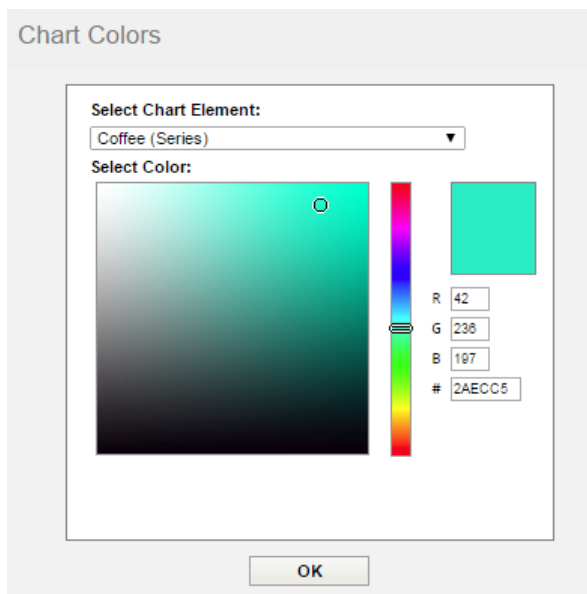



Chart Colors Dialog

To set the color for a chart element, first select it from the drop-down list at the top of the dialog. Then select a new color either by clicking on one of the swatches, entering RGB values for the color, or entering HEX code to the # field. Once you finish setting up the colors, click on the *OK* button to apply the changes.

Please note that some elements are not visible by default (e.g. data border). In this case, first make the element visible (see Section 4.3.5.4.1 - Data Properties) and then use *Chart Colors* dialog for setting a color of the element.

4.3.5.2.4. Color Gradient

You can set the color gradient for the chart by clicking on the  *Set Color Gradient* button on the toolbar. The *Color Gradient* dialog will open in the left pane.

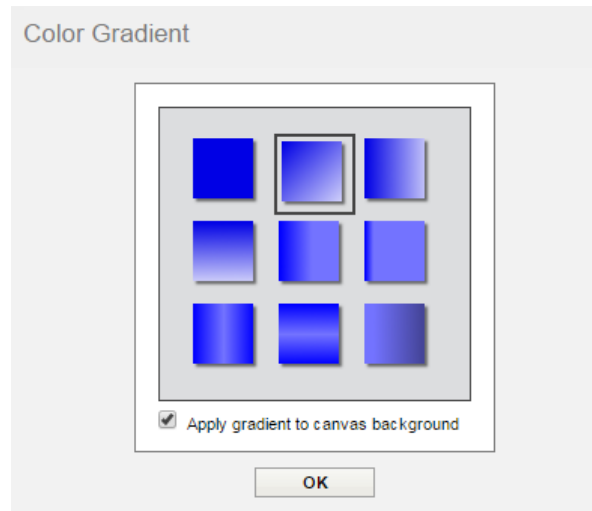


Chart Colors Dialog


Select a gradient you want to use and click on the *OK* button to apply the changes. You can apply the same gradient to the canvas background.

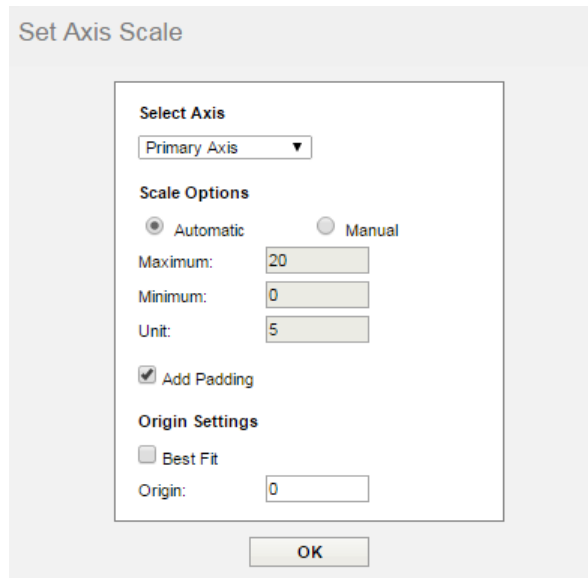
4.3.5.3. Chart Axes

In QuickDesigner Charts, you have several formatting options for the chart axes. You can customize the axis scale as well as label/axis appearance options.

4.3.5.3.1. Axis Scale

By default, the scale of any value axes in the chart is calculated to provide the *best fit* for the plotted data. This is especially useful if the data change often. However, there are many times when you may want to set the scale

of the axes manually. To modify the axis scale, click on the  *Axis Scale* button on the toolbar. A new dialog will open in the left pane.




Axis Scale Dialog

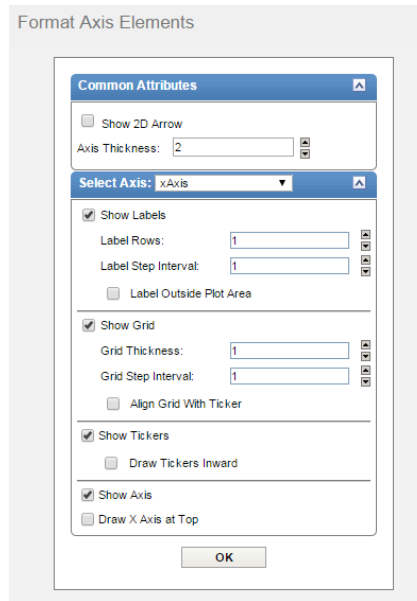
You can select the axis for which you would like to set the scale by selecting it in the drop-down list at the top of the dialog. The following scaling options are provided for each value axis:

- Automatic:** This option turns on automatic scaling for the axis. This is the default setting.
- Manual:** This option turns on manual scaling to set the axis scale to the options provided.
- Maximum:** This is the highest value on the axis scale.
- Minimum:** This is the lowest value on the axis scale.
- Unit:** This is the step size of the axis scale.
- Add Padding:** This option will add padding to the top of the automatic scale. If this option is disabled, the maximum scale will match the largest data point.
- Best Fit:** This option will automatically place the origin of the chart based on minimum and maximum values of the data.
- Origin:** This option allows you to specify where the X and Y axes should intersect. This is usually at zero.

Once you finish specifying the axis scale options, click on the *OK* button to apply the changes.

4.3.5.3.2. Axis Elements

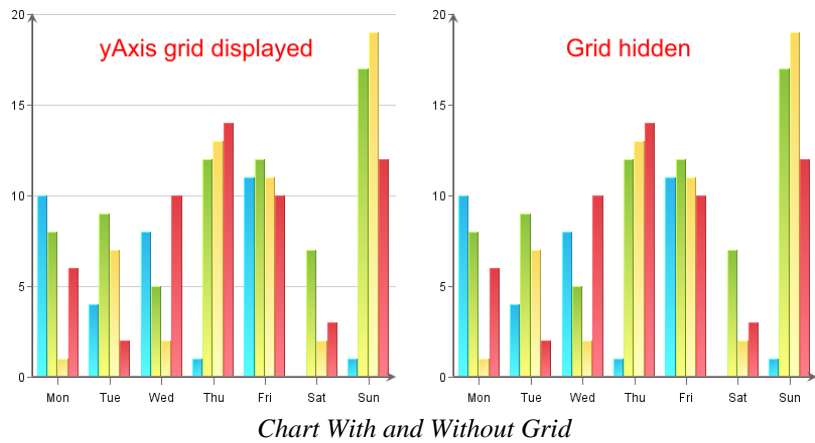
Additional appearance properties for the axes and axis labels can be set in the *Format Axis Elements* dialog. To change these settings, click on the  *Axis Options* button on the toolbar. This will open the dialog in the left pane.



Axis Elements Dialog

To change the settings for an axis, first select it in the drop-down *Select Axis* list and then set the available options below. The dialog allows you to perform the following options. Note that some options are only available for certain chart types, certain data types, and on certain axes.

- | | |
|---------------------------------|--|
| Show 2D Arrow: | This option allows you to remove or display arrowhead at the end of the axis. Note that this option applies to all chart axes and is only available for two-dimensional charts. |
| Axis Thickness: | This option allows you to set the thickness of the axis in pixels. Note that this setting is only available for two-dimensional charts and it is applied to all axes in the chart. |
| Show Labels: | This option allows you to remove or display the labels for each axis. |
| Label Rows: | This option allows labels to be displayed in alternating rows. This can prevent overlapping. This option is only available for the X-axis. |
| Label Step Interval: | This option allows you to set the label step interval for the data. For example setting this to 2 would draw the label for every other data point in the chart. |
| Label Outside Plot Area: | This option sets the labels to be placed outside of the plot area, regardless of where the axis is. This feature can be useful for category axis labels if you are plotting data with both positive and negative values. |
| Label Interval Unit: | This option allows you to select the unit to be used when sorting and representing time-based data (date, time, or timestamp). Selecting tickers will use the data as it is read by Chart Designer. |
| Show Grid: | This option allows you to remove or display the grid for each axis. |


Grid Thickness:

This option allows you to specify the thickness of any grid lines along the axis.

Grid Step Interval:

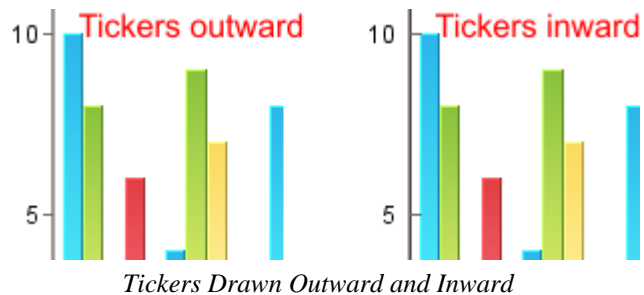
This option allows you to set the grid step interval for any grid lines along the axis.

Align Grid With Ticker:

This option aligns the grid line with the ticker instead of placing it between tickers. This places the ticker and corresponding grid line along the same line. This option only applies to the category axis of column-type charts.

Show Tickers:

This option allows you to remove or display the axis tickers and draw them inward or outward.


Draw Tickers Inward:

This option will draw the axis tickers inside the plot area instead of outside (default).

Show Axis:

This option allows you to remove or display the axis (for two-dimensional charts) or the wall (for three-dimensional charts).

Draw X Axis at Top:

This option allows the X-axis to be positioned at the top of the chart instead of the default bottom position. This option is available for two-dimensional column, bar, scatter, high-low, HLCO, bubble, and Gantt charts.

Swap Y Axis Position:

This option will swap the primary and secondary value axes. This option can only be found under 2nd Axis tab.

4.3.5.3.2.1. Axis Label

The *Format Axis Elements* dialog also allows you to format the appearance of the axis labels, depending on what type of data is plotted on the axis.

• Formatting Numeric Data:

For numeric data, there are three primary options for display formatting: locale-specific fixed point, fixed point, and scientific. Additional options will be displayed when you click on the *Format* button.

Numeric Format

☒ Locale-Specific ▼

☐ Fixed Point ▼

☐ Scientific ▼

Numeric Data Format Options

Locale-Specific:

This will change the format of the data depending on the locale in which it is being viewed. Additional formatting for this option allows you to specify whether the data should be displayed as a number, currency, or percentage. In addition, you can set the maximum and minimum number of integer digits and fraction digits. *Use Grouping* brings space as the thousands separator. Other display attributes will vary depending on locale.

Please note that if you select *Currency* as *Type*, local-specific format will automatically take currency symbol from your system settings (e.g. it will add “\$” to the label if you're in U.S.A.).

☒ Locale-Specific ▲

Type: ☒ Number ☐ Currency ☐ Percent

Maximum Integer Digits:

Minimum Integer Digits:

Maximum Fraction Digits:

Minimum Fraction Digits:

☒ Use Grouping

Locale-Specific Formatting Options

Fixed Point:

This will keep the data format consistent, regardless of locale. Additional formatting for this option allows you to set the number of decimals, rounding for digit number, unit symbols, negative sign position, decimal and thousands separator, and to use *Leading Zero For Fractions* which determines whether it should display 0 before decimals for numbers from -1 to 1 (e.g. 0.15/ .15).

☐ Fixed Point ▲

Decimals:

Units Symbol: ▼

Rounding:

Decimal Point Symbol: ▼

Units Symbol Position: ▼

Thousands Separator: ▼


Negative Sign Position: ▼

☒ Leading Zero For Fractions

Fixed Point Formatting Options

Scientific:

This will display the data in scientific notation. Additional formatting for this option allows you to set the number of decimals.



Scientific

Decimals: 2

Scientific Formatting Options

- **Formatting Date/Time Data:**

For date/time data, there are two primary options for display formatting: locale specific and standard. Additional options will be displayed if you click on the *Format* button. The available options will vary depending on the nature of your data. Date, time, and timestamp data will bring up date, time, and date & time options.



Date Format

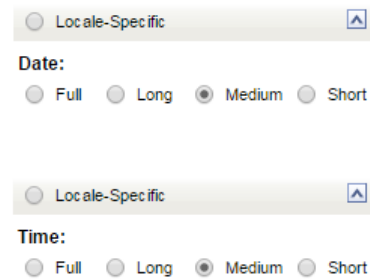
Locale-Specific

Standard

Date/Time Data Format Options

Locale-Specific:

This option will change the format of the data depending on the locale in which it is being viewed. Additional formatting for this option allows you to select full, long, medium, or short notations for date, and time information. Other displayed attributes will vary depending on your locale.



Locale-Specific

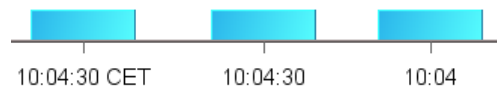
Date:

Full Long Medium Short

Time:

Full Long Medium Short

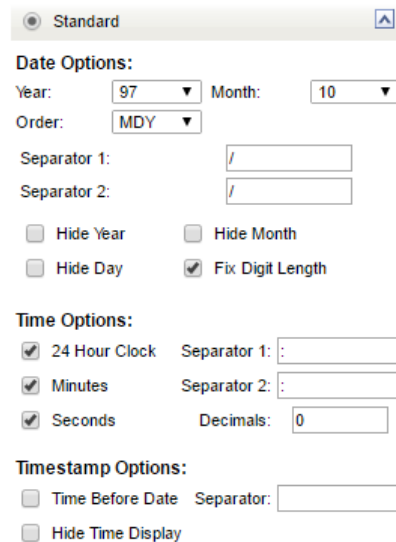
Locale-Specific Formatting Options



Example of Long/Medium/Short Time Format

Standard:

This option will keep the data format consistent, regardless of locale. Additional formatting for this option allows you to select year and month displays, as well as the order in which month, day, and year information is presented. You can also select the characters to be used as separators. Time options allow you to display hours, minutes, and/or seconds, and select the separators between them. For timestamp data, you can select to display the time before or after the date and the separator to be used between them. *Fix Digit Length* option adds zeros to days and months between 1 – 9 to keep the length of the date consistent (two digits for both day and month).



Standard

Date Options:

Year: Month:

Order:

Separator 1:

Separator 2:

☐ Hide Year ☐ Hide Month

☐ Hide Day ☒ Fix Digit Length

Time Options:

☒ 24 Hour Clock Separator 1:

☒ Minutes Separator 2:

☒ Seconds Decimals:

Timestamp Options:

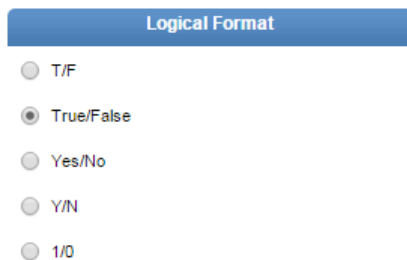
☐ Time Before Date Separator:

☐ Hide Time Display

Standard Formatting Options

- **Formatting Logical Data:**

There are five options available for displaying logical or Boolean data: T/F, True/False, Yes/No, Y/N, and 1/0.



Logical Format

☐ T/F

☒ True/False

☐ Yes/No

☐ Y/N

☐ 1/0


Logical Formatting Options

4.3.5.4. Plot/Data Elements

In QuickDesigner Charts, there are many ways to configure how data points and chart plot should be drawn.

4.3.5.4.1. Data Properties

Many of the data display options are controlled through the data properties dialog. From this dialog, you can control the size of bars/columns, set display options for null values, and specify options for data labels. To invoke this

dialog, click on the  *Data Properties* button on the toolbar. The dialog will open in the left pane.

Set Data Properties

Column Width:

☒ Show data for nulls
 ☒ Use dotted line for nulls

☒ Draw from end to end
 ☒ Show shadow on line

☐ Show null as zero
 ☐ Set translucent

☐ Show data border
 ☐ Draw Series In-Line

Top Label Properties

☒ Show primary
 ☒ Show stack section

☐ Show secondary

Top Label Position

☐ Default
 ☒ Positive side
 ☐ Negative side

Top Label Alignment

Primary:

Stack Section:

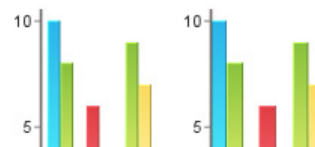
OK

Data Properties Dialog

This data properties dialog contains the following options:

Column Width:

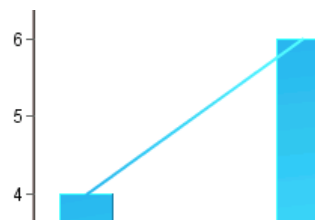
This option specifies the ratio of the bar/column width relatively to the gap between successive bars in the chart. Each unit represents 1/10th of the space between data points. Therefore, entering **9** would leave 10% of the space between data points blank, while **10** would eliminate all space between bars/columns. This option only pertains to two-dimensional bar, column, stack bar, stack column, high-low, HLCO, and Gantt charts.



Column width 9 vs 10

Show data for nulls:

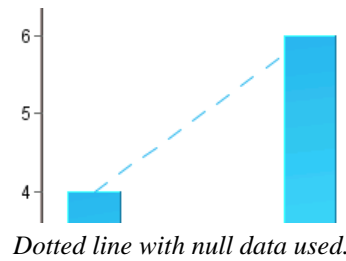
This option will connect null data with lines. For example, if you have three points and the value of point 1 is 4, point 2 is null, and the value of point 3 is 6, then a line will be drawn from 4 to 6 for the three points. This option is only available for line charts or other two-dimensional charts with lines. All other chart types will not plot null data.



Connected line with null data used.

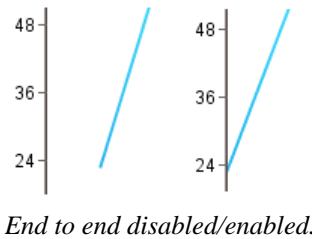
Use dotted lines for nulls:

Instead of drawing a whole line through null values in the chart, you can use this option to draw a dotted line. Like the show data for nulls option, this option is only available for lines.



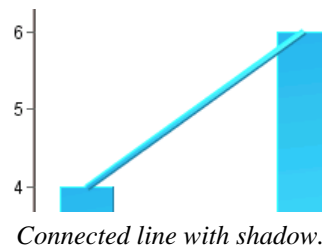
Draw from end to end:

This option allows you to draw 2D line and area charts across the entire plot area, rather than offsetting to the first and last data points on the chart.



Draw shadow for lines:

This option specifies whether to draw shadow on 2D lines. In order for shading to apply, the line must be thicker than one pixel.



Show null as zero:

This option will display 0 for null data.

Set translucent:

This option will draw all data points in translucent color, allowing overlapped elements to be visible. This is useful for radar charts and two-dimensional area charts with data series. This option is also available for bubble and Gantt charts. Note that because this feature requires Java 1.2 or higher, the translucent elements will not display if you are deploying charts in applets and the client is not using the Java plugin.

Show data border:

This option will draw a border around columns and areas.

Draw Series In-Line:

This option will display series in one line. It only pertains to three-dimensional column, stack column, percent column, and bar charts.

Show primary:

This option will display data top labels for the primary values in the chart.

Show secondary:

This option will display data top labels for the secondary values in the chart.

Show stack section:

This option will display individual labels for each stack section for stack bar, stack column, and stack area charts.

Top label position:

This option allows you to specify where the top labels should be drawn. By default, they are drawn above data points if they are positive and below data

points if they are negative. Using these options, you can force the data points to always draw to the positive or negative side of the data.


Label Alignment:

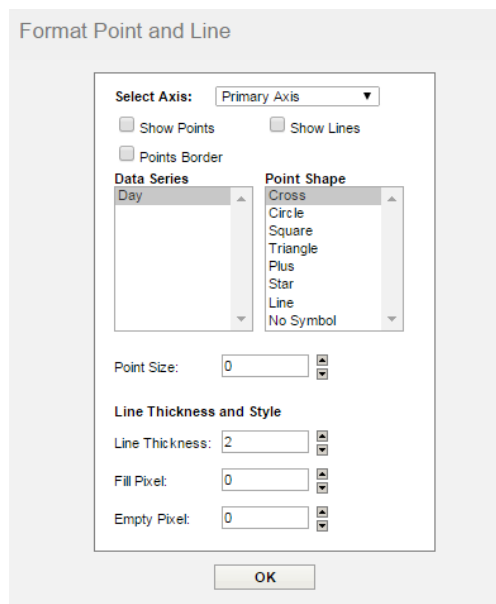
This option allows you to set the alignment for the data top labels. You can draw them at the top, bottom, or middle of the data points. In addition, you can select to draw the label inside the data point at the top or bottom. An additional option for stack charts allows you to set the alignment for stack section labels.

Once you finish setting up the options, click on the *OK* button to apply the changes.

4.3.5.4.2. Line And Point

For any two dimensional chart type, you can choose to display lines and points for all data points in the chart. Note that some chart types already use this representation (i.e. line or scatter charts). Line and point settings can be

adjusted by clicking on the  *Line and Point Settings* button on the toolbar. The dialog will open in the left pane.



Line and Point Dialog

The first three options allows you to specify whether you want to show lines, points, and points border for the chart. For radar charts, you also have the option of showing areas. The remaining options allows you to customize the line and point displays for each element in the data series.


For each data series element, you can specify the point shape you want to use. You can also control the size of the points. The default point size is 0. You can specify point sizes of -1, -2, & -3 which represent sizes of 0.75, 0.5, and 0.25. At -3 (0.25), the point will be drawn as a dot regardless of the selected point shape.

For lines you can specify the line thickness as well as customize a dash pattern. The dash pattern is created by specifying the number of filled pixels and empty pixels (between 0 and 255). The line is then drawn by dividing into segments - the number of filled pixels followed by the number of empty pixels. Setting 0 for both will result in a solid line. Setting 255 for both will result in no line being drawn.

Once you finish setting up the options, click on the *OK* button to apply the changes.

4.3.5.4.3. Trend Line

A powerful feature in QuickDesigner Charts is the ability to add trend lines to charts. Trend lines can help you show more details of a chart's data by exposing and highlighting certain trends. Trend lines settings can be adjusted by

clicking on the  *Trend Line* button on the toolbar. The dialog will open in the left pane.

Trend Line Dialog

There is a list of existing trend lines at the top of the dialog. You can select existing trend line and edit it, remove the selected trend line, and/or create a new trend line.

You can specify a label for the line as well as on which element of the data series the calculation should be based. The following types of trend lines are supported: polynomial of any degree, exponential, logarithmic, power, moving average, exponential moving average, triangular moving average, and cubic B-spline. For moving averages you will need to specify the average period and for a polynomial you will need to specify the curve order. You can also specify the thickness of the line and configure whether a label in legend and the attached text should be shown. In case the chart has data series, you can configure the trendline for a specific series.

Please note that there is a *Stack line* option in the *Trend Line* dialog. This option is only available for Stack Column, Stack Bar, and Stack Area chart. You can select whether the line will be drawn for positive or negative values.

Once you finish setting up the options, click on the *OK* button to apply the changes.

4.3.5.4.4. Plot Area

The plot area is the plane on which the data points are drawn for two-dimensional charts. You can customize the


appearance of the plot area by clicking on the  *Format Plot Area* button on the toolbar. The dialog will open in the left pane.

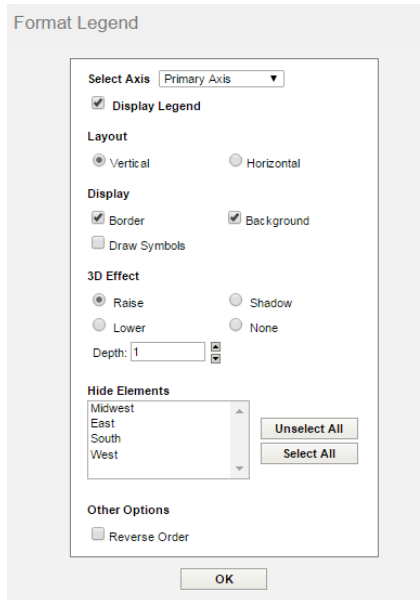
Plot Area Dialog

This dialog allows you to draw a border around the plot area or fill it to provide a background color. If you fill the area, you can also specify several 3D effects like raising, lowering, or drawing a shadow. You can emphasize the 3D effect by increasing the *Depth* option.

Once you finish setting up the options, click on the *OK* button to apply the changes.

4.3.5.4.5. Chart Legend

You can control and modify the display of the chart legend by clicking on the  *Format Legend* button on the toolbar. The dialog will open in the left pane.



Format Legend Dialog


The drop-down list at the top of the dialog allows you to select which legend (primary or secondary) you want to modify. The formatting options are as follows:

- Display Legend:** This option allows you to turn on or off the legend.
- Layout:** This option allows you to change the legend from vertical to horizontal layout.
- Display:** This option allows you to display border, background, and symbols for the legend.
- 3D Effect:** This option allows you to add a 3D effect to the legend. You can raise it, lower it, or draw a shadow. The 3D effect can be emphasized by increasing the *Depth* option.
- Hide Elements:** This option allows you to select certain category/series elements to hide in the legend.
- Other Options:** This option allows you to choose whether or not to draw the legend in reverse order.

Once you have finish seting up the options, click on the *OK* button to apply the changes.

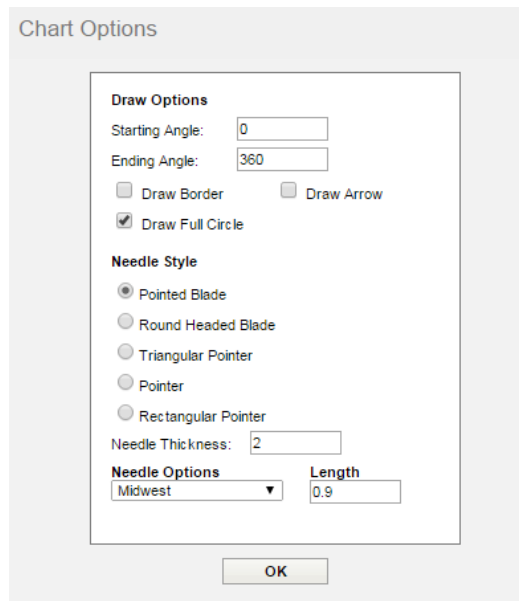
4.3.5.5. Chart-Specific Options

There are a number of formatting options that are unique to certain chart types - Pie Chart, Dial Chart, HLCO

Chart, Gantt Chart, Polar Chart, Doughnut Chart. These options can be modified by clicking on the  *Chart Type Options* button on the toolbar in these types of charts. This will bring up a dialog in the left pane that varies depending on the type of the current chart. Besides the chart types listed here, other chart types have no additional options.

4.3.5.5.1. Dial Charts

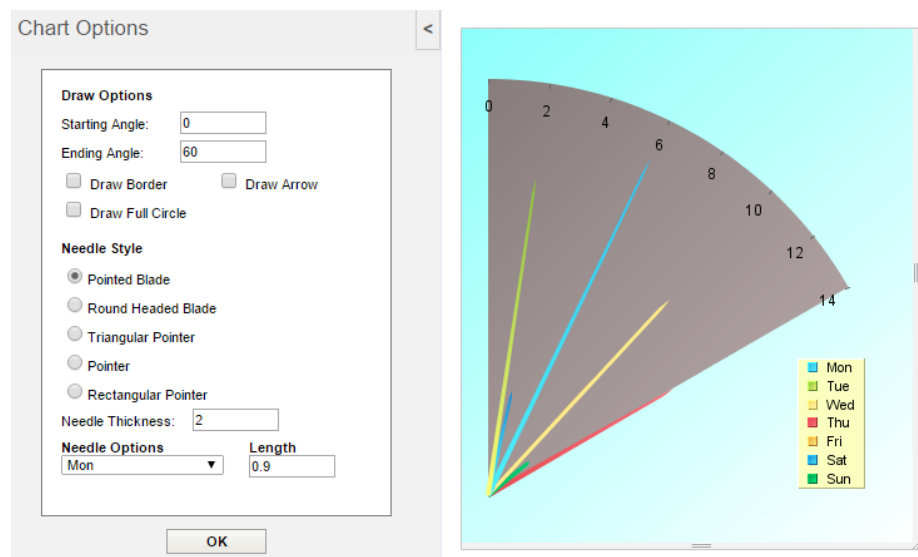
For dial charts, the following dialog is opened:



Dial Chart Options Dialog

The following options are available for dial charts:

- Starting Angle:** This option specifies the angle of the first axis label. The angle is represented in degrees and is set to 0 by default. Assuming the dial chart is a clock face, 0 degrees is 12 o'clock.
- Ending Angle:** This option specifies the angle of the last axis label. The angle is represented in degrees and is set to 360 by default. By default, the labels (and data points) encompass the entire circumference of the dial.
- Draw Border:** This option specifies whether or not to draw a border around the dial.
- Draw Arrow:** This option specifies whether or not to draw arrowheads at the end of the dial hands.
- Draw Full Circle:** This option specifies whether to draw the dial as a complete circle (360 degrees) or only draw the portion of the circle determined by the starting and ending angles.



Circular Sector

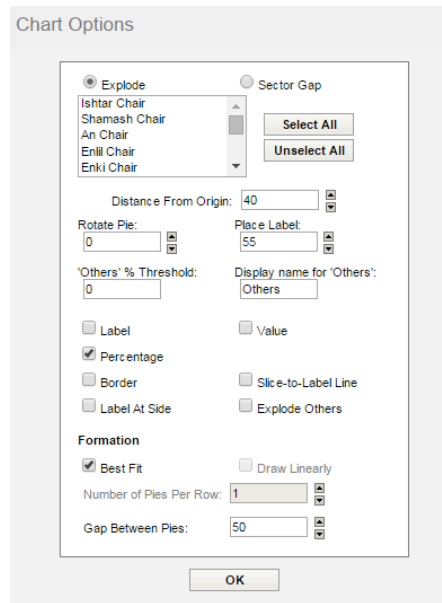
- Needle Style:** This option allows you to set style of the needle. The default one is *Pointed Blade*.

Needle Options: This option specifies the distance of the hand for each needle (if multiple needles exist) from the center of the dial. The range is from 0 (center of the dial) to 1 (end of the dial). You can adjust the needle length for each category by selecting the category from the drop-down list.

4.3.5.5.2. Pie and Doughnut Charts

Please note that doughnut chart specific options are exactly same as for pie charts, so even though the next section will only mention pie charts, the same options also apply for doughnut charts.

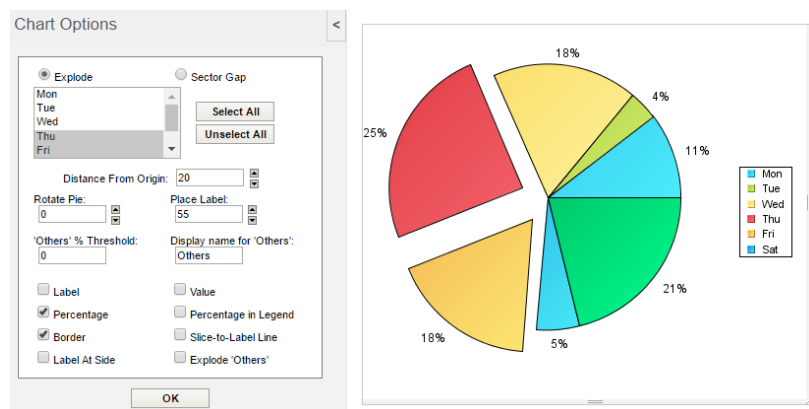
The following dialog is opened for pie charts:



Pie Chart with Series Options Dialog

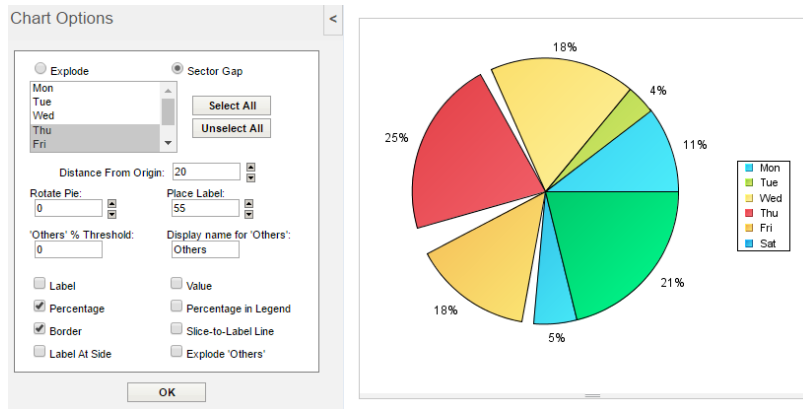
Different options will appear in this dialog depending on whether the pie chart is two-dimensional or three-dimensional and whether or not it has a series. The following options are available for pie charts:

Explode: This option allows you to pick one or more category/series elements whose sections are to be drawn at a certain distance away from the center of the pie.



Exploded Slices

Sector Gap: This option allows you to pick one or more category/series elements whose sections maintain the same distance from the center of the pie. This option creates a gap between the slices.



Sector Gap Applied on Slices

Distance From Origin:

This option allows you to specify how far the exploded sections are to be drawn away from the center. This number, represented as a percentage of the radius, indicates the distance between the center and the tip of the pie slice to be exploded. For sector gap, this option is used to define the gap between the slices representing selected categories/series.

Rotate Pie:

This option allows you to specify the number of degrees that the chart should be rotated. To rotate the pie in a clockwise direction, use negative values (i.e. -90). To rotate it in a counter-clockwise direction, use positive values (between 1 - 360).

Place Label:

This option indicates the distance of the labels from the center of the pie.

'Others' % Threshold:

This feature is useful for pie charts that have a large number of small categories. Rather than drawing a slice for each category, you can select a threshold value. Any category with less than the threshold value will be moved into "Others" slice.

Display Name for 'Others':

This option allows you to set the display name for the "Others" slice that is created for categories that fall below the set threshold value. This label will appear in the legend and/or for the slice label.

Label:

This option determines whether a category/series label should be drawn for each pie slice. By default, it only appears as legend.

Value:

This option allows you to specify whether to display the actual value of each pie slice.

Percentage:

This option allows you to display the percentage for each pie slice. The percentages are calculated by dividing the value of each section by the sum of all the values.

Percentage in Legend:

This option allows you to display the percentage in the legend represented by each slice in the pie. In case a pie slice is too thin, this can be a preferable presentation. This option is only available if the pie chart does not have a data series.

Border:

This option specifies whether to draw a border around each pie slice.

Slice-to-Label Line:

This option will draw a line from any label(s) to its corresponding pie slice.

Label At Side:

This option will place the labels for a pie chart away from the plot around the outside of the chart. When used with the "Slice-to-Label Line" option,

it gives you a way to display the pie labels for charts with many small categories without any text overlapping.

Explode Others:

This option allows you to draw the “Others” segment at a certain distance from the center of the pie chart.

Best Fit:

This option will arrange multiple pies in best configuration to fit the chart canvas. It's only available for pie charts with data series.

Draw Linearly:

This option will arrange multiple pies in a straight horizontal line. It's only available for pie charts with data series.

Number of Pies Per Row:

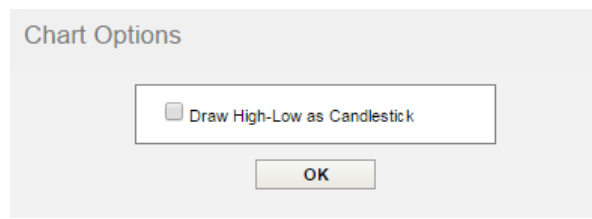
This option allows you to create a custom arrangement of multiple pies by specifying the number of pies to be drawn in each row. It's only available for pie charts with data series.

Gap Between Pies:

This option allows you to specify a gap between multiple pies. The number represent multiple of the pie radius, so the gap will adjust with the size of the chart plot. It's only available for pie charts with data series.

4.3.5.5.3. HLCO Charts

The following dialog is opened for HLCO charts:

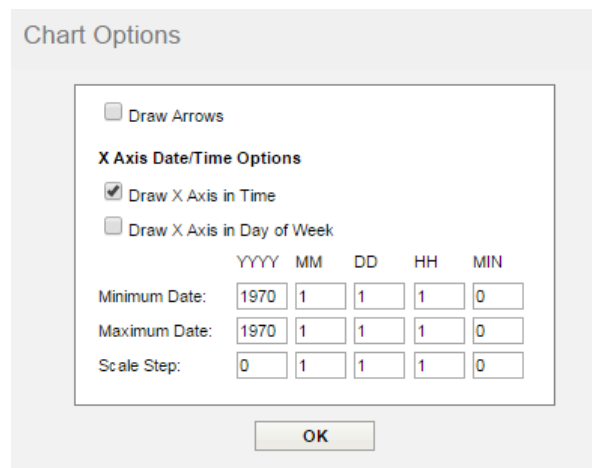


HLCO Chart Options Dialog

The *Draw Hi-Low As Candle Stick* option will turn the HLCO chart into a candle representation. A candle HLCO chart blends high, low, close, and open data into a single object that resembles a candlestick.

4.3.5.5.4. Gantt Charts

The following dialog is opened for Gantt charts:



Gantt Options Dialog

The following options are available for Gantt charts:

Draw Arrows:

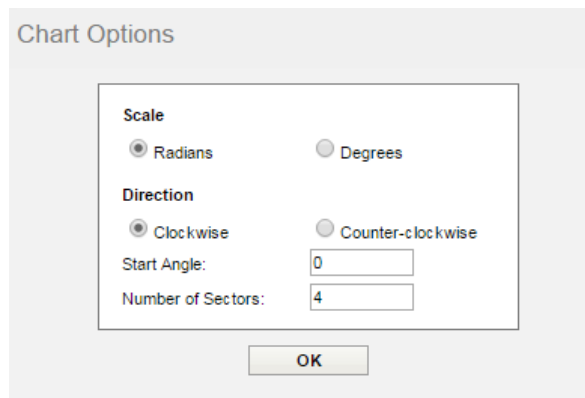
This option will draw connecting arrows between category elements of the Gantt chart. This allows you to illustrate a sequence between scheduled

events. The arrows are drawn in the order the category elements appear in the data source.

Display X-Axis in Time:	This option shows the ticker labels as time values instead of numeric values for the X-axis.
Display X-Axis in Day of Week:	This option shows the ticker labels as days of the week with the date for each Sunday shown as well.
Minimum Date:	This option specifies the beginning date for the X-axis. The format is year, month, day, hour, and minute.
Maximum Date:	This option specifies the end date for the X-axis. The format is year, month, day, hour, and minute.
Scale Step:	This option specifies the scale step for the X-axis. The format is year, month, day, hour, and minute.

4.3.5.5. Polar Charts

For polar charts, the following dialog is opened:





Polar Chart Options Dialog

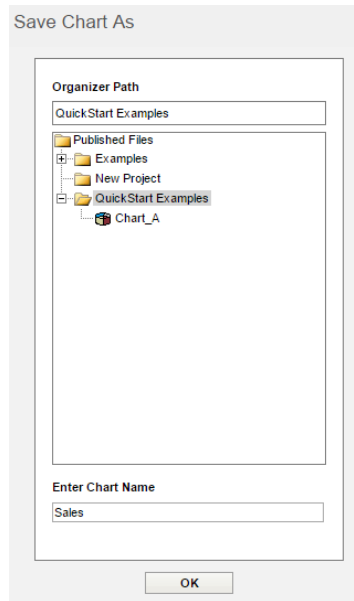
The following options are available for polar charts:

Scale:	This option allows you to specify whether the input data for the angle (θ) portion of the data points is in radians or degrees. The chart will always display angles from 0 to 360. If the input data is in radians, it will be displayed as degrees.
Direction:	This option allows you to specify whether the circular plot should be drawn clockwise or counter-clockwise.
Start Angle:	By default, the top of the polar chart plot is 0 degrees. This option allows you to specify a different angle for the top of the plot. It can be displayed either in degrees or radians, depending on the scale you chose.
Number of Sectors:	This option allows you to select the number of sectors you want to show in the chart. Sectors are created by drawing additional polar axis lines at specified angle intervals. By default, four sectors are shown.

4.3.6. Save the Chart

Once you create a chart in QuickDesigner Charts, you can share it with other users by saving it on the server and inserting it into the Organizer.

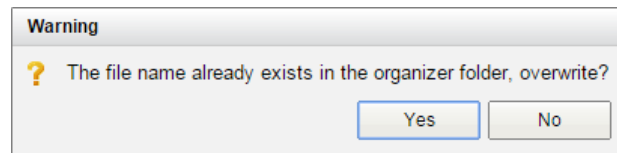
To save a chart, click on the  *Save* (or  *Save Chart As* for saving the file under a different name that is already saved) button on the toolbar. A dialog will open in the left pane.

*Save Chart Dialog*


Here you can see all Organizer projects and folders you have access to. Select the project or folder you want to save the chart into. Specify a name for the chart at the bottom of the dialog. The chart will be saved with .qch extension.

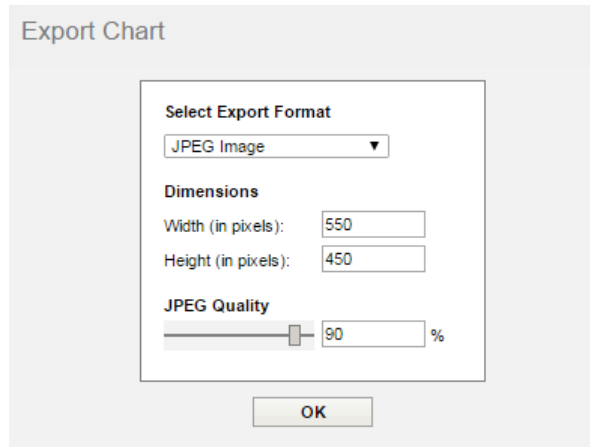
Once you finish specifying the options, click on the *OK* button to save the chart. The new file will be saved in the ChartFiles folder in the EDAB installation directory as ChartNameInOrganizer_Username_TimeStamp.qch. This should help to avoid unintentional overwriting of chart files.

If you try to save a file with the same name as another file, you will be asked to confirm overwriting it or using a different name. If you wish to save the changes in the currently opened chart, do not change the chart name and the Organizer folder in the Save Chart dialog. Click *Yes* in the dialog and the chart file will be updated.

*Chart Overwrite Dialog*

4.3.7. Export the Chart

You can export a chart in a number of different formats. To do so, click on the  *Export* button on the toolbar and a dialog will open in a new window prompting you to specify the export options.



Export Chart Dialog

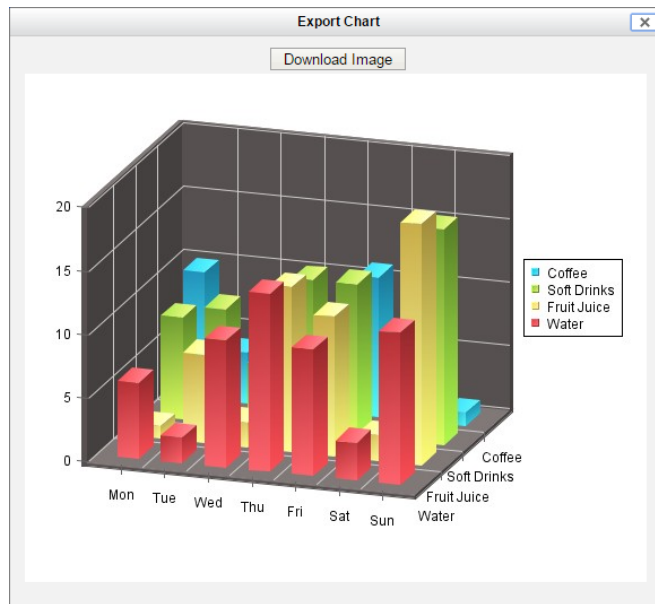
The first option allows you to specify the format in which to export the chart. The available formats are GIF, JPEG, PNG, BMP, PDF, SVG, SWF (Flash), Text, XML, and Chart Image Map.

The second option allows you to set the dimensions of the exported image. By default, these will match the canvas dimensions of your chart.

The last options allow you to set some image type-specific options. (The options appear only for appropriate format.) You can set the background transparent for GIF files (please note that the result is not good for more than 256 colors - e.g. with Color Gradient), set the quality for JPEG files, as well as specify compression for PNG files.

For more information about the chart export options, please see Section 4.1.6.2 - Exporting Charts.

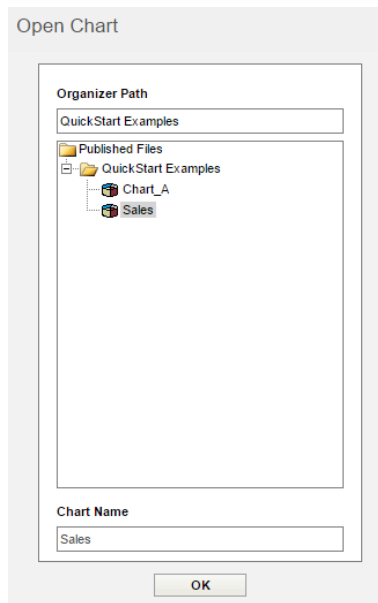
Once you finish specifying the options for the exported chart, click on the *OK* button and a new window will open containing the exported chart. You can save the generated file to your local system by clicking on the *Download Image* button.



Exported Chart

4.3.8. Open the Saved Chart



You can open a saved chart by clicking on the  *Open* icon on the main toolbar. The *Open Chart* dialog will appear.



Open Chart Dialog

Here you can see all charts created in QuickDesigner Charts. Select a chart and then click on the *OK* button to open it.

4.3.9. Exit

You can exit QuickDesigner Charts either by clicking on the  *Home* icon in the upper right corner, clicking on the *QuickDesigner Charts* title, or by clicking on the  *Logo* icon in the upper left corner. Before closing, you will be asked if you want to save an unsaved chart.

Chapter 5. Designing Maps

5.1. Introduction to EDAB Maps

EDAB Maps are designed to report geographical data from data sources. They fetch geographical data from data source and mark them on a map. There are two types of maps: Online Maps and SVG Maps.

Online Maps The **Online Maps** feature in EDAB displays geographical data using high quality on-line maps made by Google and Open Street Maps. Online Maps provides street maps and satellite maps, spanning over the entire world. By using high quality satellite imagery, a very wide zoom range is possible, making it possible to display every imaginable area of the world. Online Maps are useful for displaying exact spots on the maps. These spots are called **map points** and they are marked with **map markers**. A powerful tool provided by EDAB Online Maps is **Geocoding**. Geocoding can locate buildings/cities on map by knowing only their addresses. Usage of Online Maps is limited by Online Maps Terms of Service (To see the current version, visit Google Maps API Terms of Service [<https://developers.google.com/maps/terms.html>], Open Street Maps API Terms of Service [<https://www.openstreetmap.org/copyright/en>]).

Online Maps can contain tooltips and drilldowns. **Tooltips** are small reports or charts which are displayed on mouse over a map marker. They display brief summary information related to the selected map point. **Drilldowns** are links to parameterized charts, reports or other maps that open when you click on a map marker. They are usually used to display more detailed information about the selected map point.

SVG Maps **SVG Maps** display data on SVG (Scalable Vector Graphics) maps. They differ from EDAB Online Maps in two ways. First, SVG Maps work with geographical **areas** (e.g. a state is marked by the whole territory, not just by a point in the middle of the state). These areas can be colored according to set criteria. Second, the SVG Maps feature requires **SVG images** as the map source and do not use any online map source. The SVG Maps feature may be used to display spatial (non-geographic) and customized maps (e.g. maps of buildings, floor plans, mine maps, etc.).

5.2. Online Maps

EDAB Online Maps use two types of files: Coordinates and Map Files. **Coordinates** contain location of individual map points (e.g. cities, company branches). Coordinates have to be created before creating a map. One set of Coordinates can be shared by several maps, so you do not have to store the same file several times. It is enough to add a map point in the Coordinates and it is added to all maps that use these Coordinates. For more information about Coordinates, see Section 5.2.4.2 - Create Coordinates. **Map File** displays map markers based on Coordinates and defines Drilldowns, Tooltips, and Heatmaps.

5.2.1. Generating Google Maps API Key

It is possible to use EDAB Maps without Google API Key if you don't select either Google Street Map, Google Satellite Map or Geocoding.



Tip

You can use Open Street Maps even without the API key

Open Street Maps are a free alternative to Google Maps. Select Open Street Map in the Map Options dialog if you don't need Google Maps.

Since the time Google Maps were embedded into our products, Google has change their ToS. To keep the EDAB Maps compatible, our developers added the option to use your Google API Key in EDAB.

To use Google geocoding and Google Maps, you have to get the correct Google API key set up.

All the steps necessary for enabling Google Street Map, Google Satellite Map or Google Geocoding:

1. If you don't have a Google Cloud Platform account, create one <https://cloud.google.com/console/google/maps-apis/overview>
2. Log in To Google Cloud Platform here <https://cloud.google.com/console/google/maps-apis/overview>
3. Create a new project there
4. Select the project
5. Select APIs & Services → Credentials
6. Create credentials → API key

Then enable the services you need.

For using Google Maps without geocoding, you have to have “Maps JavaScript API” enabled

For using Geocoding, you have to have both Maps “JavaScript API” and “Geocoding API” enabled

To enable Maps JavaScript API

1. Click *APIs*
2. Click *Maps JavaScript API*
3. Click *ENABLE*

To enable Geocoding API

1. Click *APIs*
2. Click *Geocoding API*
3. Click *ENABLE*

To check quotas and billing:

- Click *APIs* again
- In the list of enabled APIs, select the *Geocoding API*
- Click *Quotas*
- Scroll down to *Requests*
- Find “Requests per day” and click the pencil *Edit* icon
- As you can see, you cannot increase the limit for more than 1 request per day unless you enable billing
- Click *Enable billing for this project*.
- Then you can either use an existing credit card information saved in Google or create a new one.

5.2.2. Data Sources

Each EDAB Online Map uses data from two data sources - the first one is used for Coordinates and the second one is for the map itself. This chapter explains where and how these data sources are used.

The Coordinates data source is used when you are creating Coordinates. It is used for obtaining all the Point IDs. The **Point ID** is an identifier of a place (e.g. city name, branch name etc.) and it is loaded from the data source where you selected to create the Coordinates. The Point ID can consist of several **Point ID fields** - each data source column creates one field (except the fields which contain longitude, latitude, or WKT). For example, if your data source contains two columns - City and State, the Point ID will consist of two fields - City and State.

Each map point should have its unique Point ID, so any map point can be distinguished from the others. The data source should be chosen wisely to fulfill this condition (each row of data should be unique, there must not be duplicates).

It is recommended not to use parameterized data source for Coordinates, because there is no way to set parameter values for Coordinates. If you use a parameterized data source for Coordinates, the default parameter values are always used. This does not apply to multivalue parameters which use all the values (e.g. if you use CustomersInStates HSQL query (Databases/Woodview/Query/CustomersInStates) to create coordinates using geocoding - all param values (states) will be used).

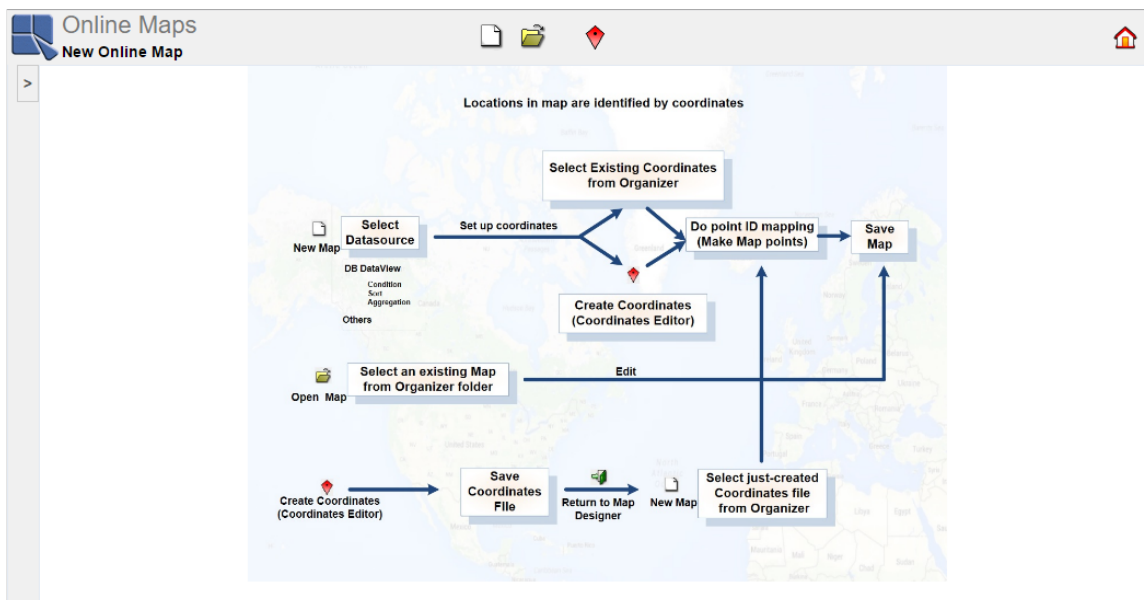
The Point ID is used for mapping map markers to the map data source in the Map File, so it should contain data which occurs in the map data source and which uniquely identifies map points. For example, if you want to display cities where your company has some customers, you should use City name and State name as the Point ID fields. The State name is useful in case when there are more cities with the same name in several different states. Optionally, you can add more Point ID fields for more detailed specification of city. For example if there are two cities with the same name in the same state, you should use more Point ID field(s) which will distinguish between these two cities, a ZIP code for instance.

The Coordinates data source can also contain coordinates (longitudes, latitudes, or WKT) of the map points. The longitudes and latitudes have to be stored in two separated columns of double data type, while WKT is a text field. These columns are not used as Point ID fields. If the coordinates are read from the data source, they are fetched every time the map is generated, so if there is a change in the Coordinates data source, it will be reflected in the map immediately. If the coordinates are not read from the data source, they are stored directly in the Coordinates file. In this case, the Coordinates data source is accessed only when you are creating or editing the Coordinates in the Coordinates Editor. When you are creating or displaying a map that is using this Coordinates, the data source is not accessed at all, because all the necessary information is in the Coordinates file, so it is not necessary for this Coordinates data source to be available when the map is created/displayed.

The map data source has two purposes. The first one is that it contains data that are used for Tooltips and Heatmaps. The second purpose is filtering. The map data source is mapped to the Coordinates as described in the Section 5.2.5 - Coordinates Mapping and only those map points from the Coordinates that have at least one associated row in the map data source are displayed. The map data source can be parameterized. In this case, the whole map is parameterized. If you create/open map that is using parameterized data source, there will be a filter icon in the map which opens parameters dialog that allows you to enter parameter values.


5.2.3. Start Online Maps

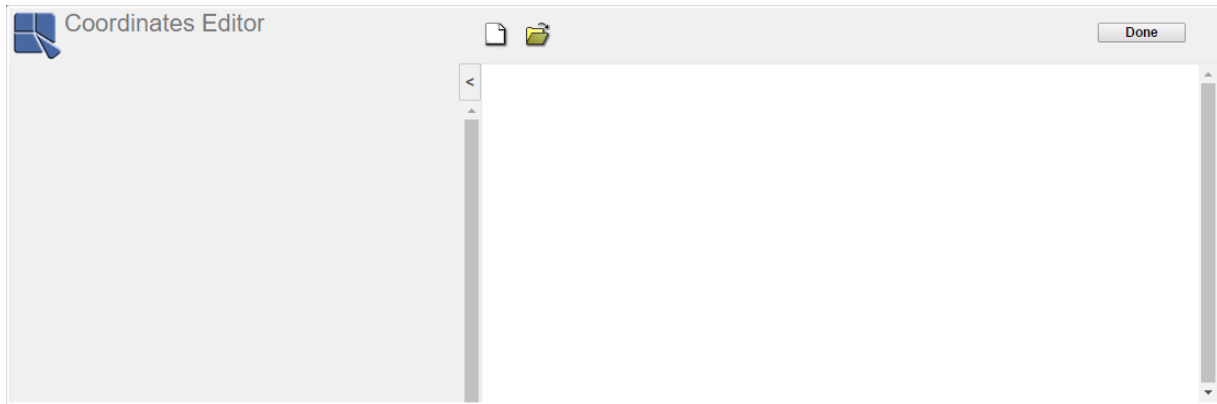
Online Maps can be started by clicking on the *Online Maps* link on the EDAB Main page. After launching Online Maps, you will see a diagram that shows how to work with Online Maps. You can choose to create a new map, open an existing one, or open the Coordinates Editor where you can create/edit Coordinates.



Online Maps Start Options


5.2.4. Coordinates Editor

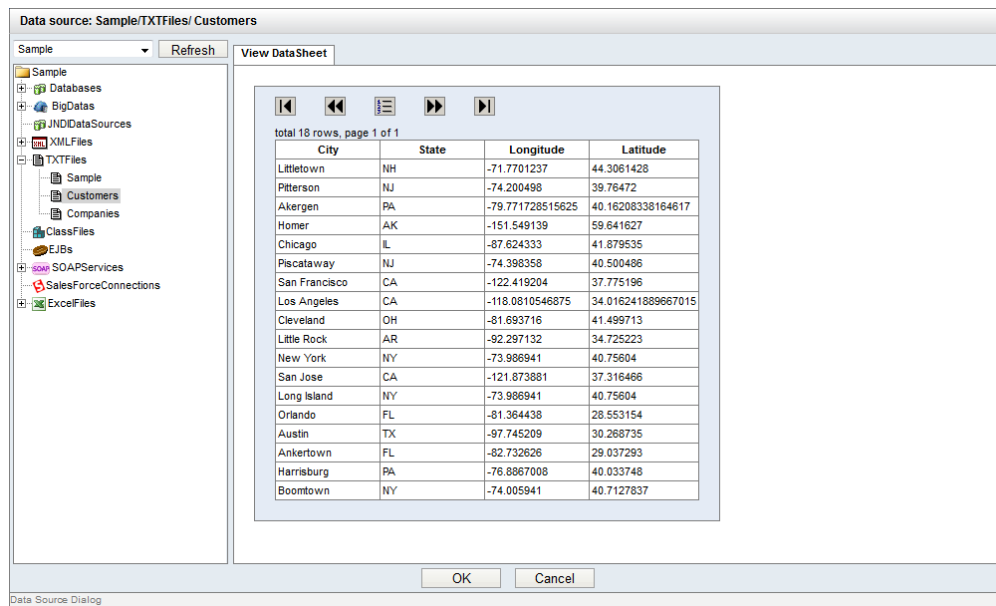
Click on the  *Coordinates Editor/Edit Coordinates* icon on the toolbar. The *Coordinates Editor* will open.



Coordinates Editor Start Options

5.2.4.1. Select Data Source

Open the *Data Source Dialog* by clicking on the  *New Coordinates* icon on the toolbar. Select the data registry in the upper left corner of the dialog and then select the desired data source (For more information about managing data registries, visit Section 3.1.1 - Managing Data Registries). A view of the data will appear in the right pane, which will allow you to check whether you chose the right data source. Click on the *OK* button to close the *Data Source Dialog*.

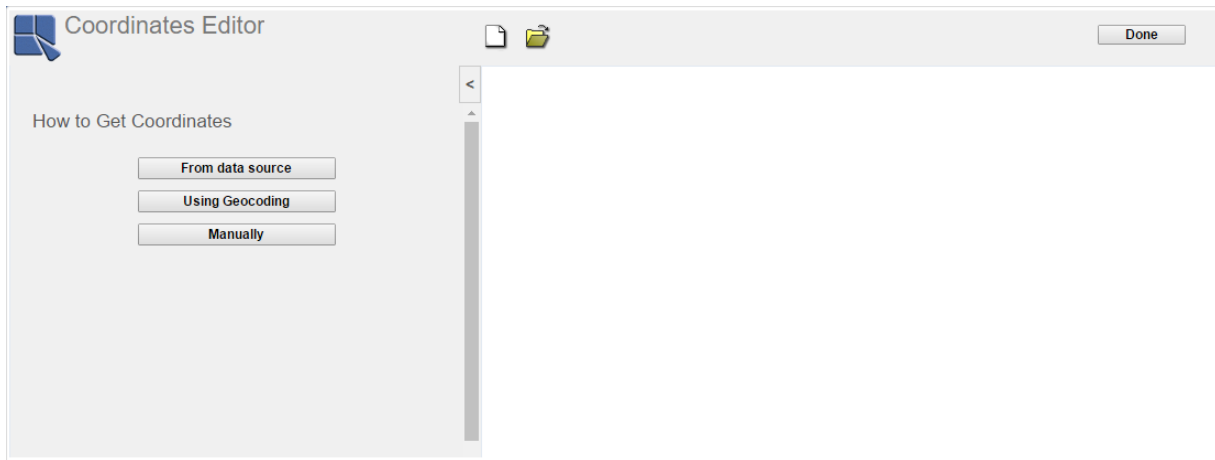


Data Source Dialog

5.2.4.2. Create Coordinates

The **Coordinates** file contains the coordinates of map points. Each Coordinates data record consists of WKT (obtained automatically or manually) and Point ID (for more information about Point ID, see Section 5.2.2 - Data Sources). Note: Existing coordinates files or data sources with longitude and latitude columns are still supported, but only WKT will be generated by geocoding or manual entry.

After selecting the data source, you will be prompted to select a method for obtaining Coordinates from the data source.

*Get Coordinates*

There are three ways how to get Coordinates, which are described below. Status of each pointID is determined by colors in the pointID table (For more information about colors, see Section 5.2.4.3 - Coordinates Editor Interface):

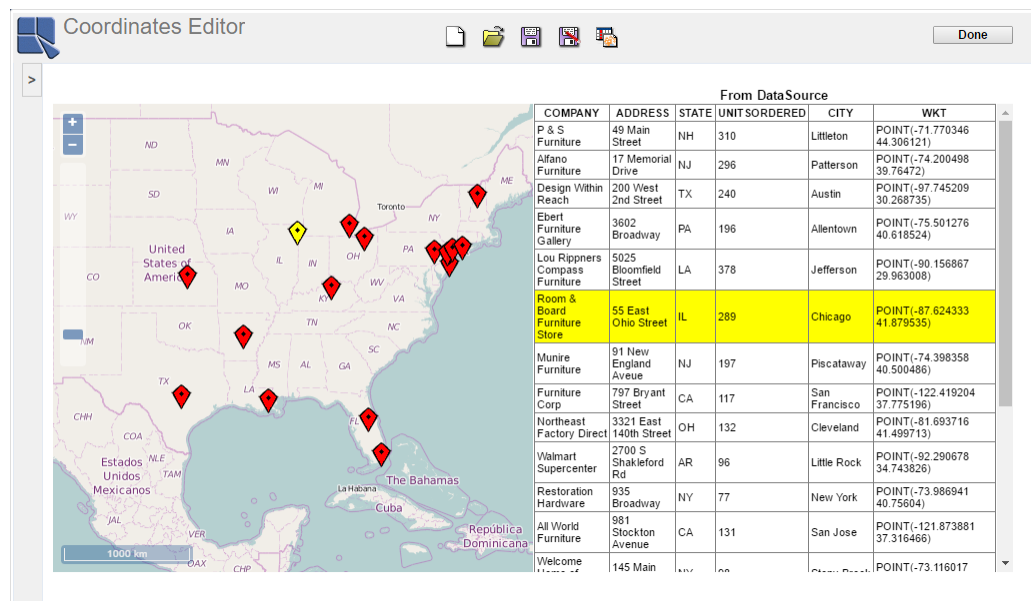
5.2.4.2.1. From Data Source

This option requires two numeral data source columns containing information about the latitude and longitude or WKT of particular map point. Latitude and longitude have to be in different columns. The data type of these columns should be double and they should contain latitude and longitude in decimal degree notation. Positive longitude means East longitude, negative means West. Positive latitude means North latitude, negative means South.

In this case, map point latitudes and longitudes are read from the data source every time a map, which is using this Coordinates, is open. If the data source changes, the change is reflected in the map (e.g. if a new map point is added, it will be displayed on the map). It is not necessary to edit the Coordinates, updating the data source is good enough.

If Geocoding or Manual insertion of the map point coordinates is used, it is necessary to edit the Coordinates every time you want to add or modify (move) a map point.

If the Coordinates are obtained from a data source, all map points will be placed automatically according to the data from the data source. These rows will have white color. In this case, the Coordinates are read-only - you can only check which map marker belongs to which Point ID by holding mouse over a map marker/Point ID. The associated Point ID/map marker will be highlighted with yellow color. If you want to change the position of a map point, you have to modify the data source directly (i.e. change the longitude and latitude or WKT in the appropriate data source column).



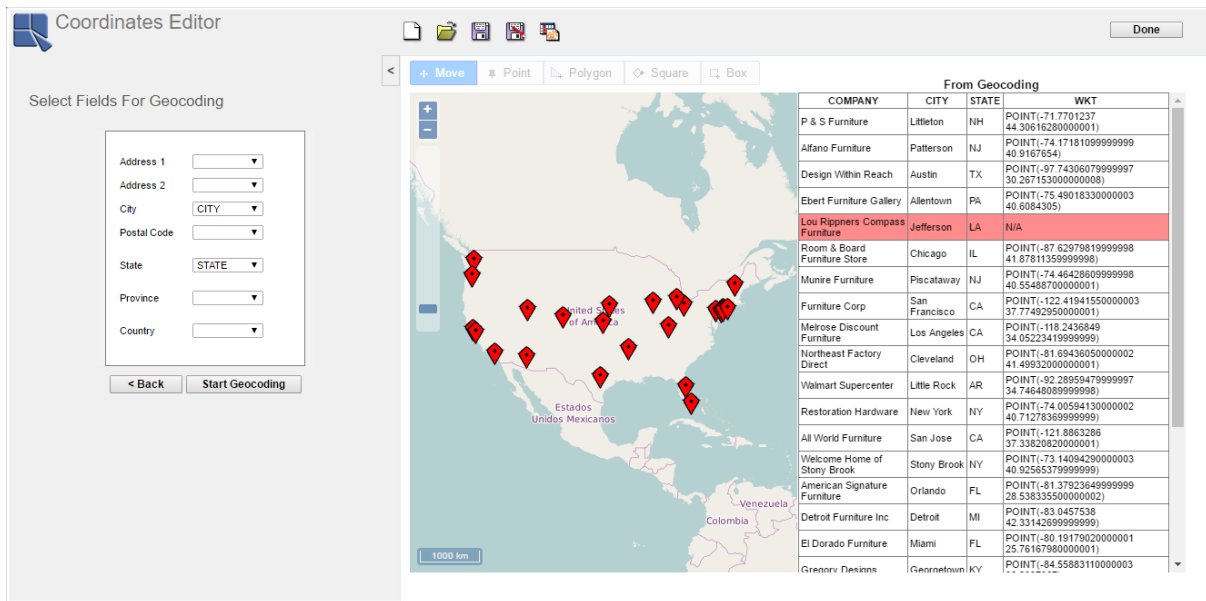
Coordinates from Data Source

5.2.4.2.2. Geocoding

Positions will be determined automatically using point addresses. The addresses can consist of several data source columns. You can select columns, which contain Address 1, Address 2, City, Postal Code, State, Province, and Country. Some of the fields may already be pre-selected when there is a convenient column in your data source (e.g. if the data source contains a column called *City*, it will be pre-selected in *City* field). You do not have to fill all the fields. If you do not fill any field, you will be automatically switched to manually option (see the next chapter). The name of the Geocoding fields are only illustrative, they do not have to be filled with exactly this information. The only rule you should keep in mind is to use the more specific information (e.g. street address) before the less specific (e.g. state). If your data source column contains more than just one field, use the first suitable field (e.g. if there is column with full address, you should use Address 1 field and leave the rest empty). Geocoding is very adaptive as it does not require any specific format of addresses and is able to find even inaccurate or incomplete addresses.

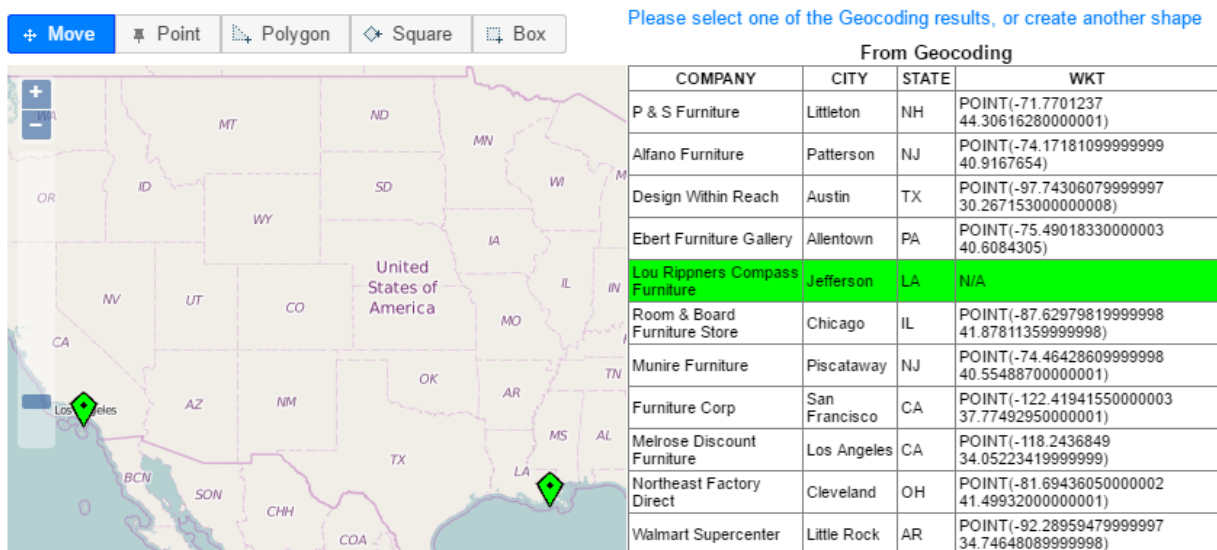
Please note that Geocoding may not be supported for all countries. To see the list of currently supported countries, visit Google Maps Coverage [<https://developers.google.com/maps/coverage>] and find your country in the list.

If you selected Geocoding, the Geocoding process is started automatically. It goes through all Point IDs and tries to determine positions of the corresponding points from their addresses. If it finds exactly one result, the map point is inserted automatically. The new point and the corresponding Point ID will turn yellow for a moment and then it will turn white, which indicates that the position is already determined. The toolbar is disabled during Geocoding, so you have to wait until it is finished before performing other actions. You can watch the Geocoding progress on the status bar. After the Geocoding is done, you should insert points that were not inserted automatically. These points had either no Geocoding results or more than one Geocoding result. The process is same as inserting the point manually and it is described below.



Coordinates by Geocoding

If you used Geocoding and you insert the rest of the points manually, the Point ID row may turn green instead of dark red after you click on it. This occurs when there is more than one Geocoding result. These results are displayed on the Online map with green markers so you can select the right one by clicking on it. If the desired map point is not among the Geocoding results, you can still add the map point manually on the correct place in the same way as manual inserting. If you hold mouse over a Geocoding result (green map marker), the full address of this map point is displayed to help you identify the correct place.



Selecting Geocoding Result

Please note that the Geocoding is not 100% accurate, so you should check all results that were inserted automatically.

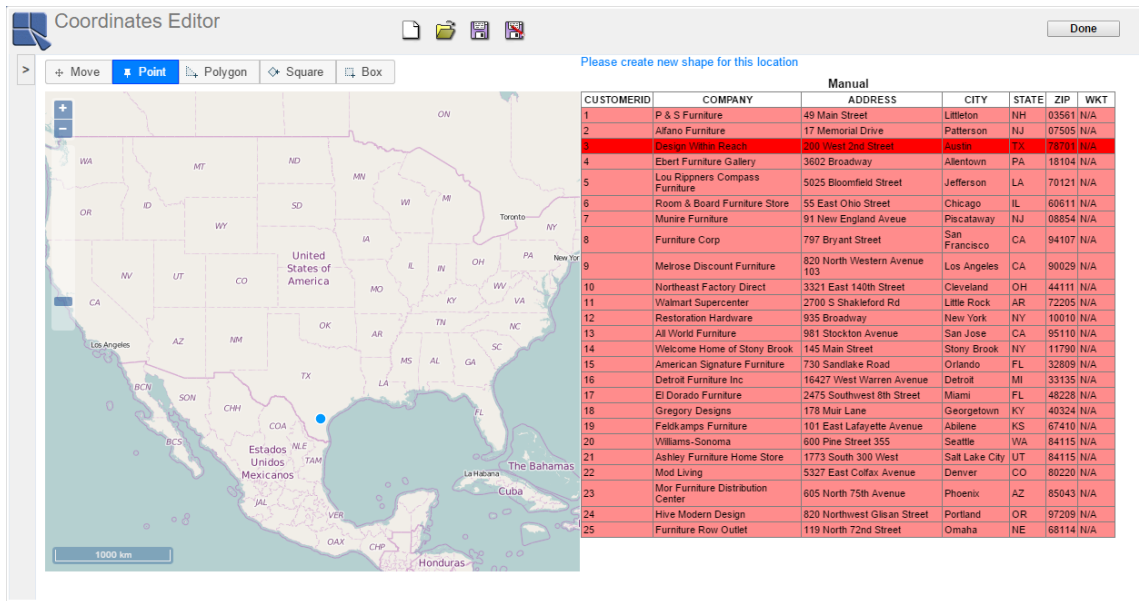
5.2.4.2.3. Manually

This option does not require any geographical data at all. Coordinates of map points are determined manually by clicking on the map.

In the *Manually* case, all the Point IDs are light red at the beginning, which means that their position is not determined. To place a map point, click on a Point ID (row in the table) first. The Point ID row will now turn dark red. *Point* option should be active as default on the map toolbar (*Point* is blue). Mouse over the map and you will see a small blue circle under your mouse pointer. Then click on the location where you want to place your map point.

Designing Maps

A new map marker will be placed on the selected map point and the appropriate row will become white. Then you can place another map point in the same way. To freely navigate across the map, select *Move* option on the toolbar. Now you can move and zoom the map any way you need. You can also use another shapes to define a location on the map - *Polygon*, *Square*, and *Box* on the map toolbar. To use these options, select the shape, place the first point on the map by clicking the mouse, move the mouse and place another point(s).



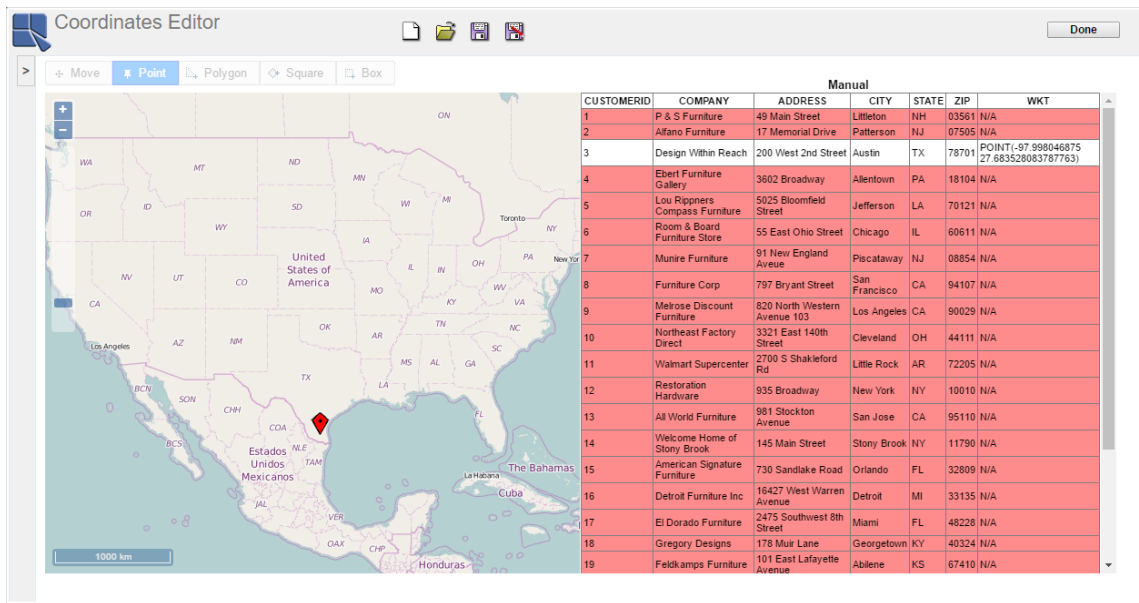
Coordinates Editor

Please create new shape for this location

Manual

CUSTOMERID	COMPANY	ADDRESS	CITY	STATE	ZIP	WKT
1	P & S Furniture	49 Main Street	Littleton	NH	03561	N/A
2	Alfano Furniture	17 Memorial Drive	Patterson	NJ	07505	N/A
3	Design Within Reach	200 West 2nd Street	Austin	TX	78701	N/A
4	Ebert Furniture Gallery	3602 Broadway	Allentown	PA	18104	N/A
5	Lou Rippners Compass Furniture	5025 Bloomfield Street	Jefferson	LA	70121	N/A
6	Room & Board Furniture Store	55 East Ohio Street	Chicago	IL	60611	N/A
7	Munire Furniture	91 New England Avenue	Piscataway	NJ	08854	N/A
8	Furniture Corp	797 Bryant Street	San Francisco	CA	94107	N/A
9	Meirose Discount Furniture	820 North Western Avenue 103	Los Angeles	CA	90029	N/A
10	Northeast Factory Direct	3321 East 140th Street	Cleveland	OH	44111	N/A
11	Walmart Supercenter	2700 S Shakelford Rd	Little Rock	AR	72205	N/A
12	Restoration Hardware	935 Broadway	New York	NY	10010	N/A
13	All World Furniture	981 Stockton Avenue	San Jose	CA	95110	N/A
14	Welcome Home of Stony Brook	145 Main Street	Stony Brook	NY	11790	N/A
15	American Signature Furniture	730 Sandlake Road	Orlando	FL	32809	N/A
16	Detroit Furniture Inc	16427 West Warren Avenue	Detroit	MI	33135	N/A
17	El Dorado Furniture	2475 Southwest 8th Street	Miami	FL	48228	N/A
18	Gregory Designs	178 Muir Lane	Georgetown	KY	40324	N/A
19	Feldkamps Furniture	101 East Lafayette Avenue	Ablene	KS	67410	N/A
20	Williams-Sonoma	600 Pine Street 355	Seattle	WA	48115	N/A
21	Ashley Furniture Home Store	1773 South 300 West	Salt Lake City	UT	84115	N/A
22	Mod Living	5327 East Colfax Avenue	Denver	CO	80220	N/A
23	Mor Furniture Distribution Center	605 North 75th Avenue	Phoenix	AZ	85043	N/A
24	Hive Modern Design	820 Northwest Glisan Street	Portland	OR	97209	N/A
25	Furniture Row Outlet	119 North 72nd Street	Omaha	NE	68114	N/A

Selecting Location



Coordinates Editor

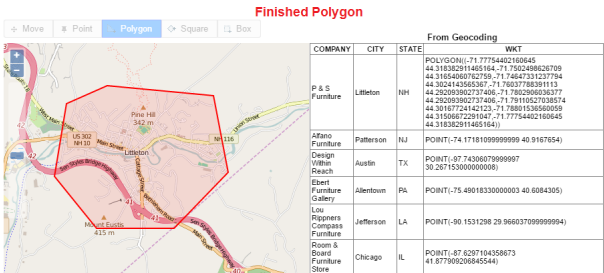
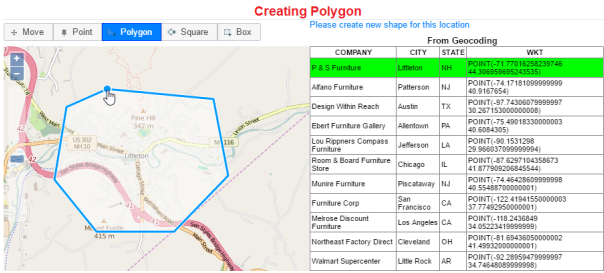
Please create new shape for this location

Manual

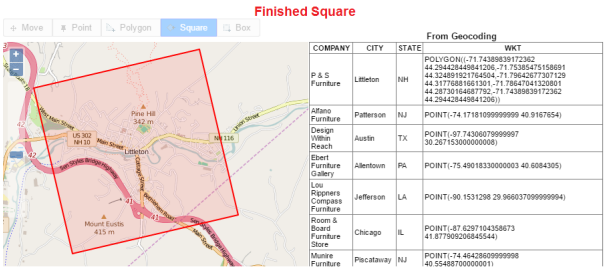
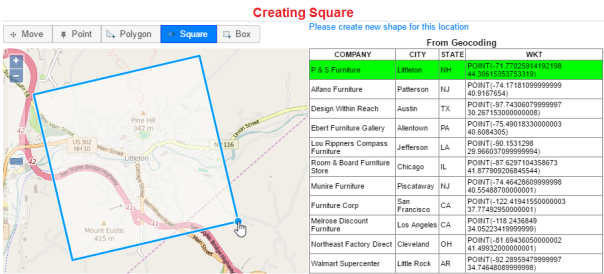
CUSTOMERID	COMPANY	ADDRESS	CITY	STATE	ZIP	WKT
1	P & S Furniture	49 Main Street	Littleton	NH	03561	N/A
2	Alfano Furniture	17 Memorial Drive	Patterson	NJ	07505	N/A
3	Design Within Reach	200 West 2nd Street	Austin	TX	78701	POINT(-97.998046875 27.683528083787763)
4	Ebert Furniture Gallery	3602 Broadway	Allentown	PA	18104	N/A
5	Lou Rippners Compass Furniture	5025 Bloomfield Street	Jefferson	LA	70121	N/A
6	Room & Board Furniture Store	55 East Ohio Street	Chicago	IL	60611	N/A
7	Munire Furniture	91 New England Avenue	Piscataway	NJ	08854	N/A
8	Furniture Corp	797 Bryant Street	San Francisco	CA	94107	N/A
9	Meirose Discount Furniture	820 North Western Avenue 103	Los Angeles	CA	90029	N/A
10	Northeast Factory Direct	3321 East 140th Street	Cleveland	OH	44111	N/A
11	Walmart Supercenter	2700 S Shakelford Rd	Little Rock	AR	72205	N/A
12	Restoration Hardware	935 Broadway	New York	NY	10010	N/A
13	All World Furniture	981 Stockton Avenue	San Jose	CA	95110	N/A
14	Welcome Home of Stony Brook	145 Main Street	Stony Brook	NY	11790	N/A
15	American Signature Furniture	730 Sandlake Road	Orlando	FL	32809	N/A
16	Detroit Furniture Inc	16427 West Warren Avenue	Detroit	MI	33135	N/A
17	El Dorado Furniture	2475 Southwest 8th Street	Miami	FL	48228	N/A
18	Gregory Designs	178 Muir Lane	Georgetown	KY	40324	N/A
19	Feldkamps Furniture	101 East Lafayette Avenue	Ablene	KS	67410	N/A

Selected Location

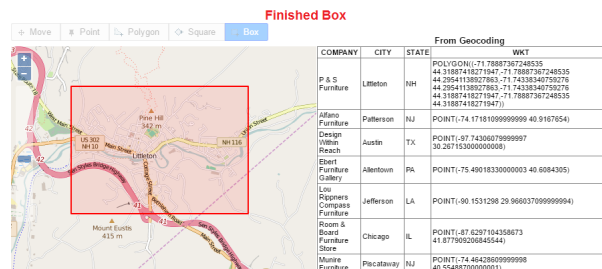
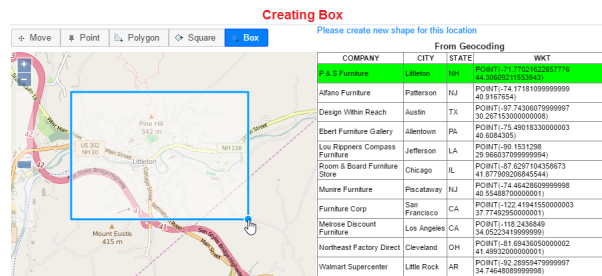
The following images show the formation of shapes (blue color) and their final form (red color).



Polygon



Square

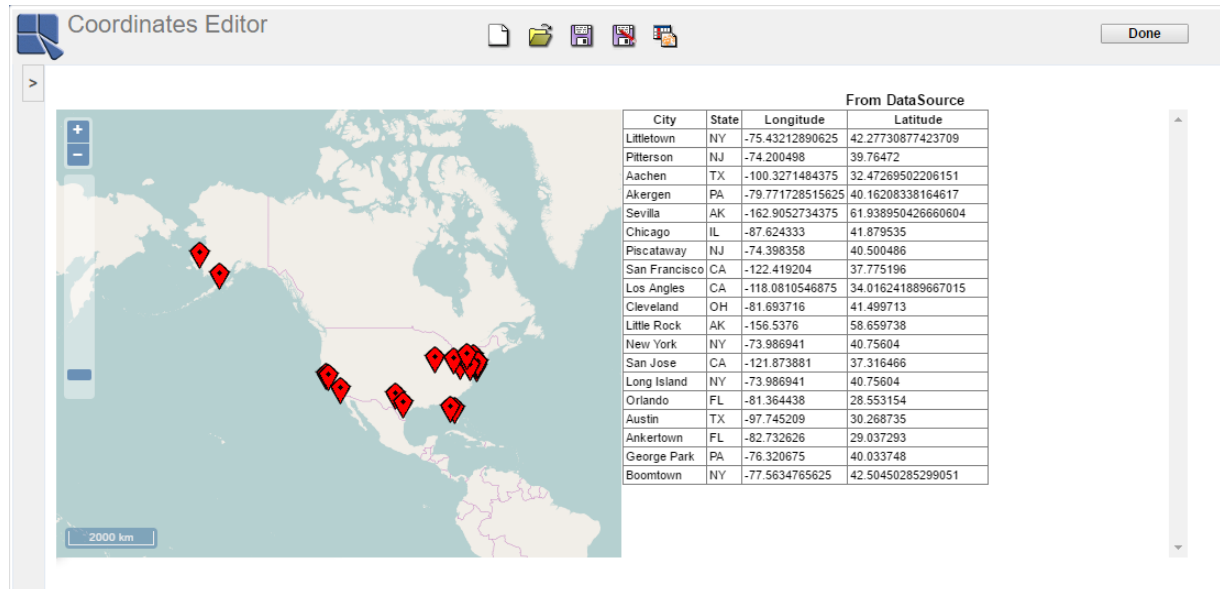


Box

You can move any of the already existing map points by dragging the appropriate map marker with mouse. You can also resize a shape by dragging the edge of a side to any direction.

5.2.4.3. Coordinates Editor Interface

After choosing a method of obtaining coordinates, you will get a map with a point ID table.



Coordinates Editor

The following buttons are on the toolbar:



New: Creates new Coordinates.



Open: Opens existing Coordinates.



Save: Saves current Coordinates to a file and Organizer. Note that all saved files will include userID and timestamp in addition to the name you provided. If an Organizer item with the supplied name already exists in the folder, you will be given a choice to override the existing item or create a new entry.



Save As: Allows you to save the existing or modified Coordinates to a different file or location.



Change Point Mapping: Allows you to change point mapping.

The left part of the window displays the Online Map, which displays individual map points using the map markers. The Online Map can be scrolled and zoomed to find a specific object.

Point ID table in the right part of the window shows all the Point IDs, which were read from the data source, and longitude and latitude of map points. Each row represents one Point ID and can have one of the following background colors:



White: Location of the corresponding map point has been successfully determined.



Light red: Location of the corresponding map point has not been determined yet. There are multiple locations for the point. This is not static status color.



Grey: Location of the corresponding map point has not been determined yet. There is no location for the point. When you click this grey-colored row, it turns into dark red. This is not static color.



Dark red: Marks the point ID, which is currently under mouse cursor. If you click on it, it will either change into green (multiple locations), or grey (No location found), waiting for users to enter the location manually.




Yellow: Highlights the row with Point ID of the map point under mouse cursor on the map (this is used to determine which Point ID belongs to which map marker).

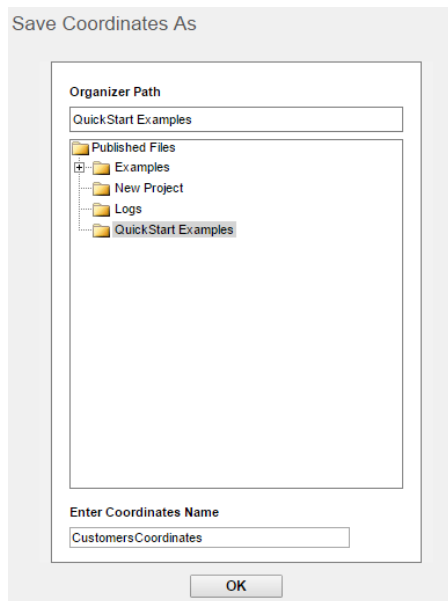


Green: Indicates that there are Geocoding matches for the selected Point ID.

5.2.4.4. Save Coordinates





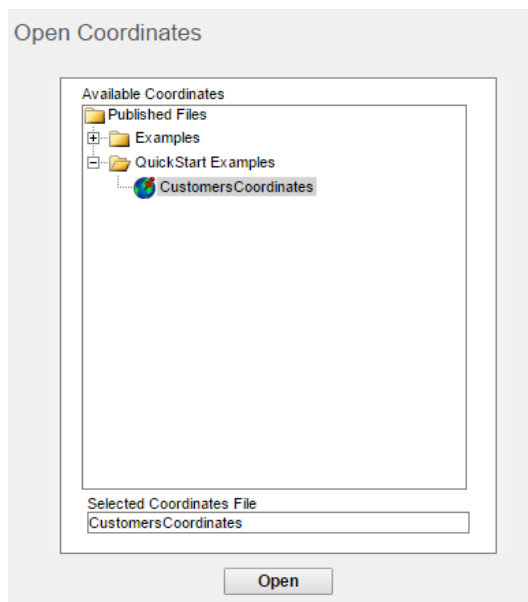
If you want to use the Coordinates for creating maps, be sure to save it to Organizer (click the  *Save* icon, enter a Coordinates name, and select any Organizer folder). Then click the *Done* button on the right side of the toolbar to close the Coordinates Editor and go back to Online Maps main window.



Save Dialog

5.2.4.5. Open, Edit Coordinates

To open a saved coordinates file, first open Coordinates Editor by clicking the  *Coordinates Editor/Edit Coordinates* icon on the main toolbar of the Online Maps designer. Then click the  *Open* icon on the main toolbar of Coordinates Editor.

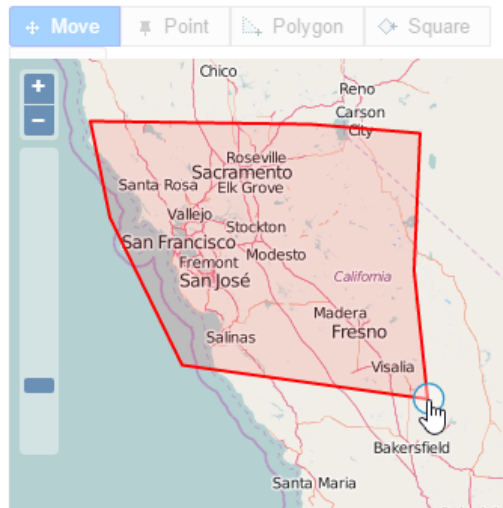


Open Coordinates

You can see all coordinates files from Organizer. Select a coordinates file and click the *Open* button.

Now you can edit coordinates obtained via geocoding or manual insertion. Coordinates obtained from data source are not possible to edit. You can change a point when you click the respective row in the pointID table - then you can either select again one of the geocoding results (applies only to geocoding coordinates) or replace a current point/shape with a new one in the map.

You can directly edit a map point using the drag and drop feature or edit a shape by stretching its sides.



Shape Editing

Please note that if the coordinates obtained via geocoding are not complete (not all rows are white), the geocoding process starts each time the file is opened.

Once you finish editing the coordinates, save the file and click the *Done* button to close the Coordinates Editor.

5.2.5. Coordinates Mapping

EDAB Online Maps needs Coordinates to map the data source records to map points. When creating Online Maps, you have to map the Point ID fields from the Coordinates file to the map data source columns. According to this mapping, data source rows are associated with map points. There can be more data source rows associated with one map point but not vice versa (because Point IDs must be unique). The rows associated with a map point are used to generate Tooltips (see Section 5.2.6.7 - Tooltips) for the appropriate map marker.

Not all the Point ID fields have to be mapped. For example, if your Point ID has three fields - City, State, and Country, you can map only Country and State. But you have to keep in mind that any map point should be uniquely determined by the mapped Point ID fields. So if there are two Point IDs in your Coordinates, which differ only in City, but have equal Country and State, you have to use the City as well to distinguish between those two map points. You will get no error message if you use wrong mapping, but the data can be displayed incorrectly.

Some map points from the Coordinates may not have any associated rows. These map points are not displayed on the map using map markers. Only the map markers, for which some data was found in the data source, are displayed. So you can have Coordinates containing all the major cities in USA, but you can display only the one you are interested in. And you can use the same Coordinates for different map, which will show different set of cities, because the data source will contain data for different cities, so you do not have to maintain too many Coordinates files if you choose suitable Point ID fields.

Example:

Your data source contains the following data and you want to display data grouped by cities on a map:

Row #	City	State	Customer Name	Sales
1	San Francisco	CA	Company A	\$152,560
2	New York	NY	Company B	\$240,468
3	Los Angeles	CA	Company C	\$335,256
4	San Francisco	CA	Company D	\$80,381

Row #	City	State	Customer Name	Sales
5	Boston	MA	company E	\$23,540
6	Chicago	IL	Company F	\$124,532

You will have to create the Coordinates first. In this case, only the City name could be used as the Point ID because it is unique, but we will use two Point ID fields instead - City and State, in case that there will be more cities with the same name in different states in the future. So we will create a data source with two columns - City and State. This data source has to contain all the listed cities (and could contain some more). We can either write a suitable query to obtain these two columns from the original data source (if using a database for the data source) or we can, for example, create a simple text file data source with these two columns.

Next, we can start Coordinates Editor and create the Coordinates. Assuming we have a data source that does not contain data for longitudes and latitudes or WKT, so we cannot use coordinates from data source but we know part of an address, we will use Geocoding. Using Geocoding, we will assign WKT to every city, so the table in the Coordinates Editor will look like this (note that there is one extra city - Orlando, which is not in the map data source):

City	State	WKT
San Francisco	CA	POINT(-122.41941550000003 37.77492950000001)
New York	NY	POINT(-74.00594130000002 40.712783699999999)
Los Angeles	CA	POINT(-118.2436849 34.052234199999999)
Chicago	IL	POINT(-87.629798199999998 41.878113599999998)
Orlando	FL	POINT(-81.379236499999999 28.538335500000002)

Now you can create a map from the data source using these Coordinates. You map City Point ID field to the City data source column and State Point ID field to the State data source column. The Point IDs will be paired with the data source rows - San Francisco with the first and fourth, and Chicago with the sixth. Orlando does not have any matching row in the data source, so it will not be displayed at all. If you add a Tooltip, the associated rows will be used to generate it. For example, the Tooltip for San Francisco will be created using data from the first and fourth row of the data source.


If you add a new city to the data source and you want to display it too, you have to edit the Coordinates data source first and add the city there. Next, you can open the Coordinates in the Coordinates Editor and there will be one new row in the table with this new city. This enables you to add the new map point for this city. You do not have to modify anything in your map. Therefore, if this Coordinates is shared among several maps, the map point is added automatically to all these maps.

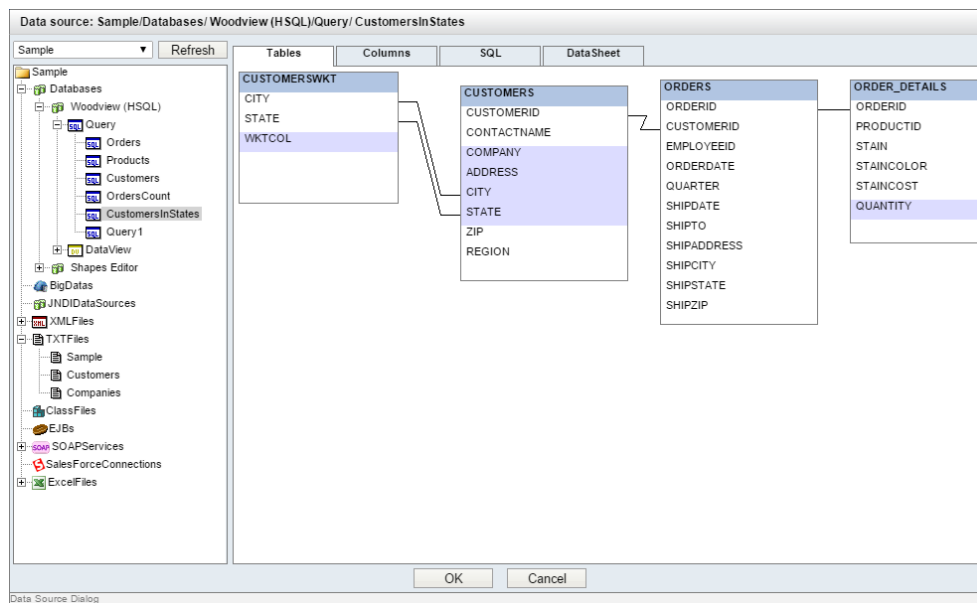
5.2.6. Create Online Maps

EDAB Online map is a special kind of report that allows displaying geographical data on map. It points out the reported geographical data on a map using map markers. Markers can be used as drill-down links to parameterized reports, charts, or maps. Markers may also have tooltips. **Tooltips** look like bubbles, containing a report or chart. When tooltips are enabled, every map marker has its own tooltip bubble, which displays data associated to this map point. Only one tooltip bubble can be displayed at the time. A tooltip appears when you move your mouse cursor over a map marker. It closes automatically when you open a different tooltip, or it can be closed by clicking on the X sign in the upper right corner of the tooltip bubble.

To create a Online Map, launch Online Maps from the EDAB main page.

5.2.6.1. Select Data Source

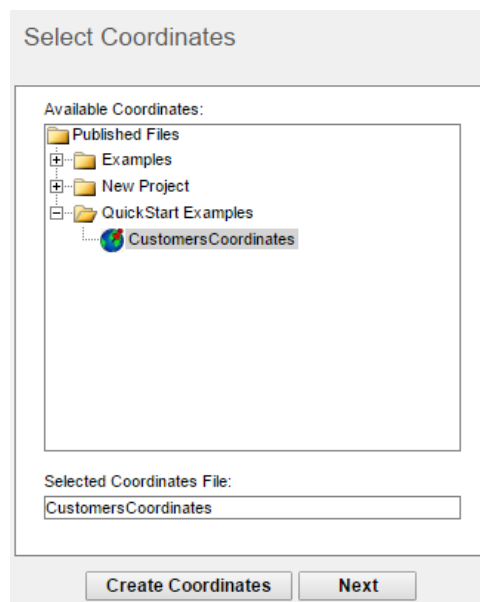
To create a new Online Map, click on the  *New Map* icon on the toolbar. The *Data Source Dialog* will appear. Select a data source for your map and click *OK*.



Data Source Dialog

5.2.6.2. Select Coordinates

Once you select a data source for your map, you will be prompted to select Coordinates file.



Select Coordinates

Select a Coordinates file and click the *Next* button. You can also launch the Coordinates Editor from this dialog by clicking the *Create Coordinates* button and creating a new coordinates file. Once the new coordinates file is saved, click the *Done* button in the upper right corner of the Coordinates Editor. You will be brought directly to *Coordinates Mapping* which is described in the next chapter.

5.2.6.3. Coordinates Mapping

You will be prompted to select point mapping, i.e. you have to pair columns that will enable EDAB Online Maps to assign records from maps record's data source to suitable Point ID fields from the Coordinates (e.g. if you want to show cities with your affiliates on map, your Coordinates may contain a column called *City* which is mapped to *Affiliates_City* from the map's data source). For more information, see Section 5.2.5 - Coordinates Mapping. Please note that Online Maps designer automatically recognizes which columns are the most suitable for Coordinates

mapping. Therefore, some of the fields may already be pre-selected when there is a convenient column in your data source (e.g. if the data source contains a column called *City*, it will be pre-selected in *City* field).

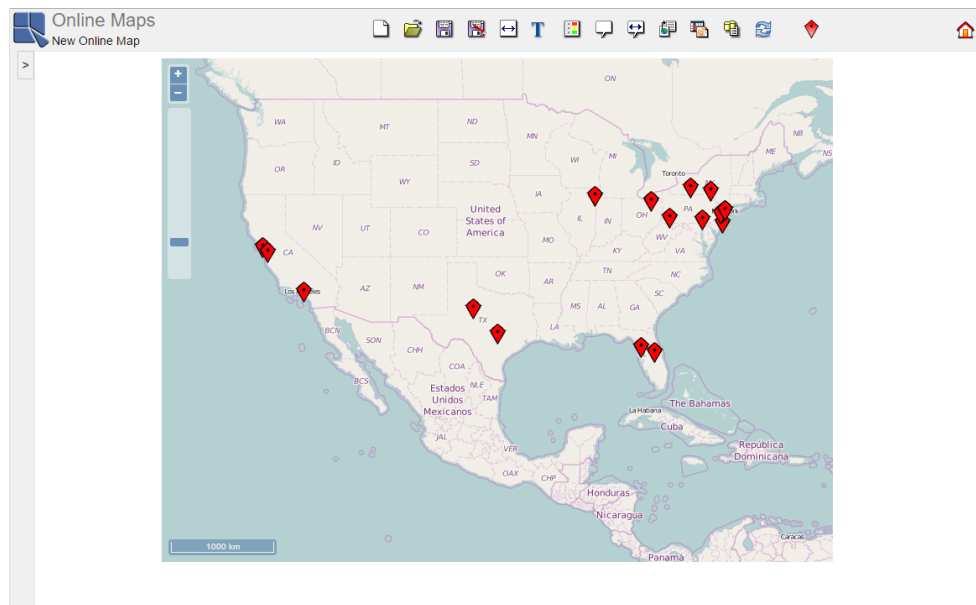
Coordinates to Map Data Mapping

Coordinates	Map Data
City	CITY ▼
State	STATE ▼


< Back Apply

Coordinates Mapping

You have to map at least one column. Then click *Apply* and the Online Map will appear.



Online Maps Designer

You can change the Coordinates mapping by clicking the  *Change Point Mapping* icon on the toolbar.

5.2.6.4. Toolbar

These buttons are on the Online Maps toolbar:



New Map: Creates a new Map.



Open Map: Opens an existing Map.



Save Map: Saves the current Map to a file and Organizer. Note that all saved files will include userID and timestamp in addition to the name you provided. If an Organizer item with the supplied name exists in the folder, you will be given a choice to override the existing item or create a new entry.



Save Map As: Allows you to save the existing or modified Map to a different file or location.



Set Map Options: Allows you to change the way the Online Map is displayed.



Map Title Options: Allows you to set map title and its options.



Set Heatmap: Allows you to color-code map markers.



Tooltip Template: Adds/Modifies report/chart templates used as tooltips.



Tooltip Options: Allows you to change dimensions of tooltip bubbles.



Drilldown Options: Configures drill-downs.



Change Point Mapping: Changes the point mapping.



Change Data Source: Changes the data source of current map. Note that changing data source will result in losing tooltip and drill-down settings.



Refresh: Refreshes the map data (if the data source changed).



Coordinates Editor/Edit Coordinates: Opens Coordinates Editor that allows you to create/edit coordinates.



Home: Closes Online Maps designer. Before closing, you will be asked if you want to save an unsaved map.

5.2.6.5. Map Options

To set map options, click on the  *Set Map Options* icon on the toolbar.

Map Options

Dimensions
Width (in pixels):
Height (in pixels):

Zoom
Minimal Zoom:
Maximal Zoom:

Map Controls
Zoom Control:
Map Type:
Map Scale Control: ☒

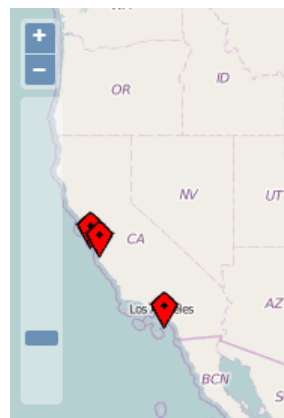
Apply

Map Options

Dimensions: Allows you to set width and height of the map (in pixels).

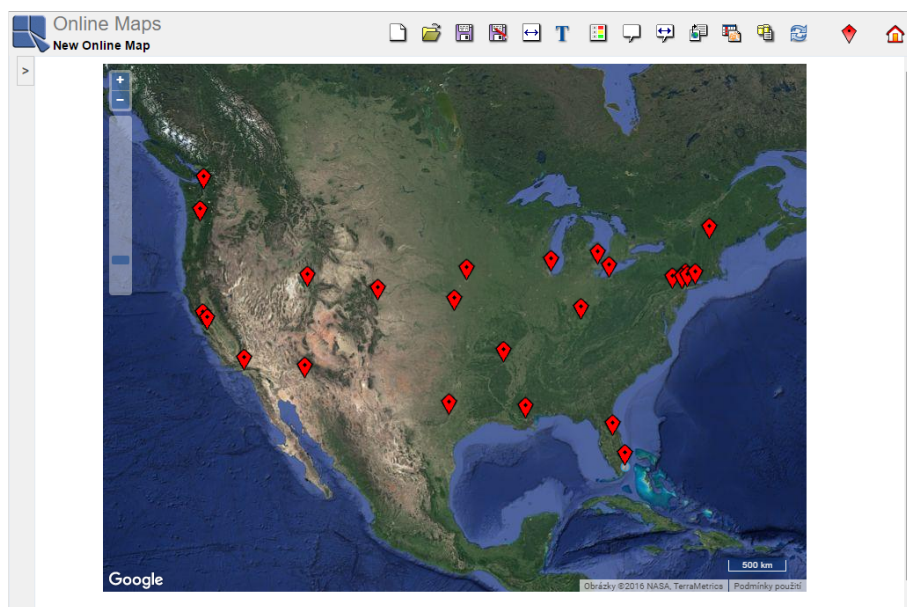
Zoom: Allows you to set minimal and maximal zoom of the map (in the range 1-17).

Map Controls: **Zoom Control** - shows/hides the zoom controls in the upper left corner (large, small, none).



Zoom Control Large

Map Type - allows you to set map type (Open Street Map, Google Street Map, Google Satellite).



Google Satellite

Map Scale Control - shows/hides the scale control in the lower right-hand corner.




Scale Control

5.2.6.6. Heatmap

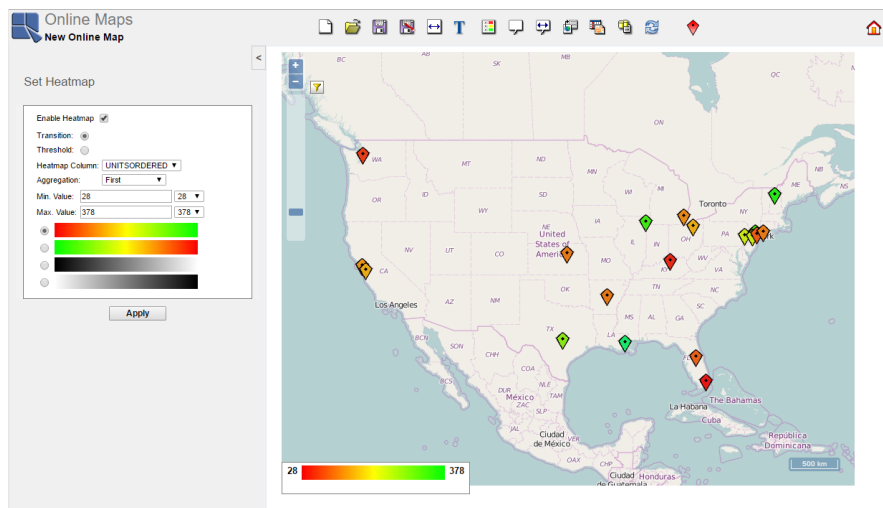
Heatmaps allows you to display map points in different colors based on values from a column in the data source. You can choose between two heatmap types - transition or thresholds.

Please note that some of the data sources have multiple records belonging to single map points in coordinates file. The *Aggregation* option allows users to choose a preferred way on how to aggregate values in the given heatmap column. The default aggregation value is set to *First* and this value is recommended only for maps where multiple records for one point in the map are not expected. In other cases, it is better to choose a more appropriate aggregation.

To set a heatmap, click on the  *Set Heatmap* icon on the toolbar. The *Set Heatmap* dialog appears in the left pane. At first check off the *Enable Heatmap* option and select a type of the heatmap - *Transition* or *Threshold*. The *Set Heatmap* dialog will be adjusted accordingly:

Transition Heatmap:

Select a *Heatmap Column* from the drop-down list, select an *Aggregation*, and set minimal and maximal value of the selected column values. You can also choose a value from the drop down lists which contain the precise values from the selected column. Select a color gradient and click *Apply* to apply your settings on the map.

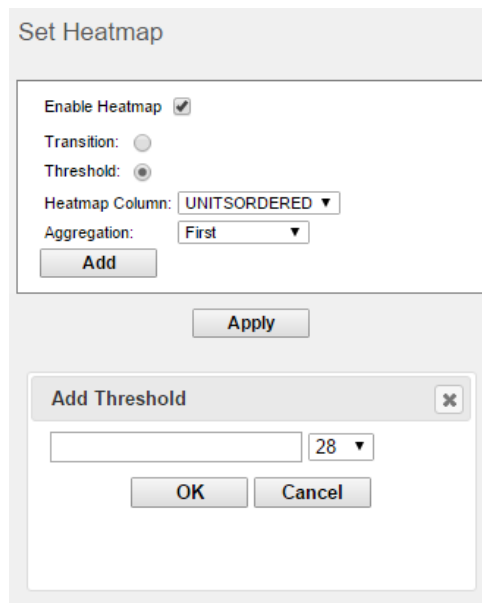


Set Transition Heatmap

You can enable/disable a heatmap by checking/unchecking *Enable Heatmap* option.

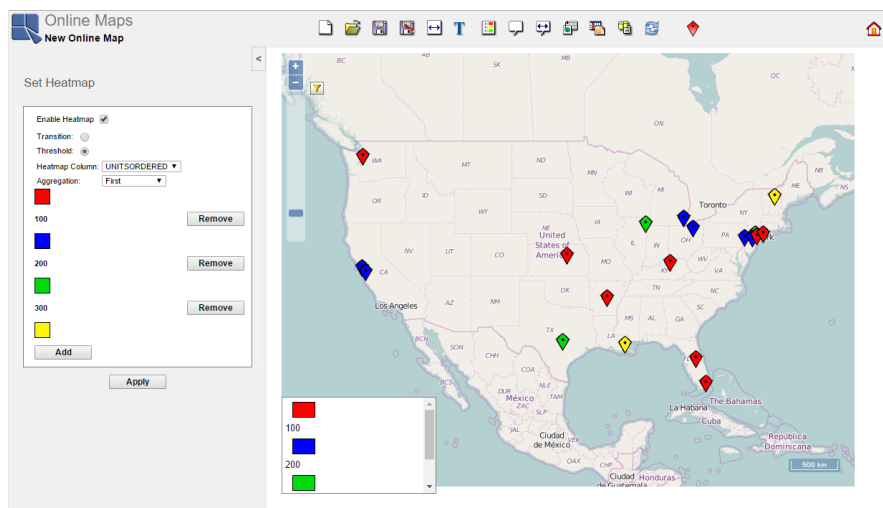
Threshold Heatmap:

Select a *Heatmap Column* from the drop-down list, select an *Aggregation*, and click the *Add* button to add a threshold. *Add Threshold* dialog will appear.



Set Threshold Heatmap

Type a threshold value or choose a value from the drop-down list which contains precise values from the selected column and click the *OK* button. Set another thresholds in the same way and set colors for each threshold by clicking on the color square and selecting the color. Then click *Apply* to apply the heatmap.

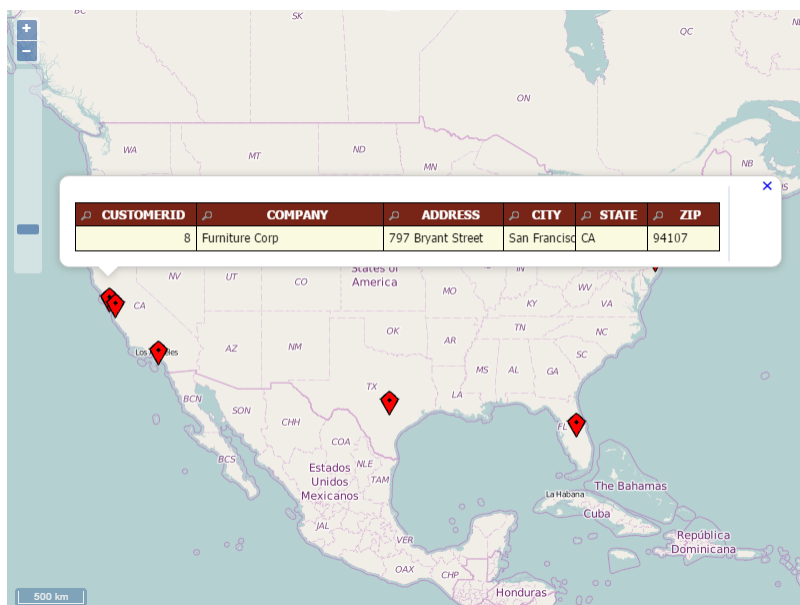


Set Threshold Heatmap


You can remove each threshold by clicking the *Remove* button next to the threshold. To enable/disable a heatmap, check/uncheck *Enable Heatmap* option.

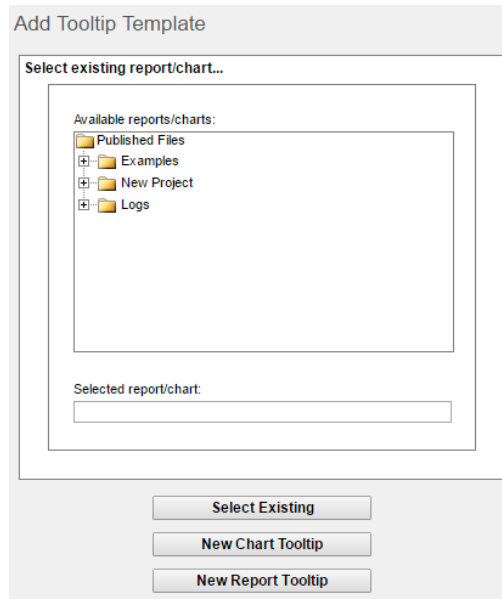
5.2.6.7. Tooltips

Tooltips may contain a report or chart that shows up when you move your mouse cursor over any map marker on the Online Map. Tooltips display single or multiple-row data that is related to the particular map point. As an example, a report tooltip will display data from the map datasource using parameter(s) from the *Map Data* of the corresponding map point. The *Map Data* is the same as the list in the *Change Point Mapping* dialog box. The report or chart file that is displayed in the tooltip bubble is called **Tooltip Template**.



Online Map with Tooltip

To enable tooltips for EDAB Online Map, open the map in Online Maps and click on the  *Tooltip Template* icon. The *Add Tooltip Template* options will appear in the left pane.



Add Tooltip Template


You can select an existing tooltip, create chart tooltip, or create report tooltip.

Select Existing:

To select an existing Tooltip Template, select it from *Available reports/charts* list and click the *Select Existing* button. Using this option, you can also use the report/chart templates created in the Report Designer/Chart Designer, so you are not limited to use just the QuickDesigner Charts or QuickDesigner Reports to design Tooltip Templates. In this case, it is recommended to use exactly the same data source for creating this Tooltip Template report/chart as for the map, otherwise there might be a problem with data mapping.


New Chart Tooltip:


Launches QuickDesigner Charts that will allow you to create a new chart file. Cre-

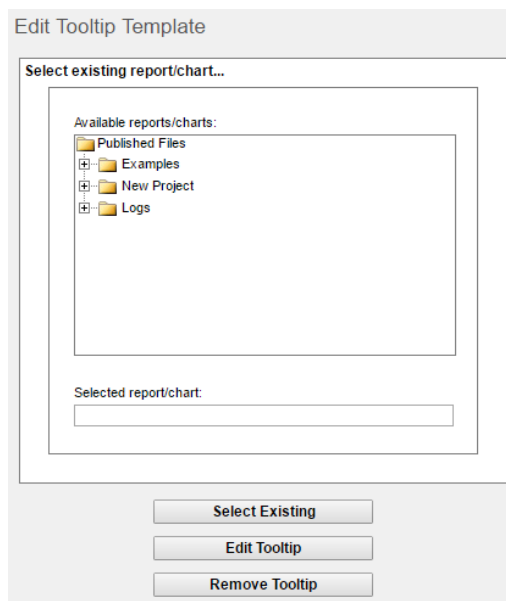
ate a chart, save it, and leave QuickDesigner Charts by clicking the  *Return to Map Designer* icon on the toolbar. (For more information about working in QuickDesigner Charts, see Section 4.3 - QuickDesigner Charts.)

New Report Tooltip:

Launches QuickDesigner Reports that will allow you to create a new report file.

Create a report, save it, and leave QuickDesigner Charts by clicking the  *Return to Map Designer* icon on the toolbar. (For more information about working in QuickDesigner Reports, see Section 4.2 - QuickDesigner Reports.)

To edit, select, or remove the current tooltip, click on the  *Tooltip Template* icon. The *Edit Tooltip Template* options will appear in the left pane. Please note that from this dialog, you can only directly edit files created in QuickDesigner Reports and QuickDesigner Charts (.qdr or .qch).



Edit Tooltip Template

Select Existing:

Allows you to select other Tooltip Templates.

Edit Tooltip:

Launches QuickDesigner Report or QuickDesigner Chart where you can edit the current tooltip template if it's a .qdr or .qch file (i.e. files from one of the QuickDesigners).


Remove Tooltip:

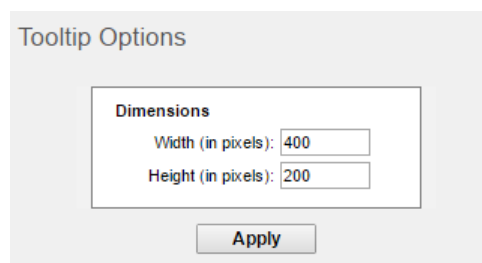
Removes current Tooltip Template.



Note

To create a new chart/report tooltip you have to remove current tooltip at first and then will be *New Chart Tooltip* and *New Report Tooltip* options available.


To configure dimensions of the tooltip bubbles, click on the  *Tooltip Options* icon on the toolbar. Set new dimensions and click *Apply*.



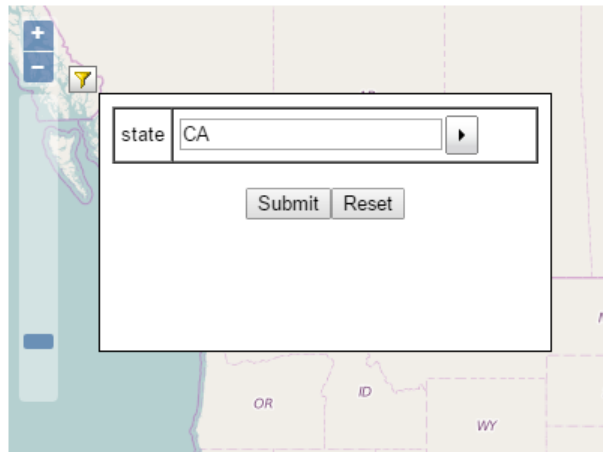
Tooltip Options

Changes will take effect after closing the current tooltip bubble (if a bubble is open, otherwise the changes will take effect immediately). It is not necessary to close the current tooltip bubble, you can just mouse over some other map point.



5.2.6.8. Parameter Setting

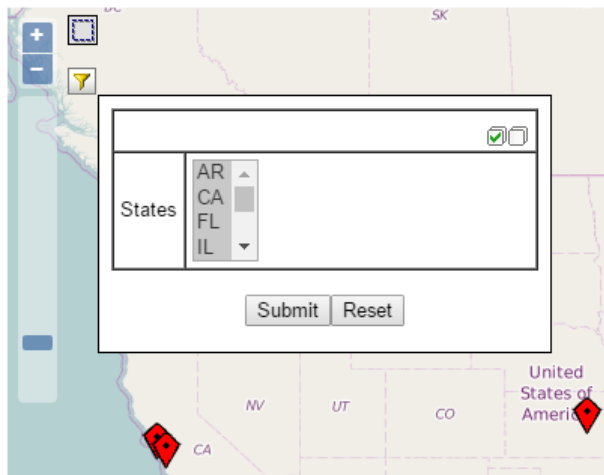
You can change parameters directly on the map by clicking the  *Set parameters* icon in the upper left corner of the map (The icon is displayed only if the data source is parameterized). A new dialog will open where you can select parameter value(s).

If the parameter is single-value, the next dialog will display a drop-down list. Select a parameter value and click the *Submit* button to apply it on the map. (You can reset your selection by clicking the *Reset* button.)




Parameter Setting

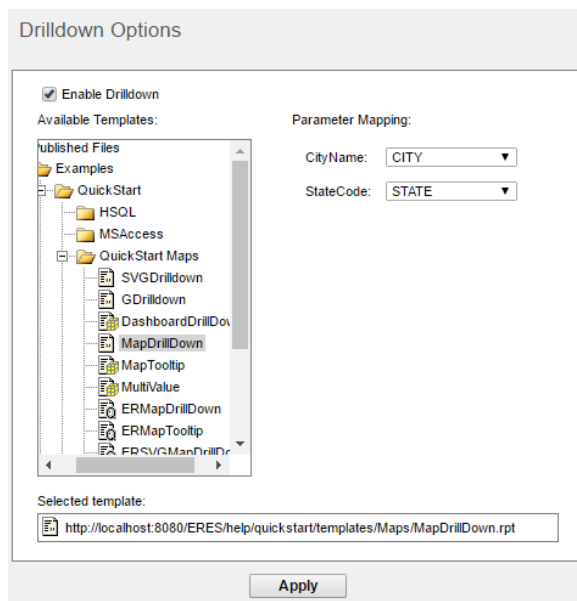
If the parameter is multi-value, the next dialog will display a list of parameter values. You can select parameter values by clicking on them (use **Ctrl+click** for multiple selection), select all parameter values by clicking the , deselect all parameter values by clicking the  *Select None* icon, and reset your selection by clicking the *Reset* button. Then apply your selection to the map by clicking the *Submit* button.



Parameter Setting

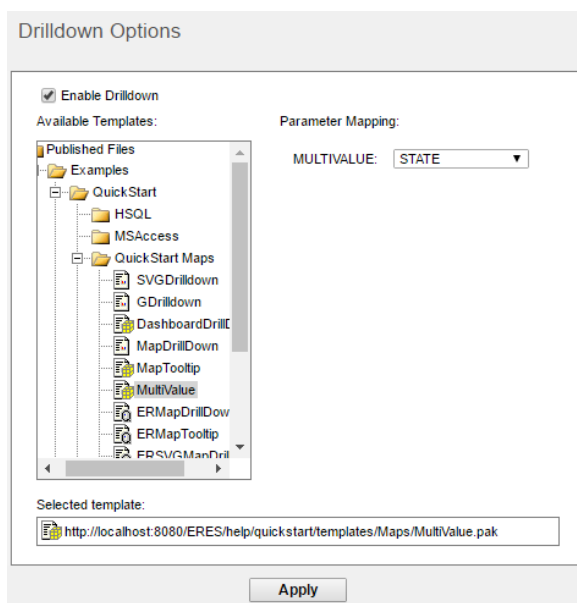
5.2.6.9. Drilldowns

To enable drilldowns in a map, click on the  *DrillDown Options* icon. Check the *Enable Drilldown* checkbox which will show you all parameterized reports, charts and maps saved in Organizer. Click on a suitable parameterized report, chart, or map and select parameter mapping. All drilldown parameters have to be mapped to some Point ID field from the Coordinates. Mapping in this case means that the Point ID values of the selected map point will be used as drilldown parameter values (each map marker has unique Point ID). When you click on a map marker on the map, the drilldown report/chart/map will open in a new browser tab.



Drilldown Options

If all the drilldown parameters are multi-value (see Section 3.1.3.2.2.1 - Multi-Value Parameters), you will be allowed to select multiple map markers at once. For example, you can use Examples/QuickStart/QuickStart Maps/MultiValue data source, map *MULTIVALUE* parameter to *STATE* column, and then click *Apply*.



Multi-value Data Source

Selecting multiple map markers can be done by clicking the *multi-value Drilldown* button (see the image below; the button is displayed only if the map contains multi-value Drilldown).



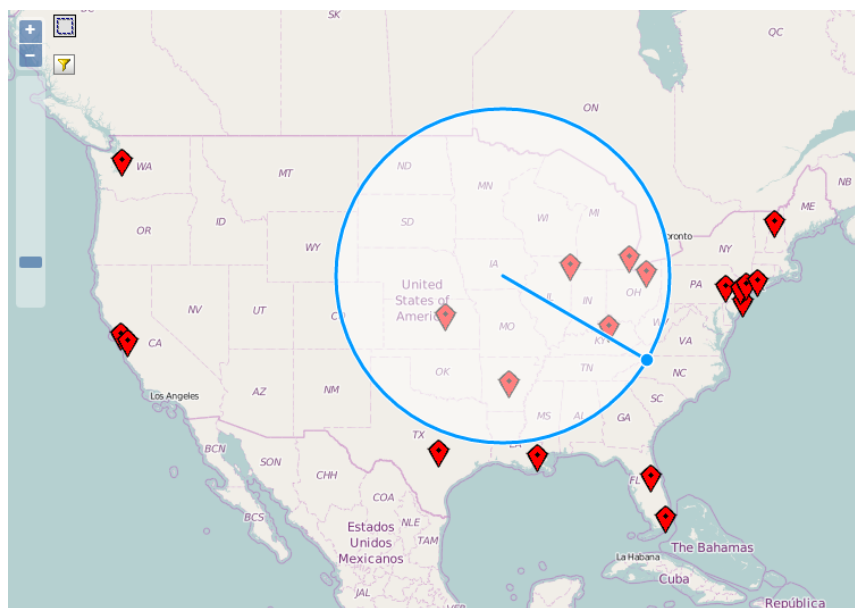
Multi-value Drilldown Button

After clicking the *Multi-value drilldown* button, the *Multivalue drilldown* dialog will appear.



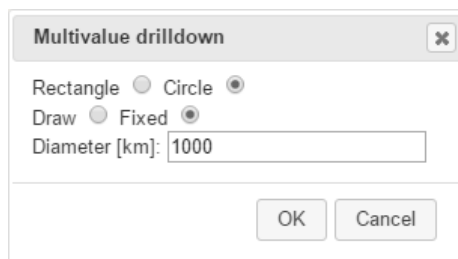
Multi-value Drilldown Dialog

You can choose a shape of the bounding box (rectangle or circle). The circle can be fixed or drawn. Once you select a shape, click the *OK* button. Then you can select multiple map markers by dragging using bounding box. Left click on the map (this determines rectangle corner or circle center), hold mouse button down and drag it, and then release the mouse button to determine size of the bounding shape.




Bounding Box

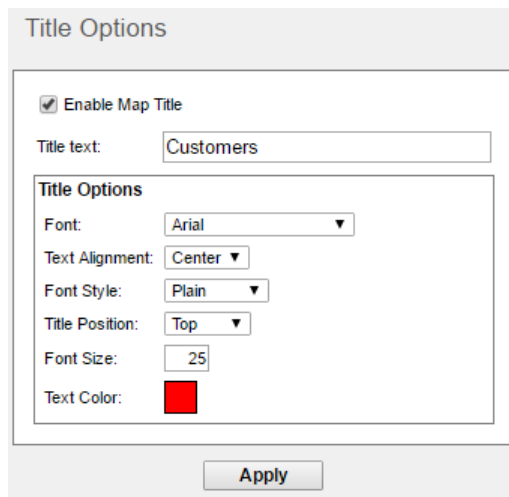
If you select fixed circle as a bounding box, you can set a diameter of the circle. Write a diameter value and click the *OK* button. You will see a circle with the diameter you just set. Then click to the map. Your click is a center of the circle bounding box. A new tab with a drilldown of your selection (markers within the circle) will appear.



Fixed Circle Bounding Box


5.2.6.10. Map Title

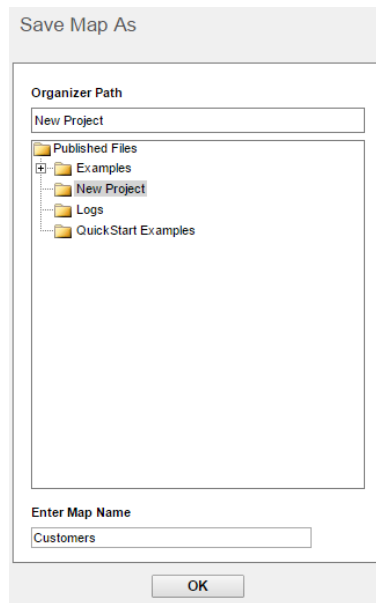
To enable map title, click on the  *Map Title Options* icon and check the *Enable Map Title* checkbox. Then you can insert the map title text and select various options - font, text alignment, font style, title position, font size (positive integers only), and text color. The color can either be chosen from swatches or inserted manually by entering red, green and blue components (all the values has to be integers from 0 to 255).



Map Title Options

5.2.7. Save Map


You can save the Online map by clicking the  *Save* button on the toolbar. This will open a dialog allowing you to specify a name for the map. Enter a name for the map, select a project where you want to save it, and click the *OK* button.

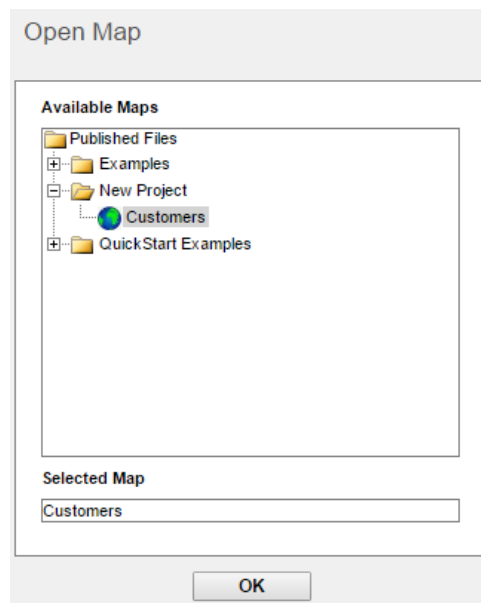


Save Dialog

After saving or opening an existing Online map, the  *Save As* button allows you to save the existing or modified map into a different name or location.

5.2.8. Open Saved Map




You can open a saved map by clicking the  *Open* icon on the main toolbar. The *Open Map* dialog will appear.



Open a Map

All Online maps created in Online Maps designer are visible in Organizer. Select a map and click the *OK* button to open it.

5.2.9. Exit

You can exit Online Maps designer either by clicking the  *Home* icon in the upper right corner, clicking the  *Online Maps* title, or by clicking the  *Logo* icon in the upper left corner. Before closing, you will be asked if you want to save an unsaved map.

5.3. SVG Maps

5.3.1. Introduction to SVG Maps

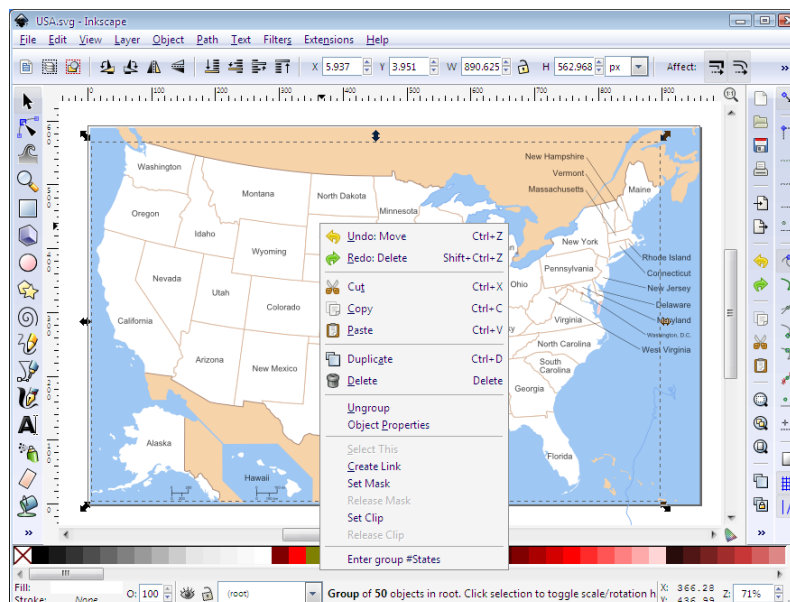
Unlike Online Maps, SVG Maps do not use map points. They use **map areas**. The map areas can be colored based on values from the map data source. Coordinates of the map areas are defined in **SVG map image**. Virtually any SVG image can be used for SVG Maps, but it usually requires some modifications, which can be done in any graphical editor with SVG support. EDAB does not contain any integrated graphical editor, so you have to use third party software. The recommended software is **Inkscape**, which can be downloaded at www.inkscape.org [<https://inkscape.org/>]. Inkscape is released under GPL license version 2 [<http://www.gnu.org/licenses/old-licenses/gpl-2.0.html>], so it is free for commercial and non-commercial use.

SVG (Scalable Vector Format) images consist of objects. These objects can be used as map area. To distinguish among individual map areas, EDAB uses **Area ID**. Each map area must have an Area ID, but unlike Point ID, Area ID does not take up more than one field (it always consists of only one field). The Area ID has to be unique for every area and they are read from the SVG Image object ID attribute. This attribute can be set in most graphical editors. Please see the next chapter for a step-by-step guide for the recommended tool - Inkscape.

EDAB is distributed with several SVG images that are already prepared to be used for creating SVG Maps without any modification. These images can be found in `<EDAB_Installation_directory>/MapFiles/SVG`. They are also included in Organizer in SVGMaps project. You can find the list of all these maps in the Appendix 5.A - List of SVG Map Images.

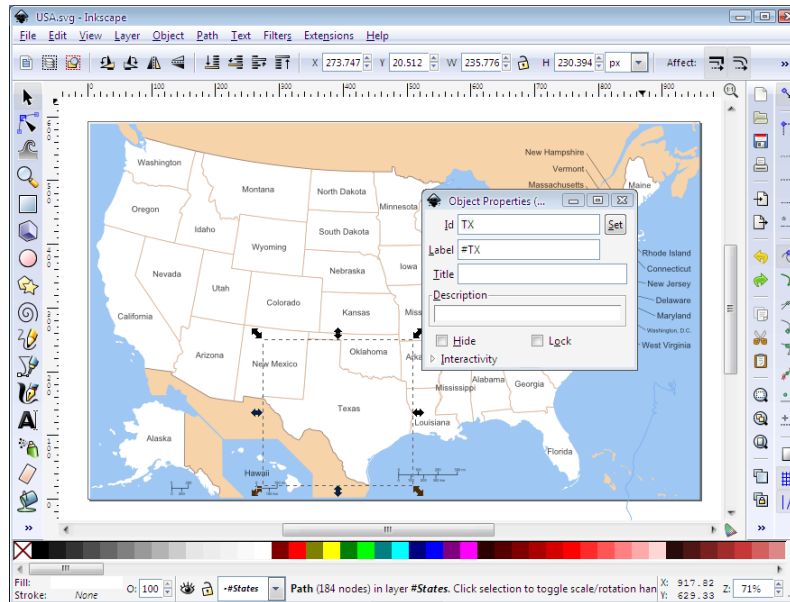
5.3.2. Set Area IDs Using Inkscape Editor

Download the Inkscape editor (<http://www.inkscape.org/> [<https://inkscape.org/>]) and install it on your computer. Open an SVG image. Complex SVG Images may consist of groups, each consisting of several elements (areas). Right click on the area you want to change the ID of. At the bottom of the pop-up menu, notice an option called *Enter group [name of the group]* (see the screenshot below) and click on it to enter the group.



Inkscape - Entering Object Group

Right click on the area you wish to change the ID of again and select the *Object properties* option. Fill in the new Area ID in the ID field and confirm it by pressing the *Set* button. Close the *Object properties* dialog. You can now set Area IDs for other objects in a similar way.



Inkscape - Editing ID Attribute

If your map area consists of several unconnected SVG objects (like Alaska and Hawaii for USA), you can group them in one group and assign the Area ID to the whole group. To do that, do not enter the group as described above, but instead go to the Properties dialog for the whole group and assign the ID there. If a group has Area ID assigned and there are objects inside this group that also have Area IDs assigned, there could be conflicts if the both Area IDs (ID of the whole group and ID of an object from this group) are present in the map data source. In this case, the behavior is undefined, so you can get unexpected results. It is OK to have groups with Area IDs that contains objects with Area IDs, unless you use both IDs simultaneously in your data source. It can make sense in some cases (e.g. you can have a world map with continent names as well as country names. You can safely use this map if you use either continent names or country names separately).

To save the changes, open File menu (upper left corner) and select Save. Note that SVG images that you want to use for creating SVG Maps have to be inserted in Organizer.

5.3.3. Area ID Mapping

SVG Map needs an SVG image that defines the map areas. All SVG objects in the SVG image can be used as map areas. The only requirement is that they must have an Area ID assigned as described in the previous section. These Area IDs are used for mapping the map data source rows to map areas (SVG objects). Unlike the Point IDs, Area IDs cannot have multiple fields, so the mapping is much simpler. It is enough to select just one data source column, which will be mapped to the Area ID. A map area is associated with the data source row, which has the value of the mapped column equal to the Area ID (comparison is case sensitive).

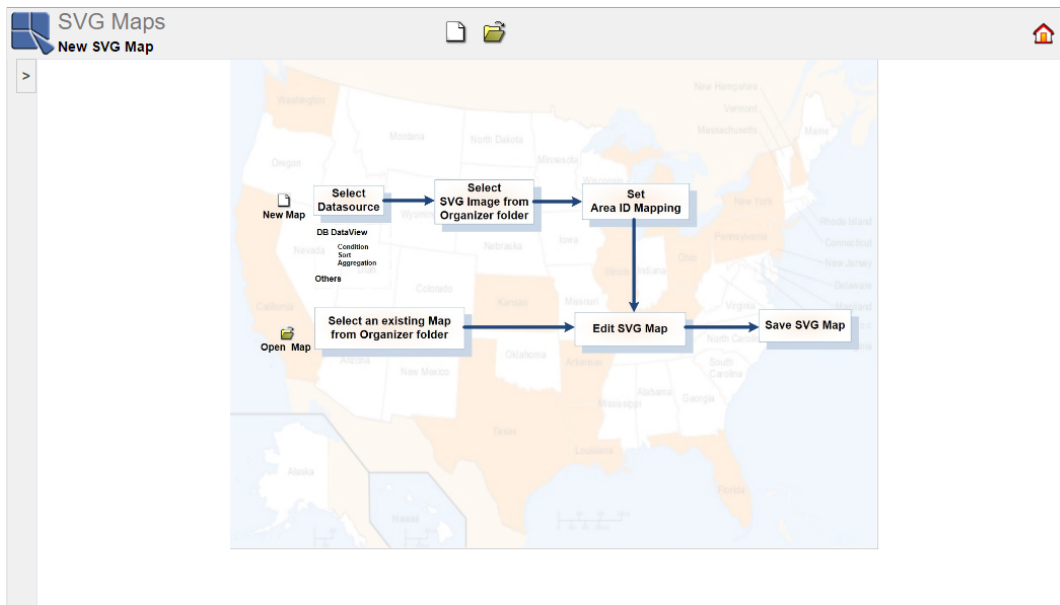
In the SVG Maps, the map data source is used for coloring areas according to numerical value of a data source column. It brings one more limitation for the map data source compared to Online Maps. There has to be at least one matching data source row for one Area IDs. If there is no matching row, the area will have original (unchanged) color. If there is exactly one, the fill color of the matching map area will be changed according to Thresholds (see Section 5.3.4.5 - Thresholds). If there is more than one matching row, the behavior is undefined. There will be no error message, but the data displayed may not make sense.

5.3.4. SVG Maps Designing

5.3.4.1. Start


SVG Maps can be started from the EDAB Start page. If you have logged in as a user with design privileges, then you can follow SVG Maps link to begin using SVG Maps.

When the SVG Maps designer launches, the first page that appears prompts you to select between opening an existing map, or creating a new one.

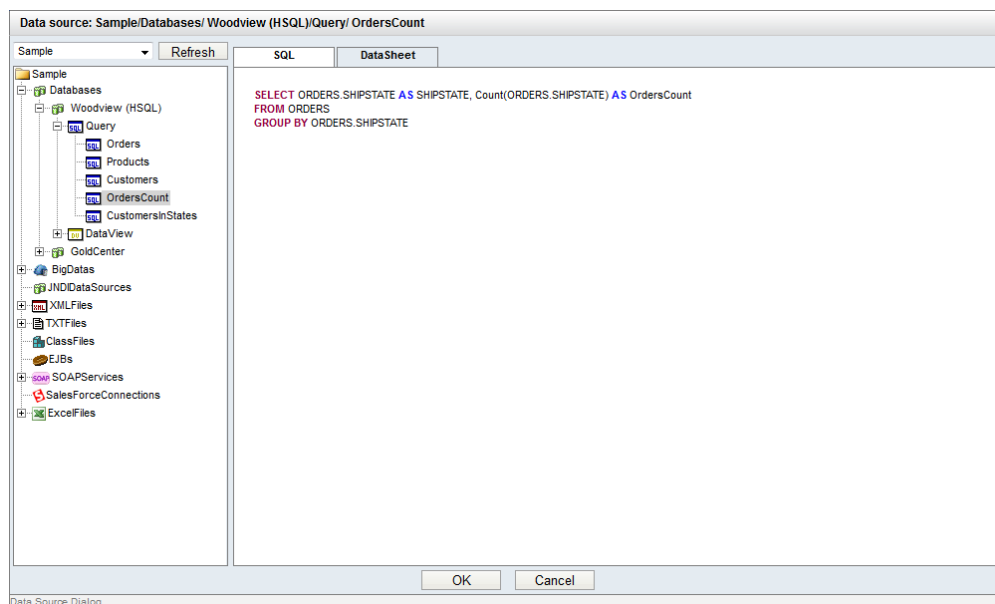


SVG Maps Start Options

5.3.4.2. Select Data Source

If you want to create a new SVG map, click on the  *Create New Map* icon on the toolbar. The *Data Source Dialog* appears and prompts you to select data registry and data source you want to use with SVG Maps. In order to use SVG Maps, a user must have read privileges to one of the registries defined in the Organizer. For more about creating and managing data registries, please see Section 3.1.1 - Managing Data Registries.

Select data registry from the drop-down menu in the upper left corner. The tree-list bellow displays the content of the selected registry. There are all of the data sources that have been defined in the registry that the user has access to. Select a data source you want to use for the SVG map.




Data Source Dialog

Unlike Chart Designer where the user can select between creating new data sources or modifying the existing ones, the only option is to select a data source for the map. The only exception to this is for Data Views or Data View

Queries. If data sources of this type are selected, you will see the *DataView Builder* on the right side of the Data Source Dialog. The *DataView Builder* allows you to build or modify queries against the View. For more information about managing data sources, please see Section 3.2 - Data in QuickDesigners and Maps.

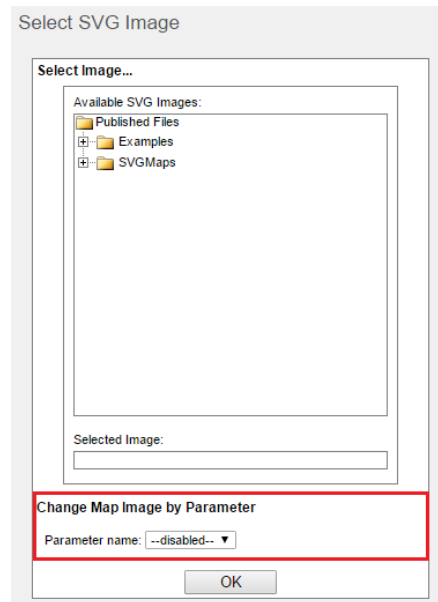
Once you finish selecting a data source, click the *OK* button to close the *Data Source Dialog*.

5.3.4.2.1. Change Data Source


To change a data source (for SVG maps that have already been created), click the  *Change Data Source* icon on the toolbar. The *Data Source Dialog* will open and you will see current data source. Select other data source from the tree list and then click the *OK* button to apply it.

5.3.4.2.2. Parameterized Data Source

You can use parameterized data source for SVG maps. For more information about parameterized data source in SVG maps, please see Section 5.3.5 - Dynamic SVG Maps.

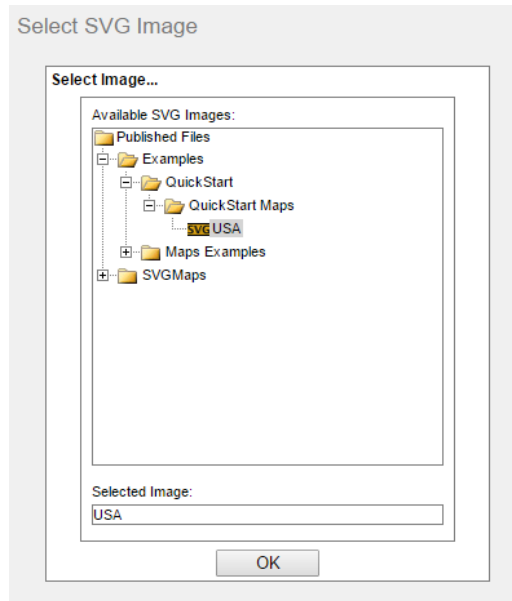


Selecting Parameterized Data Source

To change a parameter from the toolbar of the SVG Maps designer, click the  *Refresh/Set Parameters* icon on the toolbar. You will see available parameters. Set the parameter(s) and click *Submit* to apply this setting.

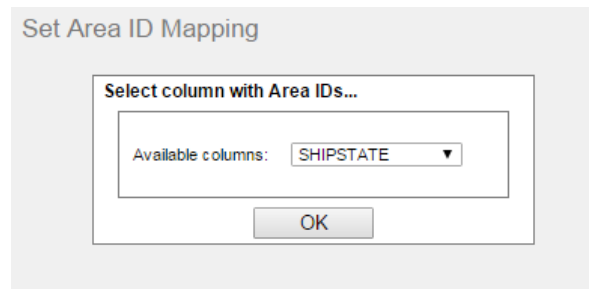
5.3.4.3. Select SVG Image

Now you can see the *Select SVG Image* dialog on the left side of the SVG Maps designer. This dialog prompts you to select a Map Image. Map Images are SVG image files with data structures containing geographic data (see Section 5.3.3 - Area ID Mapping for more information about required image format) and must be inserted into the Organizer before they can be used.



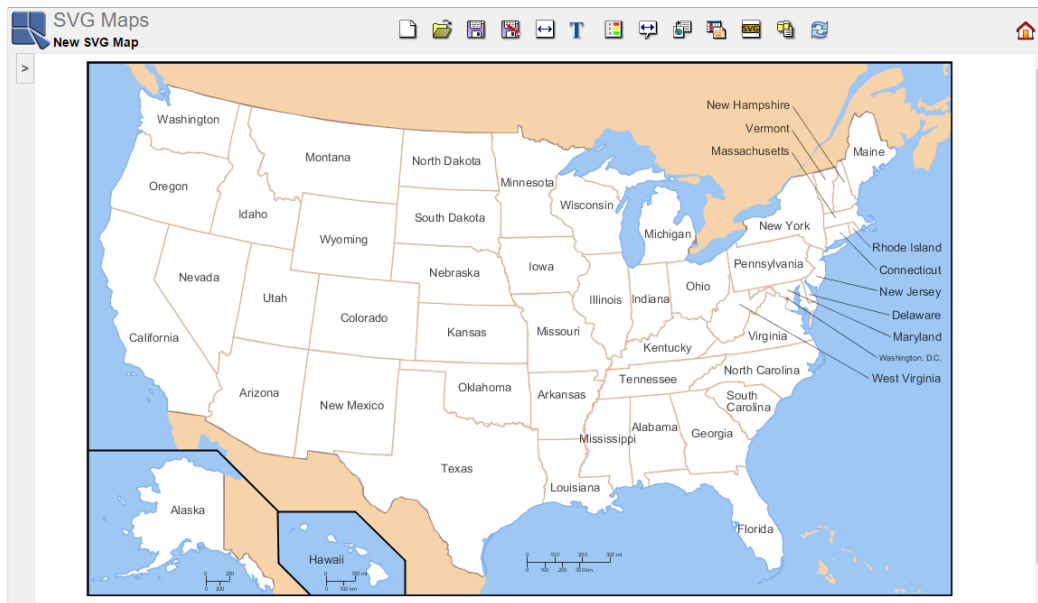
Select SVG Image Dialog

The *Select SVG Image* dialog contains a tree-list showing all projects, folders, and SVG images inserted in the Organizer. Select an SVG image you want to use as the map image and click the *OK* button. The *Set Area ID Mapping* dialog appears. Select a data source column that should be mapped to the Area IDs and then click the *OK* button.



Select Column with Area IDs


SVG map will open (*Set Area ID Mapping* dialog is collapsed on the screenshot).

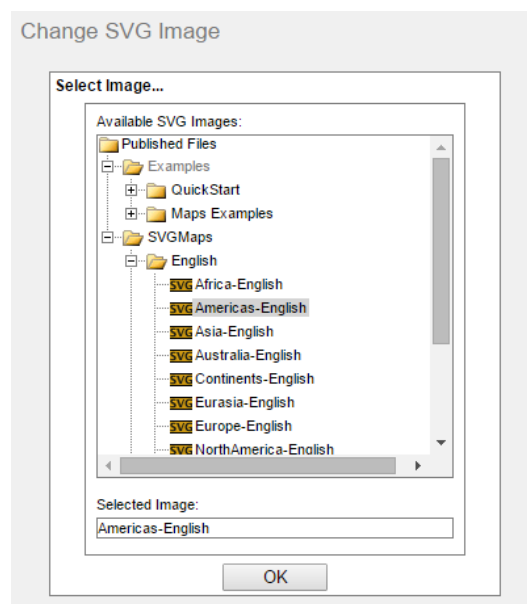


SVG Maps Interface

As you can see, right now we have only the basic map with no data highlighted. There are two ways of adding data to the SVG Map: Thresholds and Drilldowns.

5.3.4.3.1. Change SVG Image

To change an SVG image (for SVG maps that have already been created), click the  *Change SVG Image* icon on the toolbar. Select an image from the tree list and then click the *OK* button to apply the image.













Changing SVG Image

5.3.4.4. Toolbar

A lot of formatting actions for reports in SVG Maps are accessed through the toolbar. The icons perform the following actions:

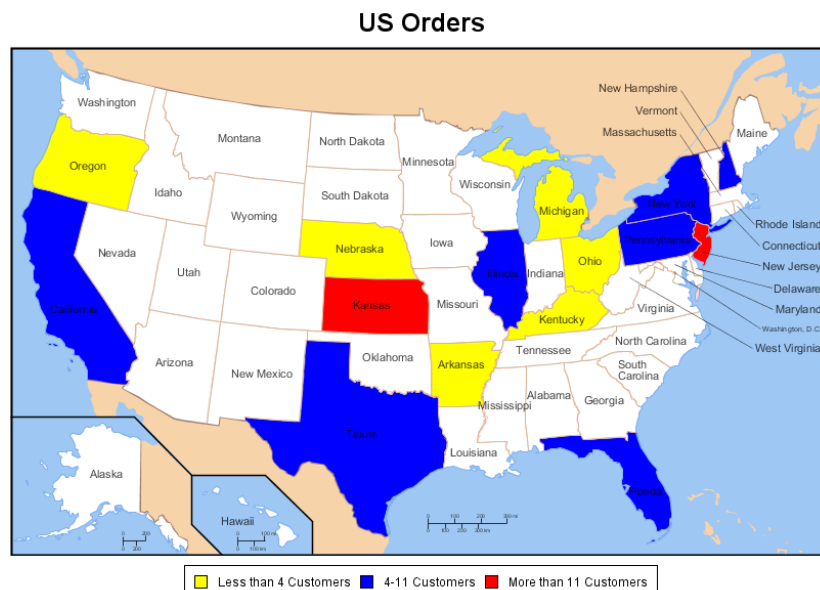


Start a new SVG map



-  Open an existing SVG map
-  Save the current SVG map
-  Save the SVG map as
-  Set a map size
-  Insert/Edit a map title
-  Set thresholds
-  Insert/Edit a tooltip
-  Insert a drilldown
-  Change area ID mapping
-  Change SVG image
-  Change data source
-  Refresh/Set parameters

5.3.4.5. Thresholds

Thresholds allow you to differentiate areas by color based on value in a particular data source column. If a threshold is set, all the map areas (i.e. objects with a record in the map data source) are colored with color assigned to the appropriate value range.



Example: SVG Map with Thresholds

To set the thresholds, click on the  *Set Thresholds* icon on the toolbar and select a threshold column (i.e. column that will be compared to the ranges defined below). Only numerical column can be selected. You can add a threshold value(s) in actual values or percentage unit. To add a threshold value, click the  button next to the empty table field. A dialog box will open prompting you for the new threshold value.

Absolute Values:

Enter threshold value [absolute]

or select from available values:

▼

OK
Cancel

Percentage:

Enter threshold value [%]

or select from available values:


▼

or enter a number of intervals:

OK
Cancel

Entering Threshold Values

You can either enter it manually or select one of the suggested values from the drop-down list at the bottom of the dialog. This list contains all the values from the selected threshold column rounded to integers (There is a third way to enter threshold values for percentage units. It is an option to enter a number of intervals). After adding the first threshold value, two ranges are added - one for values below and one for values above the given threshold. Any threshold value added after that will create only one new range by splitting one of the old ranges in two. You can edit any of the existing thresholds by clicking on the appropriate range (the first column in the table).

To change the color assigned to a value range, click on the empty table field next to the value range you want to change the color of. You can then select one of the predefined colors or choose any color by entering its red, green and blue component values (each value is an integer within the 0-255 range). To remove a threshold value, click on the red  button next to the value you want to remove. Changes will take effect after you confirm the *Set Threshold* dialog by clicking the *OK* button.

To remove all thresholds and disable coloring of areas, delete all threshold values or select an empty field for the threshold column.

Set Threshold

Threshold column:

Threshold mode: ☒ Absolute Values ☐ Percentage

Thresholds

range	color	
<= 4	Yellow	X
4 - 11	Blue	X
> 11	Red	X
		+

Legend

☐ Enable legend

OK

Annotations in image:
 - Click here to change threshold value (points to range cell)
 - Click here to remove threshold (points to X icon)
 - Click here to add new threshold (points to + icon)
 - Click here to change color (points to color cell)

Set Threshold Dialog

5.3.4.5.1. Legend

The *Set Threshold* dialog also allows you to add a legend to describe the value ranges. To enable it, check the *Enable Legend* checkbox. Then you can select a layout and position of the legend. If the legend is enabled, legend input fields next to the value ranges will appear. Insert a description of the ranges in these fields. If the legend is too large to fit one column (for vertical layout) or row (for horizontal layout), it will be split into several columns/rows. If it still does not fit the map canvas even after splitting, it will not be displayed at all. If this ever happens, you can try to resize the map or change the position or layout of the legend.

Set Threshold

Threshold column:

Threshold mode: ☒ Absolute Values ☐ Percentage

Thresholds

range	color	legend
<= 4	Yellow	Less than 4 Customers
4 - 11	Blue	4-11 Customers
> 11	Red	More than 11 Customers

Legend

☒ Enable legend

Legend layout:


Legend position:

OK

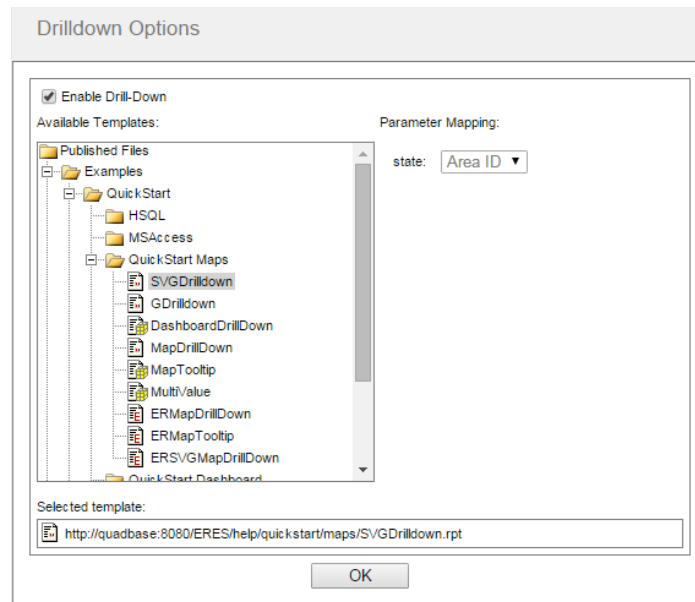
Annotation in image:
 - Fill in the legend text here (points to legend text field)

Setting Legend

5.3.4.6. Drilldowns

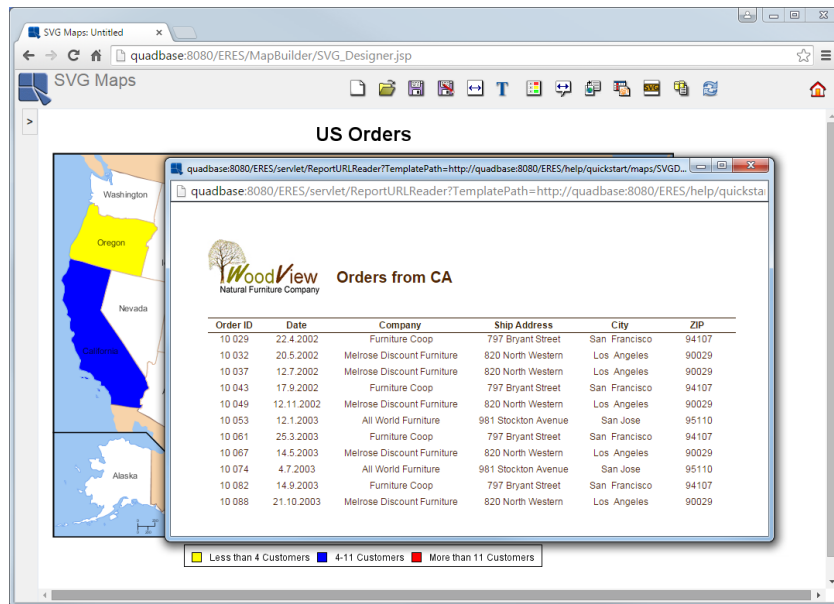
Click on the  *Drilldown Options* icon on the toolbar and select *Enable Drilldown* option. The Organizer structure will show up in the *Available Templates* treeview. Select a parameterized report, chart, or map. All parameters will be automatically mapped to the Area ID. This means that all parameters of the drilldown report, chart or map will be set to the Area ID of the selected area. It is not possible to change the mapping. SVG map also does not

support multi-value drilldowns (multi-value parameters are treated as single-value). Once you select a file, click the *OK* button to open it.



Drilldown Options

When you click in an area matching the ID mapping (e.g. a colored state), a drilldown report will open in a new window.



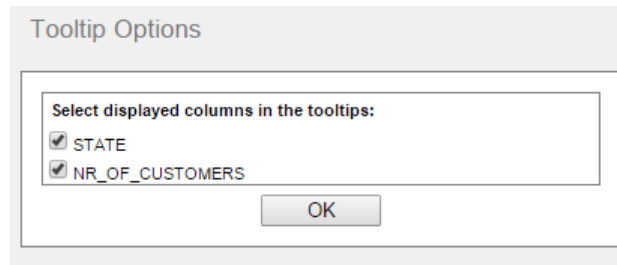
Map with Drilldown

5.3.4.7. Tooltips

Tooltips in SVG maps are different than tooltips in Online maps. In SVG maps, there are no reports in tooltips, just data from data source fields associated with particular map area.



To set up tooltips in a SVG map, click on the *Tooltip Options* icon on the main toolbar. The following dialog will appear:



Tooltip options dialog

On this dialog, you can choose which data source fields will be displayed in tooltips, which will pop up in yellow background when you move your mouse cursor over a map area.



Note

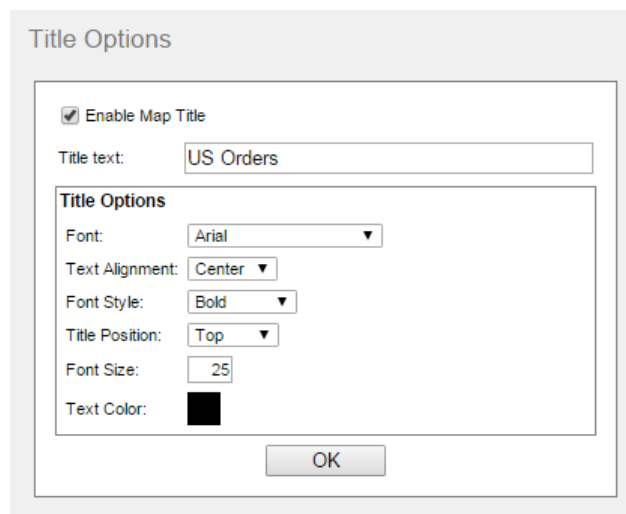
If you deselect all options on the *Tooltip options* dialog, default browser tooltips will be displayed (in grey background), showing SVG area IDs.



SVG Map with a Tooltip

5.3.4.8. Map Title

To insert a map title, click the **T** *Map Title* icon on the toolbar. Check the *Enable Map Title* checkbox. Then you can insert the map title text and select various options - font, font style, font size (positive integers only), text alignment, title position, and text color. The color can be either choosed from swatches or inserted manually by entering red, green and blue components (all values has to be integers from 0 to 255). To apply your settings, click the *OK* button.



Title Options

☒ Enable Map Title

Title text:

Title Options

Font:

Text Alignment:

Font Style:


Title Position:

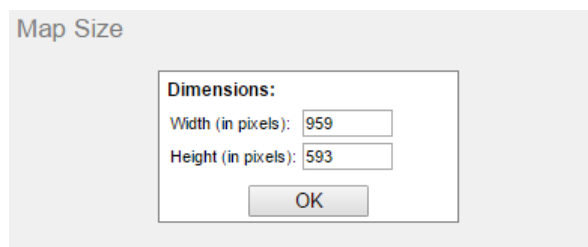
Font Size:

Text Color:

Map Title

5.3.4.9. Map Size

To set a map size, click the  *Set Map Size* icon on the toolbar. Enter dimensions and then click the *OK* button to apply your setting.



Map Size


Dimensions:

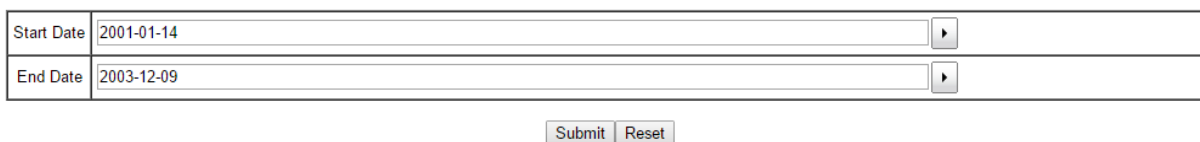
Width (in pixels):

Height (in pixels):

Map Size Setting

5.3.4.10. Change Parameter Values

To change parameter values (for maps with parameterized data source) or refresh the map, click the  *Refresh* icon on the toolbar and the Parameter setting dialog will appear. Select a parameter value(s) and then click the *Submit* button to apply parameter values on the map (You can reset your selection by clicking the *Reset* button).




Start Date	<input type="text" value="2001-01-14"/>	<input type="button" value="▶"/>
End Date	<input type="text" value="2003-12-09"/>	<input type="button" value="▶"/>

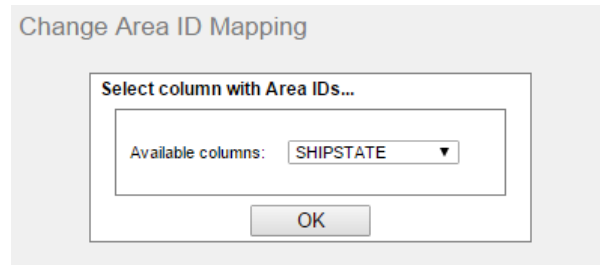
Parametr Setting Dialog for Single-value Parameters


States	<input type="text" value="AR"/> <input type="text" value="CA"/> <input type="text" value="FL"/> <input type="text" value="IL"/>	<input type="button" value="▶"/>
--------	--	----------------------------------

Parameter Setting Dialog for Multi-value Parameters

5.3.4.11. Change Area ID Mapping

To change area ID mapping, click the  *Change Area ID Mapping* icon on the toolbar. Select a column from the drop down menu and then click the *OK* button to apply your setting (For more information about area ID mapping, please see Section 5.3.3 - Area ID Mapping).




Changing Area ID Mapping

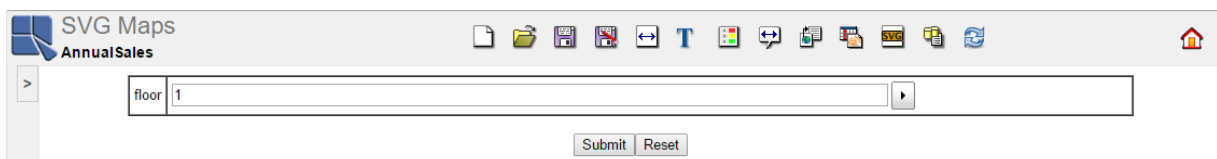
5.3.5. Dynamic SVG Maps

EDAB Dynamic SVG Maps allow changing SVG image according to a parameter, i.e. you can use several SVG images in a single SVG map. Dynamic SVG map can be used also as a drilldown, so you can create useful SVG map with drilldown - for example continent to country, country to state, state to county, etc.

There are some rules and recommendations for Dynamic SVG Maps:


- SVG images have to be in format: `imageName_PARAMETER.svg`, where `imageName` is the same for all SVG images and `PARAMETER` is used as the parameter (e.g. `worldmap_political`, `worldmap_physical`, `worldmap_topographic`,...)
- Data source has to be parameterized where one of parameters takes the same values at the end of the SVG images file names (after the underscore character).
- SVG images have to be saved inside a single folder.
- One of the SVG images has to be in Organizer. This image is used as default for creating a map and it is also used in case the SVG image is not available due to a wrong parameter, missing file, etc.
- It is recommended to have all SVG images in the same size.

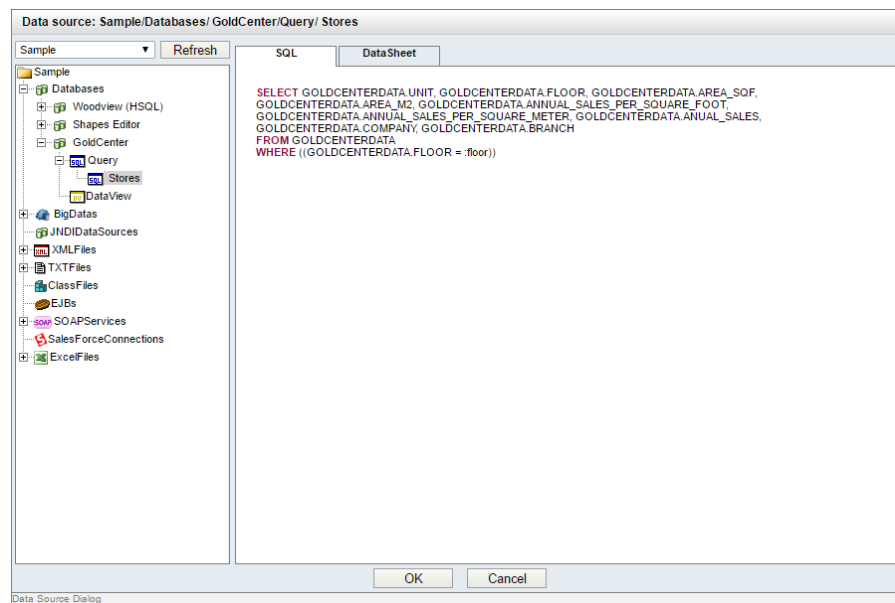
It is possible to change SVG Images by clicking the  *Refresh/Set Parameters* icon on the toolbar. You will see a drop down menu for selecting a parameter. Select a parameter and click the *Submit* button. The corresponding image will be used for the map. All settings of the map (thresholds, legend, title, tooltips, drilldowns) will be applied to each SVG image.



Parameter Selection

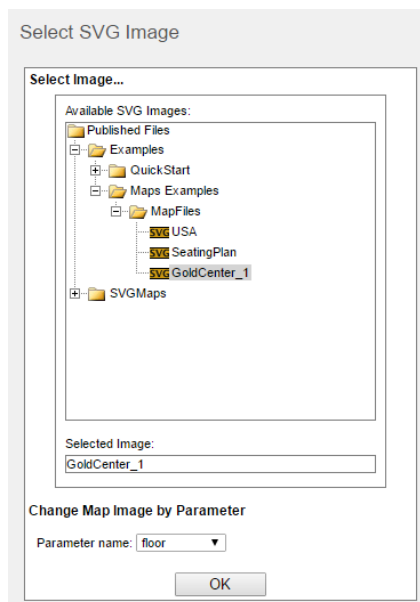
Let's take a look at the following example of a shopping center with three floors. We want to display the status of annual sales per square foot of stores. Open SVG Maps designer (from the EDAB main page) and click the

 *Create New Map* icon on the toolbar. *Data Source Dialog* will appear. Select *Sample* data registry and then select *Databases/GoldCenter/Query/Stores* query as a data source. Click the *OK* button to close the *Data Source Dialog*.



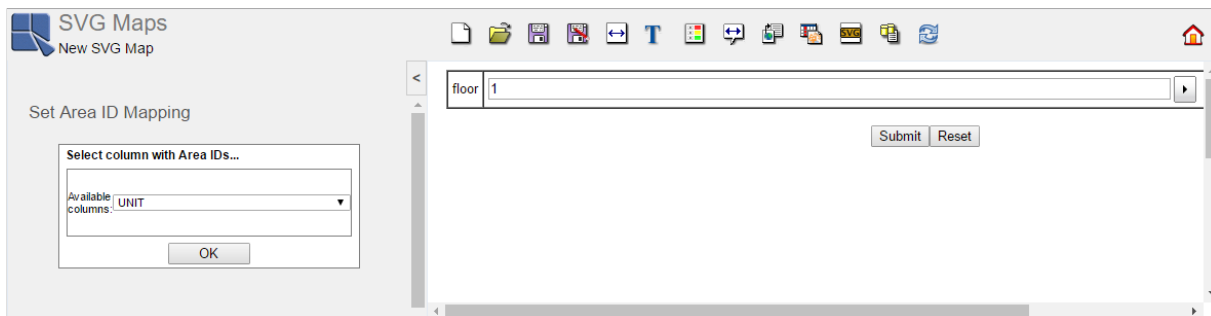
Select Data Source

Now you are prompted to select an SVG image. Select `Examples/Maps Examples/MapFiles/GoldCenter_1` SVG image. We have three SVG images for this example (`GoldCenter_1.svg`, `GoldCenter_2.svg`, and `GoldCenter_3.svg` in the `EDAB/help/examples/Maps/MapExample4` folder), but you can see only one of them in the *Select SVG Image* dialog because only one was inserted into Organizer (it is not necessary to insert all SVG images to Organizer). Next select a parameter for changing SVG images in the *Change Map Image by Parameter* dropdown menu. Select the parameter `floor` and click the *OK* button.



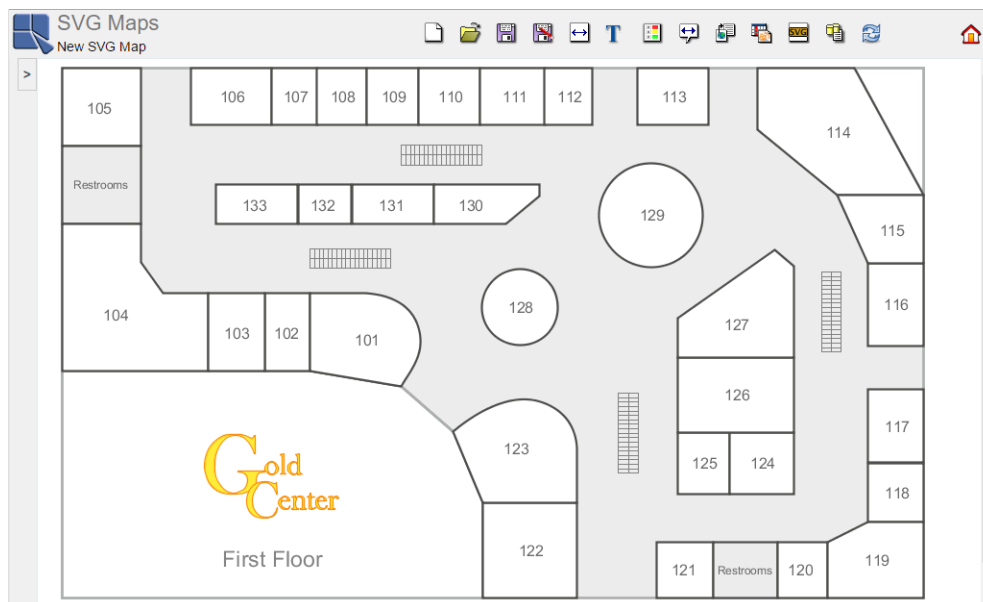
Select SVG Image

Next step is to map IDs to the data source column. Select `UNIT` and click the *OK* button. You can also select a parameter, a number of floor in our case, in this step. Leave the first floor selected and click the *Submit* button to open a map.




Set Mapping and Parameter

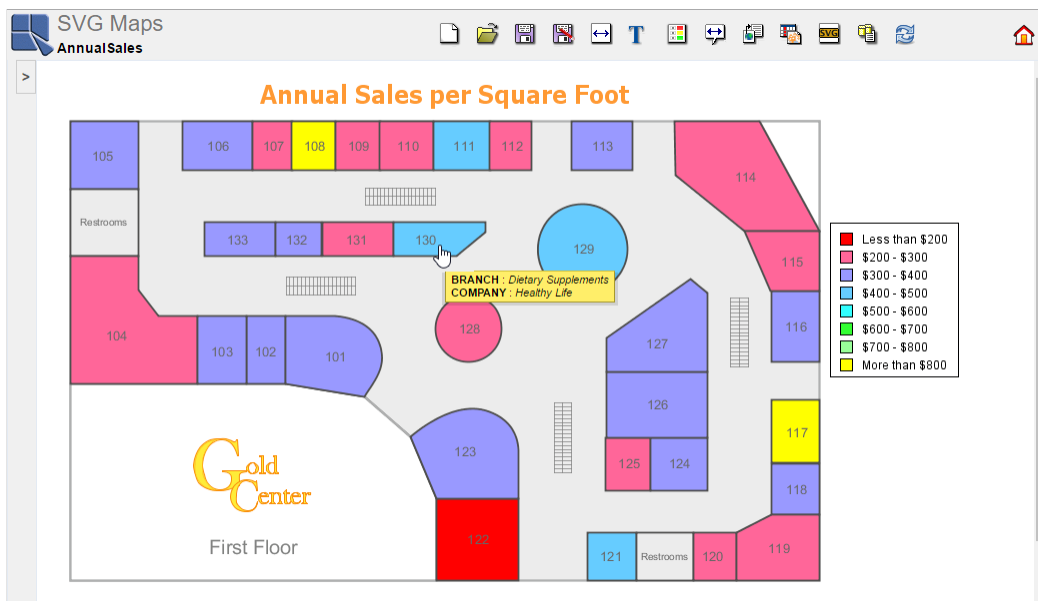
You can now see a map of the first floor (Left pane is collapsed in the next image).



Set Mapping and Parameter

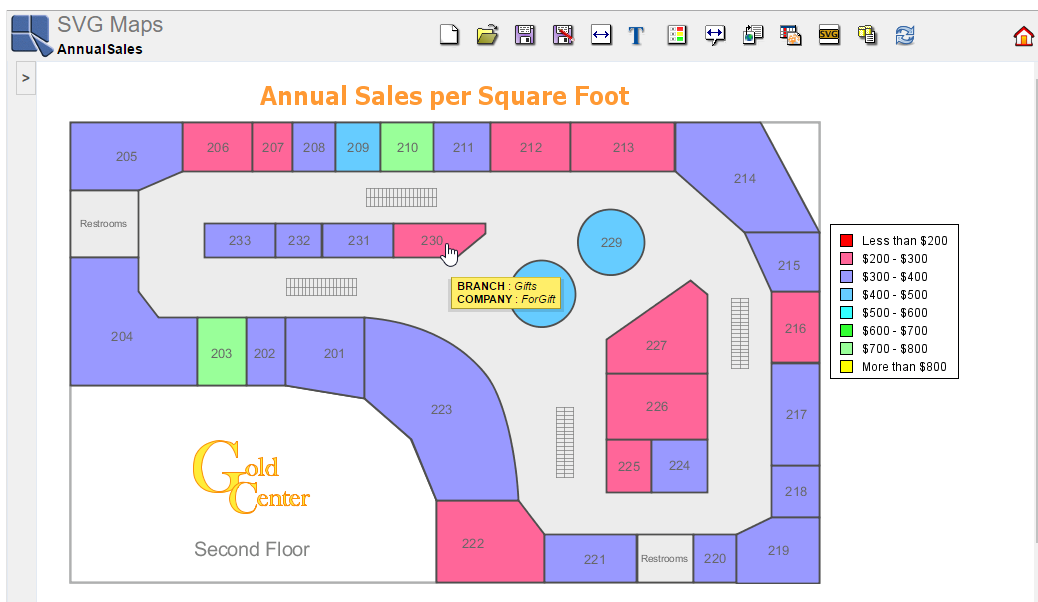
To change the SVG image (the other floor of the shopping centre in our case), you only need to change the parameter value by clicking the  *Refresh/Set Parameters* icon and selecting parameter value from the dropdown menu.

The next image shows the finished SVG map (you can find this SVG map in `Examples/Maps Examples/MapFiles/AnnualSales`). There are set thresholds (for more information, see Section 5.3.4.5 - Thresholds), a legend (for more information, see Section 5.3.4.5.1 - Legend), a title (for more information, see Section 5.3.4.8 - Map Title), and a tooltip (for more information, see Section 5.3.4.7 - Tooltips).



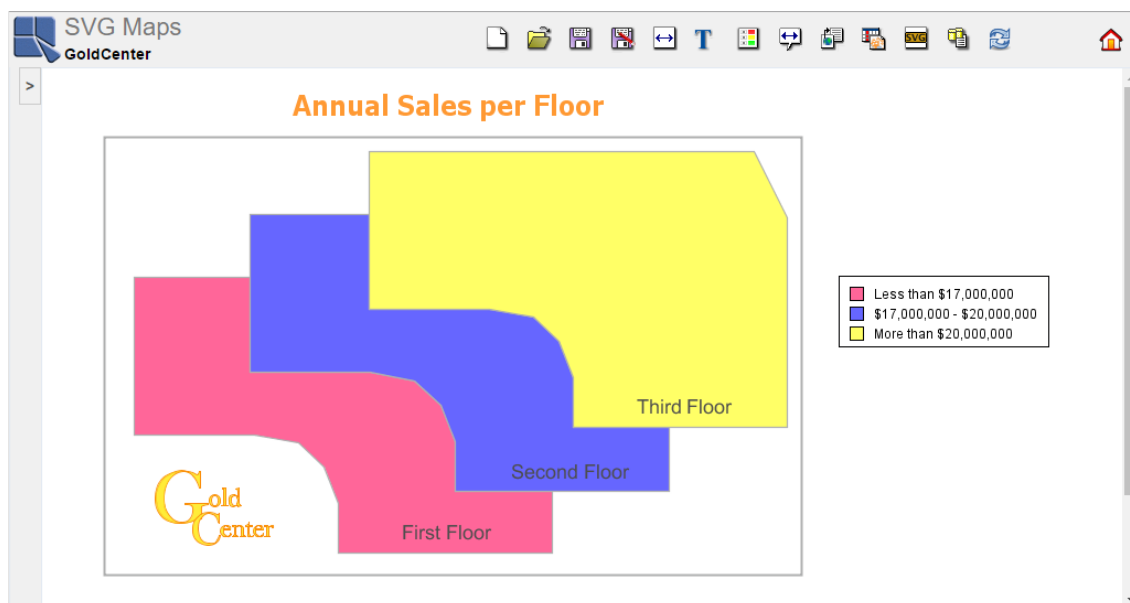
Finished Map

The next image shows the SVG map for the parameter value 2, which is the second floor of the shopping center. All adjustments are also valid for the SVG image of the second floor.




Changed SVG Image

Dynamic SVG map can also be used as a drilldown. For example, open GoldCenter SVG map from Examples/Maps Examples/MapFiles directory. You should see this map:




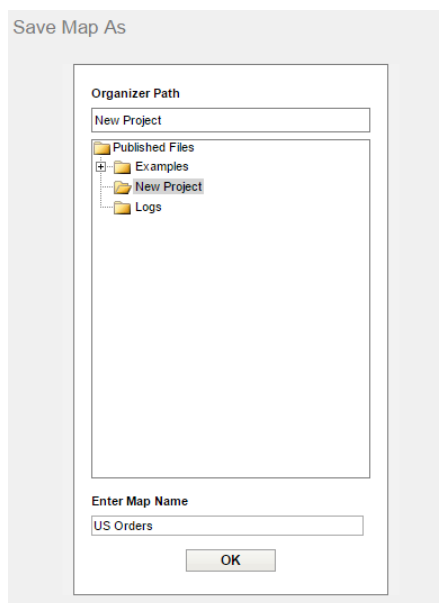
Basal SVG Map

Add a drilldown by clicking the  *Drilldown Options* button, select *Examples/Maps Examples/MapFiles/AnnualSales* file in the *Drilldown Options* dialog and click the *OK* button. Then you can click on the area of some floor to open the drilldown.

You can also open the finished SVG map with drilldown *GoldCenterWithDrilldown* under *Examples/Maps Examples/MapFiles* node.

5.3.6. Save Map


You can save the SVG map by clicking the  *Save* button on the toolbar. This will open a dialog allowing you to specify a name for the map. Enter a name for the map, select the project where you want to save it and click the *OK* button.

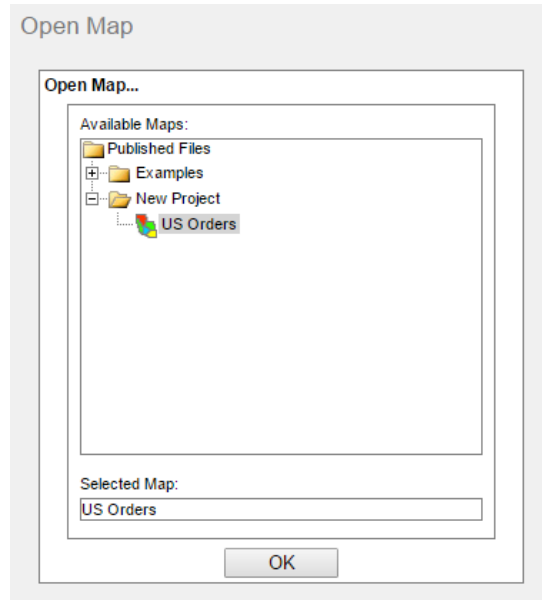


Save Dialog

After saving or opening an existing SVG map, the  *Save As* button allows you to save the existing or modified map into a different name or location.

5.3.7. Open Saved Map

You can open a saved map by clicking the  *Open* icon on the main toolbar. The *Open Map* dialog will appear.




Open the SVG Map

All SVG maps created in SVG Maps designer are visible in Organizer. Select a map and click the *OK* button to open it.

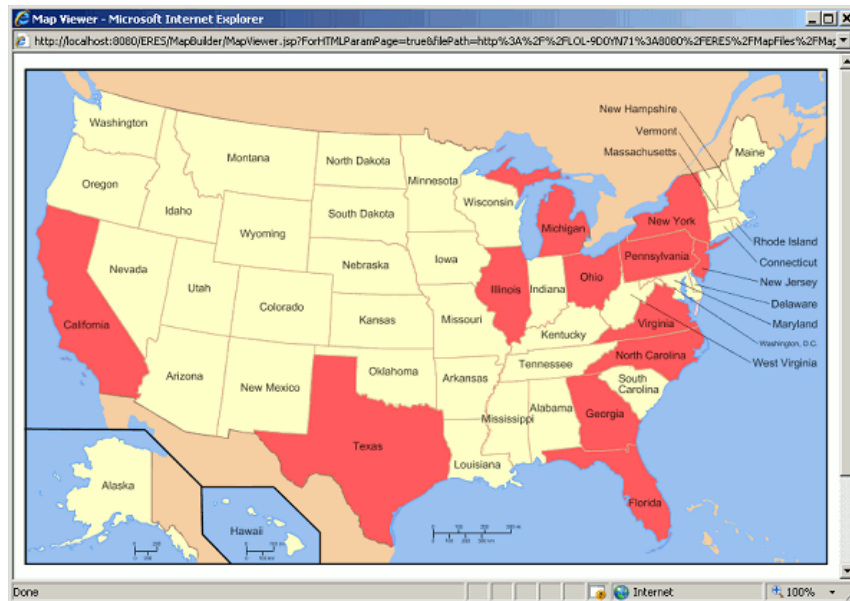
5.3.8. Exit

You can exit SVG Maps designer either by clicking the  *Home* icon in the upper right corner, clicking the *SVG*

Maps title, or clicking the  *Logo* icon in the upper left corner. Before closing, you will be asked if you want to save an unsaved map.

5.4. Map Viewer

Map Viewer is a JSP page used for viewing maps. It is located in `http://machine:port/<EDAB_CONTEXT>/MapBuilder/MapView.jsp` and it can be used for both Online Maps and SVG Maps. If you open a map from Menu Page, it is automatically opened in the Map Viewer (If you open a map from Organizer, it is opened in corresponding map designer). Another way to use the Map Viewer is to write your own map URL. This is described in the next chapter.



EDAB Maps Viewer (showing SVG Maps with threshold)

5.4.1. Writing Map Viewer URL

You can write your own Map Viewer URL. The URL always begins with the location of the Map Viewer JSP: `http://<machine>:<port>/<EDAB_CONTEXT>/MapBuilder/MapViewer.jsp?`. Following the question mark you need to specify Map Viewer parameters. Parameters are separated by the “&” character.

The most important Map Viewer parameter is **filePath**. It specifies the path to the map you want to display. The value can be either URL or file path on the server. The file path can be absolute or relative to the EDAB installation directory. This parameter is mandatory and must always be specified.

The remaining Map Viewer parameters are used for setting map parameters. These parameters have no function in non-parameterized maps, so they are ignored if the map is not parameterized. The parameters are:

ForHTMLParamPage: This is a Boolean flag that indicates whether or not to display parameter page before displaying the map. If this argument is set to true then the URL will return the parameter page and ignore any parameter values passed into the URL. This argument is set to false by default.

QueryParamName: This parameter is used to specify parameter for parameterized map. QueryParamName specifies the name of the parameter you will be supplying a value for (Parameter names are specified when you create the query. For more information about this, see Section 3.1.3.2.2 - Parameterized Queries). It is always followed by QueryParamSize and QueryParamValue parameters.

QueryParamSize: This parameter is used to specify the number of values that will be passed into a particular map parameter. As described in Section 3.1.3.2.2.1 - Multi-Value Parameters, some queries can have multi-value parameters. This URL parameter allows you to specify if the parameter (indicated by the previous QueryParamName parameter) takes multiple values. This parameter is always followed by one or more QueryParamValue arguments for each of the parameter values for a particular parameter. If you do not specify this parameter, it will assume that the number of the values to be passed in is one.

QueryParamValue: This parameter is used to specify a parameter value you want to pass to the query parameter specified by the QueryParamName argument. For multi-value parameters, a separate QueryParamValue argument must be supplied for each distinct parameter value that you are passing in. For more information about creating parameterized queries, please see Section 3.1.3.2.2 - Parameterized Queries.

Example:

To open GoogleMapParam (one of the map examples - parameterized Online Map) with parameter State set to CA, use this URL:

```
http://machine:port/<EDAB_CONTEXT>/MapBuilder/MapView.jsp?filePath=help/
examples/Maps/MapExample1/GoogleMapParam.gxml&QueryParamName=State&QueryParamSize=1&QueryParamValue=CA
```

Please replace machine with your machine name or IP, port with the port Tomcat is running on (8080 by default) and <EDAB_CONTEXT> with your EDAB context (EDAB by default).

5.5. Migrating Maps

Often you may need to move maps from one location to another. For example, maps have to be moved between the server and the develop machine. On the develop machine the tooltips/drill-downs/svg images locations can change and may not be located from the data stored in the map .gxml or .sxml file. Therefore, EDAB uses SPAK (for SVG Maps) and GPAK (for Online Maps) files to move or archive maps along with its components. To learn how to create a SPAK or a GPAK file, see Section 2.1.4.4 - Dashboard and Map Packages.



Note

The SPAK and GPAK files can be viewed in the MenuPage, but they cannot be opened in the SVG or Online map designers. To edit a map from a SPAK or a GPAK file, you have to unpack it first (to learn how to unpack DPAK or SPAK files, see Section 2.1.4.4 - Dashboard and Map Packages).

5.A. List of SVG Map Images

The following SVG map images are distributed with EDAB. They are prepared to be used for creating SVG Maps. They all have uniform look - the land is gray and borders and oceans are white. You can change the colors in any SVG editor (e.g. Inkscape) or you can edit them directly in the SVG files (this requires at least basic knowledge of the SVG format).

The maps are split in two directories according to the Area IDs naming convention. The maps in the <EDAB_Installation_directory>/MapFiles/SVG/English directory use English country (continent) names, while the maps in the <EDAB_Installation_directory>/MapFiles/SVG/ISO directory use ISO 3166-1-alpha-2 [<https://www.iso.org/iso-3166-country-codes.html>] code. All Area IDs in these maps are in UPPER CASE. The English names are exactly the names from this table [<https://www.iso.org/obp/ui/>], only the spaces are replaced with underscores (_), because spaces are not supported in the SVG objects IDs. However, underscores are automatically converted to spaces before comparing them to the data source columns, so you should still use spaces in your data source.

These SVG map images are already inserted in the Organizer in the SVGMap project. Please note that this project is hidden in the Menu Page, but it can still be displayed and modified in Organizer and the files from this project can be used for creating maps.

This is the list of all SVG map images distributed with EDAB with short description. The <NAMING> placeholder can be one of “English” or “ISO”, depending on the naming convention used.

Africa-<NAMING>.svg	Map of all countries in Africa
Americas-<NAMING>.svg	Map of all countries in North and South America
Asia-<NAMING>.svg	Map of all countries in Asia
Australia-<NAMING>.svg	Map of all countries in Australia and Oceania
Continents-English.svg	Map of the whole world divided by continents. The Area IDs are English names of the continents. This map is not available with ISO naming convention because there are no codes for the continents.

Eurasia-<NAMING>.svg	Map of all countries in Europe and Asia
Europe-<NAMING>.svg	Map of all countries in Europe
NorthAmerica-<NAMING>.svg	Map of all countries in North America
SouthAmerica-<NAMING>.svg	Map of all countries in South America
WorldCompact-<NAMING>.svg	Map of all countries in the world except Antarctica.
WorldCompact-Small-Countries-<NAMING>.svg	Same as WorldCompact-<NAMING>.svg, but very small countries are represented by circles, so they are more visible.
World-<NAMING>.svg	Map of all countries in the world including Antarctica.
World-SmallCountries-<NAMING>.svg	Same as World-<NAMING>.svg, but very small countries are represented by circles, so they are more visible.

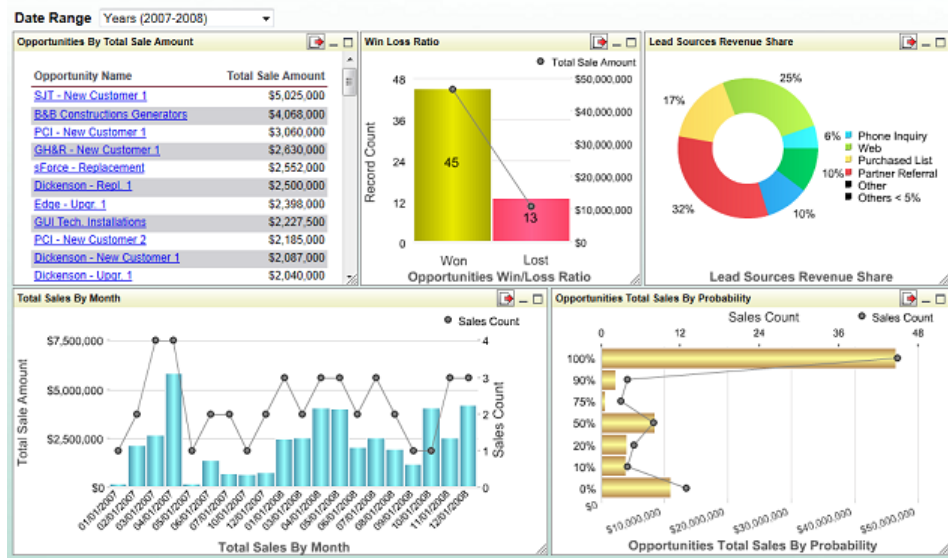
Chapter 6. Designing Dashboards

6.1. Introduction to Dashboards

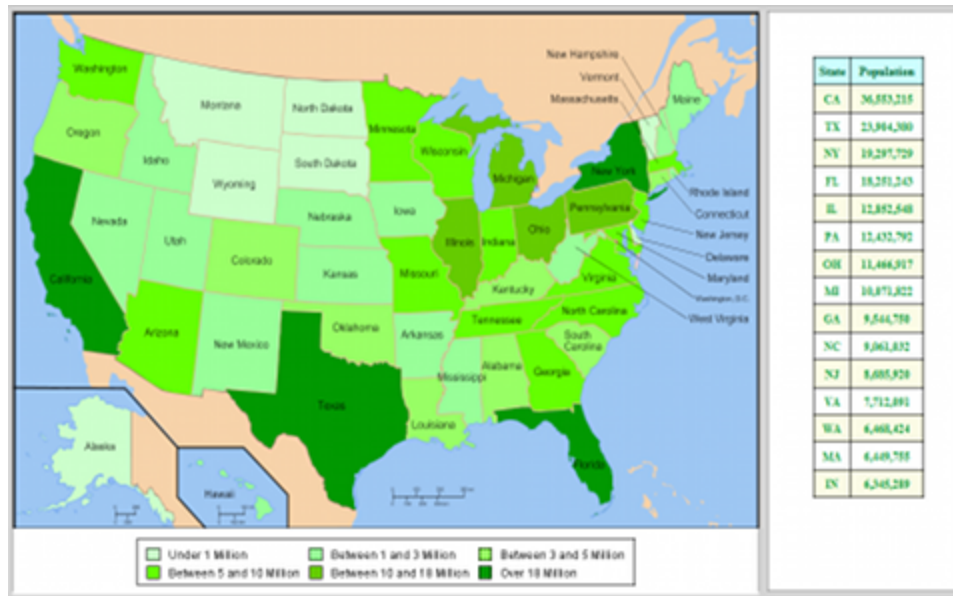
Dashboards allow users to easily consolidate summarized information in a single page or briefing. EspressoDashboard requires no special skill or administrative support. Simple thin-client interfaces allow all users to easily create customized information presentations.

Here are some sample dashboard presentations.

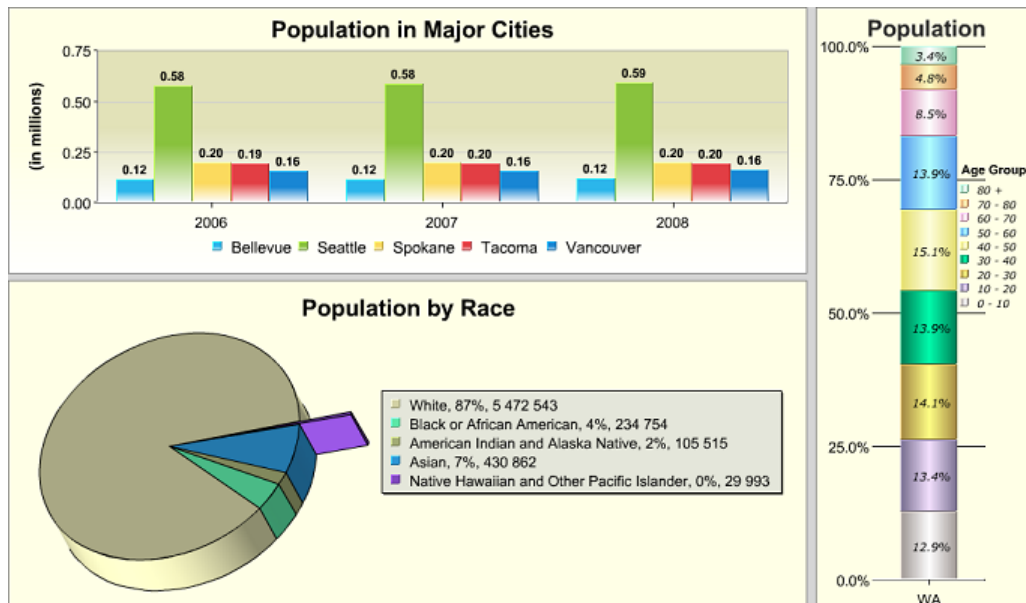
SALES DASHBOARD



Sales Dashboard



Summary Dashboard



Population Dashboard

The top-level or main presentation consists of a set of charts, reports or maps arranged on the page. You can add shared parameters in a panel, or list the values of parameters so that the dashboard users can apply filters to refresh the applicable charts/reports/maps.

Individual options for each chart or report allow users to apply additional filters as well as export the charts, reports and maps to various formats.

Users can also add layers of drilldown to each chart/report/map in the main presentation. The drilldown can point to a report, chart or map. This allows each piece of summarized information in the presentation to be immediately actionable.

6.2. Create Dashboard

Dashboards in EDAB can be created using the Dashboard Builder interface. This easy-to-use thin-client interface allows you to easily create dashboard presentations using charts, reports and maps. You can simply insert dashboard components (charts, reports and maps) that are already deployed in the Organizer to the dashboard. Alternatively, you can create them from scratch directly in the Dashboard Builder. The newly created item will be automatically inserted in the dashboard. One convenient feature is that you can edit a chart, report or map in the Dashboard Builder interface without leaving it.

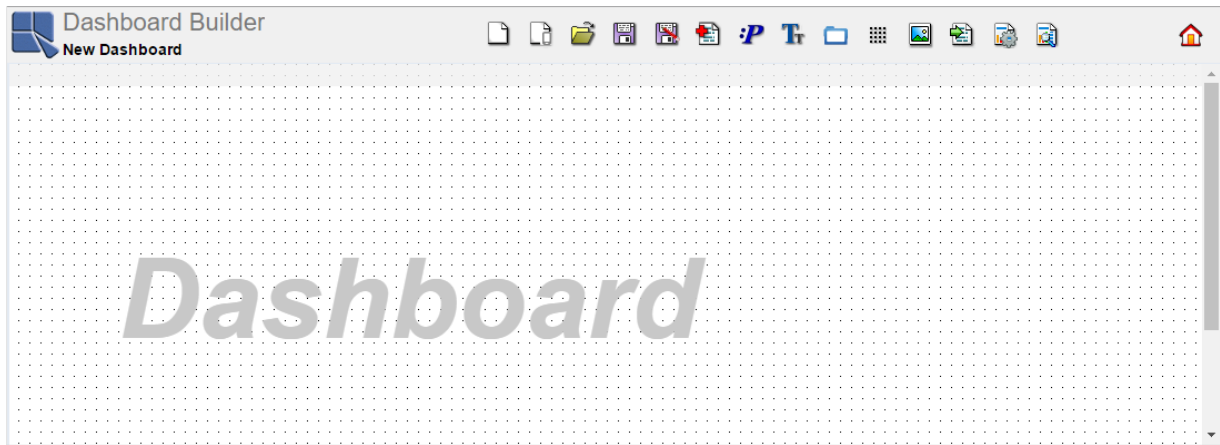
The dashboard layout can be either static or responsive. As you may know, responsive layout is great for displaying content on small devices such as mobile phones. In a static dashboard, the width is dependent on the size and layout of its components. If the dashboard is wider than the screen, all the components stay in place and a horizontal scrollbar appears. Unlike the static dashboard, the responsive layout, using Responsive Dashboard feature, optimizes utilization of the display without exceeding the display width. Therefore the layout is automatically rearranged. The width of the responsive dashboard can also be set manually by dragging a limiter, a vertical line on the right side of the Dashboard Builder working space (in case you want the dashboard to be narrower than the screen).



Note














It is not possible to display static dashboards as responsive and it is not possible to change them to responsive and vice versa.

The Dashboard Builder can be launched from the EDAB Start page. If you've logged in as a user with design privileges, you can follow the Dashboard Builder link to open the interface.

*Dashboard Builder Interface*

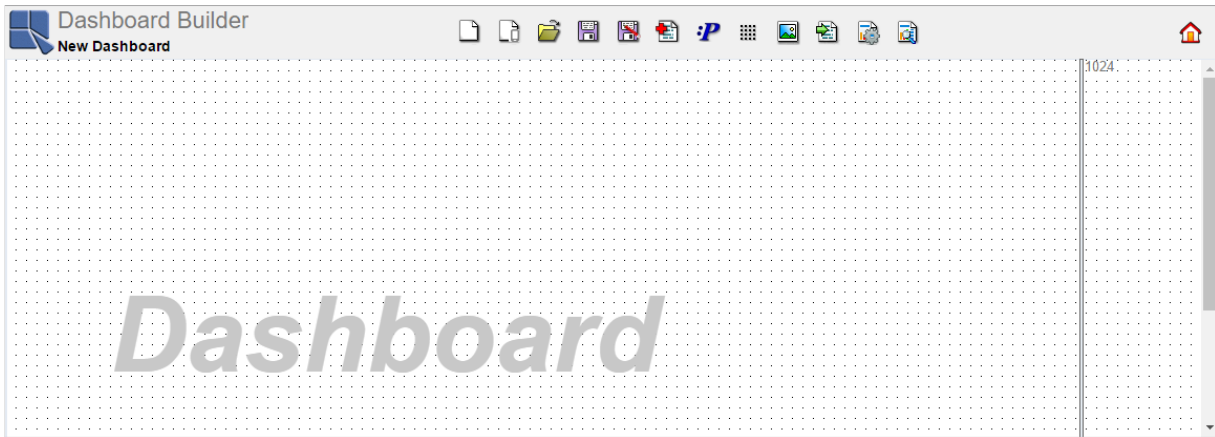
6.2.1. Toolbar

The top part of the Dashboard Builder interface contains a small toolbar that allows you to initiate the following actions:

-  Start a new static dashboard
-  Start a new responsive dashboard
-  Open an existing dashboard
-  Save the current dashboard
-  Add report/chart/map to the current dashboard
-  Add share parameter(s)
-  Insert a label into the current dashboard
-  Insert Folder
-  Hide/Show Grid
-  Add dashboard background
-  Migration
-  Dashboard additional options
-  Preview the current dashboard

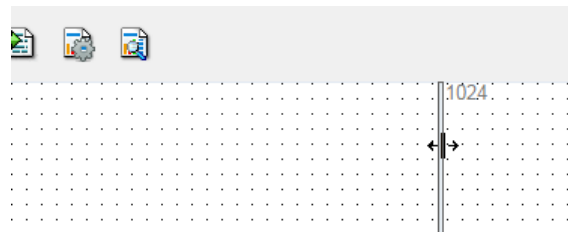
6.2.2. Responsive Dashboard

To create a new responsive dashboard, click the  *New Responsive Dashboard* icon on the toolbar.



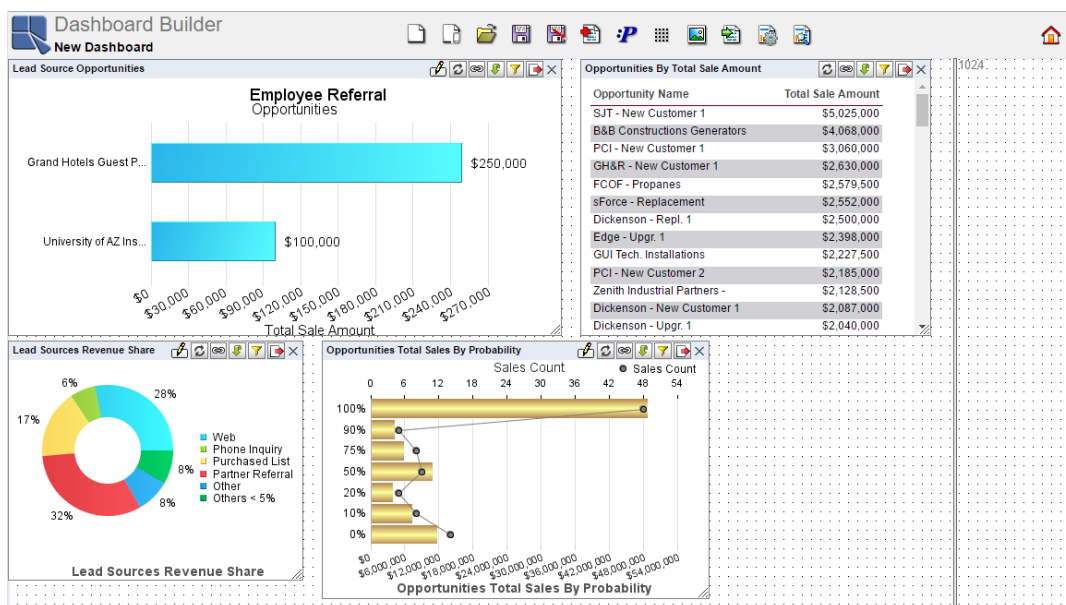
New Responsive Dashboard

You can click and drag the limiter to set a dashboard width.



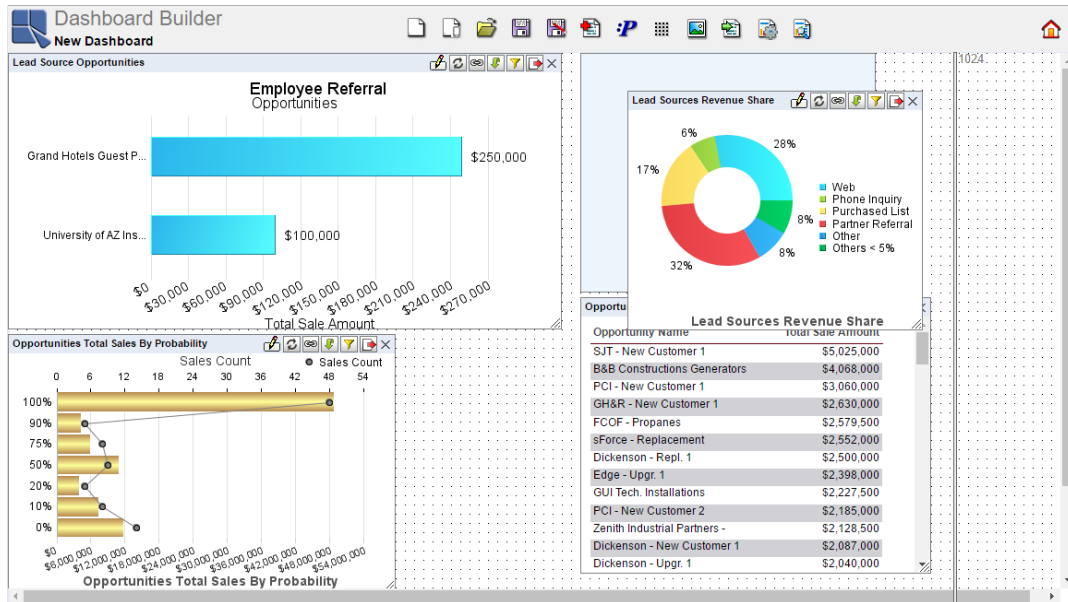
Dashboard Limiter Setting

Now you can add reports/charts/maps and resize them. The limiter (or the screen width) keeps the width of the dashboard. The layout of reports/charts/maps is arranged automatically.



Dashboard With Limiter

To change the layout manually, click on the header of the report/chart/map, hold the mouse button and move it. If there is enough space for the component, a blue rectangle will appear. Release the mouse button. The moved component is placed instead of the blue rectangle and the other components are arranged as best as possible.



Dashboard Layout Changing



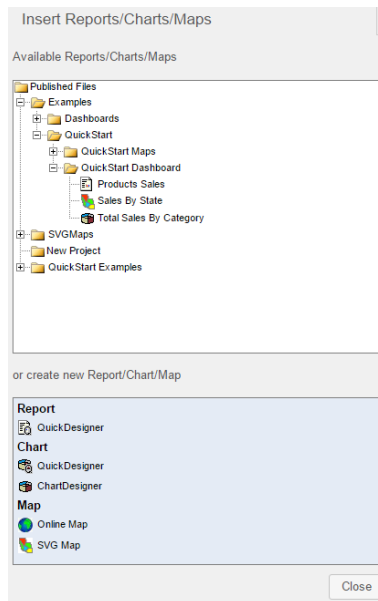
Note

It is not possible to insert labels and folders (see Section 6.2.9 - Folders) into responsive dashboards. Also, you cannot show component header bar (see Section 6.2.7 - Additional Options). Hence dynamic charts are not supported in preview and published dashboards.

6.2.3. Add Charts, Reports and Maps




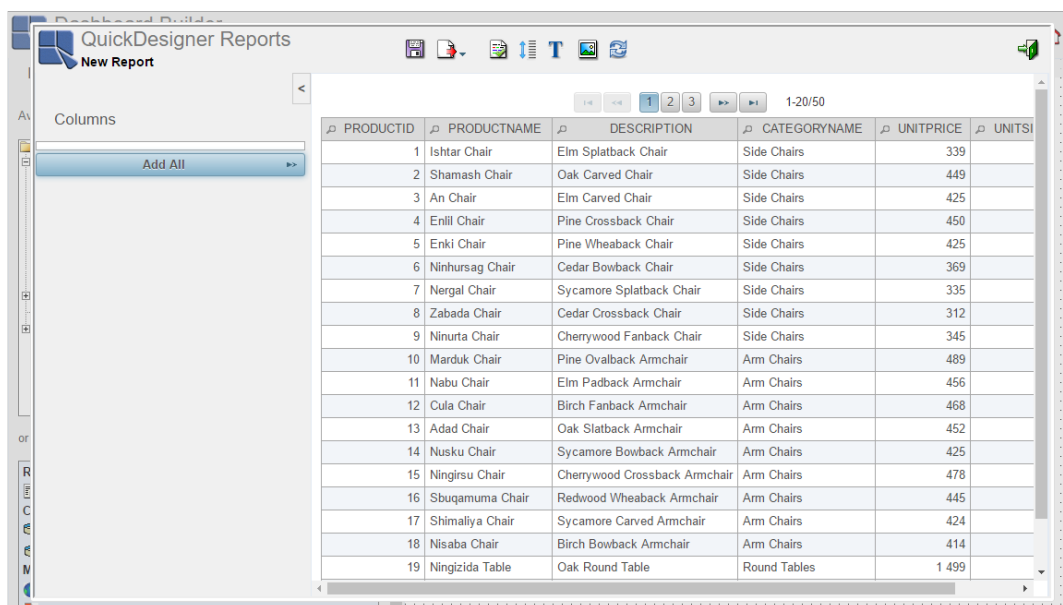
To add a chart, report or map to the dashboard, click the **Add Report/Chart/Map** button on the toolbar. After you do so, the *Insert Reports/Charts/Maps* dialog will appear in the left pane. The dialog contains a tree that mirrors the folder structure in the Organizer. All charts, reports and maps to which you have access to are listed here. In order to insert a chart, report or map into the dashboard, simply click on the chart or report entry in the tree. The chart, report or map will then be added to the dashboard.



Insert Report/Chart/Map Dialog

You can also create a new report/chart/map directly from the Dashboard Builder. To do this, click the relevant link in the lower pane of the *Insert Reports/Charts/Maps* dialog (titled *create new Report/Chart/Map*) and you will be taken to the corresponding designer. Create a new report/chart/map in the designer and save it. Then close

the designer or click the exit icon  to return to the Dashboard Builder. Your new component is added to the dashboard. For more information about working in designers, please see Section 4.1 - Chart Designer, Section 4.2 - QuickDesigner Reports, Section 4.3 - QuickDesigner Charts, Section 5.2 - Online Maps, Section 5.3 - SVG Maps.

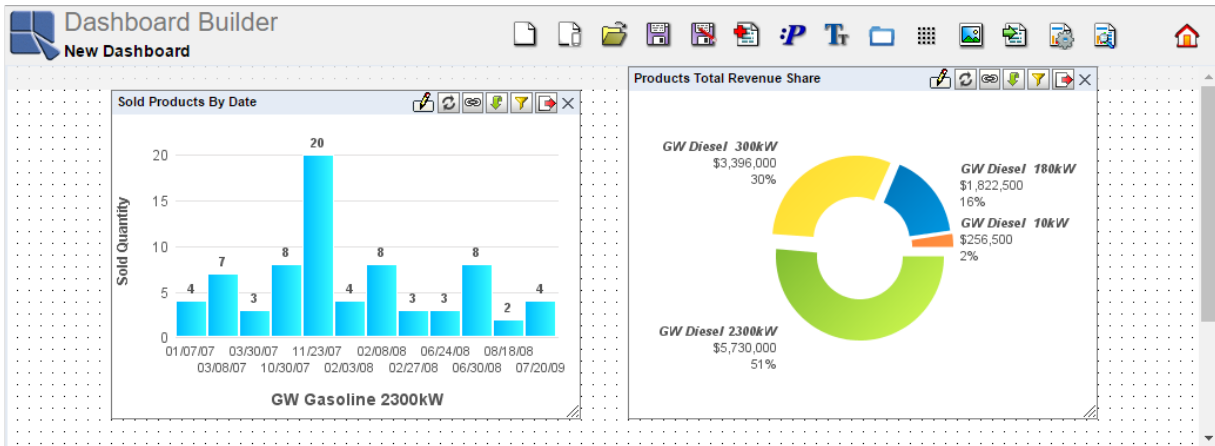


QuickDesigner Reports Opened from Dashboard Builder

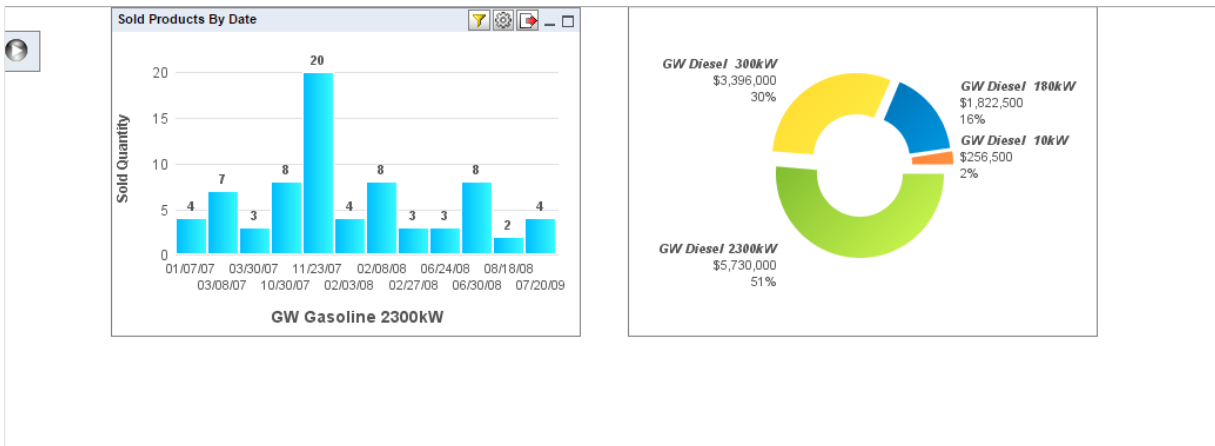
6.2.3.1. Move and Resize Charts, Reports and Maps

Charts, reports and maps in the dashboard can be moved or resized in free-form. You can resize and move charts, reports and maps using the mouse. To move a chart/report/map, simply click on the header bar of the chart/report/map window and drag it. To resize a chart/report/map, click on the lower right corner of the chart/report/map window and drag on the sizing handle that appears in the lower right corner of the window.


You can see a gray stripe under the Dashboard Builder toolbar. This stripe allows you to place a report/chart/map so that its header bar isn't visible in the preview window. You can see the difference in two images below.



Report/Chart/Map Placement

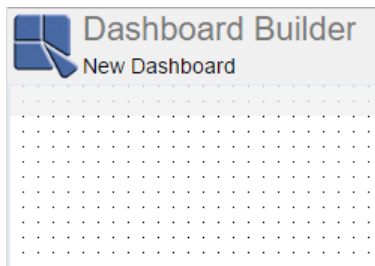


Report/Chart/Map in Preview



If a report, chart or map contains alerts, *Watch alerts* icon  appears in the header bar. To learn more about alerts in dashboards, please visit Section 11.3 - Alert types.

6.2.3.2. Snap to Grid

You can see a grid of dots in Dashboard Builder that helps you to align charts, reports and maps in the dashboard. The grid is visible by default and it snaps all dashboard elements (reports, charts, maps).



Dashboard Grid

You can hide/show the grid by clicking the  *Hide/Show* icon on the toolbar. To position charts/reports/maps completely free, open the *Dashboard Options* dialog by clicking the  *Options* icon and disable the option *Snap to Grid* under *Other* section.

6.2.3.3. Chart/Report/Map Toolbar

Each chart/report/map in the dashboard has its own toolbar that allows you to initiate the following actions:



Edit Template - This button opens appropriate designer that allows you to edit a template.



Refresh Template - This button refreshes a template.



Add/Modify Link - This button allows you to use data from one chart/report/map as a parameter in another. (see Section 6.2.10 - Template Linkage)



Add/Modify Drilldown - This button allows you to add/modify drilldowns. (see Section 6.2.8 - Drilldown)



Filter Data - This button allows you to specify filters/parameters for a template. (see Section 7.4.2 - Preview Options)



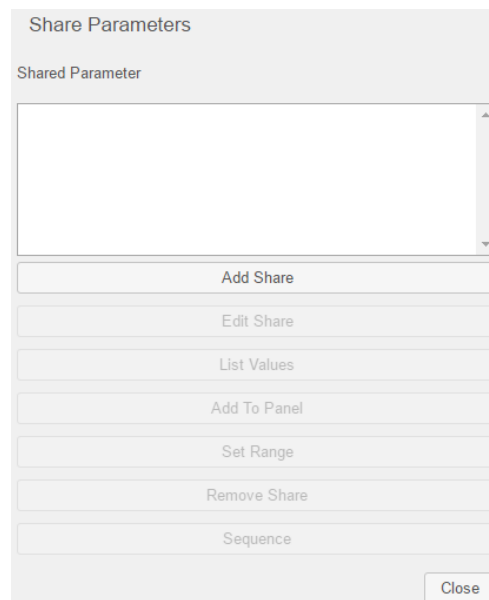
Export - This button allows you to export chart/report/map. (see Section 7.4.2 - Preview Options)

6.2.4. Shared Parameters

Shared parameters allows you to group common parameters from all charts/reports/maps into a single parameter. You can apply a common filter to some or all of the dashboard items at the same time. All of the dashboard's shared parameters are listed in the *Shared Parameter* list in the *Share Parameters* dialog that can be opened by clicking

the  *Share Parameters* icon on the Dashboard Builder toolbar.

For example, create a dashboard by adding all files from the *Examples/Dashboards/CategorySales-Dashboard* node (OrdersCatRegion, ProductSales, SalesRegionalSuccess, TimeSales). Then click the *Share Parameters* icon. You will now see *Share Parameters* dialog in the left pane.



Share Parameters Dialog

To create a shared parameter, click the *Add Share* button in the Share Parameters dialog. This will open a dialog that allows you to set options for the share.

Shared Parameter Dialog

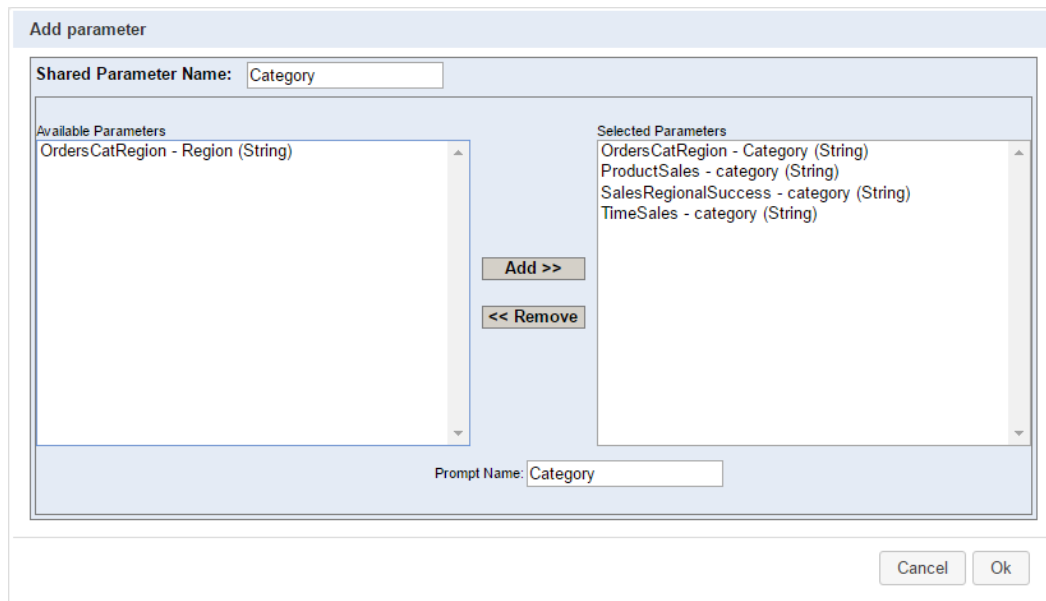
The left side of the dialog shows all parameters for all the selected charts/reports/maps in the dashboard. You can select which parameters you want to add to the share by selecting them in the left side window and clicking the *Add* button. Note that all parameters in the share must have the same data type.

The parameter mapping options (whether the parameter is mapped to a database column) is based on the first parameter selected for a share. For more information about parameter mapping, see Section 3.1.3.2.2.2 - Initializing Query Parameters.

You can enter a prompt name for each shared parameter. Once you finish entering all informations for the shared parameter, be sure to specify a name for it in the field at the top and then click the *Ok* button to save the changes. The parameter will then be added to the *Shared Parameters* list in the share parameters dialog.

You can edit or remove a shared parameter by selecting it in the list and clicking the *Edit Share* or *Remove Share* buttons.

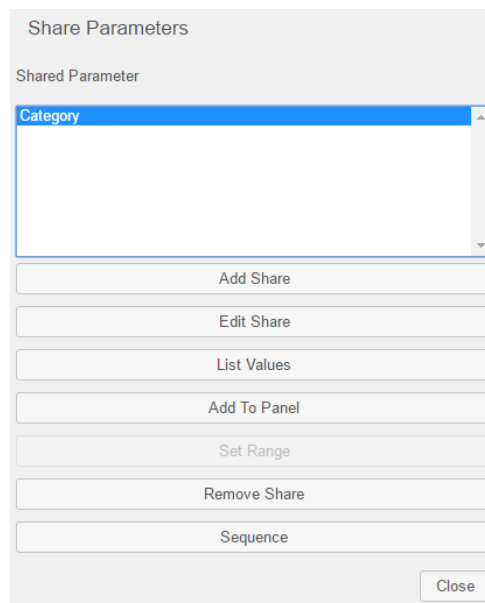
For our example, write *Category* as the *Shared Parameter Name* and the *Prompt Name* and select all available parameters *Category*. Click the *Ok* button to create a shared parameter.



Shared Parameter Dialog

6.2.4.1. Arrange Shared Parameters

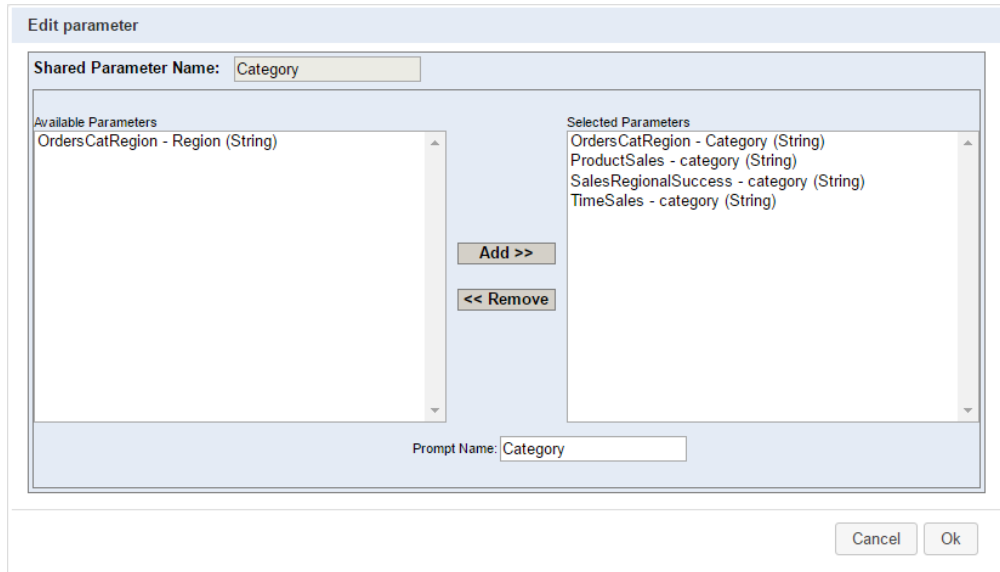
In addition to adding/editing/removing shared parameters, you can also list values of a shared parameter into a set of buttons in the dashboard or arrange a shared parameter to be displayed in the panel that allows you to enter parameter value right from the dashboard. Please note that in order to list the values of a parameter, the shared parameter can be either single or multi-valued and must have predefined selectable values. To add the shared parameters to the dashboard, simply click the *List Values* or *Add to Panel* button in the *Share Parameters* dialog.



Share Parameters Dialog

Edit Share

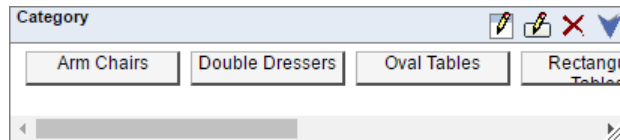
Allows you to edit a shared parameter. To do this, select a shared parameter and click the *Edit Share* button. This will open *Edit parameter* dialog.



Edit Shared Parameter

List Values

If the parameter has a small number of values, you can choose to list the values as buttons or images. For example, let's say your dashboard shows sales data for your products and you have a relatively small number of products. You can select this option to make a list which makes it easier for the users to pick a value for the filter. To list the values for a shared parameter, first select it from the shared parameters list and click the *List Values* button. This will open the parameter value-list panel that contains the parameter values in buttons.



Parameter Value-list Panel

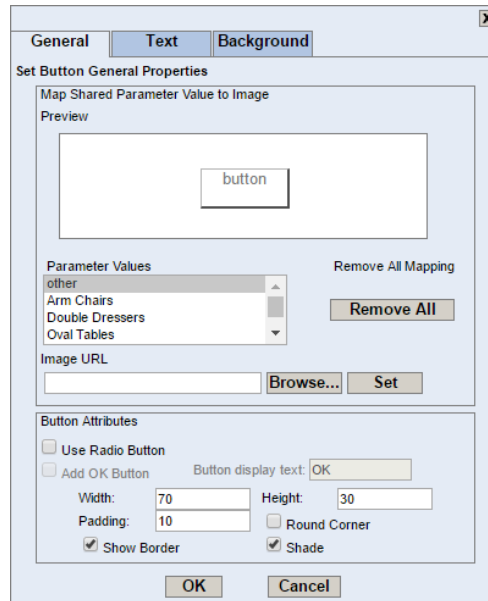
There are four buttons available in the panel header bar.



Modify Panel Attributes: This button will open a new dialog that allows you to modify panel attributes such as background color and panel border. For more information about the dialog, please see Section 6.2.5 - Insert Labels.



Modify Button Attributes: This button will open the following dialog that allows you to modify button attributes.

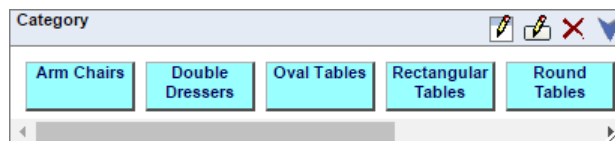


Set Button Properties Dialog

The first option in this tab allows you to map shared parameter value to an image. To map a shared parameter value, first select it either from the left panel, specify the image URL or browse for it using the *Browse* button and click the *Set* button. In order to remove image URL mapping, select appropriate parameter value, clear the image URL text box and press the *Set* button or use the *Remove All* button to remove all mapping. The other options in the dialog allows you to specify button width, height and padding. You can also choose whether to show the buttons border or whether the buttons should be shaded or not. Another option allows you to display the boxes with a radio button. This option is not available for multi-valued parameters, which are automatically given checkboxes.

The other tabs of the Set Button Properties dialog allows you to set the button text and background. The tabs are basically the same as for labels and they are described in the following Section 6.2.5 - Insert Labels.

The next screenshot shows our example with a modified view of buttons.




Modified Buttons




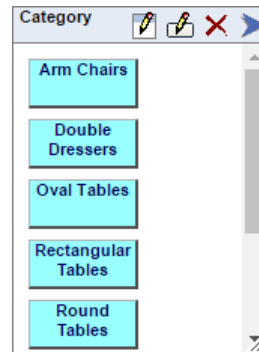
Delete: This button will delete the parameter value-list panel.



Set Buttons Alignment: These buttons allows you to set vertical or horizontal alignment of buttons. By default, buttons are aligned horizontally and you will only see

the  *Vertical Align* button. When you click on it, the alignment will be changed

to vertical and the  *Horizontal Align* button will appear.



Vertical Buttons Alignment

Add To Panel

To arrange the layout of shared parameters by adding them to a panel, first select it from the shared parameters list and click the *Add to Panel* button. This will open a parameter panel with the parameter's list box added to the panel.



Parameter Panel

In the Dashboard Builder, you can move/resize the panel or move the small parameter box inside the panel. If you close the panel, you can still use the shared parameter with the preview toolbar (see Section 7.4.1 - Preview Toolbar). If you change the parameter value, the dashboard will be automatically refreshed.

There are three buttons available in the panel header bar:



Disable Auto Rearrangement: This button will disable/enable auto rearrangement.



Modify Detach Panel Attributes: This button will open a new dialog that allows you to modify the parameter panel attributes such as adding an *OK* button, changing panel layout, text label properties, background color and border for the panel. For more information about the dialog, please see Section 6.2.5 - Insert Labels.



Delete: This button will delete the parameter panel.

Set Range

This option allows you to define a range across two shared parameters. For more information, please see Section 6.2.4.2 - Parameter Range.

Remove Share

This option allows you to remove shared parameter. To do this, select the shared parameter and click the *Remove Share* button.

Sequence

This option allows you to set an order of shared parameters. For more information, please see Section 6.2.4.3 - Cascading Parameters.


6.2.4.2. Parameter Range

This function allows you to define a range across two shared parameters. To use this function, select two related shared parameters (such as start and end date, or integer objects representing a range) and then click the *Set Range* button. This will bring you to the *Range Param Attributes* dialog. To create a user defined range variable, simply fill out the *Range Variables Detail* dialog box. To see an example, please see the QuickStart Guide (Section Q.9.1.1 - Create a Dashboard). The next screenshot shows other setting of parameter range for the example created in QuickStart Guide.

In addition to creating your own range variables, you can also create range variables using *Predefined Range Variables* as described below.

Parameter Range Dialog

The dialog has these following options:

- Range Name:** This field allows you to enter a title for the parameter range window.
- Prompt Text:** This text is displayed next to the selected combo parameter dropdown menu.
- Range Variable Detail:** This window has several sub-fields:
 - Option Name:** This is the display name in the *Selected Range Variables* list.
 - First Param:** This is the first parameter in the combined parameter, usually the starting value.
 - Second Param:** This is the last parameter in the combined parameter, usually the ending value.
 - Expression:** There are two expression fields, one for each parameter.
For date parameters, you can click on the *Date calculator* button  next to the expression field.

In this screen, you can enter one of the three keywords: *CurrentDate*, *CurrentTime*, and *Current-*

DateTime, or one of the function names, such as FIRSTOFTHISYEAR. Details about functions are listed in the list of functions.



You can add or subtract time units from the current date/time, allowing you to have a dynamic date range. For example, a report may have the following default values:

StartDate: CurrentDate - 1 WEEK

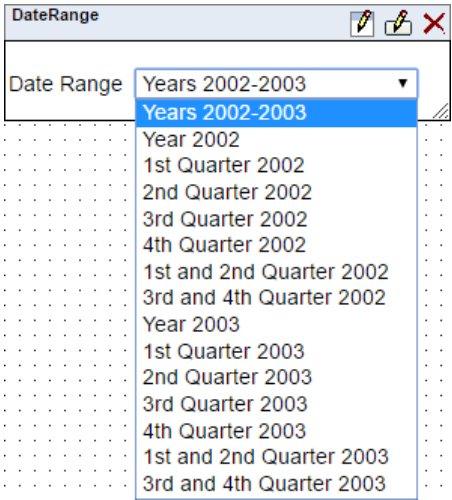
EndDate: CurrentDate

This would indicate that every time the report is run, the default prompt should be one week ago of the current date. Other supported options are YEAR, MONTH, DAY, HOUR, MINUTE, and SECOND. This feature only supports single addition or subtraction and it does not support multi-value parameters.

Selected Range Variables:

This window displays all range variables that have been defined. Clicking on any range variable in this list allows you to modify it, delete it or copy it. Once the variable is selected, you can see its detail in the *Range Variable Detail* window where you can modify it. You can move the selected range variable up or down in the list by clicking the   arrows right next to the window.

The next screenshot shows a dropdown menu in the dashboard that contains defined range variables (range variables from the *Selected Range Variables* window).



Range Variables in Dashboard

Add:

This button has two modes: *Change* or *Add*. When the text of the button reads *Add*, the parameter information in the *Range Variable Detail* window will be added into the *Selected Range Variables* list.

Range Variable Adding

Change:

This button has two modes: *Change* or *Add*. Clicking on an item in the *Selected Range Variables* list will change the text to *Change*. When in this mode, clicking the button will save any changes made in the *Range Variable Detail* window to the currently selected range variable.

Range Variable Change

Copy:

This button will copy the selected item and add it to the *Selected Range Variables* list under a new name. The new name is created automatically by adding *_0*, *_1*, *_2*, etc. or you can change it in the *Range Variable Detail* window. To copy a variable under your name, select it, change a name in the *Range Variable Detail* window and then click the *Copy* button.

Copy can help you to create range variables more quickly. For example, you can select a range variable, change only the year and then click the *Copy* button.

Range Variable before Copy

Range Variable Copied

Remove:

This button will remove the selected item from the *Selected Range Variables* list.

Predefined Range Variables:

This window contains predefined date range variables for your convenience. It contains commonly used date range expressions such as *This Year*, *This Month*, *This Week*, *This Quarter*, and many more. To use a predefined variable, simply select the needed predefined range variable and click the *Add* button. You can add more than one expression and all of them will be listed under the *Selected Range Variables* window.

Allow Custom Range Settings:


This option allows you to specify a parameter range by using calendar and parameter input boxes in the parameter range panel. The following screenshots show the difference between with and without custom range.

Enabled Custom Range Settings
*Disabled Custom Range Settings***Enable Submit Button**

If you allow *Custom Range Settings*, you can add *Submit Button*. This button allows you to apply custom range when both parameters are set. Without this button, each parameter is applied immediately when it is set. You can also determine a text of this button.

Once you finish setting up the parameter range in the *Range Param Attributes* dialog, click the *Ok* button to close the dialog. A rectangle will follow your mouse pointer around the dashboard design window. Position the rectangle and click. A parameter range panel will appear.

Parameter Range Panel with Submit Button

You can select range variables from the dropdown menu where are all variables from the *Selected Range Variables* window, or use customer text boxes (text boxes are available only when *Allow Custom Range Settings* option was checked). The text boxes represent the first and second parameters in the combined parameter. To use customer text boxes, check the radio button next to them and insert parameter values. Note that you can use the calendar to specify the date parameters. To open the calendar, simply click the *Calendar* button  next to the textbox.

There are three buttons available in the panel header bar:



Modify Panel Attributes: This button will open a new dialog that allows you to modify panel attributes such as text label properties, background color and border for the panel. For more information about the dialog, please see Section 6.2.5 - Insert Labels.



Modify Range Param Attributes: This button will open the *Range Param Attributes* dialog that allows you to edit range variables.



Delete: This button will delete the range parameter panel.

**Note**

If you delete this panel, all range variables that were defined in the *Range Param Attributes* dialog will be lost.

6.2.4.3. Cascading Parameters

By default, you are prompted to enter all report parameters at once in the prompt dialog. However, this configuration may not be the best solution in case some parameters are mapped to database columns with a significant number

of distinct values. It can be difficult to go through a very large list and you may select values that don't return any data, depending on the parameter combination.

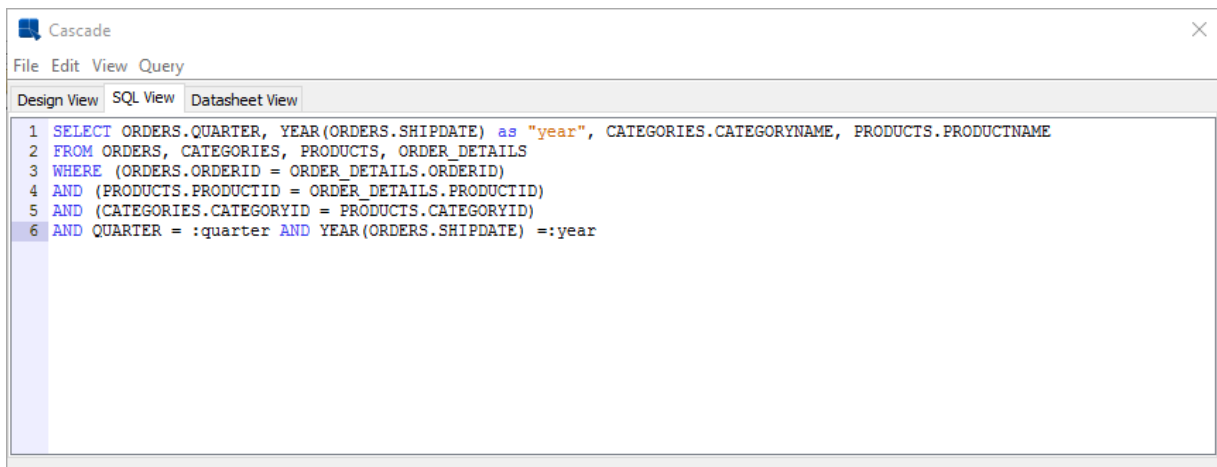
To assist with these problems, EDAB provides a feature that allows you to configure the order in which the parameters should be entered. With this feature enabled, you can enter parameters in the dialog in a pre-defined order. As such each selection will be applied as a filter to the next parameter prompt(s). Using cascading parameters can limit the number of distinct values presented and can prevent selecting invalid parameter combinations.

First, create a query named e.g. Cascade with parameter sequence in Woodview HSQL database that is in EDAB.



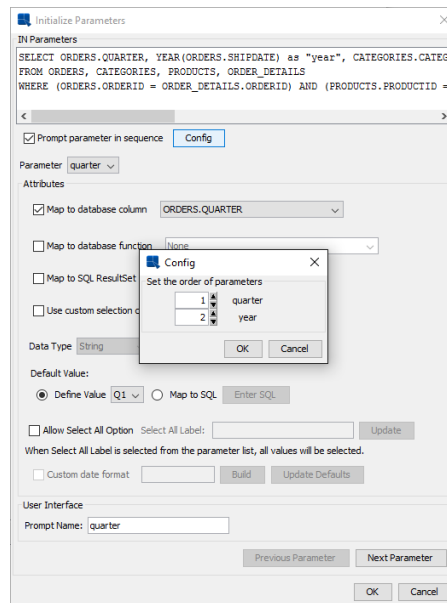
To do this, open Organizer, click the *Manage Data Sources* icon to open the *Data Registry Manager* dialog, select a data registry and click the *Edit* button. Select *Queries* under *Databases/Woodview (HSQL)* node and click the *ADD* button to create a new query. Type *Cascade* as a name and for example, use the following query:

```
SELECT ORDERS.QUARTER, YEAR(ORDERS.SHIPDATE) as "year",
       CATEGORIES.CATEGORYNAME, PRODUCTS.PRODUCTNAME
FROM ORDERS, CATEGORIES, PRODUCTS, ORDER_DETAILS
WHERE (ORDERS.ORDERID = ORDER_DETAILS.ORDERID)
AND (PRODUCTS.PRODUCTID = ORDER_DETAILS.PRODUCTID)
AND (CATEGORIES.CATEGORYID = PRODUCTS.CATEGORYID)
AND QUARTER = :quarter AND YEAR(ORDERS.SHIPDATE) =:year
```



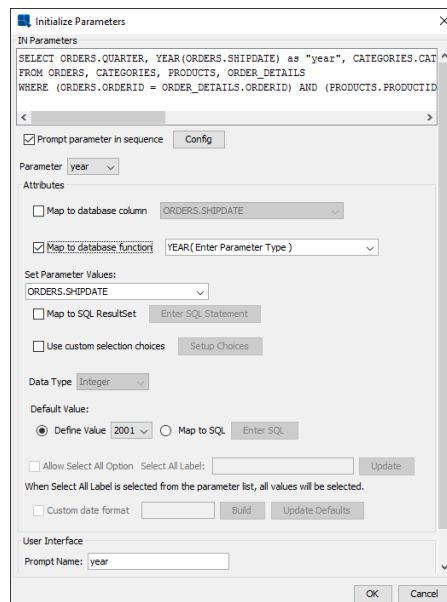
Cascading Parameters Query

Now initialize query parameters. Check *Prompt parameter in sequence* and click the *Config* button. Set the parameter sequence according to the image below, i.e. the *quarter* parameter will be in the first level and *year* in the second level. Then map the first parameter *quarter* to database column *ORDERS.QUARTER*.



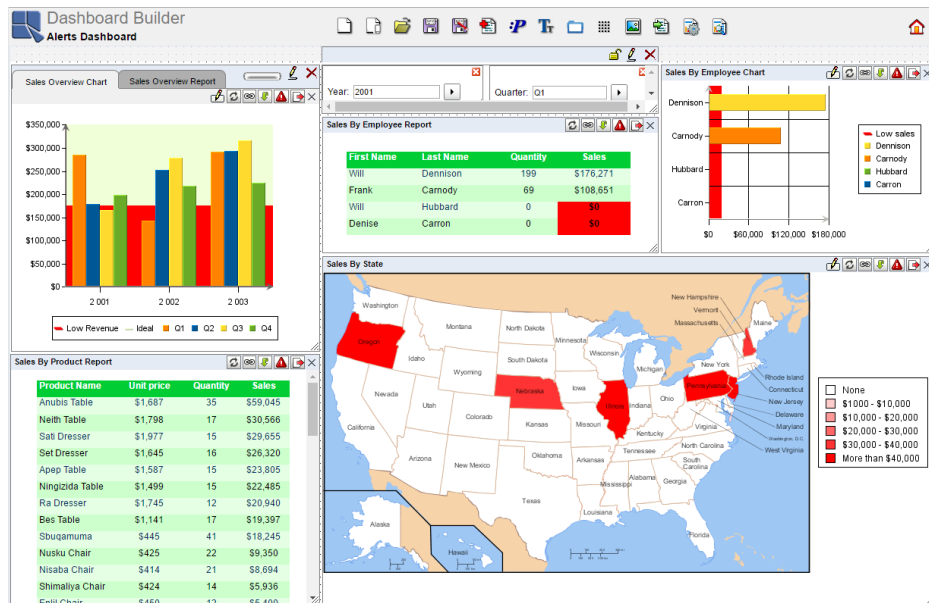
Initialize First Parameter

Map the second parameter *year* to database function *YEAR* and *Set Parameter Values* to *ORDERS.SHIPDATE*. Then click *OK* to save the query.




Initialize Second Parameter

Go to Dashboard Builder and open the *Alerts* dashboard (Published Files/Examples/Dashboard-s/Alerts Dashboard).



Alerts Dashboard

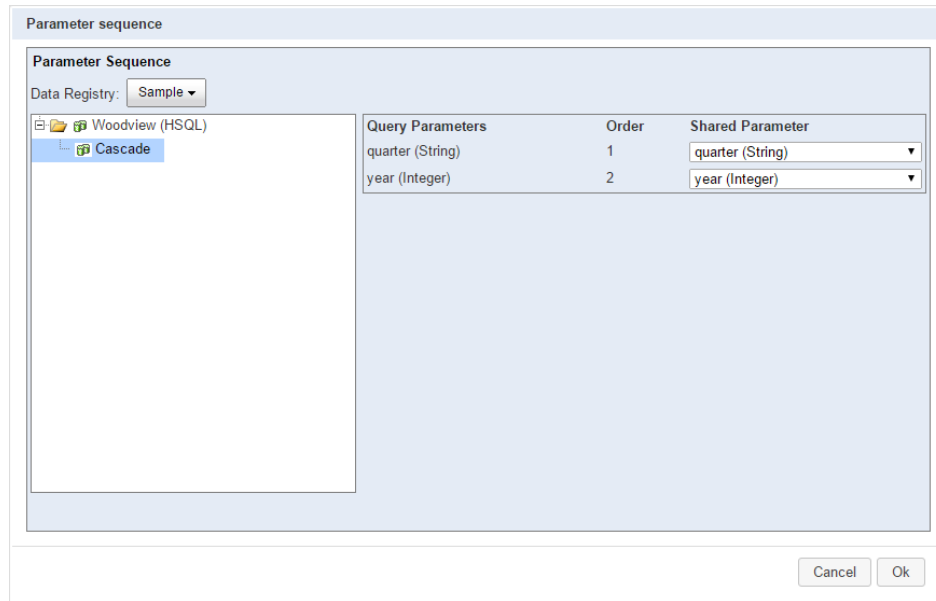
Then click the  *Share Parameters* icon on the Dashboard Builder toolbar to show the list of shared parameters in the dashboard (Notice that there are two shared parameters *quarter* and *year* already added in a dashboard. You can remove them to make room for the cascading parameters). From the dialog, select one of parameters to activate options and click the *Sequence* button at the bottom.

The screenshot shows the Share Parameters dialog box with the following components:

- Shared Parameter:** A list box containing 'year' and 'quarter'. 'year' is selected.
- Buttons:** Add Share, Edit Share, List Values, Add To Panel, Set Range, Remove Share, Sequence, and Close.

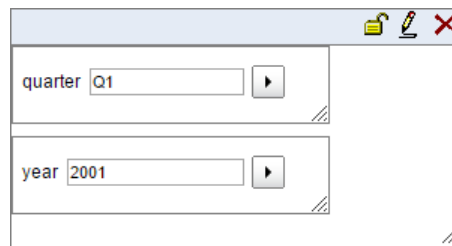
Shared Parameters Setting

The *Parameter Sequence* dialog will then appear. From the query tree on the left, select *Cascade* query you have created in previous steps. It should automatically map query parameters to shared parameters in the dashboard.



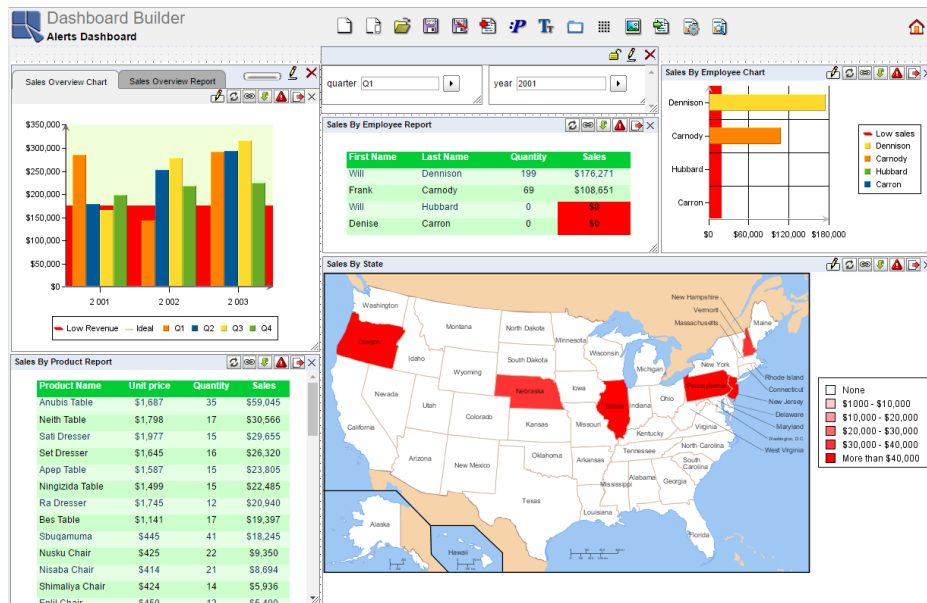
Parameter Sequence Dialog

Click the *OK* button and place the parameter panel into a dashboard.




Parameters Panel

The image below shows inserted cascade parameters panel in the dashboard. Now you can specify a parameter value for the first and second level. Once you specify the last parameter in the sequence, the dashboard templates should refresh according to the selected values.



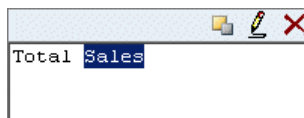
Dashboard with Cascade Parameters Panel

6.2.5. Insert Labels

You can insert a label into a dashboard by clicking the *Insert Label* button  on the dashboard builder toolbar (Please note that labels are not available for responsive dashboards). After you click the button, a small rectangle will follow your mouse pointer around the builder interface. Position the rectangle where you want to insert the label and click. The label will be inserted into the dashboard. You can edit the label text by double-clicking on the label body.



Inserting a Label



Editing a Label

Labels can be moved or resized in the same way as charts, reports and maps. To move a label, simply click on the header bar of the label window and drag it. To resize a label, click on the lower right corner of the label window and drag on the sizing handle that appears around the lower right edge of the window.

There are three additional buttons available for a label in the label header bar.



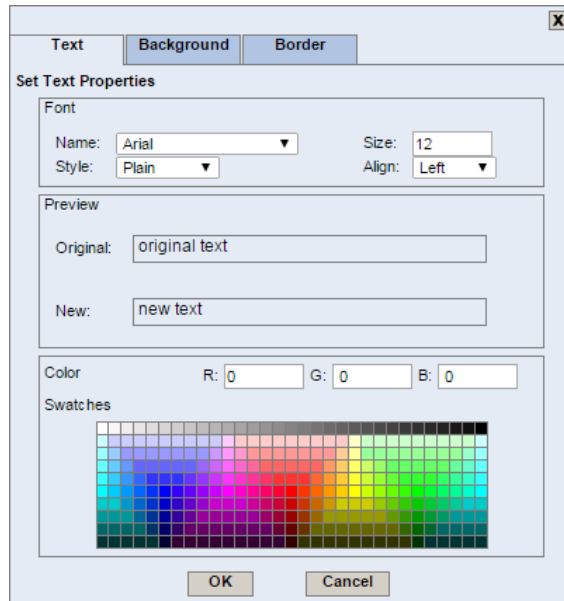
Move to Back: This button will move the label to back.



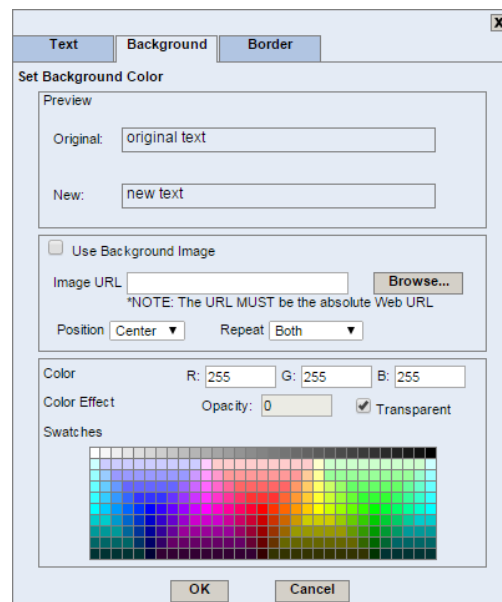
Edit: This button will open a new dialog that allows you to specify text label properties, background color and border for the label.

Text Label Properties:

The first tab of the edit window allows you to adjust text label properties. You can adjust the font type, font style, font size and alignment of the label as well as changing font color using swatches or RGB values. In the preview section you can still see the original and new text.

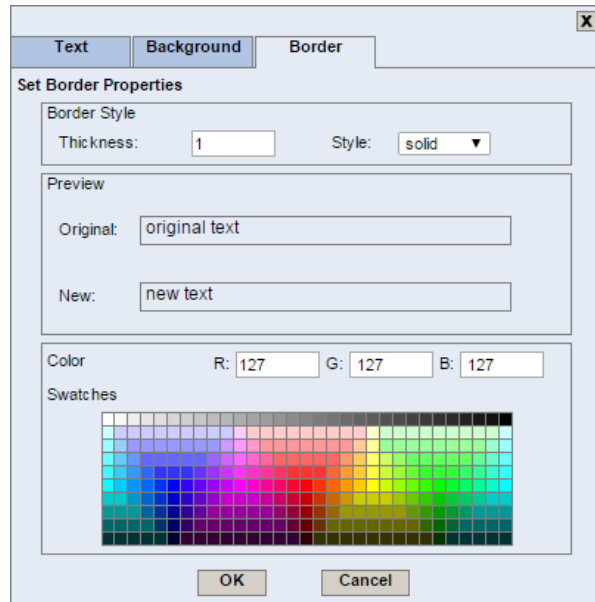
*Label Text Properties***Label Background Color:**

The second tab of the edit window allows you to set label background color or to use background image. The background color can be adjusted using swatches or RGB values. You can also specify the label opacity or choose whether or not to set the label background transparent. If you want to use a label background image, simply check the *Use Background Image* checkbox and enter the image URL or use the *Browse...* button to browse to the appropriate background image. Please note that the URL must be absolute Web URL. In addition to adding background images, you can also specify the background image position (available positions are: center, left, right, top, bottom and fixed) or choose whether to set the dashboard background image to be repeated (available options are: both, horizontal, vertical and none).


*Label Background Color*

Border Properties:

The third tab of the edit window allows you to set label border style and border color. The label border can be adjusted using swatches or RGB values. If you want to use border style, specify the border thickness and select appropriate border style (available options are: solid, dotted, dashed, double and groove). In the preview section you can still see the original and new text and border.




Label Border Properties

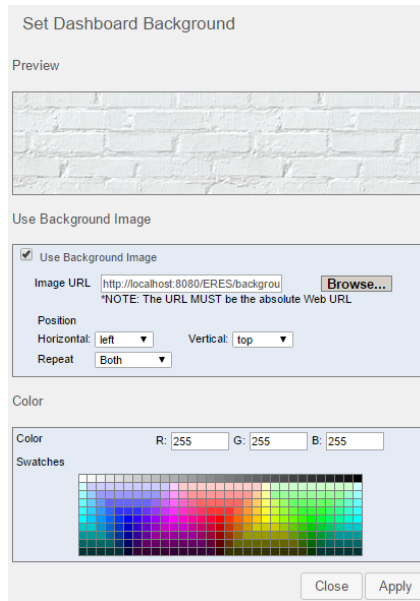
 **Delete:** This button will delete the label.

Please note that the labels can also contain any HTML tags, which may override external settings from the dialog.

For example, let's say you have a label with the font type set to Arial. However, once you enter HTML tag `Label content` into the label, Arial font will be overridden with Garamond font for the Label content text.

6.2.6. Add Background

You can also add dashboard background by clicking the *Add Dashboard Background* button  on the dashboard builder toolbar. This will open a dialog that allows you to set the dashboard background.




Dashboard Background Dialog

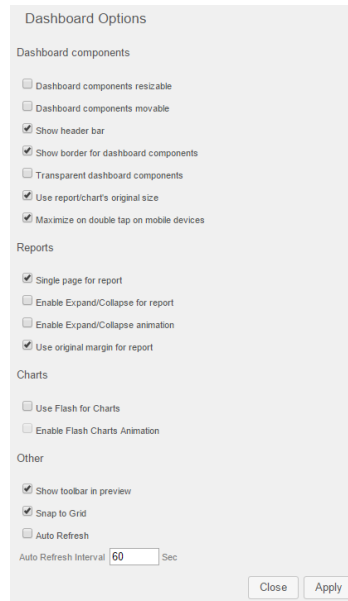
The first option in the dialog allows you to use a background image for a dashboard. If you want to use a background image for the dashboard, simply check the *Use Background Image* checkbox and enter the image URL or use the *Browse* button to browse to the appropriate background image. Please note the URL must be the absolute Web URL. In addition to adding background images, you can also specify the background image position (available positions are: center, left, right, top and bottom) or choose whether to set the dashboard background image to be repeated (available options are: both, horizontal, vertical and none). In the preview you can still see updated dashboard background.

The second option in the dialog allows you to set dashboard background color. The background color can be adjusted using swatches or RGB values.

Once you finish setting up the dashboard background options, click the *Apply* button to apply the changes. You can close this dialog by clicking the *Close* button or by clicking the *X* sign in the upper right corner.

6.2.7. Additional Options

You can also specify additional dashboard options by clicking the *Options* button  on the dashboard builder toolbar. This will open a dialog that allows you to set the dashboard components resizable/movable and other options.



Dashboard Options Dialog

Dashboard components resizable:

This option allows you to resize the dashboard components. It doesn't affect the Dashboard Builder as it works in dashboard viewer (i.e. when you open a dashboard from MenuPage etc.) and dashboard preview only.

Dashboard components movable:

This option allows you to move the dashboard components. Like the previous option, this one also affects deployed dashboards only.

Show header bar:

Shows/hides header bars for templates in the dashboard. Affects deployed dashboards only. In Dashboard Builder, headers are always visible.

Show border for dashboard components:

Shows/hides borders for templates in the dashboard. Affects deployed dashboards only. In Dashboard Builder, borders are always visible.

Transparent dashboard components:

Enable transparent background of dashboard components (e.g. transparent background of chart/report/map title, legend).

Use report/chart's original size:

This option only affects charts/reports at the moment you add them to the dashboard. If this option is enabled, charts/reports will be added to the dashboard using their original size settings (i.e. canvas size for charts and page size for reports). If this option is disabled, all reports/charts will be resized to standard size and their original size settings will be ignored. This option doesn't affect charts/reports after they have been added to the dashboard so you can resize them as you like.

Maximize on double tap on mobile devices:

Enable this option to maximize a dashboard template by double tapping on mobile devices.

Single page for report:

This option allows you to set whether reports will be displayed in single page or multi-page format

Enable Expand/Collapse for report?

Enables Expand and Collapse function for reports.

Enable Expand/Collapse animation:

Enable animation for Expand and Collapse function.

Use original margin for report:


This option will set reports to be displayed with their original margins as defined in the Page Setup in the Report Designer. If this option is disabled, report margins are set to 0.2 inches.

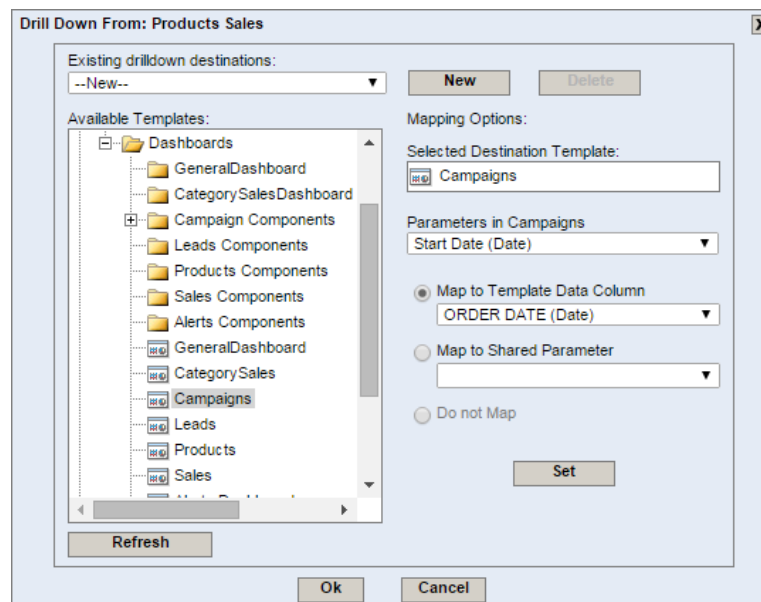
Use Flash for Charts:	Use Flash for charts instead of the default PNG format.
Enable Flash Charts Animation:	Enable/disable animation for Flash charts.
Show toolbar in preview:	Hides/displays toolbar in deployed dashboards. For more information about the preview toolbar, please see Section 7.4.1 - Preview Toolbar.
Snap to Grid:	Snaps components to grid when resizing/moving.
Auto Refresh:	Enables auto refresh for deployed dashboards (this option doesn't affect the Dashboard Builder - it affects dashboard viewer and dashboard preview only). In already deployed dashboards, auto-refresh can be also enabled/modified from the preview toolbar.

Once you finish setting up the dashboard options, click the *Apply* button to apply the settings. You can close this dialog by clicking the *Close* button or by clicking the *X* sign in the upper right corner.

6.2.8. Drilldown

You can add a drilldown to any chart/report/map in the dashboard. The drilldown feature allows you to map the chart data points, report columns, and/or shared parameters to the lower-level chart or report parameters. This creates a relationship where an end-user can either click on a data point in a dashboard chart, or row in a table and be taken to a lower-level report or chart that shows detailed information for the selected point.

To add a layer of drilldown, click the  *Add/Modify Drilldown* button in the chart/report/map header bar. A dialog will open that allows you to configure drilldown options for the chart, report or map.



Drilldown Options Dialog

The first option in the dialog, Existing drilldown destinations, is a dropdown selection box that will contain a list of the drilldowns that have been set in the template. The first element is always *--New--*. You can select a drilldown from the list and the component information will be filled automatically. If you choose *--New--*, all dialogs will be cleared and new entries into the dialogs can be entered. The items in the list are names of the drilldown targets with an icon that can help you to identify the type of the file (see Section 2.1.4.3 - File Identification Icons). The *Delete* button will only be enabled when you select an existing drilldown from the *Existing drilldown destinations*. When you hit this button, the current selected drilldown will be removed from the list and the whole dialog will be reset.

The field on the left side of the dialog contains a tree that mirrors the Organizer folder structure. All of the reports/charts/maps and dashboards to which you have access to will be listed. Please note that drilldowns to dashboards will be discussed in the next section.

To select a chart, report or map as the drilldown destination (lower-level), click on it in the list. The dialog will update to show the name of the currently selected drilldown report/chart/map. The first drop-down list below the name will be populated with all of the parameters in the selected report/chart/map.

Each parameter in the destination report/chart/map can be unmapped (i.e. it does not pass the parameter value and shows the drillable link no matter where you click on the report/chart) or mapped to either a column (data element) in the top-level chart/report/map, or one of the shared parameters defined in the dashboard. To set an unmapped link, simply select the *Do not Map To Column*. Note that this option is only available if the next drillable link is a dashboard. To set the parameter, first select the destination parameter in the drop-down list. Then select the radio button to indicate whether it should be mapped to the top-level chart/report/map or a shared parameter. Then select the element/field or shared parameter from the appropriate drop-down list.

Note that a column selected for the drilldown in the *Map to Template Data Column* can only be used once, i.e. you can have only one drilldown per column. A shared parameter in the *Map to Shared Parameter* can be used multiple times in the dashboard. This is because a shared parameter isn't tied to a specific template in the dashboard and it behaves like a parameter value provider for the drilldown destination template.

If all destination template parameters are mapped to shared parameters and if the parent template is a report, the first column of the report is used as the anchor for the drilldown link, although it won't pass the element value to the destination template. If the parent template is a chart or a SVG map, the data points become the anchor for the drilldown links.

You can click the *Set* button to save the drilldown settings that is currently being worked on. If the drilldown is new, a new entry in the *Existing Drilldown destinations* will be added, else the drilldown settings are updated in the browser memory. To make the saved settings permanent, click the *OK* button.

The *Refresh* button reloads the folder tree with latest organizer folder list. Note that all folders will be shown in a collapsed view.

Once you finish selecting the options, click the *Ok* button to save the changes. You will be returned back to the Dashboard Builder interface.

6.2.8.1. Drilldown to Dashboard

In addition to drilldown to any chart/report/map in the dashboard, you can also map the chart data points, report columns, and/or shared parameters to another dashboard. The dashboard builder supports both parameterized and non-parameterized dashboard as the destination of the drilldown. If the target dashboard has no parameter or you don't setup any parameter mapping, the template from which the drilldown link starts will become clickable and the result of the click will popup the target dashboard in a new window without any default parameter setup.


In case you want to set a parameterized dashboard to be a drilldown destination, you will need to map shared parameter(s) from the target (child) dashboard to chart data points/report columns or shared parameters in the mother dashboard. It will behave like setting up a drilldown link for a report, chart or map, as mentioned before. The corresponding field(s) will then be changed to a hyperlink and when you click the link, the target dashboard will pop up and the shared parameter of the target dashboard will be updated.

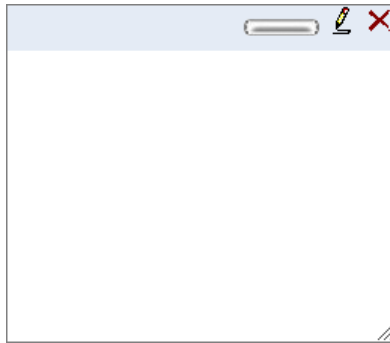
You can set up a drilldown link without any parameter mapping to another dashboard. If the template has both unmapped and mapped drilldowns and is a report or chart, the drilldown that appears depends on which point on the report/chart was clicked. If it is a datapoint that is used as a drilldown link for a mapped drill-down, a mapped drillable link will appear. If not, an unmapped drillable link will appear.

The other drilldown functionalities are basically the same as mentioned in the previous section. For more details about setting drilldowns and the drilldown options dialog, please navigate to the previous Section 6.2.8 - Drilldown.

6.2.9. Folders

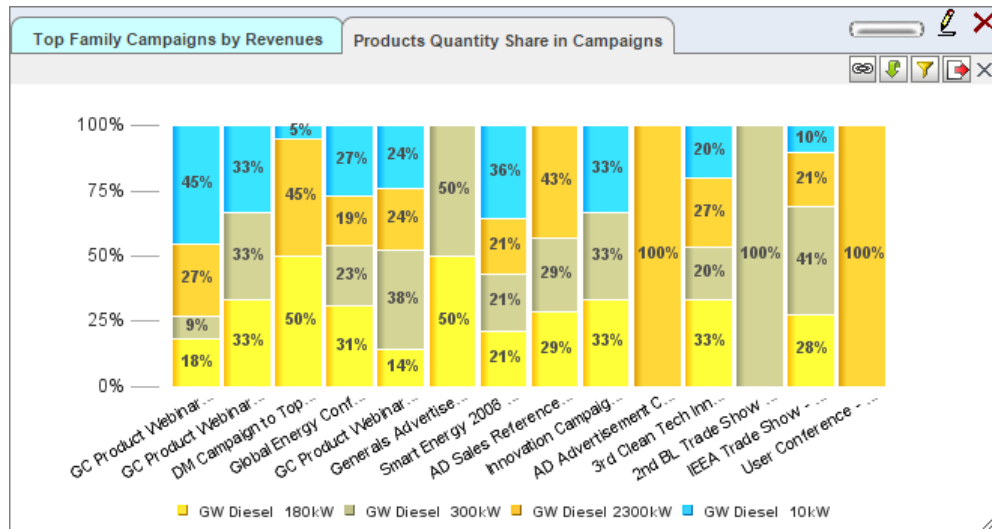
Folders allow you to place tabbed objects into your dashboard. Folders can contain charts, reports or maps, and use a tabbed interface that allows you to switch between current objects to display. You can insert a folder into

a dashboard by clicking the *Insert Folder* button  on the dashboard builder toolbar. After you do so, a small rectangle will follow your mouse pointer around the builder interface. Position the rectangle where you want to insert the folder and click. The folder will be inserted into the dashboard. To move a folder, click and drag your mouse over the right side of the folder's header bar.



New Folder

After adding a folder to the dashboard, you can drag and drop reports, charts and maps into it. After you add an object to a folder, a new tab will be visible inside the folder, showing the names of the objects within it. To switch between objects, simply click on the tab of the object you wish to view. Objects can be removed from the folder by clicking and dragging on the object's title bar.



Folder with objects and style formatting

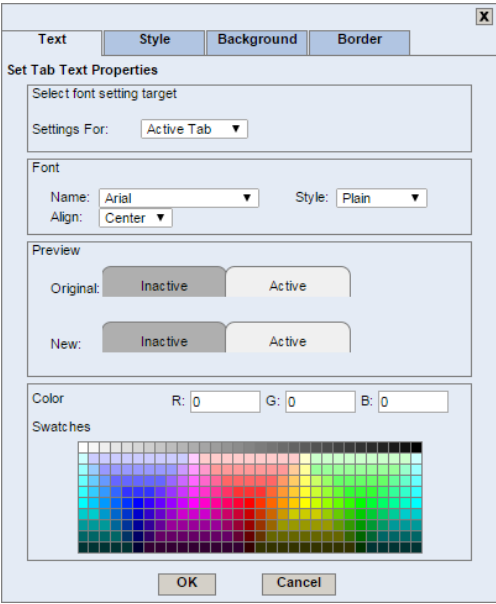
There are two additional buttons available for a folder in the folder header bar:



Edit: This button will open a new dialog that allows you to specify text label properties, style, background color and border for the folder tabs.

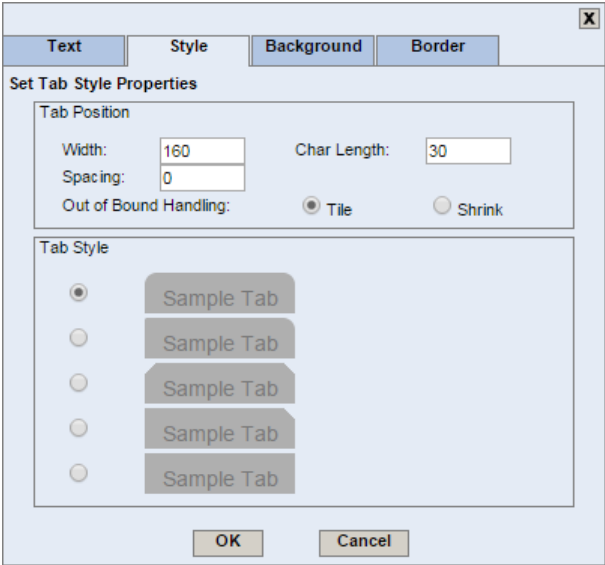
Text Properties:

The first tab of the edit window allows you to adjust text label properties. You can adjust the font type, font style, font size and alignment of the label as well as changing font color using swatches or RGB values.



Style Properties:

The second tab of the edit window allows you to set style properties. You can adjust tab shape and position.



Width: This option allows you to set a width of tabs in pixels.

Char Length: This option allows you to set character length. Shortened file names will end with three dots (included in the length).

Spacing: This option allows you to set a space between tabs in pixels.

Out of Bound Handling: **Tile -** The width of the tabs will be kept even if the

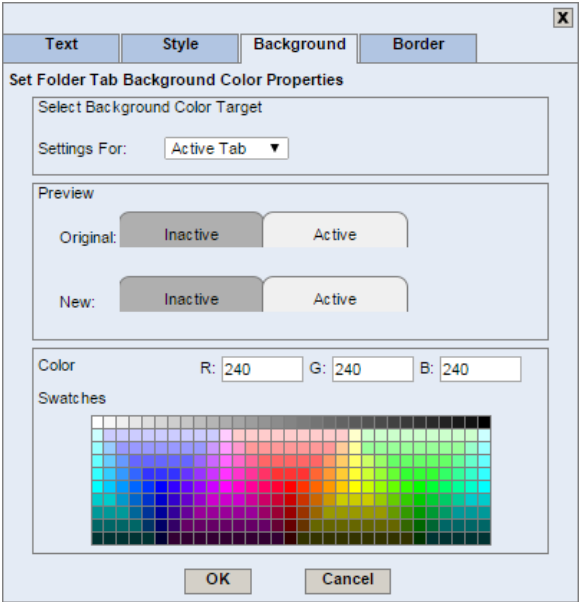
whole tabs are not visible.



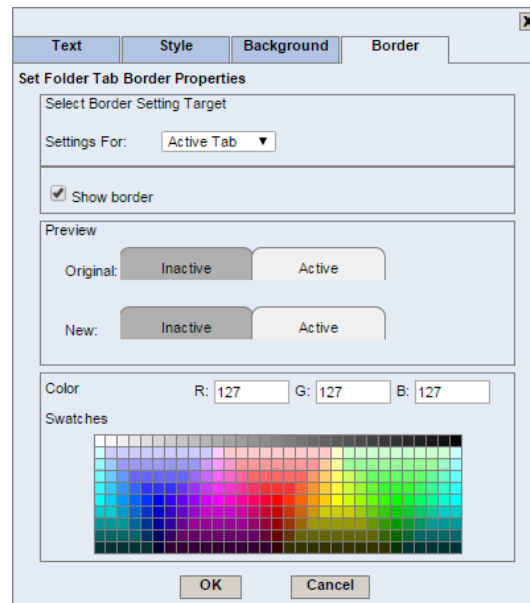
Shrink - The width of the tabs will adjust to make the whole tabs visible.



Background Properties: The third tab of the edit window allows you to set the background color for either active or inactive tabs. The background color can be adjusted using swatches or RGB values.




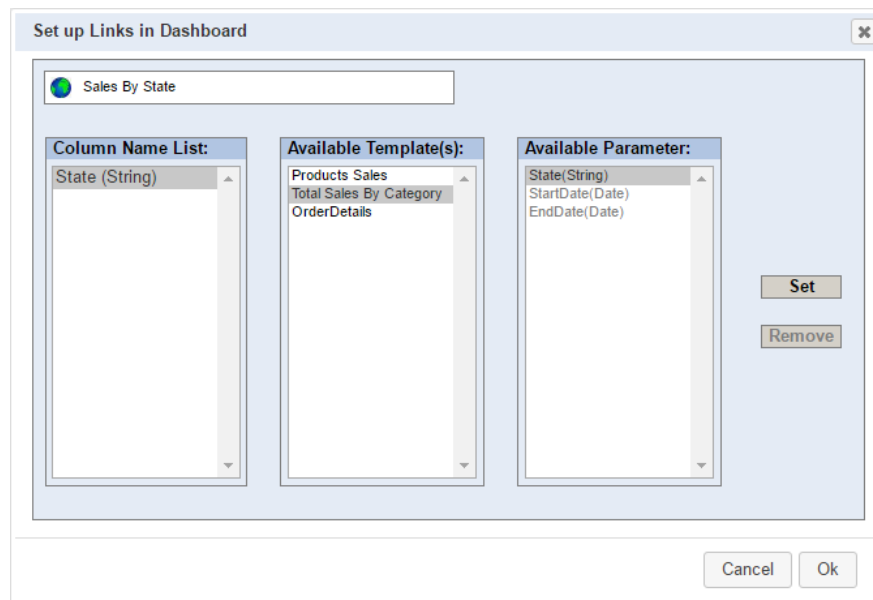
Border Properties: The fourth tab of the edit window allows you to set the tab border color. The tab border can be adjusted using swatches or RGB values.



 **Delete:** This button will delete the folder.

6.2.10. Template Linkage

Template Linkage allows you to use data from one chart, report or map as a parameter in another. To use this feature, click the *Add/Modify link* button  on the source chart, report or map to bring up the *Set up Links in Dashboard* screen.



On this screen, there are three columns:

Column Name List -


This is a list of available columns and their data type in the source template. Click on a value from this column to use it as a parameter in another template.

Available Template(s) -	This lists the other templates that are open in the current dashboard. Click on a template name to select it. While a template is selected, it will be highlighted in the dashboard.
Available Parameter -	After selecting a column name and template, this list of available parameters will appear. Select a parameter you want to map to the selected column name in the first column.

When you are done selecting values in each column, click the *Set* button. You can then select new fields if you wish to add additional links. Once a link has been established, clicking on a value in the specified source column (or chart object) will cause the target templates to be refreshed using data from that column.

To remove a link, select it and then click the *Remove* button.

6.2.11. Dashboard Preview

At any point in the design process, the dashboard can be previewed by clicking the *Preview* button  on the dashboard builder toolbar. This will open a new window showing the dashboard. Dashboards that have been deployed to the Organizer can also be viewed in the Menu Page.

For more information about dashboard preview options, please see Section 7.4 - Dashboard Viewer.

6.3. Dashboard Migration

Often you may need to move dashboards from one location to another. For example dashboards may move between the server and the develop machine. On the develop machine the reports/charts/maps locations can change and may not be located from the data stored in the dashboard .dsb file. Therefore, EDAB uses the DPAK files to move or archive dashboards along with it's components. To learn how to create a DPAK file, see the Section 2.1.4.4 - Dashboard and Map Packages.



Note

The DPAK files can be viewed in Published Files, but they can't be open in the Dashboard Builder. To edit a dashboard from a DPAK file, you have to unpack it first (to learn how to unpack a DPAK file, see the Section 2.1.4.4 - Dashboard and Map Packages).

6.3.1. Migrating from previous EDAB versions

DPAK files can't be created in EDAB version 6.3 and older. If you want to move a dashboard from an old EDAB version (6.3 or older) to a new EDAB version (6.6 or newer), you will have to use the “Dashboard Migration” feature.

To use this feature, you will need to copy the following files from source EDAB server to the corresponding files on the destined EDAB server :


- dashboard file from <InstallDirectory>/DashboardFiles
- report files from <InstallDirectory>/ReportFiles (can be a stand-alone report or behave as a drilldown target)
- chart files from <InstallDirectory>/ChartFiles (can be a stand-alone chart or behave as a drilldown target)
- drilldown reports/charts from <InstallDirectory>/DrillDown (for drilldown templates that were created in the drilldown wizard)
- charts embedded in reports from <InstallDirectory>/chart
- map files from <InstallDirectory>/MapFiles with all the files used by this map (i.e. Coordinates file, Tooltip template and DrillDown template for Google Maps or SVG Image and DrillDown template for SVG Maps). If you are not sure about what files are used by your map, you can view the map file and search for filenames. Map files are XML files, so they are human-readable. Please note that all the files used by your map should be copied in the same subdirectories of the EDAB installation directory on the destined EDAB server. Otherwise, you will have change the file links in the map file manually.
- necessary images which are used in dashboard, but not available from the destined machine

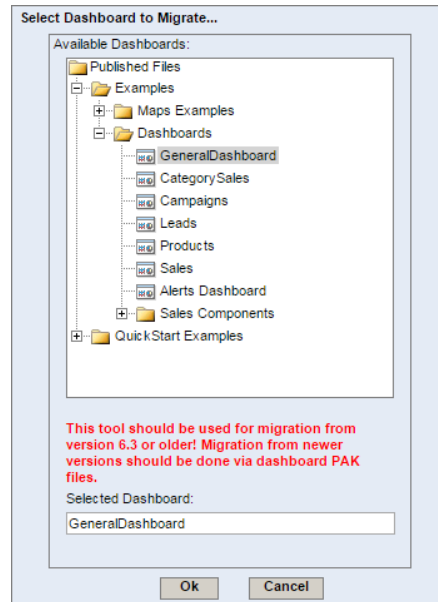


Note

The directories that are mentioned in the above list are the default directories that are used for saving certain file types. If you saved a template to a different directory or if you inserted some files into Organizer from a custom directory, you will have to search for the files in their original locations.

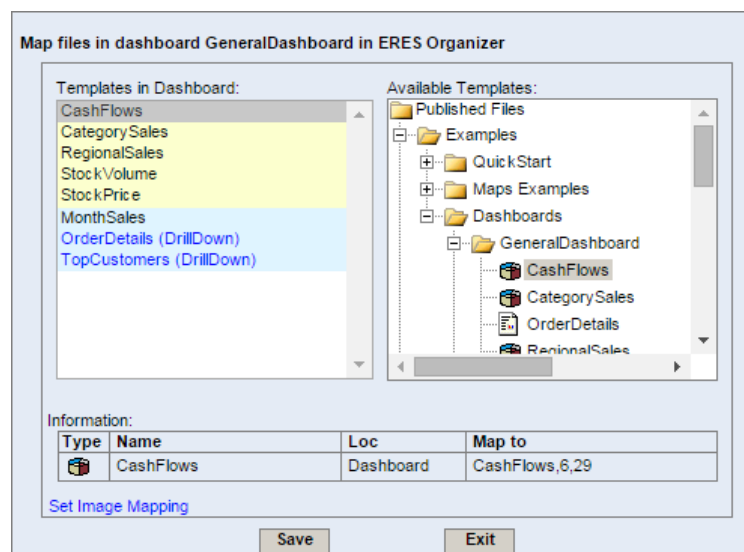
Next, you will need to add previously mentioned files to the destined Organizer and then launch the dashboard builder.

To migrate a dashboard from the Dashboard Builder, click the *Migration* button  on the Dashboard Builder toolbar. After you have clicked the button, the following dialog will then appear.



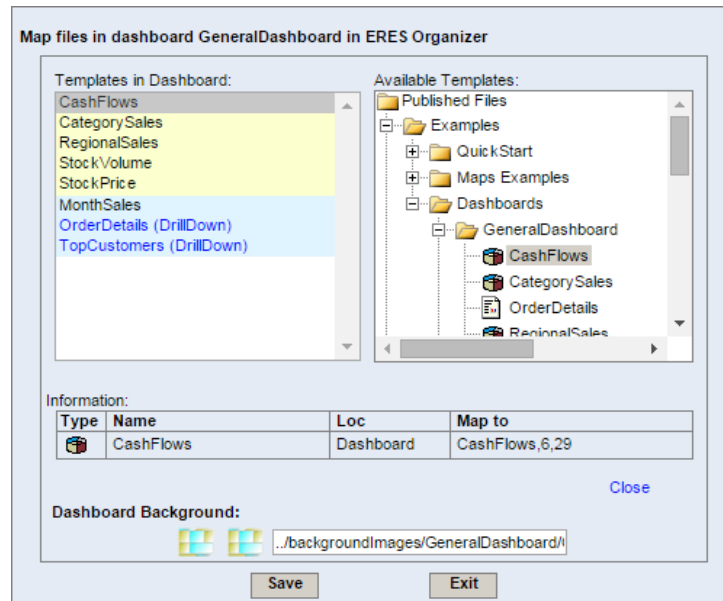
Dashboard migration dialog

The dialog contains a tree that mirrors the folder structure in the Organizer. All of the dashboards to which you have access are listed. In order to select a dashboard to migrate, simply select it in the tree and click *Ok*. The migration mapping dialog will then appear.



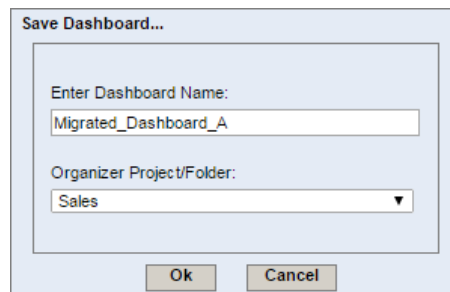
Dashboard migration dialog

From this dialog you can map the original reports/charts/maps to the reports/charts/maps in the destined Organizer. Please note when your dashboard uses images (dashboard background, shared param panels background, shared param value list button icons, etc.), which are not available from the destined machine, you need to set image mapping as well. To set the image mapping click *Set Image Mapping* link at bottom of the dialog. The dialog will then extend to allow you specifying/changing the image mapping.



Dashboard migration dialog


All the images that were used in the original dashboard will be listed with their URLs. You can see the preview of the original (the first image) and current mapped image (the second image). If an image is not correctly mapped, the second preview image will be blank. Once you have finished specifying the mapping for the dashboard files/images click *Save*. It will open the following dialog prompting you to enter the name for the migrated dashboard.

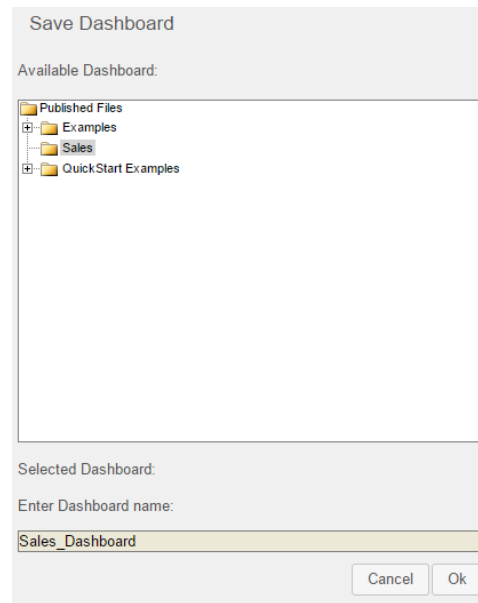


Dashboard migration dialog

Once you have finished specifying the options click *Ok* and the dashboard will then be saved.

6.4. Save Dashboard

Once you have finished setting up all the dashboard definitions, you can save the dashboard by clicking the *Save* button  on the toolbar. A new dialog will open prompting you to specify the name for the dashboard.

*Save Dashboard Dialog*

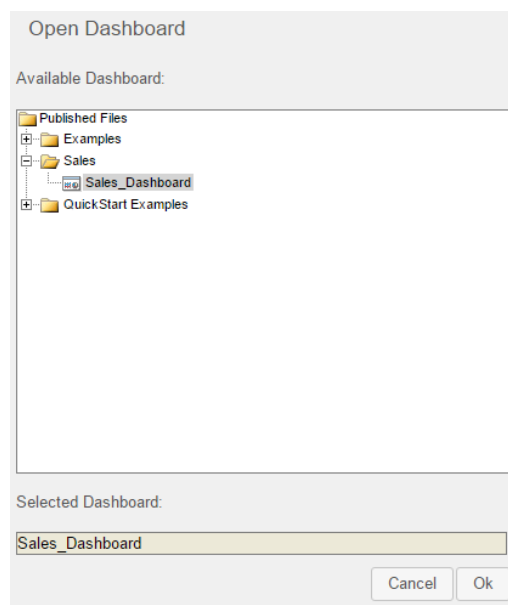
The first option allows you to specify a name for the dashboard. The dashboard will be saved with a `.dsb` extension in the `/DashboardFiles/` directory of your EDAB installation.

The second option allows you to insert the dashboard into the Organizer. The drop-down list contains all of the projects and folders in Organizer to which you have access. Select the project or folder in Organizer where you'd like to place the dashboard.

Once you have finished specifying options, click the *Ok* button to save the file.



6.5. Open Saved Dashboard

You can open the saved dashboard by clicking the  *Open* icon on the main toolbar. *Open Dashboard* dialog will appear in the left pane.

*Open the Dashboard*

You can see all dashboards from Organizer created in Dashboard Builder. Select a dashboard and click *Ok* to open it.

6.6. Exit

You can exit Dashboard Builder by clicking the  *Home* icon in the upper right corner, by clicking the *Dashboard Builder* title, or by clicking the  *Logo* icon in the upper left corner. Before closing, you will be asked if you want to save an unsaved dashboard.

Chapter 7. Publishing (Menu & URLs)

7.1. The Menu Page

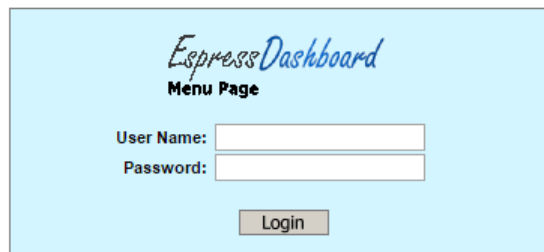
In EDAB, all reports, charts, maps and dashboards created or added to the Organizer are automatically published. By logging into the menu page, users can access and run any files to which they have privileges in the Organizer. The menu page is included with EDAB and deploys without any code.

7.1.1. Launching the Menu Page

The menu page can be launched from the EDAB start page. Once you've successfully logged in, click the *Published Files* link.

The menu page can be also accessed directly by going to the menu page login: `Menu_Login.jsp`. If you installed EDAB with Tomcat, the URL to the login is `http://machinename:port/EDAB/Menu_Login.jsp`. This will bring up a page allowing you to login to the menu page directly.

 Home

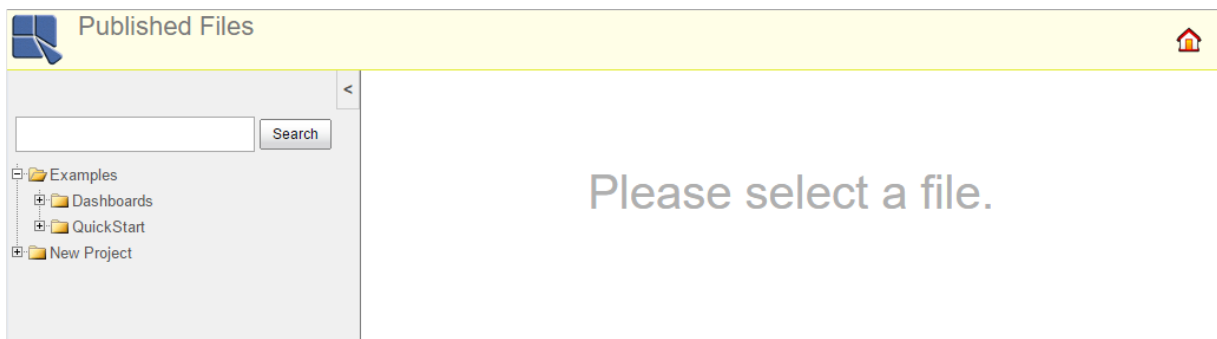


The image shows a login form for the 'Espress Dashboard Menu Page'. It has a light blue background. At the top, it says 'Espress Dashboard' in a stylized font, followed by 'Menu Page' in a smaller font. Below this, there are two input fields: 'User Name:' and 'Password:'. A 'Login' button is positioned below the password field.

Direct Access to Menu Page

7.1.2. Using the Menu Page


When the menu page first loads, it will show a collapsed tree-list of all the reports, charts, maps, dashboards, and other files in the Organizer which you have privileges to see in a tabular format.





Menu Page

7.1.2.1. Viewing Reports and Charts

To open a file, expand respective project/folder nodes (in the tree-list on left side) to locate the file and click on the filename. The file will load in the right *DHTML Viewer* panel.

You can also search for specific files. To do so, enter search terms in the upper left dialog and click the *Search* button. The tree-list will reload, showing only relevant files. To cancel the search filter, click the  *Close Search* icon next to the search field. Note that the *Close Search* icon will not be visible if there is no filter applied to the list at the moment.

You can hide the left panel by clicking on the  *Collapse* icon located in the top-right corner of the navigation panel.

To close the MenuPage and to return to the EDAB start page, click the  button in the top-right corner of the MenuPage.

7.1.2.1.1. Viewing Reports with Encrypted Data

If your report contains encrypted data, you need to do two things in order to view the data.

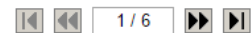
1. You need to create an XML file that gives the database URL, database driver, name of column to be decrypted, and the function to be applied when the data is being retrieved.
2. You need to include the *ReplaceColumnInfoList* option in the command line, i.e. `servercommand.txt` before you start EDAB server.

For more details and an example for viewing encrypted data, please see Section 3.2.2.1.2 - Querying Encrypted Data.

7.1.2.2. Using the DHTML Report Viewer

If you click on a report in the menu page tree-list, the report will open in the DHTML report viewer. This interface is a thin-client component that allows you to view and interact with the report. In the DHTML report viewer, the first page will show as soon as it is ready, while the rest of the report continues to be processed on the server side. This means that for larger reports, navigating to a page that is not yet exported will result in delay. Before a report is finished processing, the toolbar options will not be available and clicking on any of them will result in a warning message. The main window of the interface provides HTML view of the report with a small toolbar in the top-left corner of the page. Each option on the toolbar will bring up a dialog allowing you to make changes to the report.

The main window also contains a page navigator that contains first page, previous page, next page, last page buttons and a textbox that allows you to jump to any specific page number. The textbox displays the current page number in the form <Current Page Number> / <Total Page Number>. To jump to a specific page, simply type the page number into the textbox and press **Enter**. You can also enter the value in the same fractional format that is used in the display. For example, to jump to page 7 of a 22 page report, you can enter 7 or 7 / 22.



Company	Invoice	Order Date	Product	Quantity	Price	Total
Alfano Furniture	10059	12.3.2003	Nisaba Chair	17	\$414.00	\$7,038.00
		12.3.2003	Geb Table	18	\$1,215.00	\$21,870.00
	10075	9.7.2003	Anahita Dresser	1	\$1,987.00	\$1,987.00
		9.7.2003	Adad Chair	7	\$452.00	\$3,164.00
		9.7.2003	Tiamet Dresser	4	\$1,861.00	\$7,444.00
	10092	20.11.2003	Renpet Dresser	2	\$1,861.00	\$3,722.00
		20.11.2003	An Chair	18	\$425.00	\$7,650.00
		20.11.2003	Nusku Chair	6	\$425.00	\$2,550.00
All World Furniture	10054	16.1.2003	Zabada Chair	7	\$312.00	\$2,184.00
		16.1.2003	Ningizida Table	4	\$1,499.00	\$5,996.00

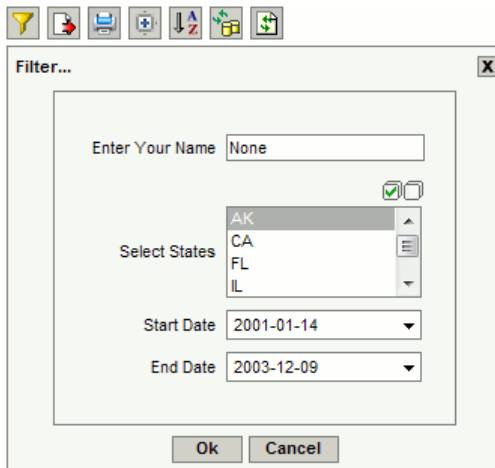
DHTML Viewer Interface

7.1.2.2.1. The DHTML Report Viewer Toolbar

The options in the DHTML report viewer toolbar are as follows:



Filter Report: This option is only available if the report is parameterized. It will bring up a dialog containing all parameters defined in the report.


 A dialog box titled "Filter..." with a toolbar at the top containing icons for filter, save, print, undo, redo, and zoom. The main area contains:

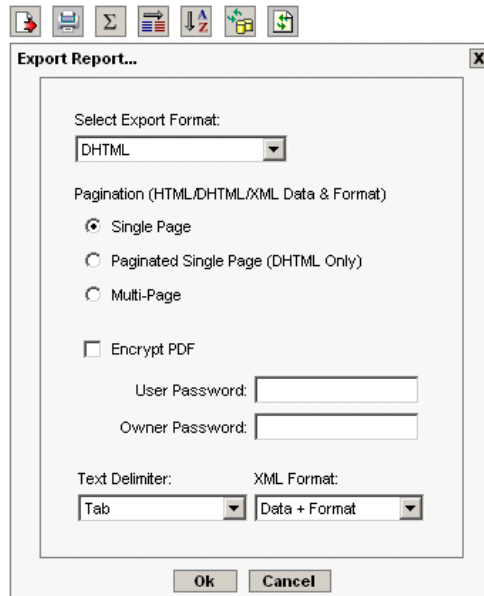
- "Enter Your Name" with a text field containing "None".
- "Select States" with a list box containing "AK", "CA", "FL", and "IL".
- "Start Date" with a date field containing "2001-01-14".
- "End Date" with a date field containing "2003-12-09".
- "Ok" and "Cancel" buttons at the bottom.

Filter Dialog

Select new values and click the *Ok* button to update the report. There is a validation system for unmapped parameters that will check entered values against expected datatype and alert you if it does not match.



Export Report: This option allow you to export the report in a variety of file formats. It will bring up a dialog prompting you to select export options.


 A dialog box titled "Export Report..." with a toolbar at the top containing icons for save, print, sum, list, filter, undo, redo, and zoom. The main area contains:

- "Select Export Format:" with a dropdown menu showing "DHTML".
- "Pagination (HTML/DHTML/XML Data & Format)" with three radio buttons: "Single Page" (selected), "Paginated Single Page (DHTML Only)", and "Multi-Page".
- "Encrypt PDF" checkbox (unchecked).
- "User Password:" and "Owner Password:" text fields.
- "Text Delimiter:" dropdown menu showing "Tab".
- "XML Format:" dropdown menu showing "Data + Format".
- "Ok" and "Cancel" buttons at the bottom.

Export Dialog

The first option in the dialog allows you to specify in which format you want to export the report. Available options are HTML, PDF, CSV, Excel (XLS), Excel 2007 (XLSX), DHTML, text, XML and rich text.

Other options allow you to set single or multi-page export for HTML, DHTML, and XML exports. PDF format allows you to enable encryption. For text export, you can select delimiter to be used, and for XML export, you can select whether to export only report data or XML description of the report and data.

Once you finish specifying options for the exported report, click the *Ok* button. A new window will open containing the exported report. The toolbar will be active on this window, allowing you to save the generated file to your local system or print the report locally.



Print Current Report: This option allows you to print current page of the report on user's local printer, as displayed inside the right-side panel.

1. By default, all browsers don't print background colors and images to save ink. You have to change print settings to allow that.

How to set it in different browsers:

Chrome: Check the *Background colors and images* option. It is in the *Print* dialog that opens automatically.

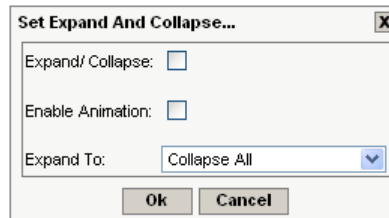
IE 11: Click the gear icon in the upper right corner → *Print* → *Page Setup* → check *Print Background Colors and Images*

Firefox: *Print* → *Page Setup* → check *Print Background (colors & images)*

2. Reports wider than page might be cut off. If you need to print wider reports, please export them to PDF and print the PDF.



Expand and Collapse View: This option allows you to add expand and collapse controls to any grouped report.



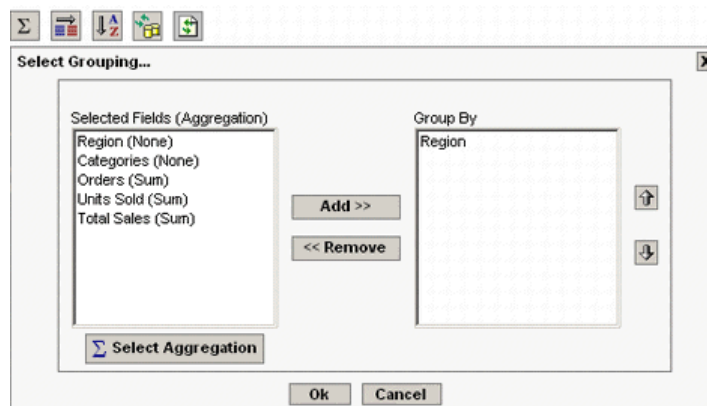
Expand and Collapse Dialog

The first option in the dialog allows you to turn the control for expanding and collapsing on. Enable Animation allows the grouping to fade in and out as it is expanded and collapsed. Expand To specifies the initial presentation of the report, i.e. collapsed completely, expanded completely or expanded to a specific group level.

Once you finish specifying the options, click the *Ok* button. The controls will now be added to the report.



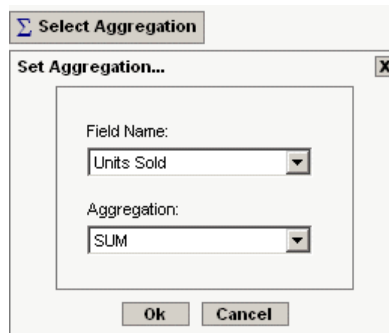
Change Grouping/Summaries: This option is only available if the report was created using QuickDesigner. It opens a dialog that allows you to edit the report grouping and aggregation.



Grouping Dialog

To add or remove grouping, select a column and click the *Add* or *Remove* button. If you select more than one level of grouping, you can change the nesting order using the arrow buttons on the right side of the dialog.

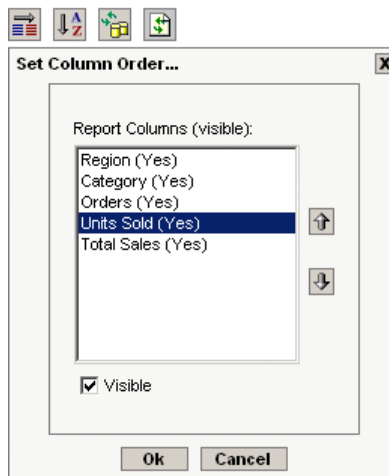
To set column aggregation, click the *Select Aggregation* button below the column list on the left side. This will open a new dialog allowing you to change the aggregation for the columns in the report.

*Aggregation Dialog*

The top drop-down list contains all selected columns for the report and the lower list contains all available aggregation options. To set aggregation for a column, select it in upper list and set the desired aggregation in lower list. Once you finish setting up the aggregation, click the *Ok* button and the dialog will close. The new aggregations will be reflected in the column list.



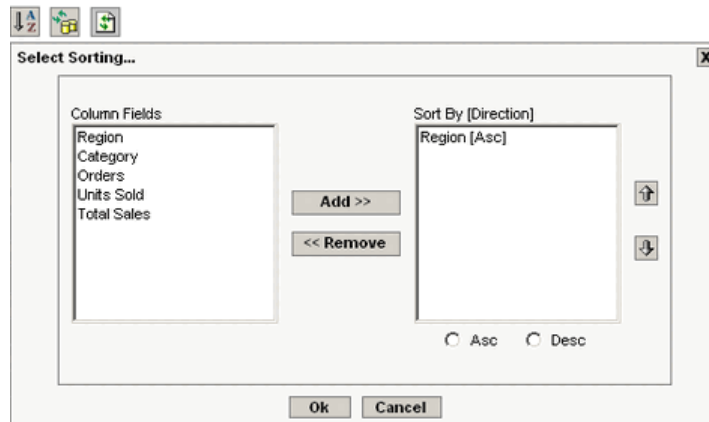
Change Column Order/Visibility: This option is only available if the report was created using Quick-Designer. It opens a dialog that allows you to change the column order in the report as well as show/hide columns.

*Column Order Dialog*

The dialog contains a list of all columns in the report. To change the column order, select a column in the list and click on the *up* or *down arrow* buttons to change the order. The order from top to bottom in the dialog represents the column order from left to right in the report. To show or hide a report column, check or uncheck *Visible* option in the bottom of the dialog.



Change Sorting Options: This option allows you to change the sorting in the report. It will bring up a dialog prompting you to set sorting options.



Sorting Dialog

The left side of the dialog contains all columns in the report. To sort it by a column, select it in the left side and click the *Add* button. You can control the sort priority by selecting a sort-by column on the right side and clicking on the *up* or *down arrow* buttons to change its position. You can control the sort direction by selecting a sort-by column on the right side and clicking on the *Asc* or *Desc* radio buttons to indicate the direction.



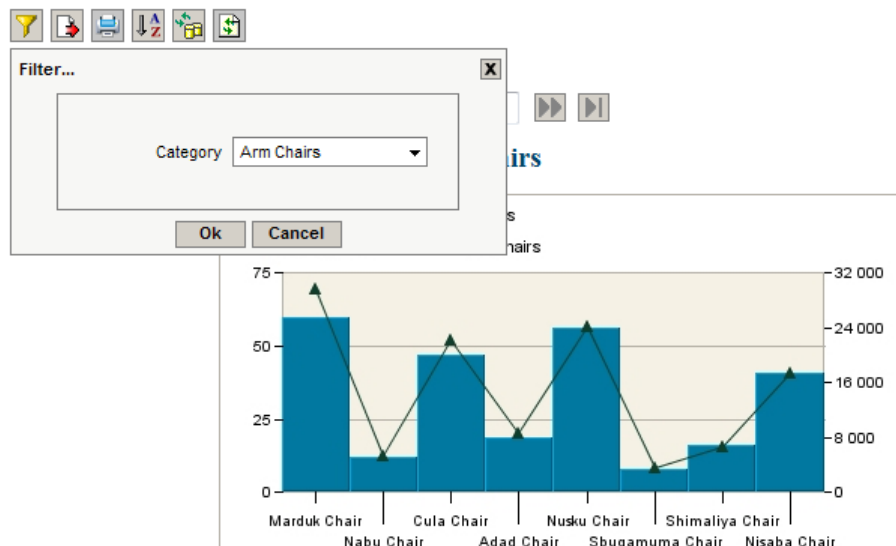
Refresh Data: The DHTML Viewer caches the report and therefore when you close and reopen the same report, the data is not updated. This option will refresh the data in the report.



Reload Report: When you run a report in the DHTML Viewer, make some changes to this report and then try to open this report again in DHTML Viewer, you'll notice that the report does not reflect the new changes. This is because the DHTML Viewer caches the report to improve performance. To see the changes, use the *Reload Report* button to have the DHTML Viewer read the report structure again and also refresh the data.

7.1.2.2.2. Parameterized Reports and Reports with Unmapped Drilldown Parameters

When viewing a parameterized report in the DHTML Viewer, the parameters are initially set to their default values. You can then change the parameter value using the *Filter* icon described above. The DHTML Viewer can also be used to view drill-down reports with unmapped parameters. Similar to a regular parameterized report, the drilldown report will display using default values, but you can adjust them by using the *Filter* icon.



Drilldown with Unmapped Parameter

7.1.2.3. Using the DHTML Chart Viewer

If you click on a chart in the menu page tree-list, the chart will open in the DHTML chart viewer. This interface is a thin-client component that allows you to view and interact with the chart. The DHTML chart viewer displays the chart using default parameter value (if the chart is parameterized). The parameter value can be changed using the toolbar options. The main window of the interface provides you with HTML view of the chart with a small toolbar in the top-left corner of the page. Each option on the toolbar will bring up a dialog allowing you to modify the chart's look and feel, and navigate through the chart.

7.1.2.3.1. The DHTML Chart Viewer Toolbar

The options in the DHTML chart viewer toolbar are as follows:



Filter Parameter : This option is only available when the chart is parameterized. Clicking on the icon will bring up a dialog that allows you to specify parameter values for the chart. You can choose/select the values and then click the *OK* button to update the chart. For unmapped parameter, the value is validated against the expected data type and alerts you in case of a mismatch.



Export Chart : This option allows you to export the chart to a variety of formats such as PDF, PNG, GIF, JPEG and SVG. The exported chart is then displayed in a new window from where it can be saved to your local system or printed.



3D Display Option : This option is only available when the chart is three dimensional. Three-dimensional charts have several additional options. These options can be accessed by clicking the *3D Display Options* button on the toolbar. This will open a dialog in a new window. Available options include:

- Arrow buttons at the top of the dialog allows you to adjust the viewing angle of the chart. The 3D chart can be rotated horizontally and vertically.
- The first three sliders allows you to control the chart scaling of the chart in the X, Y, and Z directions. The scaling features control the proportion of the chart in each direction. The final slider controls the thickness of the data points.
- The remaining options allow you to toggle wire frame mode for the chart, show/hide the border around the chart, and draw the series inline (instead of on the Z axis) for charts with data series. The last option also enables 3D approximation for the chart. 3D approximation is useful when dealing with charts with a large number of data points. The approximation can improve 3D rendering performance when a large amount of data points exists on the chart. This feature is automatically enabled when the chart contains more than 100 data points.

Once you finish setting the options, click the *Ok* button to save the changes.



Zoom Option : The Zoom option dialog is only available for a chart with zoom feature enabled. You can modify the zoom properties such as lower bound, upper bound and zoom scale, as well as change the period layout to linear. Then click the *Ok* button to apply the new zoom changes.



Refresh Data : You can use the refresh data option to fetch the latest data for the chart.



Reload Chart : Changes in the chart are not always reflected in the DHTML chart viewer. This is because the DHTML chart viewer caches the chart to improve performance. You can click on the *Reload* icon to display the changes and refresh the data.



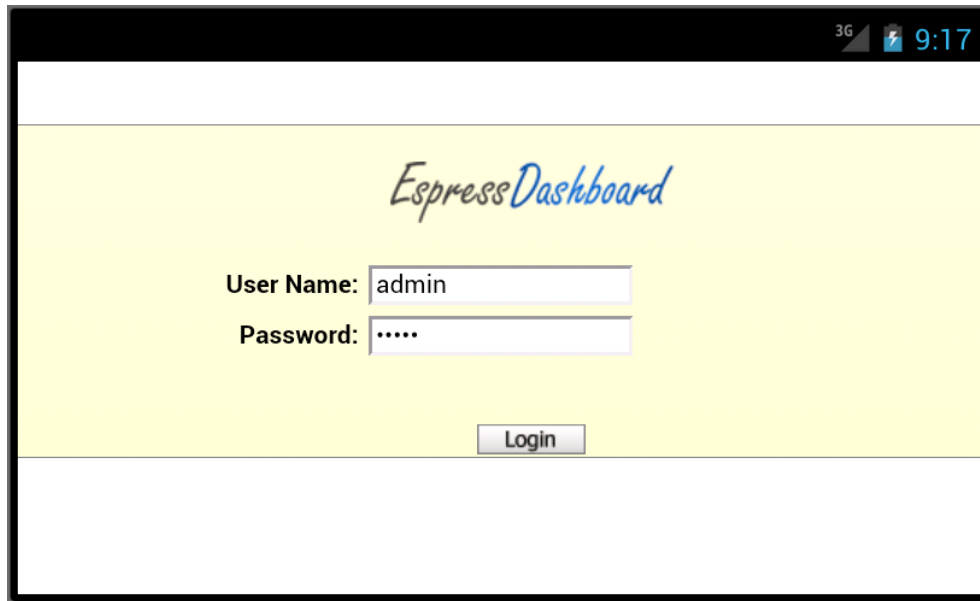
Back Button : The *Back* button appears if you are in a sub-level chart. Clicking the *Back* button will take you back to the previous level.

The DHTML chart viewer also supports drill-down. You can drill-down to the next level chart (if the chart is a drill-down chart) by clicking on the desired data point.

The DHTML chart viewer also pops up a hintbox that displays the data information/hyperlink information for the specific data point. If a hyperlink is embedded, clicking on the data point will direct the user to the linked page.

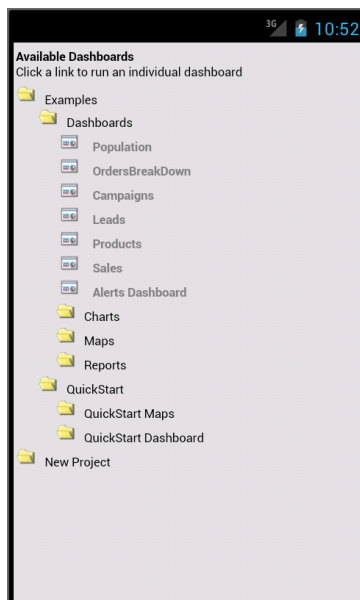
7.1.3. Mobile MenuPage

If you access EDAB main page from a mobile device, a special simplified EDAB start page will be displayed.

*EDAB mobile start page***Note**

The mobile features don't work in so-called *PC view*, which can be enabled in some mobile web browsers. In *PC view*, the standard EDAB start page and MenuPage will be displayed.

Enter your user name and password and tap on the *Login* button. You will be redirected directly to a special mobile MenuPage which allows you to view reports, charts, maps and dashboards.

*Mobile MenuPage*

The mobile MenuPage is easy to use - scroll/swipe up or down to locate a template or a dashboard and tap on its name to open it.

Mobile Report Viewer

If you open a report from the mobile MenuPage, standard DHTML Viewer will open. To learn more about the DHTML Viewer, see Section 7.1.2.2 - Using the DHTML Report Viewer.

*DHTML Viewer***Mobile Chart and Map Viewer**

Chart/map viewer has no control elements.

If the chart/map has any parameters, a parameter prompt will be displayed first. Enter any parameter values and tap on the *Submit* button to display the chart/map.

State: AK

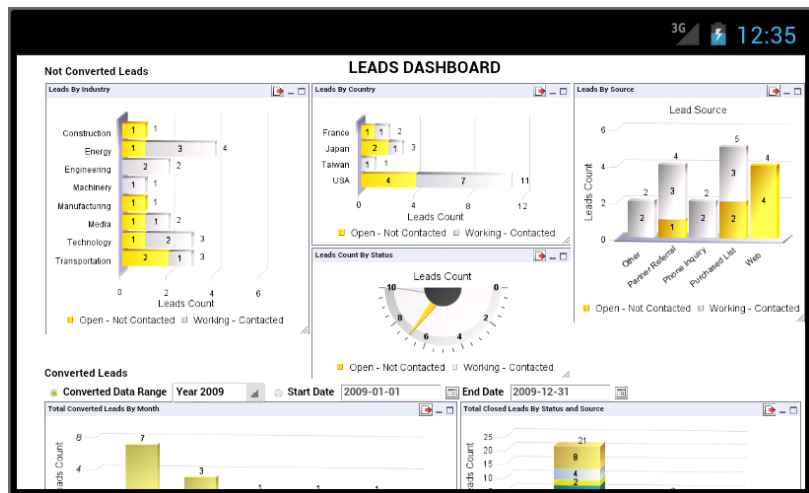
StartDate: 2001-01-01

EndDate: 2003-12-31


Submit Reset

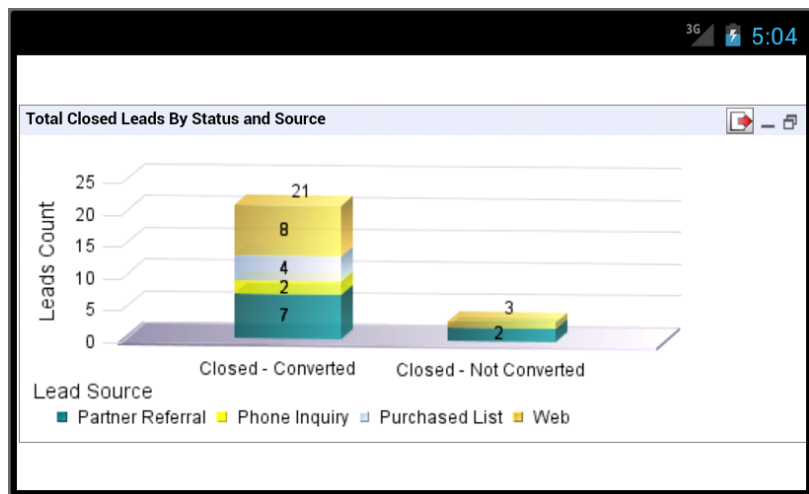
*Parameter prompt***Dashboards**

If you open a dashboard on a mobile device, you can swipe the dashboard to navigate in it.



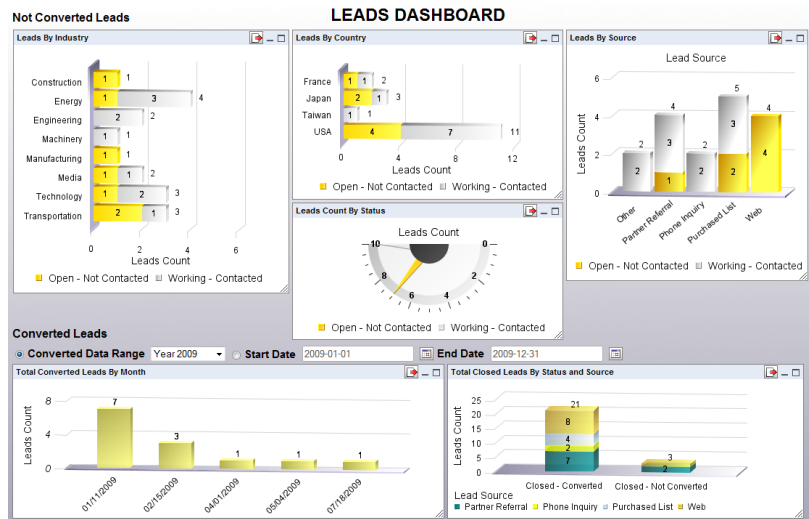
However, on smaller devices, navigating through large dashboards may require extensive zooming and scrolling. In such cases, you can maximize a dashboard template to display the dashboard in a more mobile-friendly way.

To maximize a template, click on its  maximize icon, or double-tap/pinch on the template.

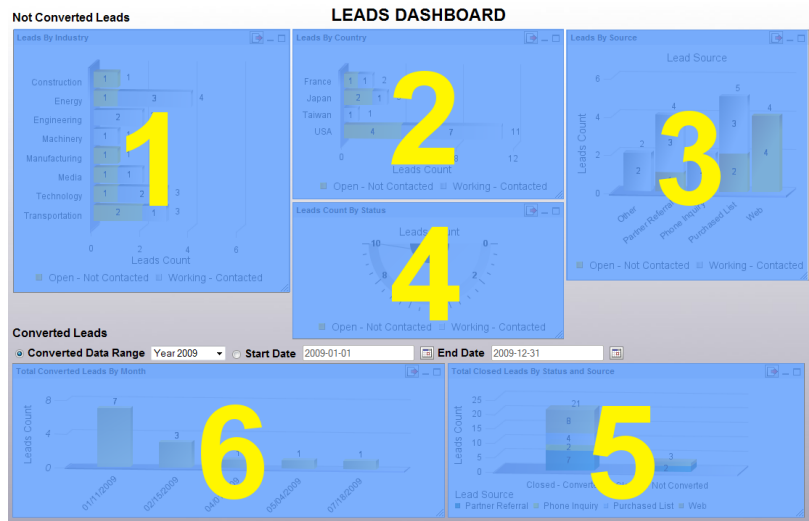


When navigating in maximized-mode, all dashboard templates will be arranged in a single loop, so you don't have to remember the dashboard layout at all. If you swipe on a maximized template to the left, the next template will be displayed.

For example: Templates from the following dashboard:



Would be displayed in the following order:

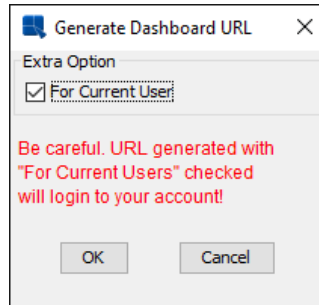


7.2. Dashboard URLs

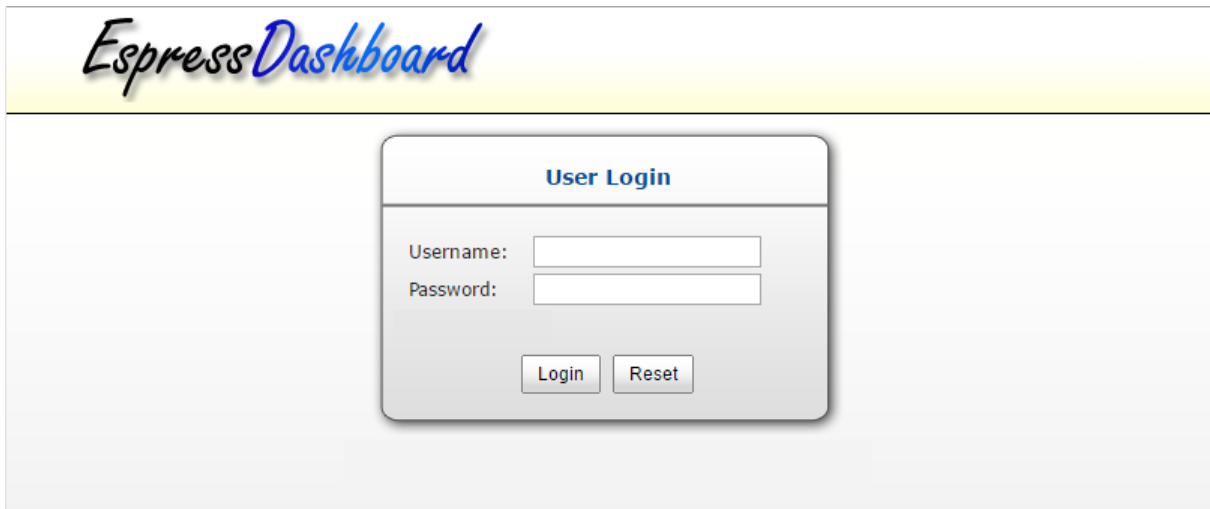
In addition to the menu and the API, EDAB also allows you to deploy dashboards using URLs. Dashboard URLs allow dashboards to be called via http call to the server. Dashboard URLs are whole URLs that are run by pointing the web browser to the URL address, or by specifying a hyperlink to the dashboard URL in another Web page.

7.2.1. Generating URLs in Organizer

The easiest way to generate dashboard URLs is to have the Organizer do it for you. To generate a dashboard URL in the Organizer, first select the dashboard file (.dsb) that you want to use. Then you can select *Generate Dashboard URL* from the Publish menu, click the *Generate Dashboard URL* button on the toolbar, or right-click and select *Generate Dashboard URL* from the pop-up menu. A new dialog will appear allowing you to specify user option for the generated dashboard URL.

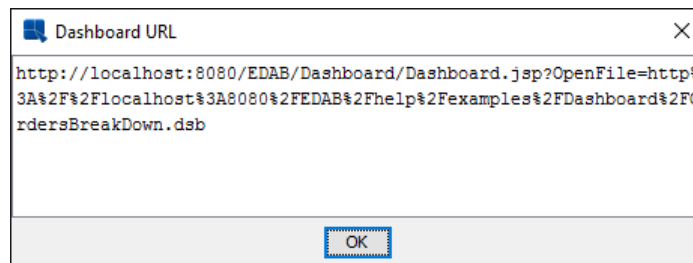
*Generate Dashboard URL Dialog*

Dashboard URLs can have username and password passed in as arguments in the URL. If the *For Current User* option is checked, the current user will be embedded in the URL. If no username or password is supplied in the URL (*For Current User* option is unchecked) when it is run, the user will be re-directed to a login page before viewing the dashboard.

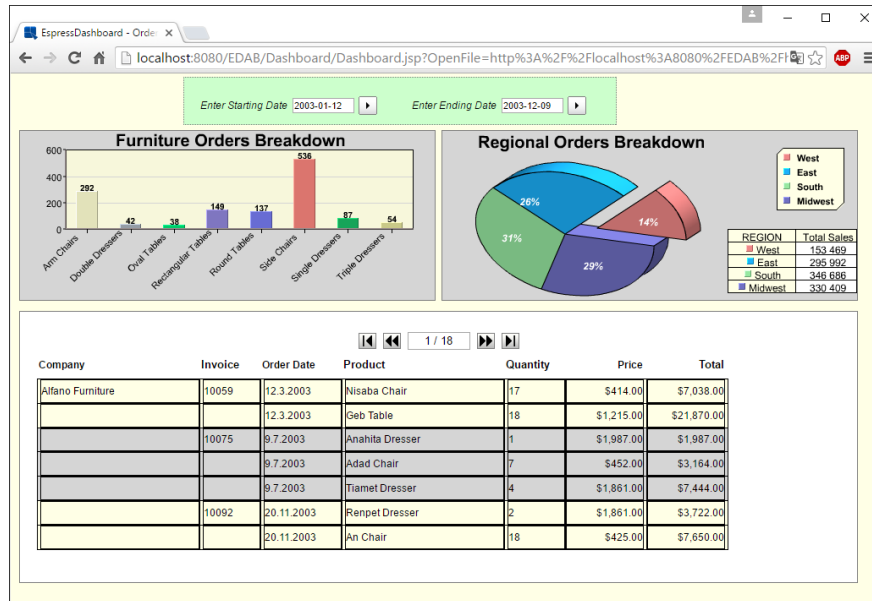
*User Login Page*

7.2.1.1. Running Dashboard URLs

Once you specify the *For Current User* option and click the *OK* button, a new window will open containing the generated URL string. From this window, you can copy the dashboard URL.

*Generated Dashboard URL*

In order to view a dashboard, just copy and paste the URL into the browser window.



View Dashboard URL

7.2.2. Writing Dashboard URLs

Rather than having the Organizer generate the dashboard URL, you can write your own by using the URL syntax below.

7.2.2.1. URL Syntax

The syntax for dashboard URLs is fairly simple. Every URL begins with a call to the Server: `http://<MACHINE_NAME>:<PORT>/<CONTEXT_ROOT>/Dashboard/Dashboard.jsp?`. The `MACHINE_NAME` is either the URL, machine name, or ip address of the server machine. The `PORT` is the port number for the application server, the default for Tomcat is 8080. The `CONTEXT_ROOT` is the context root for the EDAB. By default, the context root is EDAB, but this can be changed by administrator. Following the question mark, users need to specify either the `USERNAME` and `PASS` parameters, or the `USESESSION` parameter instead, as well as the dashboard file location.

The following URL will open the `Population.dsb` dashboard file for the specified user.

```
http://<MACHINE_NAME>:<port>/<CONTEXT_ROOT>/Dashboard/Dashboard.jsp?
USERNAME=qMFq-5dab27h&PASS=qMFq-5dab27h&OpenFile=http://
<MACHINE_NAME>:<port>/<CONTEXT_ROOT>/help/examples/Dashboard/Population.dsb
```

For example, if the server was set up on `www.quadbase.com` with port 8080 and the context root is EDAB, then the dashboard URL will look like this.

```
http://www.quadbase.com:8080/EDAB/Dashboard/Dashboard.jsp?
USERNAME=qMFq-5dab27h&PASS=qMFq-5dab27h&OpenFile=http://
www.quadbase.com:8080/EDAB/help/examples/Dashboard/Population.dsb
```

7.2.2.2. URL Parameters

URL parameters allows you to specify the dashboard file you want to run as well as specify whether to use username and password that are stored in the browser session or not.

OpenFile: This allows you to specify the dashboard file that you want to run.


```
&OpenFile=http://machinename:port/EDAB/DashboardFiles/
TestDashboard.dsb
```

**Note**

You can use both absolute and relative paths for the dashboard file location.

USERNAME: This option allows you to specify a username. If you generate an URL in Organizer and select to use an user, the username will be encoded in the generated URL. You can also specify the username as plain text.

```
&USERNAME=user
```

PASS: This option allows you to specify a user password. If you generate an URL in Organizer and select to use an user, the password will be encoded in the generated URL. The password can also be specified using plain text.

```
&PASS=password
```

USESESSION: This allows you to have the server read the username and password from the session. If this option is enabled, the information will be retrieved from session parameters USERNAME and PASS. This option is only available if you will be deploying the URL within a servlet or JSP application.

```
&USESESSION=true
```

7.3. Menu Page Listener

The menu page and URL publishing options provide a convenient deployment vehicle for reports and charts. Using these interfaces, users can easily deploy charts and reports to the Web without any coding. However, the interfaces themselves do not provide any significant run-time customization capability. Report and chart templates are run in a pretty much “as is” format.

For users that would like to provide some additional logic, or run-time customization to reports and charts, EDAB provides the Menu Page Listener interface which is part of the EDAB extension package. Using this method, users can implement listeners that will intercept the report or chart prior to export, and use the APIs to modify it.

7.3.1. EDAB Listener Manager

The EDAB Listener Manager is a user-implemented class that is used to manage several listeners that monitor events on the server, including the Menu Page Listener. In order to implement the Menu Page Listener, users must implement the Listener Manager. Below is a sample Listener Manager class:

```
package extensionClasses;

import quadbase.reportorganizer.ext.*;

public class MyEdabListenerManager extends DefaultListenerManager {

    public MyEdabListenerManager() {}

    public EdabSchedulerListener getSchedulerListener() {
```

```

        return new MyEdabSchedulerListener();
    }

    public MenuPageListener getMenuPageListener() {

        return new MyMenuPageListener();
    }
}

```

Users can implement the `EDABListenerManager` interface or extend `DefaultListenerManager`. The above code implements two listeners, the Scheduler Listener and the Menu Page Listener. The Scheduler Listener takes effect when a schedule job executes.

7.3.1.1. Deploying the Listener Manager

You can specify the Listener Manager class as a server option for EDAB. You can set this in one of two places. The first option is the *Admin Console*. You can specify the class in the *Server Options* tab. For more information about server configuration options, see Section 1.4.1.3 - Server Options. You can also specify the class by modifying the `QB.properties` under `<EDABInstallDir>/WEB-INF/classes`. However, editing configuration files directly is not recommended and should be done only in case when the EDAB server cannot be started because incorrect values have been provided through the Admin Console.

7.3.2. Using the Menu Page Listener

The following code shows a sample implementation of the Menu Page Listener:

```

package extensionClasses;

import java.lang.*;
import java.awt.*;
import quadbase.reportorganizer.ext.*;
import quadbase.reportdesigner.ReportAPI.*;
import quadbase.reportdesigner.ReportElements.*;
import quadbase.reportdesigner.util.*;
import quadbase.ChartAPI.*;
import quadbase.reportdesigner.report.Formula;

public class MyMenuPageListener implements MenuPageListener {

    public QbReport modifyBeforeRun(QbReport report, String username) {

        System.out.println("Calling Menu Page Listener...");

        try {

            // Create a new label to show user that is running the
            report

            ReportCell userLabel = new ReportCell();
            userLabel.setText("Report Run By: " + username);
            userLabel.setFont(new Font("Arial", Font.PLAIN, 8));
            userLabel.setAlign(AlignConstants.ALIGN_RIGHT);
            userLabel.setHeight(0.2);
            userLabel.setWidth(1.6);
            userLabel.setY(0);
            userLabel.setX(5.90);

```

```

        // Create a new formula to show the report run date/time
        ReportCell runDate = new ReportCell();
        Formula runDateFormula = new
Formula("ReportRunDate", "\"Report Run Date: \" +
printDateTime(getCurrentDateTime(), \"MMM dd, yyyy h:mm a\")");
        report.addFormula(runDateFormula);
        runDate.setFormulaObj(runDateFormula);
        runDate.setFont(new Font("Arial", Font.PLAIN, 8));
        runDate.setAlign(IAlignConstants.ALIGN_RIGHT);
        runDate.setHeight(0.2);
        runDate.setWidth(2.5);
        runDate.setY(0.2);
        runDate.setX(5.0);

        // Add new cells to page header section
        report.getPageHeader().addData(userLabel);
        report.getPageHeader().addData(runDate);

        // Adjust section height if necessary
        if (report.getPageHeader().getHeight() < 0.4)
            report.getPageHeader().setHeight(0.4);

    } catch (Exception ex) {

        ex.printStackTrace();

    }

    return report;
}

public QbChart modifyBeforeRun(QbChart chart, String username) {

    System.out.println("Calling Menu Page Listener");

    try {

        // Define an array of chart colors
        Color gold = new Color(227,215,130);
        Color salmon = new Color(199,85,90);
        Color burgandy = new Color(125,18,66);
        Color slate = new Color(81,119,156);
        Color teal = new Color(130,203,217);
        Color blue = new Color(26,42,103);
        Color beige = new Color(188,204,177);
        Color[] chartColors = {gold, salmon, burgandy, slate,
teal, blue, beige};

        // Apply new colors to the chart
        chart.gethDataPoints().setColors(chartColors);

    } catch (Exception ex) {

        ex.printStackTrace();

    }
}

```

```


        return chart;
    }
}

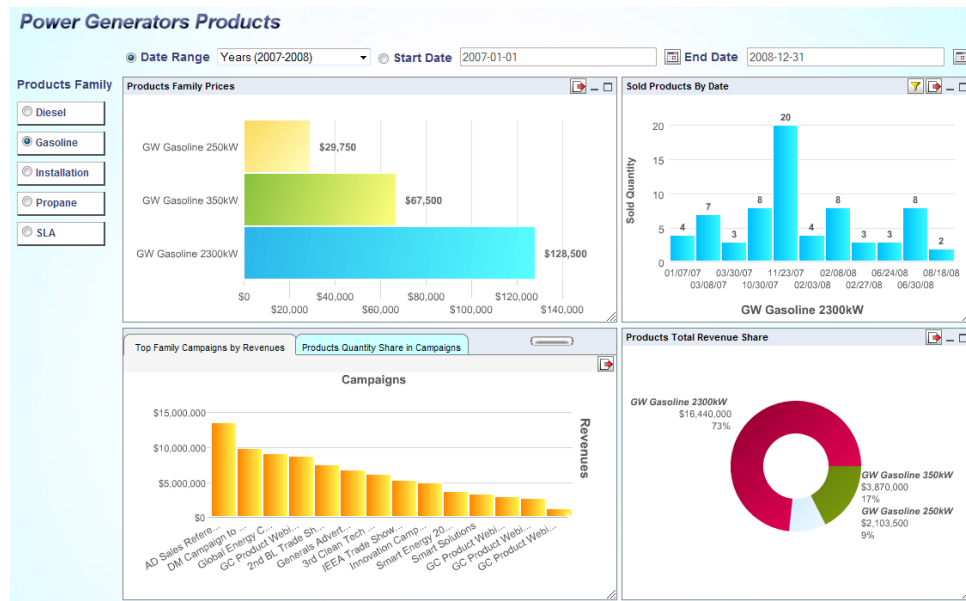
```

This example implements to modifyBeforeRun methods, one for charts and one for reports. With this code deployed, everytime a report is run, the username and the run time is added to the report header. Everytime a chart is run, the chart colors are modified to use the palette defined in the method. Note that this is only one example. Any report/chart API code can be used in these interfaces to modify the reports and charts prior to export.

7.4. Dashboard Viewer

Dashboard Viewer is an applet that enables you to view and manipulate a dashboard dynamically through a web browser. Viewer reads the file (in .dsb format) as outputted by Dashboard Builder and then displays the dashboard.

The dashboard can be previewed by clicking the *Preview* button  on the Dashboard Builder toolbar. This will open a new window showing the dashboard.



Dashboard

7.4.1. Preview Toolbar

The top part of the Preview interface contains a small toolbar that allows you to initiate the following actions:



Set shared parameter(s)



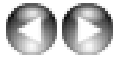
Refresh a dashboard



Set auto refresh

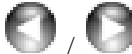



Export Dashboard to PDF (Note: All objects in dashboard including Online maps are exported, however, if map type is set to Google street map or Google satellite, map imagery will be missing in the exported file due to its license restrictions.)




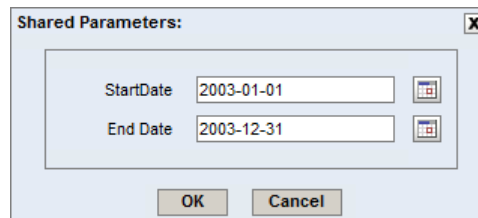
Pack/Unpack the preview toolbar

7.4.2. Preview Options

This section gives you description of options available from the preview toolbar and additional options available from the chart/report/map header bar. You can hide/show the toolbar by clicking the  /  Pack/Unpack icons.


Set Shared Parameters:

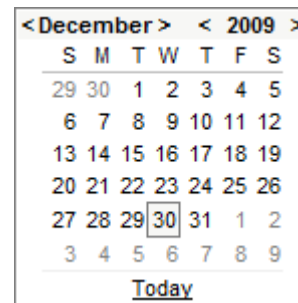
To set shared parameters, click the *Set Shared Parameters* button  on the preview toolbar. After you click the button, the shared parameters dialog will then appear. The dialog allows you to specify values for shared parameters.



The dialog box titled "Shared Parameters:" contains two input fields: "StartDate" with the value "2003-01-01" and "End Date" with the value "2003-12-31". Each field has a small calendar icon to its right. At the bottom are "OK" and "Cancel" buttons.

Set Shared Parameters Dialog

Note that you can use calendar for specifying date parameters. To open the calendar, simply click the *Calendar* button  in the shared parameters dialog.




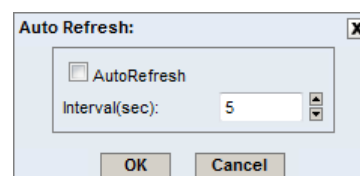
A calendar widget showing the month of December 2009. The days of the week are listed at the top: S, M, T, W, T, F, S. The dates are arranged in a grid. The date 30 is highlighted with a blue border. At the bottom, there is a "Today" link.

Calendar

Once you finish specifying the shared parameters, click the *Ok* button to save the changes. You will be taken back to the Preview.

Auto Refresh:

If you want your dashboard to be refreshed periodically, click the *Auto Refresh* button  on the preview toolbar. After you do so, the *Auto Refresh* dialog will then appear. You have to check off the *AutoRefresh* checkbox to enable the auto refresh feature. The dialog allows you to set the refresh interval in seconds. For example, if you specify the refresh interval for 10 seconds, the dashboard will refresh every 10 seconds.



The dialog box titled "Auto Refresh:" contains a checkbox labeled "AutoRefresh" which is currently unchecked. Below it is a label "Interval(sec):" followed by a text input field containing the value "5" and a small up/down arrow control. At the bottom are "OK" and "Cancel" buttons.

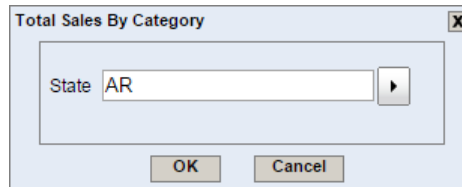
Auto Refresh Dialog

Once you finish setting up the auto refresh options, click the *Ok* button to save the changes. You will be taken back to the Preview.

In addition to the options on the preview toolbar, there are additional options available for a chart/report/map that can be triggered by clicking the small buttons in the chart/report/map header bars.



Filter: This option will open a new dialog that allows you to specify additional filters/parameters for the chart/report/map, if it has parameters that aren't associated with one of the dashboard's shared parameters.

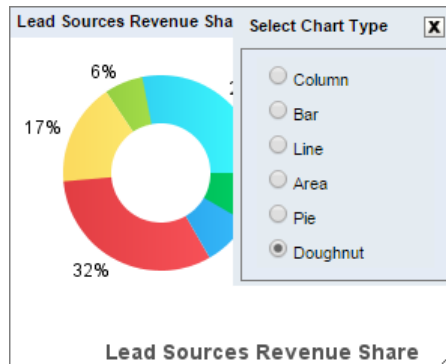


Dashboard Filter Chart/Report/Map Dialog

You can select parameter values in this dialog. Once you complete your selection, click the *Ok* button. The dashboard will refresh and the chart/report/map will reflect the new parameter values.



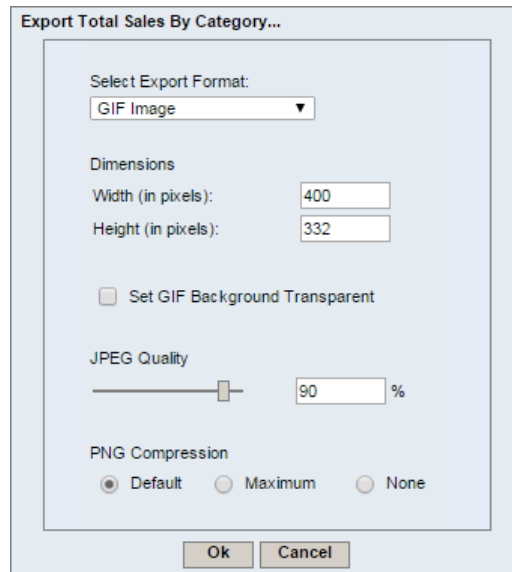
Change Chart Type: This option will open a new dialog that allows you to change a chart type. The chart type is changed only in the Preview. It is not possible to save the changed view of the chart, but you can export it.



Select Chart Type Dialog



Export: This option will open a new dialog that allows you to export the chart/report/map in a variety of formats. For charts the following dialog is displayed:

*Dashboard Export Chart Dialog*

The first option allows you to specify the format in which to export the chart. The available formats are GIF, JPEG, PNG, PDF, SVG, Excel Image (XLS), Flash, Text Data File, XML Data File and MAP Data File (HTML Image Map).

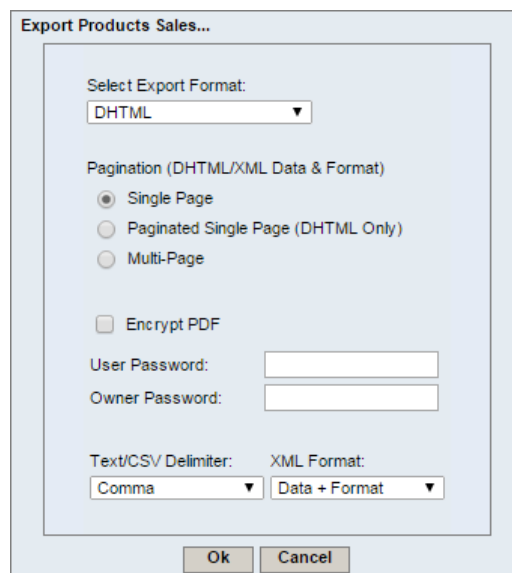
The second option allows you to set dimensions of the exported image. By default, these will match the canvas dimensions of your chart.

The last set of options allows you to set some image type-specific options. You can set the background transparent for GIF files, quality for JPEG files, and specify encoding for PNG files.

For more information about chart export options, please see Section 4.1.6.2 - Exporting Charts.

Once you finish specifying the options for exporting the chart, click the *Ok* button. A new window will open containing the exported chart.

For reports the following dialog is displayed:

*Dashboard Export Report Dialog*

The first option in the dialog allows you to specify in which format you want to export the report. Available options are PDF, CSV, Excel (XLS), Excel 2007 (XLSX), DHTML, text, XML and rich text.

Other options allows you to set single or multi-page export for DHTML and XML exports. You can enable encryption for PDF format. For text export you can select to use the delimiter and for XML export you can select whether to export only report data, or XML description of the report and data.

Once you finish specifying the options for exporting the report, click the *Ok* button. A new window will open containing the exported report.

Online Maps and SVG Maps support only one export format, PDF, so no export format settings dialog is displayed for maps. Please note that Online Maps cannot be exported if Google street map/Google satellite is used as map type, due to license restrictions. In this case, map imagery will be missing in the exported file.



Maximize: This option maximizes the report/chart/map.



Minimize: This option minimizes the report/chart/map (only template header bar is visible).



Restore: This option resizes the report/chart/map to its original size.

Chapter 8. Programming

8.1. Managing Users and Groups

8.1.1. Introduction

User and group administrator can be done programmatically via the API instead of always going to the EDAB home page. API calls are available to create, edit, and delete users and groups.

Both `EDABOrganizer.jar` and `EDABServer.jar` (in the `<EDAB-installation-directory>/lib` directory) needs to be added to the `CLASSPATH` for any code using EDAB Organizer. The following snippet shows how to connect to EDAB Organizer:

```
QbOrganizer.setServletRunner("http://localhost:8080");
QbOrganizer.setServletContext("EDAB/servlet");
QbOrganizer organizer = new QbOrganizer(null, "admin", "admin");
```

The above code sets the connection information to connect to EDAB Server and provides the username and password (in the above example, `admin` and `admin`) for EDAB Organizer.

All examples and code given in the manual assume that EDAB server is running locally (i.e. on your local machine) and on port 8080. You can change this by going to the source code (you can download the source code by clicking on the *Full Source Code* link in the corresponding chapter), editing the code to enter the new connection information and recompiling it.

Also, note that if you have applets that use EDAB Report API, the browser must have JVM plugin at least version 1.5 or higher.

8.1.2. Users and Groups

You do not have to use the EDAB Admin console to create, edit, and/or delete user and group information. You can modify any user and/or group information via the API. Note that only `admin` user can call the API methods.

To modify the user and group information, you must get a handle to `UserGroupProperties` using the following call:

```
UserGroupProperties userGroupProperties = new
    UserGroupProperties(organizer);
```

Please note that the code must log to EDAB as the `admin` user, otherwise the user/group information will remain unchanged. Any username and/or group name must be in lowercase. Uppercase and mixed case names are converted to lower case automatically.

8.1.2.1. Creating Users and Groups

Use this following method to create a new user:

```
public void createUser(String userName, String fullName, String email,
    String password, String role, String securityLevel)
```

For example:

```
userGroupProperties.createUser("jdoe", "John Doe", "jDoe@somedomain.com",
    "123", IUser.ROLE_DESIGNER, "manager");
```

Use this following method to create a new group:

```
public void createGroup(String newGroup, String description, String[] users)
```

For example:

```
userGroupProperties.createGroup("testinggroup", "For testing purpose only",  
    new String[]{"admin", "jDoe"});
```

Full Source Code [<https://data.quadbase.com/Docs71/help/manual/code/src/CreatingUserGroup.zip>]

Result Screenshot [<https://data.quadbase.com/Docs71/help/manual/images/CreatingUserGroup.gif>]

The above code creates a new user `jdoe` along with a new group `testinggroup`. The user profile information as well as group information is also passed via the code.

8.1.2.2. Deleting Users and Groups

You can also delete any users/groups through the code in the same manner.

Use this following method to delete a user:

```
public void deleteUser(String[] users)
```

For example:

```
userGroupProperties.deleteUser(new String[] { "jdoe" });
```

Use this following method to delete a group:

```
public void deleteGroup(String[] groups)
```

For example:

```
userGroupProperties.deleteGroup(new String[] { "testinggroup" });
```

Full Source Code [<https://data.quadbase.com/Docs71/help/manual/code/src/DeletingUserGroup.zip>]

Result Screenshot [<https://data.quadbase.com/Docs71/help/manual/images/AdminConsole.gif>]

The above code deletes `jdoe` user and `testinggroup` group. Note that the user and group have to exist before they can be deleted.

8.1.3. Single Sign-On

You can also have your own custom login page (for example, login to your portal) and then pass the required information to EDAB (rather than having to login twice, once for your application and the other for EDAB).

To pass in the login information, you would need to pass the following parameters to `authenticate.jsp` (located in the root EDAB install directory):

origPage : Page to be redirected to if login is successful.

loginPage : Page to be redirected to if login is not successful.

username : username to be passed.

password : password to be passed.

The following example shows the possible contents of a jsp page that is passing the information to `authenticate.jsp`. Note that in the example, username and password is inputted, although you can specify them before calling `authenticate.jsp`:

```
<html>

    <head>
    </head>
    <body>

        <form name="Login" action="authenticate.jsp" method="POST">

            <input type="hidden" name="origPage" value="MenuPage.jsp">
            <input type="hidden" name="loginPage"
value="myLogin.html">

            <table width=100% cellpadding=2 cellspacing=0>

                <tr>
                    <td width=80 valign="center" align="right">User
Name: </td>
                    <td align="left"><input type="text"
name="username"></td>
                </tr>

                <tr>
                    <td width=80 valign="center" align="right">Password:
</td>
                    <td align="left"><input type="password"
name="password"></td>
                </tr>

            </table>

            <input type="image" src="Web_Component/STARTUP/Login.gif"
border=0></td>

        </form>

    </body>

</html>
```

The recommended way to run the above code is to go to the root EDAB directory and copy the contents to an empty `.jsp` page. Note that `myLogin.html` has to be created as well.

8.1.4. Login Listener

You can also create additional code that changes the datasource (only if the datasource is a database) of the report/chart based on the user logged into EDAB at the time. This code extends the `LoginListener` class and specifies the database connection information based on the user.

Please note that after creating the code, you need to change `QB.properties` (in the `<EDAB-installation-directory>/WEB-INF/classes` directory) and add the following argument to the `ServerCommands` line:

-LoginListenerClass:<name of class file extending LoginListener>

The following code shows how to use the LoginListener class:

```
import javax.servlet.http.HttpServletRequest;
import javax.servlet.http.HttpSession;

import quadbase.auth.WebLogin;
import quadbase.auth.bean.Login;
import quadbase.reportorganizer.ext.LoginListener;

public class ExampleLoginListener extends LoginListener {

    public void setUserDatabaseConnection(HttpServletRequest req,
String userName) throws Exception {

        HttpSession session = req.getSession();
        WebLogin wl =
(WebLogin)session.getAttribute(Login.WEB_LOGIN);
        if (wl == null || wl.getUser() == null) {

            return;

        }

        String user = wl.getUser();
        if (user.equalsIgnoreCase("user1"))
        {

            session.setAttribute(USER_DBURL, "someDatabaseURL1");

            session.setAttribute(USER_DBDRIVER, "someDatabaseDriver1");

            session.setAttribute(USER_DBUSERNAME, "someDatabaseUsername1");

            session.setAttribute(USER_DBPASSWORD, "someDatabasePassword1");

        } else if (user.equalsIgnoreCase("user2"))
        {

            session.setAttribute(USER_DBURL, "someDatabaseURL2");

            session.setAttribute(USER_DBDRIVER, "someDatabaseDriver2");

            session.setAttribute(USER_DBUSERNAME, "someDatabaseUsername2");

            session.setAttribute(USER_DBPASSWORD, "someDatabasePassword2");

        }

    }

}
```

To use the above code, add the following to `QB.properties` (located in the `/WEB-INF/classes` directory) to the `ServerCommands=` line:

```
-LoginListenerClass:ExampleLoginScheduler
```

Make sure that `ExampleLoginScheduler` is in the `CLASSPATH`. The code will switch the datasource information to a different database, depending on whether `user1` or `user2` has logged in. If it is a different user, the original datasource will be used.

8.1.5. Javadoc

Javadoc for the entire API is provided along with EDAB. It is located here (server functions) [<https://data.quadbase.com/Docs71/edab/help/EDABapidocs/index.html>].

8.1.6. Summary

EDAB API provides an easy-to-use and powerful API that can be used in your custom code to administer and manage users and groups in EDAB.

Please note that the API requires JDK 1.8 or higher. The EDAB API has been tested on Windows, Solaris, Linux, AIX, HP, and Mac platforms.

8.2. EDAB Menu API Overview

8.2.1. Introduction and Setup

A Menu is typically a jsp page that shows all published reports and charts. Depending on the options available, a Menu may allow the user to choose the format in which the report and/or chart are to be presented. The Menus discussed here are analogous to those described in the Menu chapter.

EDAB provides an easy-to-use application programming interface (API) that allows users to create and customize Menus to show in their servlet/jsp applications. This API is written in 100% Pure Java and can be run on any platform with no or few minimal changes. Any and every part of the Menu is customizable using the API.

The main package for EDAB Menu API is the `quadbase.reportorganizer.MenuAPI`. Five main classes are associated with this package: `MenuNode`, `MenuTableRow`, `QbMenu`, `ScheduleJob` and `Util`. The remainder of this chapter will give you an overview of these various classes and their usage. Please note that the complete documentation is located at help/EDABapidocs/index.html.

The jar files for using EDAB Menu API are located in the `EDAB/lib` directory. These jar files should already be present in your application's `CLASSPATH` when you set up EDAB.

Currently, you can only use EDAB Menu API in a servlet/jsp environment. The servlets/jsps must also run in the same servlet context as EDAB server.

8.2.2. Using the API

This section discusses the usage of EDAB Menu API with code examples and contains descriptions of the main API classes.

8.2.2.1. Creating and Customizing an EDAB Menu

The sections below describe various steps involved in creating and customizing an EDAB Menu:

8.2.2.1.1. Connecting to EDAB Server

The first step is to connect to EDAB server. After successfully connected to the server, a `QbMenu` object can be allocated. `QbMenu` is the main class containing contents for a Menu page.

The EDAB server is simply an object that resides in the `ServletContext` of the web application. Since you will want to put most Java code in your Java bean, you will need to pass the `HttpServletRequest` and `ServletContext` objects from jsp to bean. The following jsp and bean code illustrates this:

Example jsp (Menu_login.jsp):

```
<!-- import the bean class -->
<%@page language="java" import="help.examples.Menu.exampleMenuLogin"%>

<!-- include/instantiate the bean -->
<jsp:useBean id="exampleMenuLogin" scope="session"
  class="help.examples.Menu.exampleMenuLogin" />

<!-- pass the HttpServletRequest and ServletContext objects to the bean -->
<% exampleMenuLogin.processRequest(request, application); %>
```

where the jsp passes the request (HttpServletRequest) and application (ServletContext) to the Java bean MenuLogin's processRequest method. Next, you will need to implement code in the bean that allocates the QbMenu object from the ServletContext:

Example bean (MenuLogin.java):

```
public void processRequest(HttpServletRequest request, ServletContext
  context) throws IOException {

    QbMenu qbMenu = new QbMenu(context.getAttribute(QbMenu.QBMENUDATA));

}
```

where context.getAttribute(QbMenu.QBMENUDATA) is set when EDAB server started.

8.2.2.1.2. Getting User Menu Page

The second step uses the QbMenu object to authenticate the user. This is easy since all security and permission checks are performed by EDAB server. If login is successful, you can store the QbMenu object for the session so that for the duration of the session, the user will not have to login again. The following code illustrates this:

```
boolean loginSuccess = qbMenu.login(userName, password);
if (loginSuccess) {

    //store the QbMenu object in session

    session.setAttribute("QBMENUOBJ", qbMenu);

}
```

where "QBMENUOBJ" is the name that you can use later for retrieving the stored QbMenu. This means that after storing the QbMenu in the session, you can retrieve it in other jsp pages by doing session.getAttribute("QBMENUOBJ"). The QbMenu.login method returns true if and only if the user name and password match. If login failed, the user can be prompted again. Please note that login MUST succeed for the QbMenu object to return a valid Menu.

8.2.2.1.3. Getting all Visible Nodes

After authenticating the user, you can get a list of authorized Menu nodes, which contain reports or charts from the QbMenu. A node is either a project or a folder in EDAB Organizer.

```
MenuNode[] MenuNode = qbMenu.getMenuNodeList(null);
```

Passing in null to the `getMenuNodeList` method results in getting all the available Menu nodes. If you pass in a string, the method will return only those nodes items that contain the string.

Please note that the path for the Menu node will be a complete one i.e., `Project1/Folder1/SubFolder2`.

8.2.2.1.4. Getting the Node Details

Once you have retrieved the Menu nodes, the next step is to get the node details, namely, the files contained in the node. Conceptually, each item in the node is a row and is represented by the `MenuTableRow` object. The following code iterates through each file in all nodes:

```
for (int i = 0; i < MenuNode.length; i++) {
    for (int j = 0; j < MenuNode[i].getRowItemCount(); j++) {
        MenuTableRow MenuTableRow = MenuNode[i].getMenuTableRowAt(j);
    }
}
```

8.2.2.1.5. Getting the File Details

You can obtain all the information of the file (such as file name, description, whether it is a chart or report, etc.) using the `MenuTableRow` class.

The following methods are used to obtain the file details:

```
String getDescription()
Returns the description for the row item

String getName()
Returns the name for the row item

String getPath()
Returns the path for the row item

String getSecurityLevel()
Returns the security level for the row item

String getURL()
Returns the URL for the row item

boolean isChart()
Returns whether the row item is a chart

boolean isReport()
Returns whether the row item is a report

boolean isDashboard()
Returns whether the row item is a dashboard
```

Using the above methods, you can get the various file details.

8.2.2.2. Exporting Reports/Charts using the LookupServlet Servlet

Depending on your needs and architecture, you can setup the Menu to show reports and charts in various static formats (such as DHTML, PDF, JPEG, GIF, PNG, etc.) or in an interactive applet. You can code your own servlet using the API's provided in the EDAB Reporting API Overview and EDAB Charting API Overview chapters to get the templates and export them to the format desired.

EDAB also comes with a ready-made servlet that can take different parameters to produce the report and/or chart in the format desired without you having to write any additional servlet code. This servlet is the `LookupServlet` servlet and is generally in the `EDAB/servlet` context (For example, `http://192.168.0.1:8080/EDAB/servlet/LookupServlet`). For more details on this servlet and the various parameters it requires, please refer to the Designer guide.

8.2.3. Javadoc

Javadoc for the entire API is provided along with EDAB. It is located here (server functions) [<https://data.quadbase.com/Docs71/edab/help/EDABapidocs/index.html>].

8.2.4. Summary

EDAB already provides a Menu for any user-created reports and/or charts. This Menu can be reached by logging into the Organizer and choosing View Published Dashboards. In addition to this, EDAB also provides an easy-to-use, yet powerful library to create your own Menu in your own styles to display any reports and/or charts.

The jsp code discussed in this chapter is also available in the `EDAB/help/examples/Menu` directory (this directory contains both the jsp pages and the css styles). The `EDAB/WEB-INF/classes/help/examples/Menu` directory contains the source and class files for the beans. This example demonstrates the steps given above.

Please note that the Menu API requires JDK 1.8 or higher. The EDAB Menu API and jsp pages have been tested with Mozilla (1.3 and above), Firefox (0.91 and above) and Microsoft Internet Explorer (5.x and above) on Windows NT/2000, Solaris, Linux, AIX, and HP platforms.

8.3. Organizer

8.3.1. Introduction

Using the API, the EDAB Organizer can also be called with custom choices instead of the default options provided. It can be customized in terms of look, feel, and function. Report and Chart Designer, called from the Organizer, can also be customized via the API.

Both `EDABOrganizer.jar` and `EDABServer.jar` (in the `<EDAB-installation-directory>/lib` directory) needs to be added to the CLASSPATH for any code using EDAB Organizer. The following snippet shows how to connect to EDAB Organizer:

```
QbOrganizer.setServletRunner("http://localhost:8080");
QbOrganizer.setServletContext("EDAB/servlet");
QbOrganizer organizer = new QbOrganizer(null, "admin", "admin");
```

The above code sets the connection information to connect to EDAB Server and provides the username and password (in the above example, admin and admin) for EDAB Organizer.

EDAB Organizer can then be shown by getting a handle to the Organizer user interface and setting it visible.

```
JFrame frame = organizer.getOrganizerUI();
frame.setVisible(true);
```

Please note that you need to copy the contents of the images, orgimages, reportimages, and backgroundImages directories to the working directory of the `.class` file.

All examples and code given in the manual assume that EDAB server is running locally (i.e. on your local machine) and on port 8080. You can change this by going to the source code (you can download the source code by clicking on the *Full Source Code* link in the corresponding chapter), editing the code to enter the new connection information and recompiling it.

Also note that if you have applets that use EDAB Report API, the browser must have at least 1.5 JVM plugin.

8.3.2. Look and Feel

You can specify a look and feel to EDAB Organizer before getting a handle to the user interface. By default, the EDAB Organizer uses the system's look and feel. The look and feel can be set using the following method in `QbOrganizer`:

```
public static void setLookAndFeel(javax.swing.LookAndFeel newLookAndFeel);
```

8.3.3. Inserting and Removing Items

You do not have to show EDAB Organizer in order to insert and remove items. You can add and delete items from Organizer via the API.

The following method shows how to add an item to the Organizer:

```
public void insertNewItem(String name, String description, String location,
    String url, String folderName);
```

For example:

```
organizer.insertNewItem("Orders_Breakdown", "This is an example report
    added to Organizer", "OrdersBreakdown.grp", "http://localhost:8080/EDAB/
    OrdersBreakdown.grp", "/New Project");
```

Full Source Code [<https://data.quadbase.com/Docs71/help/manual/code/src/InsertingItemEDAB.zip>]

Result Screenshot [<https://data.quadbase.com/Docs71/help/manual/images/InsertingItem.gif>]

The above code inserts a new item called `Account_Report` in the Organizer under the `/ReportFiles` project folder. The URL and physical location of the file is also passed along with a short description. Please note that to run the above code successfully, you will need to create a project called `ReportFiles` in Organizer. The recommended way to run the above code is to compile the source code, then go to the root EDAB directory and enter the following command:

```
java -classpath "../../src/er/bin;./lib/EDABOrganizer.jar;./
    lib/EDABServer.jar;./lib/qblicense.jar;./lib/
    hsqldb.jar;.>PATH_TO_COMPILED_CLASS_FILE<" InsertingItem
```

You can also remove items from the Organizer using the following method:

```
public void removeItem(String name, String folderName);
```

For example:

```
organizer.removeItem("Orders_Breakdown", "/New Project");
```

Full Source Code [<https://data.quadbase.com/Docs71/help/manual/code/src/RemovingItem.zip>]

Result Screenshot [<https://data.quadbase.com/Docs71/help/manual/images/RemovingItem.gif>]

The above code deletes `Account_Report` from the `/ReportFiles` project folder in EDAB Organizer. The recommended way to run the above code is to compile the source code, then go to the root EDAB directory and enter the following command:

```
java -classpath "../../src/er/bin;./lib/EDABOrganizer.jar;./lib/EDABServer.jar;./lib/qblicense.jar;./lib/hsqldb.jar;>PATH_TO_COMPILED_CLASS_FILE<" RemovingItem
```

8.3.4. Javadoc

Javadoc for the entire API is provided along with EDAB. It is located here (server functions) [<https://data.quadbase.com/Docs71/edab/help/EDABapidocs/index.html>].

8.3.5. Summary

EDAB API provides an easy-to-use and powerful API to customize and show Organizer interface as well as customize Report Designer and Chart Designer interfaces. You can also write code to insert and delete items in Organizer.

Please note that the API requires JDK 1.8 or higher. The EDAB API has been tested on Windows, Solaris, Linux, AIX, HP, and Mac platforms.

8.4. Menu Page Listener

The menu page and URL publishing options provide a convenient deployment vehicle for reports and charts. Using these interfaces, users can easily deploy charts and reports to the Web without any coding. However, by themselves these interfaces do not provide any significant run-time customization capability. Report and chart templates are run in a pretty much “as is” format.

For users that would like to provide some additional logic or run-time customization to reports and charts, EDAB provides the Menu Page Listener interface, which is part of the EDAB extension package. Using this method, users can implement listeners that will intercept the report or chart prior to export and use the APIs to modify it.

8.4.1. EDAB Listener Manager

The EDAB Listener Manager is a user-implemented class that is used to manage several listeners that monitor events on the server, including the Menu Page Listener. In order to implement the Menu Page Listener, users must implement the Listener Manager. Below is a sample Listener Manager class:

```
package extensionClasses;

import quadbase.reportorganizer.ext.*;

public class MyEdabListenerManager extends DefaultListenerManager {

    public MyEdabListenerManager() {}

    // not used. Always returns null
    public EdabSchedulerListener getSchedulerListener() {

        return null();
    }

    public MenuPageListener getMenuPageListener() {

        return new MyMenuPageListener();
    }

}
```

Users can implement the `EDABListenerManager` interface or extend `DefaultListenerManager`. The above code implements the `Menu Page Listener` (note that the `Scheduler Listener` always returns null).

8.4.1.1. Deploying the Listener Manager

You can specify the `Listener Manager` class as a server option for EDAB. You can set this in one of two places. The first option is the Admin Console. You can specify the class in the *Server Options* tab. You can also specify the class by modifying the `QB.properties` under `<EDABInstallDir>/WEB-INF/classes`. For more information about server configuration options, see Section 1.4.1.3 - Server Options.

8.4.2. Using the Menu Page Listener

The following code shows a sample implementation of the `Menu Page Listener`:

```
package extensionClasses;

import java.lang.*;
import java.awt.*;
import quadbase.reportorganizer.ext.*;
import quadbase.reportdesigner.ReportAPI.*;
import quadbase.reportdesigner.ReportElements.*;
import quadbase.reportdesigner.util.*;
import quadbase.ChartAPI.*;
import quadbase.reportdesigner.report.Formula;

public class MyMenuPageListener implements MenuPageListener {

    // Note that ... is not valid and is just an example
    String databaseURL = "...";
    String databaseDriver = "...";
    String databaseUID = "...";
    String databasePwd = "...";

    public QbReport modifyBeforeRun(QbReport report, String username) {

        System.out.println("Calling Menu Page Listener...");

        try {

            // Change the datasource of the report
            report.getInputData().setAllDatabaseInfo(databaseURL,
            databaseDriver, databaseUID, databasePwd);

        } catch (Exception ex) {

            ex.printStackTrace();

        }

        return report;

    }

    public QbChart modifyBeforeRun(QbChart chart, String username) {

        System.out.println("Calling Menu Page Listener");

        try {
```

```

        // Change the datasource of the chart
        quadbase.ChartAPI.DBInfo newChartDataSource = new
quadbase.ChartAPI.DBInfo(databaseURL, databaseDriver, databaseUID,
databasePwd, chart.gethInputData().getDatabaseInfo().getQuery());

chart.gethInputData().setDatabaseInfo(newChartDataSource);

    } catch (Exception ex) {

        ex.printStackTrace();

    }

    return chart;

}

}

```

This example implements two “modifyBeforeRun” methods, one for charts and one for reports. With this code deployed, everytime a report or chart is run, the datasource is changed to the one specified in the code. Note that for best results, the report and/or chart should be using a database datasource originally and the new datasource must return a result set for the original queries.

This example shows the most common scenario where the Listeners are used. Note that any report/chart API code can be used in these interfaces to modify the reports and charts prior to export. You can refer to the java documentation [<https://data.quadbase.com/Docs71/edab/help/apidocs/index.html>] to learn more about the API available.

8.5. Class File Data Source

For maximum flexibility, you can retrieve and prepare the dataset in any way you want and pass it to the report engine. To pass in your class file as the data source, your class file must implement the IDataSource [<https://data.quadbase.com/Docs71/edab/help/apidocs/quadbase/reportdesigner/util/IDataSource.html>] interface.

8.5.1. Non Parameterized

Below is a simple example that implements IDataSource and contains no parameters:

```

public class CustomClassData implements IDataSource {

    // Setting DbData for passing data as arguments
    String dataType[] = { "string", "String", "double" };
    String fieldName[] = { "Destination", "Time", "Price" };
    String records[][] = { { "Mayfair", "13:43", "3.50" },
        { "Bond Street", "13:37", "3.75" },
        { "RickmansWorth", "13:12", "5.25" },
        { "Picadilly", "13:24", "3.00" } };
    DbData data = new DbData(dataType, fieldName, records);

    public IResultSet getResultSet()
    {
        return data;
    }

}

```

Full Source Code [<https://data.quadbase.com/Docs71/help/manual/code/src/CustomClassDataEDAB.zip>]

The example above creates data (DbData instance) and stores it in memory. When `getResultSet()` method is called, it returns DbData object which implements `IResultSet`. Keep in mind that it is not necessary to create your data in this manner. As long as you are able to return an object that implements `IResultSet`, you can obtain the data from any data source.

Please note that if you are passing in your own class file as the data source, the class file must be accessible from the CLASSPATH of the EDAB Server.

8.5.2. Parameterized

You can also pass in a parameterized class file as the data source for the report. The parameter is obtained at run-time from the user and the data is then fetched and used to generate the report. Here is an example of a parameterized class file.

```
public class ParamClassFile implements IParameterizedDataSource
{
    String url = "jdbc:hsqldb:help/examples/DataSources/database/woodview";
    String driver = "org.hsqldb.jdbcDriver";
    String username = "sa";
    String password = "";

    // Specify what the parameter properties are.
    public IQueryInParam[] getParameters()
    {
        SimpleQueryInParam[] params = new SimpleQueryInParam[2];
        // SimpleQueryInParam(name, promptString, mapToColumn, TableName,
        // ColumnName, dataType, defaultValue, actualValue)
        params[0] = new SimpleQueryInParam("price", "Max price:", false, null,
        null, Types.DOUBLE, new Double(500.00), null);
        params[1] = new MySimpleQueryInParam("popular", "Popular Items Only:",
        false, null, null, Types.VARCHAR, new String("NO"), null);
        return params;
    }

    private class MySimpleQueryInParam extends SimpleQueryInParam implements
    IQueryParamValuesProvider
    {
        public MySimpleQueryInParam(String paramName, String promptName, boolean
        mapToColumn,
        String tableName, String columnName, int sqlType, Object defaultValue,
        Object value) {
            super(paramName, promptName, mapToColumn, tableName, columnName, sqlType,
            defaultValue, value);
        }
        public Vector getSelectionChoices() {
            Vector choices = new Vector();
            choices.add(new String("Yes"));
            choices.add(new String("No"));
            return choices;
        }
    }

    // Specify what data is to be returned
    public IResultSet getResultSet(IQueryInParam[] params)
    {
        double price = 500.00;
        int units = 999;

        if ((params != null) && (params.length >= 1))
```

```

{
    Object obj = params[0].getValue();
    if ((obj != null) && (obj instanceof Double)) price =
((Double)obj).doubleValue();
    obj = params[1].getValue();
    if ((obj != null) && (obj instanceof String) &&
((String)obj).equalsIgnoreCase("yes")) units = 15;
}

try{
    Class.forName(driver);
    Connection conn = DriverManager.getConnection(url, username, password);
    String query = "SELECT ProductName, Description, UnitPrice, UnitsInStock
FROM Products WHERE UnitPrice < "
+ price + " AND UnitsInStock < " + units;
    Statement stmt = conn.createStatement();

    ResultSet rs = stmt.executeQuery(query);
    QueryResultSet qry = new QueryResultSet(rs);

    return qry;

} catch(Exception e) {
    e.printStackTrace();
}

return null;
}
}

```

Full Source Code [<https://data.quadbase.com/Docs71/help/manual/code/src/ParamClassFile.zip>]

The parameterized class file must implement `IParameterizedDataSource` [<https://data.quadbase.com/Docs71/edab/help/apidocs/quadbase/reportdesigner/util/IParameterizedDataSource.html>]. The above example obtains data from woodview (HSQLDB) database and the query contains two parameters. The first parameter allows the user to set the maximum price for the query results. The parameter is of type double and the default is set to 500.00. The second parameter is a custom parameter and its purpose is to give the user a yes or no option to determine whether or not to show popular items only. To construct a custom parameter, create a class that extends the `SimpleQueryInParam` [<https://data.quadbase.com/Docs71/edab/help/apidocs/quadbase/reportdesigner/util/SimpleQueryInParam.html>] class, implements the `IQueryParamValuesProvider` [<https://data.quadbase.com/Docs71/edab/help/apidocs/quadbase/reportdesigner/util/IQueryParamValuesProvider.html>] interface and overwrites the `getSelectionChoices()` method. If the user selects yes, only items with less than 15 `UnitsInStock` will be returned in `getResultSet()`.

8.5.3. Custom Selection Choices

When using parameterized class file as the data source, keep in mind that you can not map it to a column directly (i.e. set the table name and column name in the `SimpleQueryInParam` constructor). In order to map a parameter to a column, you will have to use a custom parameter similar to the one shown above. However, the `getSelectionChoices()` method will look as follows:

```

public Vector getSelectionChoices() {

    System.out.println("getSelectionChoices called");
    try {
        Class.forName("jdbc:hsqldb:help/examples/DataSources/database/
woodview");
    }
}

```

```
String url = "org.hsqldb.jdbcDriver";
Connection conn = DriverManager.getConnection(url, "sa", "");
Statement stmt = conn.createStatement();
String query = "SELECT DISTINCT " + getColumnName() + " FROM " +
getTableName();
ResultSet rs = stmt.executeQuery(query);
Vector v = new Vector();

while (rs.next()) {
    switch (getSqlType()) {
        case Types.INTEGER:
            v.add(new Integer(rs.getInt(1)));
            break;
        case Types.VARCHAR:
            v.add(rs.getString(1));
            break; } }

stmt.close();
conn.close();

return v;

} catch (Exception ex) {
    ex.printStackTrace(); } return null;

}
```

Full Source Code [<https://data.quadb.com/Docs71/help/manual/code/src/ClassDataMapToColumn.zip>]

In addition to mapping the parameter to a column, this example also uses a multi-value parameter. Although the process is similar to using a single value parameter, there are some places where the code deviates from the norm. Please see the comments in the above source code for details.

Chapter 9. Configuration

9.1. Using Other Databases

As discussed in Section 1.3.2.1 - The EDAB Database, EDAB uses a database as a persistent storage mechanism for users, groups, permissions, data sources, and published dashboards. By default, EDAB uses an HSQL database that comes with the installation. However, since this database is generally inappropriate for production environments, users can easily configure EDAB to use a different one.

9.1.1. Create Table Scripts

The first step in configuring EDAB to run with another database is to setup the tables and fields that are used to store application information. Several scripts for different databases are available in the `<EDABInstallDir>/data` directory. The scripts contain the create table statements for the database.

In addition to database specific scripts, a generic script is also provided that can be modified to run with other databases. The `quadbasedb_jdbc.sql` file contains default scripts that are used to create the HSQL database.

9.1.1.1. Database Specific Notes

For each of the supported databases, there can be a couple additional configuration steps that needs to be taken to correctly setup the tables.

9.1.1.1.1. DB2

The script to create the tables for DB2 is named `quadbasedb_db2.sql`. For DB2, you will need to first create a TABLESPACE for the new tables. To do so, use the following steps from the command line:

1. connect to your database using `db2admin` using this command:

```
db2 connect to <DBNAME> user <DB2USER> using <Password>
```

Be sure to substitute your database username and password when making the connection.

2. Create buffer pool with the command below:

```
db2 CREATE BUFFERPOOL BP32k SIZE 100 PAGESIZE 32k
```

3. Create tablespace with the command below:

```
db2 "CREATE REGULAR TABLESPACE DATASPACE1 PAGESIZE 32 K MANAGED BY
    DATABASE USING ( FILE 'c:\db2\db2data\dataspace1_01.dbf' 1000 )
    EXTENTSIZE 4 OVERHEAD 10.5 PREFETCHSIZE 4 TRANSFERRATE 0.33 BUFFERPOOL
    BP32k"
```

9.1.1.1.2. MS SQL Server

The script to create the tables for Microsoft SQL Server is named `quadbasedb_sqlserver.sql`. There is no additional setup needed to run the script for this database. However, the database connection cannot be established to SQL Server with the default settings. The requirements may vary with different SQL Server versions.

MS SQL Server 2005

Download the latest MS SQL Server 2005 JDBC Driver from: [msdn.microsoft.com \[https://learn.microsoft.com/en-us/sql/connect/jdbc/download-microsoft-jdbc-driver-for-sql-server?view=sql-server-ver16 \]](https://learn.microsoft.com/en-us/sql/connect/jdbc/download-microsoft-jdbc-driver-for-sql-server?view=sql-server-ver16)

Unpack the driver and copy the `sqljdbc.jar` file into the `<EDAB>/WEB-INF/lib` directory.

Recommended *Admin Console* entries (see Section 9.1.2 - Specifying the Database Connection):

```
Database URL: jdbc:sqlserver://hostname:port;databaseName=<databaseName>;
```



```
Database Driver: com.microsoft.sqlserver.jdbc.SQLServerDriver
```

EDAB will connect to the SQL server instance using TCP/IP protocol, so you will have to determine the TCP port number of your SQL Server instance. To do that, launch *SQL Server Configuration Manager*, expand the *SQL Server 2005 Network Configuration* node and click *Protocols for <YourSQLServerInstanceName>* (for example *Protocols for MSSQLSERVER*). Double click on the *TCP/IP* protocol. First of all, make sure that the TCP/IP protocol is enabled. Look at the *Enabled* option in the *Protocol* tab, it should be set to *Yes*, if it is not, please do so. Then look at the *Listen All* option, there are two possibilities:

1. **Listen All is set to Yes** (default) : Go to *IP Addresses* tab and scroll down to the *IP All* section. The port number you are looking for is below the *IP All* section in the *TCP Dynamic Port* option.
2. **Listen All is set to No** : Go to *IP Addresses* tab and find the section that has the *IP Address* option set to the IP address you will be connecting to. The port number can be found one row below the particular IP address option (*TCP Dynamic Ports* option).

EDAB connects to the SQL database using SQL Server Authentication mode, so you have to have this function enabled in your MS SQL Server. You can do that in MS SQL Server Management Studio. Right click on the server name in *Object Explorer* and click *Properties*. Go to the *Security* page and select the *SQL Server and Windows Authentication mode* option. Also the user that you will use to connect to the SQL Server has to have the SQL Server authentication mode set. Be sure that the user has sufficient right to use the database.

MS SQL Server 2000

Download the latest tar version of MS SQL Server 2000 JDBC Driver from: [download.microsoft.com \[https://learn.microsoft.com/en-us/sql/connect/jdbc/download-microsoft-jdbc-driver-for-sql-server?view=sql-server-ver16 \]](https://learn.microsoft.com/en-us/sql/connect/jdbc/download-microsoft-jdbc-driver-for-sql-server?view=sql-server-ver16)

Unpack the `mssqlserver.tar` file. Go to the unpacked directory and unpack the `msjdbc.tar` file. Go to the `msjdbc/lib` directory. You should see three `jar` files in the `lib` folder. Copy them into the `<EDAB>/WEB-INF/lib` directory.

Recommended *Admin Console* entries (see Section 9.1.2 - Specifying the Database Connection):

```
Database URL: jdbc:microsoft:sqlserver://  
hostname:port;databaseName=<databaseName>;
```

```
Database Driver: com.microsoft.jdbc.sqlserver.SQLServerDriver
```

9.1.1.1.3. MySQL

The script to create the tables for MySQL is named `quadbasedb_mysql.sql`. There is no additional setup needed to run the script for this database.

Please make sure that your MySQL database uses case-insensitive identifiers (see MySQL Identifier Case Sensitivity [<https://dev.mysql.com/doc/refman/8.0/en/identifier-case-sensitivity.html>]).

9.1.1.1.4. Oracle

The script to create the tables for Oracle is named `quadbasedb_oracle.sql`. There is no additional setup needed to run the script for this database.

9.1.2. Specifying the Database Connection

Once you have successfully created the tables in your database, you can configure EDAB to use that database for storage. To setup the database connection, log in as *Admin* and enter the *Admin Console*. The database information can be entered under Setting Info → EDAB Repository.

Replace the *URL*, *driver*, *username*, and *password* with values of your database. EDAB uses JDBC to connect to the database, so you will need to make sure to enter the correct URL and JDBC driver class for the connection.

Once you finish making the changes, save the file. Next you will need to copy the JDBC driver for your database (typically one or several Jar files) to the `<EDABInstallDir>/WEB-INF/lib` directory. For application servers other than Tomcat, please refer to the instructions in Section 9.3 - Using Other Application Servers.

You can also specify a connection to the EDAB database using JNDI. To do so, you must first edit `<EDABInstallDir>/WEB-INF/web.xml` and uncomment the JNDI tag (remove the `<!--` and `-->` tags). You can also change the JNDI reference name (in the `<res-ref-name>` tag) from `jdbc/EDABdb` to another name. Then login as *Admin*, enter *Admin Console* and pass in the JNDI details (such as JNDI context factory, JNDI provider URL and JNDI name) under Setting Info → EDAB Repository. The values can be entered after the JNDI radio button is selected. Note that depending on the application server and the mapping used, the JNDI context factory and JNDI provider URL may not be necessary. For example, if EDAB is installed on a Tomcat or WebSphere Application Server, connecting to the JNDI server only requires supplying the JNDI name. For Oracle 10g, you have the choice of either omitting the context factory and provider URL to use the default environment, or you can specify one of context factories they provide: `oracle.j2ee.rmi.RMIInitialContextFactory`, `oracle.j2ee.naming.ApplicationClientInitialContextFactory`, or `oracle.j2ee.iiop.IIOPInitialContextFactory`. Due to the wide range of application servers and JNDI implementations, please see the documentation for your JNDI server for configuration options and details. When the JNDI name is specified, the database information (URL, driver, username, password) is ignored.

With the connection information set, restart the server. The EDAB server should start connected to the new database. Note that this new database will be blank, so any information about old users, privileges, or reports and charts will not be available. In addition, several key pieces of information will need to be manually setup in order to successfully deploy and run reports in EDAB with the new database.

- Web Root:** You will need to enter the correct Web Root for your application server/servlet container. This information is supplied in the *Admin Console*. For more information, see Section 1.4.1.2 - Setting Info.
- Servlet Context:** You may need to re-set the servlet context information. For more on this, please see Section 1.4.1.2 - Setting Info.
- URL Mapping:** You will need to add a virtual directory in Organizer that maps the Web path to your EDAB installation. For more information, see Section 2.1.5 - URL Mapping.

9.1.3. Migrating the EDAB Database

Sometimes it may be necessary to migrate the EDAB database from one platform to another. For example, initial development on an implementation may be done using EDAB in default configuration with the HSQL database. However, since HSQL is not recommended for production environments, users may need to migrate data to another database. To accomodate this, EDAB provides a database migration utility for all supported databases.



Warning

Since version 6.3, this tool can be used for database migration only. Database upgrade using this tool is no longer supported. Please use `dbupgrade` (`DBUpgradeGUI` or `UpgradeAppl`) application instead (Section 1.3.2.1.3 - Upgrading EDAB Database from previous version of EDAB).

Before starting the database migration process, you will have to prepare the destination database, which must contain empty EDAB 6.6 tables. To create the tables, run the appropriate SQL script from the `<EDABInstallDir>/data` directory. To learn more about the scripts, see the Section 9.1 - Using Other Databases chapter.

Please make sure that both the source and the destination databases belong to EDAB version 6.6. If not, use the `dbupgrade` application to fix this problem.

The program for the utility is in `<EDABInstallDir>/data/converter` directory. It is a Java application that is run from the command line (the source code is also provided). To run the converter, modify `FromDB.properties` file to use the JDBC connection information for your source database. Then modify `ToDB.properties` file to use the JDBC connection information for your destination database (sample connection files are provided for different databases in the directories under `<EDABInstallDir>/data/converter`).

At the command line, navigate to `<EDABInstallDir>/data/converter` directory and execute the following command:

```
java -classpath ".;driver1;driver2" CopyEDABDB
```

For example:

```
java -classpath ".;hsqldb.jar;jdbc_mysql.jar" CopyEDABDB
```

Within the `-classpath` argument, you will need to specify the archives/classes that contain the JDBC drivers for the source and destination databases.

Running this program will copy all data from the source (FROM) EDAB database tables to the destination (TO) EDAB database tables.

Once the conversion is complete, you need to modify the database connection information in the *Admin Console* as described in Section 9.1.2 - Specifying the Database Connection, before restarting the server.

Please note that EDAB log records can not be transferred during the database migration.

9.2. Integrating Existing Users/Groups

By default, the EDAB database provides persistent storage for users, groups, and their relationships. However, many users already have a set of users and groups, as well as persistent storage mechanism in place. In these cases, duplicating users and groups in the new reporting environment is inefficient, time-consuming, and requires extra administration overhead.

To prevent these issues, EDAB provides three different ways on how users can integrate their existing user/group framework into the reporting system. The first option is to retrieve users and groups from an LDAP server. This connection and setup is detailed in Section 1.4.1.4 - LDAP Settings. The second option is to retrieve users and groups from an existing database and the third option is to use the `UserSecurityProvider` interface which provides an open API allowing users to retrieve/implement users and groups from any source.

9.2.1. Using a Database

If your existing users are in a database, EDAB provides a simple implementation of the `UserSecurityProvider` that can be configured using a properties file. This implementation allows you to supply the connection information for the database as well as SQL statements for all the user/group related actions. Note that this implementation will not work in all circumstances. If your schema cannot retrieve/set the proper information with a single SQL statement, you may need to provide your own implementation of `UserSecurityProvider`.

The properties file is named `user-database.properties`, and should be placed in the `<EDABInstallDir>/WEB-INF/orguserdb` directory. A sample properties file is shown below:

```
#####
#Whether to use custom database for user/group
custom.user.database=true
#####

#####
#If custom.user.database=true, then use these information to
#connect to the database.
custom.user.database.url=jdbc:hsqldb:C:/EDAB/data/quadbasedb
custom.user.database.jdbcDriver=org.hsqldb.jdbcDriver
custom.user.database.login=sa
custom.user.database.password=
#####

#####
#Whether each of the methods in UserSecurityProvider
#is enabled.
#-----
getUsers.enable=true
```

```

getUsers.sql=Select username, fullname, email, password, user_role, id,
  securitylevel From users Order
  By username
#-----
getGroups.enable=true
getGroups.sql=Select name, description, id, securitylevel From groups Order
  By name
#-----
getGroupRelations.enable=true
getGroupRelations.sql=SELECT parent_g.name, child_g.name FROM
  group_relations r, groups parent_g, groups
  child_g WHERE r.parent_id = parent_g.id AND r.child_id = child_g.id
#-----
getGroupUserRelations.enable=true
getGroupUserRelations.sql=Select g.name, u.username From
  group_user_relations gr, groups g, users u
  Where gr.group_id=g.id And gr.user_id=u.id Order By g.name
#-----
getUserParents.enable=true
getUserParents.sql=SELECT r.group_id, g.name FROM group_user_relations r,
  groups g WHERE r.group_id =
  g.id AND r.user_id=?
#-----
getGroupParents.enable=true
getGroupParents.sql=Select parent_id From group_relations Where child_id=?
#-----
deleteUser.enable=true
deleteUser.sql=Delete From users Where username=?
#-----
deleteGroup.enable=true
deleteGroup.sql=Delete From groups Where name=?
#-----
createUser.enable=true
createUser.sql=Insert Into users (id, username, fullname, email, password,
  user_role, securitylevel)
  Values (?, ?, ?, ?, ?, ?, ?, ?)
#-----
createGroup.enable=true
createGroup.sql1=Insert Into groups (id, name, description, securitylevel)
  Values (?, ?, ?, ?)
createGroup.sql2=Delete From group_relations Where parent_id=?
createGroup.sql3=Delete From group_user_relations Where group_id=?
createGroup.sql4=Insert Into group_relations (parent_id, child_id) Values
  (?, ?)
#-----
updateUser.enable=true
updateUser.sql=Update users Set username=?, fullname=?, email=?, password=?,
  user_role=?,
  securitylevel=? Where username=?
#-----
updateGroup.enable=true
updateGroup.sql1=Update groups Set name=?, description=?, securitylevel=?
  Where name=?
updateGroup.sql2=Delete From group_relations Where parent_id=?
updateGroup.sql3=Delete From group_user_relations Where group_id=?
updateGroup.sql4=Insert Into group_relations (parent_id, child_id) Values
  (?, ?)
#-----

```

```

addGroupUserRelation.enable=true
addGroupUserRelation.sql=Insert Into group_user_relations (group_id,
    user_id) Values (?, ?)
#-----
setSecurityLevels.enable=true
setSecurityLevels.sql1=Update users Set securitylevel=? Where username=?
setSecurityLevels.sql2=Update groups Set securitylevel=? Where name=?
#-----
resetSecurityLevel.enable=true
#-----
getGroupsBySecurityLevel.enable=true
getGroupsBySecurityLevel.sql=Select name From Groups Where securitylevel=?
#-----
getUsersBySecurityLevel.enable=true
getUsersBySecurityLevel.sql=Select username From users Where securitylevel=?
#-----
#####

```

The question marks in the SQL statements will be supplied by the EDAB engine when it runs queries. For any method that isn't enabled, EDAB will save that information in the EDAB database. Depending on your configuration, saving some information in the EDAB database and some in your own may not be able to work correctly.

9.2.2. Implementing UserSecurityProvider

For maximum flexibility, EDAB allows users to provide their own implementation of users and groups. The UserSecurityProvider interface provides open APIs into all the methods that set/retrieve user and group information. Implementing UserSecurityProvider allows you to retrieve user/group information from any source, and allows you to integrate your user database if the schema prevents you from retrieving the correct information with a single query. Below is a sample UserSecurityProvider implementation:

```

import java.util.*;

import quadbase.reportorganizer.data.*;
import quadbase.reportorganizer.ext.*;

/**
 * A sample implementation of UserSecurityProvider that provides hard-coded
 * initialized users/groups/security that does not have any data persistent
 * mechanism.
 */
public class MyUserSecurityProvider implements UserSecurityProvider {

    User[] users;
    Group[] groups;
    GroupRelation[] gr;
    GroupUserRelation[] gur;

    /**
     * Initializes the provider with dummy users/groups/security
     */
    public MyUserSecurityProvider() {
        users = new User[6];
        users[0] = createUser("admin");
        users[1] = createUser("jason");
        users[2] = createUser("ricky");
        users[3] = createUser("jordan");
        users[4] = createUser("oneal");
        users[5] = createUser("jerry");
    }

```

```

        groups = new Group[6];
        groups[0] = createGroup("nba");
        groups[1] = createGroup("sports");
        groups[2] = createGroup("testers");
        groups[3] = createGroup("managers");
        groups[4] = createGroup("marketing");
        groups[5] = createGroup("engineers");

        gr = new GroupRelation[3];
        gr[0] = new GroupRelation("sports", "nba");
        gr[1] = new GroupRelation("managers", "marketing");
        gr[2] = new GroupRelation("engineers", "testers");

        gur = new GroupUserRelation[6];
        gur[0] = new GroupUserRelation("managers", "jason");
        gur[1] = new GroupUserRelation("nba", "jordan");
        gur[2] = new GroupUserRelation("marketing", "ricky");
        gur[3] = new GroupUserRelation("nba", "oneal");
        gur[4] = new GroupUserRelation("testers", "jerry");
        gur[5] = new GroupUserRelation("testers", "jason");
    }

    public User[] getUsers() throws Exception {
        return users; }

    public Group[] getGroups() throws Exception {
        return groups; }

    public GroupRelation[] getGroupRelations() throws Exception {
        return gr; }

    public GroupUserRelation[] getGroupUserRelations() throws Exception {
        return gur; }

    public String[] getUserParents(String username) throws Exception {
        Vector v = new Vector();
        for (int i = 0; i < gur.length; i++)
            if (gur[i].getUser().equals(username))
                v.add(gur[i].getGroup()); return
toStringArr(v); }

    public String[] getGroupParents(String groupname) throws Exception {
        Vector v = new Vector();
        for (int i = 0; i < gr.length; i++)
            if (gr[i].getChild().equals(groupname))
                v.add(gr[i].getParent()); return
toStringArr(v); }

    public void deleteUser(String username) throws Exception {
        if (users.length == 0) return;
        User[] users2 = new User[users.length-1];
        int j = 0;
        for (int i = 0; i < users.length; i++) {
            if (j == users.length-1 && !
users[i].getName().equals(username))
                return; if (!users[i].getName().equals(username)) {

```

```

        users2[j] = users[i];
        j += 1; } }
    users = users2; }

/**
 * Delete a Group. Also deletes the group relationships and
 * group-user relationships
 * that references this GROUPNAME.
 * @param groupname the group name
 */
public void deleteGroup(String groupname) throws Exception {
    if (groups.length == 0) return;
    Group[] groups2 = new Group[groups.length-1];
    int j = 0;
    for (int i = 0; i < groups.length; i++) {
        if (j == groups.length-1 && !
groups[i].getName().equals(groupname))
            return; if (!groups[i].getName().equals(groupname))
{
            groups2[j] = groups[i];
            j += 1; } }
    groups = groups2;

    Vector v = new Vector();
    for (int i = 0; i < gr.length; i++) {
        if (gr[i].getParentName().equals(groupname) ||
            gr[i].getChildName().equals(groupname)) continue;
v.add(gr[i]); }
    GroupRelation[] gr2 = new GroupRelation[v.size()];
    for (int i = 0; i < gr2.length; i++)
        gr2[i] = (GroupRelation) v.get(i); gr = gr2;

    v = new Vector();
    for (int i = 0; i < gur.length; i++) {
        if (gur[i].getGroupName().equals(groupname)) continue;
v.add(gur[i]); }
    GroupUserRelation[] gur2 = new GroupUserRelation[v.size()];
    for (int i = 0; i < gur2.length; i++)
        gur2[i] = (GroupUserRelation) v.get(i); gur = gur2; }

public void createUser(User user) throws Exception {
    User[] users2 = new User[users.length+1];
    System.arraycopy(users, 0, users2, 0, users.length);
    users2[users2.length-1] = user;
    users = users2; }

public void updateUser(User user) throws Exception {
    for (int i = 0; i < users.length; i++) {
        if (users[i].getName().equals(user.getName())) {
            users[i] = user;
            return; } } }

/**
 * Create a new Group.
 * @param parentGroup create this group
 * @param childUsers set the group to have these child users
 * @param childGroups set the group to have these child groups
 */

```

```

    public void createGroup(Group parentGroup, User[] childUsers, Group[]
childGroups) throws Exception {
        addGroup(parentGroup);

        if (childUsers != null && childUsers.length > 0) {
            for (int i = 0; i < childUsers.length; i++) {
                addGroupUserRelation (parentGroup.getName(),
childUsers[i].getName()); } }

        if (childGroups != null && childGroups.length > 0) {
            GroupRelation[] gr2 = new GroupRelation[gr.length
+childGroups.length];
            System.arraycopy(gr, 0, gr2, 0, gr.length);
            int j = 0;
            for (int i = gr.length; i < gr2.length; i++) {
                gr2[i] = new GroupRelation(parentGroup.getName(),
childGroups[j].getName());
                j++; }
            gr = gr2; } }

/**
 * Update a Group. This simple implementation simply calls
 * deleteGroup(parentGroup.getName()) and calls
 * createGroup(parentGroup, childUsers, childGroups);
 *
 * @param parentGroup update this group
 * @param childUsers set the group to have these child users
 * @param childGroups set the group to have these child groups
 */
    public void updateGroup(Group parentGroup, User[] childUsers, Group[]
childGroups) throws Exception {
        deleteGroup(parentGroup.getName());
        createGroup(parentGroup, childUsers, childGroups); }

    public void addGroupUserRelation(String groupname, String
username) throws Exception {
        GroupUserRelation[] gur2 = new GroupUserRelation[gur.length+1];
        System.arraycopy(gur, 0, gur2, 0, gur.length);
        gur2[gur2.length-1] = new GroupUserRelation(groupname,
username);
        gur = gur2; }

    // SECURITY
    LEVEL //////////////////////////////////////////

/**
 * Sets the security level(s) for a group or user of the specified
name(s)
 * Also sets the security level(s) for group or user that is not in
NAME
 * but with the same SECLEVEL to -1 (Can simply call
 * resetSecurityLevel(secLevel) in the first line of this method
 * to achieve this).
 * @param secLevel the result security level number
 * @param name an array of name(s) to change
 * @param isGroup whether the name[i] is a group (isGroup[i]==true)
 * or a user (isGroup[i]==false).
 */

```

```

    public void setSecurityLevels(int secLevel, String[] name, boolean[]
isGroup) throws Exception {
        resetSecurityLevel(secLevel);
        for (int i = 0; i < name.length; i++) {
            if (isGroup[i]) {
                for (int j = 0; j < groups.length; j++)
                    if (groups[j].getName().equals(name[i])) {
                        groups[j].setSecurityLevel(secLevel);
                        break; } } else {
                for (int j = 0; j < users.length; j++)
                    if (users[j].getName().equals(name[i])) {
                        users[j].setSecurityLevel(secLevel);
                        break; } } } }

    /**
     * Resets (set to -1) the security level number of all groups
     * and users whose security level number is SECLEVEL.
     * @param secLevel the security level number of the group/user
     */
    public void resetSecurityLevel(int secLevel) throws Exception {
        for (int i = 0; i < users.length; i++)
            if (users[i].getSecurityLevel() == secLevel)
                users[i].setSecurityLevel(-1);
        for (int i = 0; i <
groups.length; i++)
            if (groups[i].getSecurityLevel() == secLevel)
                groups[i].setSecurityLevel(-1); }

    /**
     * @param secLevel the security level number of the group
     * @return the Group names of all groups with SECLEVEL
     */
    public String[] getGroupsBySecurityLevel(int secLevel) throws
Exception {
        Vector v = new Vector();
        for (int i = 0; i < groups.length; i++)
            if (secLevel == groups[i].getSecurityLevel())
                v.add(groups[i].getName());
        String[] result = new
String[v.size()];
        for (int i = 0; i < result.length; i++)
            result[i] = (String) v.get(i);
        return result; }

    /**
     * @param secLevel the security level number of the user
     * @return the User names of all users with SECLEVEL
     */
    public String[] getUsersBySecurityLevel(int secLevel) throws Exception
{
        Vector v = new Vector();
        for (int i = 0; i < users.length; i++)
            if (secLevel == users[i].getSecurityLevel())
                v.add(users[i].getName());
        String[] result = new
String[v.size()];
        for (int i = 0; i < result.length; i++)
            result[i] = (String) v.get(i);
        return result; }

    User createUser(String name) {

```

```

if(name.equalsIgnoreCase("admin")){
return new User(name, //name
name + " full name", //full name
name, //password
name + "@example.com", //email
User.ADMIN, //role
-1); //security level
}else{
return new User(name, //name
name + " full name", //full name
name, //password
name + "@example.com", //email
User.DESIGNER, //role
-1); //security level
}
}

Group createGroup(String name) {
return new Group(name, //name
name + " desc", //description
-1); //security level
}

void addGroup(Group g) {
Group[] g2 = new Group[groups.length+1];
System.arraycopy(groups, 0, g2, 0, groups.length);
g2[g2.length-1] = g;
groups = g2; }

static String[] toStringArr(Vector v) {
String[] p = new String[v.size()];
for (int i = 0; i < p.length; i++)
p[i] = (String) v.get(i); return p; }
}

```

In the previous sample, the users and groups are defined as simple arrays in the code. As with the database properties file, any method that you do not implement in the `UserSecurityProvider` interface will use the default EDAB database as the storage mechanism.

9.2.2.1. Deploying UserSecurityProvider

The `UserSecurityProvider` class can be set as a server option in EDAB. You can set this in one of two places. The first option is the *Admin Console*. You can specify the class in the *Server Options* tab. For more information about server configuration options, see Section 1.4.1.3 - Server Options. You can also specify the class by modifying the `QB.properties` under `<EDABInstallDir>/WEB-INF/classes`. However, editing configuration files directly is not recommended and should be done only in case when the EDAB server cannot be started because incorrect values have been provided through the *Admin Console*. If you still want to edit the files manually: edit `user-database.properties` to use the "users/groups" other than ERES database, `QB.properties` to set ERES database not only "users/groups", but also ERES setting, report/chart/etc/file paths, schedules, email settings, extension classes. It has higher priority than the database defined by `user-database.properties`. Compiled, `MyUserSecurityProvider.class`, if used, should be located in `<EDABInstallDir>/WEB-INF/classes` directory.

9.3. Using Other Application Servers

EspressDashboard deploys as servlet and JSP collection in an application server. As detailed in Section 1.3.1 - Installing EDAB, EDAB comes bundled with the Tomcat servlet container. Users have an option to install Tomcat

(with EDAB automatically deployed) when running the installer. This option is recommended for new users and evaluators because EDAB will work “out-of-the-box” in this configuration.

However, EDAB can also be deployed on most popular application servers. This chapter provides step-by-step setup instructions for Tomcat, Resin, WebLogic, WebSphere, JBoss, Oracle, and Sun servers.

Throughout this chapter, the following naming conventions will be used.

<EDAB_INSTALL_DIR> :	This is the location where EDAB is installed.
<EDAB_DEPLOY_DIR> :	This is the final deployment directory of EDAB. It might not be same as your installation directory (<EDAB_INSTALL_DIR>).
<EDAB_CONTEXT> :	This is the context path used to access EDAB web application components.
<EDAB_SERVLET_CONTEXT> :	This is the servlet context of EDAB. EDAB uses servlets for many of the back-end functions. There is a <code>web.xml</code> file under the <code><EDAB_INSTALL_DIR>/WEB-INF</code> directory that maps all servlet classes packaged with EDAB. By default, every servlet is mapped to <code>/servlet/<servlet name></code> . After deploying EDAB, you may need to change the servlet context.
<server> :	This is your application/web server name. It can be the name of the server machine or IP address.
<port> :	The port number on which your application/web server is listening.
EDAB Start Page :	This is the main/home page for EDAB. It should be accessed by the following URL: <code>http://<server>:<port>/<EDAB_CONTEXT>/index.jsp</code> .

In addition, instructions in this chapter refer to starting and stopping the EDAB server, as well as features/options in the *Admin console*. If you are using *AUTOSTART* feature, you need to set correct port in *Server Options - Auto-Start* in *Admin Console* page. For details about these actions, please see Section 1.3.2.2 - Starting the Server and Section 1.4 - Administration.

As the result of moving to a different application server, the `<EDAB_INSTALL_DIR>` location may move or the name of the EDAB context may be changed. When this happens, you will need to modify the URL mapping in Organizer in order to create/modify/run reports and charts. For more information about setting URL mapping, see Section 2.1.5 - URL Mapping.



Note

These instructions assume you are running EDAB with the default (HSQL) database. If you are using a different database, you need to add the JDBC driver classes for your database to your servlet container's classpath. All JAR files are under `<EDAB_INSTALL_DIR>/WEB-INF/lib`, which is used by the servlet container to load classes required by the EDAB web application. Therefore, if you are adding JDBC driver that is needed by the EDAB web application, you can simply copy the JAR file to `<EDAB_INSTALL_DIR>/WEB-INF/lib`.

You can also deploy EDAB as a war file. To make an EDAB war file, you must:

- Go to `<EDAB_INSTALL_DIR>/WEB-INF` directory and edit `web.xml`. Remove the two instances of the `<!-- WAR Deployment and WAR Deployment -->` references and then save the file.
- Open command prompt and `cd` to `<EDAB_INSTALL_DIR>` directory. Then run following command (make sure `jdk/bin` is in your `PATH`):

```
jar -cvf0 EDAB.war *
```

- The `EDAB.war` file is now available under the `<EDAB_INSTALL_DIR>` and can be deployed to an application server.

Please note that when deploying the war file, the `<EDAB_INSTALL_DIR>` is still being used to store all templates and user files, so it must be valid and accessible. Also, the instructions, given above, must be carried out again when an update is performed on the `<EDAB_INSTALL_DIR>` directory and the newly created war file should be deployed again. Note that certain application servers requires additional files (such as JBoss needing a `jbosswar.xml` file) before the war file can be successfully deployed.

9.3.1. Tomcat 4.1/5.x/6.0.x/7.0.x

The following instructions show how to deploy EspressoDashboard under Tomcat 4.1/5.x/6.0.x/7.0.x. The instructions assume that Tomcat has been installed on the system. The location of the Tomcat installation is referenced as `<TOMCAT_INSTALL_DIR>`.

For reference, please note that Tomcat's default port is 8080, the default working directory is `<TOMCAT_INSTALL_DIR>/bin` and the default Web root is `<TOMCAT_INSTALL_DIR>/webapps/ROOT`.

1. Modify the `<TOMCAT_INSTALL_DIR>/config/server.xml` file and add the following entry: `<Context path="/EDAB" docBase="<EDAB_INSTALL_DIR>" debug="0"/>`. Path is the context path of EDAB web application and docBase is the absolute path or relative path of the EDAB installation. Please substitute `<EDAB_INSTALL_DIR>` with your actual path.



Note

Please use double quotes.

2. Now start Tomcat from `<TOMCAT_INSTALL_DIR>/bin/startup.bat / .sh`.
3. With Tomcat running, go to EDAB Start page `http://<server>:<port>/<EDAB_CONTEXT>/index.jsp`. With the above configuration, `EDAB_CONTEXT` is `EDAB`. Start the EDAB Server.
4. From the EDAB Start page, login as administrator and launch the *Admin Console*.
5. From *Admin Console* page, go to *Setting Info* tab and change the EDAB Servlet Context entry to `/EDAB/servlet`. This change requires you to restart the EDAB Server. You can also change the webroot entry to `<TOMCAT_INSTALL_DIR>/webapps/ROOT`, but it is not necessary.
6. Stop and restart the EDAB Server.

9.3.2. Resin

9.3.2.1. Resin 3.x/4.x

Deploying EspressoDashboard to Resin 3.x/4.x can be accomplished using the instructions provided below. The location of the Resin installation is referenced as `<RESIN_INSTALL_DIR>`.

For reference, please note that Resin's default port is 8080, the default working directory is `<RESIN_INSTALL_DIR>` and the default Web root is `<RESIN_INSTALL_DIR>/docs`.

1. You may need to move `tools.jar` file from your Java development kit to the `<RESIN_INSTALL_DIR>/lib` directory (for JSP compilation) or add a library path entry for the file.
2. Modify `<RESIN_INSTALL_DIR>/conf/resin.conf`. Add the following entry for context path `/EDAB` `<web-app id="/EDAB" document-directory="<EDAB_INSTALL_DIR>" />` between `<host-default>` and `</host-default>` tags. The `id` in the entry is the context path of EDAB, and `document-directory` is the absolute path of the EDAB installation. Please substitute `<EDAB_INSTALL_DIR>` with your actual EDAB installation directory.

3. In Resin 3.X, start your Resin server by running `<resin-installation-dir>/bin/httpd`. In Resin 4.X, first navigate to the `<RESIN_INSTALL_DIR>` directory and then start the server by running the following command: `java -jar lib/resin.jar start`.
4. Now from your web browser, browse to the EDAB Start page to start the server (`http://<server>:<port>/<EDAB_CONTEXT>/index.jsp`). For the above configuration, the context path is EDAB.
5. From the EDAB Start page, login as the administrator and launch the *Admin Console*.
6. From *Admin Console* page, go to the *Setting Info* tab. Change the EDAB Servlet Context entry to `/EDAB/servlet`. You can also change the webroot to `<RESIN_INSTALL_DIR>/doc`, but it is not necessary.
7. Stop the EDAB Server.
8. Stop and re-start Resin if necessary.
9. Go to the EDAB Start page and start the EDAB server.

9.3.2.2. Resin 2.1.12

The following instructions show how to deploy EspressoDashboard under Resin 2.1.12. The instructions assume that Resin has been installed on the system. The location of the Resin installation is referenced as `<RESIN_INSTALL_DIR>`.

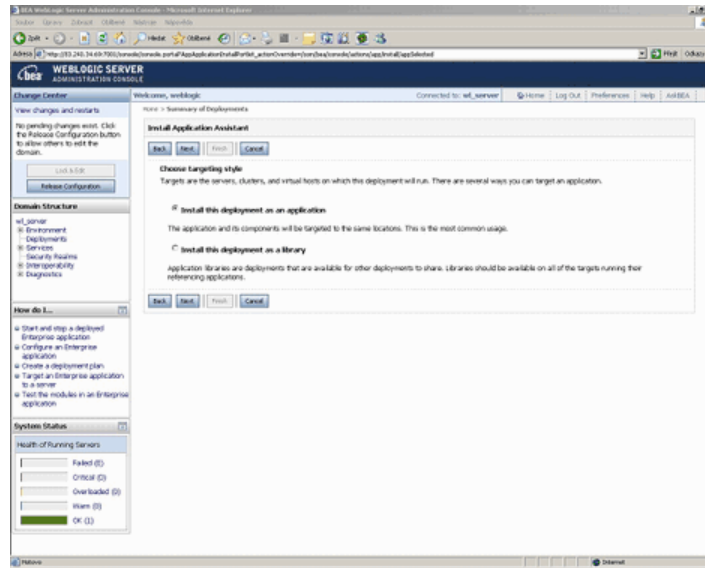
For reference, please note that Resin's default port is 8080, the default working directory is `<RESIN_INSTALL_DIR>` and the default Web root is `<RESIN_INSTALL_DIR>/docs`.

1. You may need to move `tools.jar` file from your Java development kit to the `<RESIN_INSTALL_DIR>/lib` directory (for JSP compilation), or add a library path entry for the file.
2. Modify `<RESIN_INSTALL_DIR>/conf/resin.conf`. Add the following entry for context path `/EDAB`
`<web-app id="/EDAB" app-dir="<EDAB_INSTALL_DIR>"/>` Where `id` is the context path of EDAB, and `app-dir` is the absolute path of the EDAB installation. Please substitute `<EDAB_INSTALL_DIR>` with your actual EDAB installation directory.
3. Start your Resin server by running `<resin-installation-dir>/bin/httpd`.
4. Now from your web browser, browse to the EDAB Start page to start the server (`http://<server>:<port>/<EDAB_CONTEXT>/index.jsp`). For the above configuration, the context path is EDAB.
5. From the EDAB Start page, login as administrator and launch the *Admin Console*.
6. From *Admin Console* page, go to *Setting Info* tab and change the EDAB Servlet Context entry to `/EDAB/servlet`. You can also change the webroot to `<RESIN_INSTALL_DIR>/doc`, but it is not necessary.
7. Stop the EDAB Server.
8. Stop and restart Resin if necessary.
9. Go to the EDAB Start page and start the EDAB server.

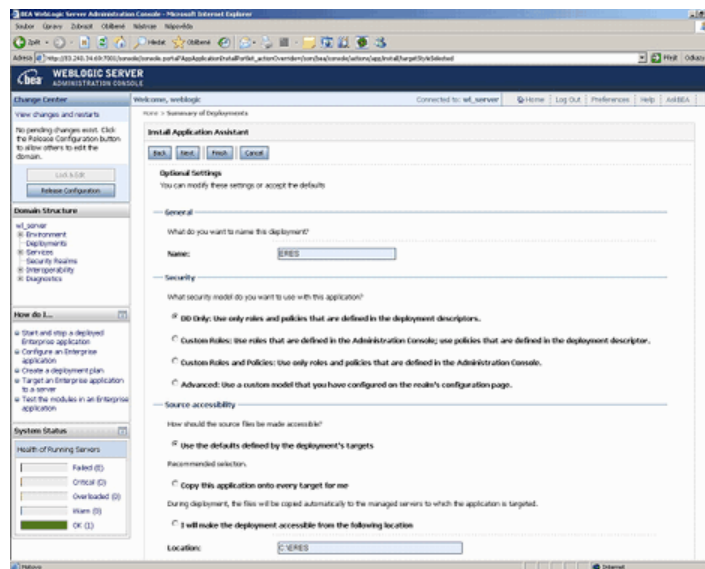
9.3.3. WebLogic Server

9.3.3.1. WebLogic Server 9.2/10.3

The following instructions show how to deploy EspressoDashboard under WebLogic Server 9.2/10.3. The instructions assume that you have WebLogic 9.2/10.3 Server installed on the system and will be deploying EDAB under the `wl_server` that comes with WebLogic. The location of the WebLogic installation will be referenced as `<WL_INSTALL_DIR>`.

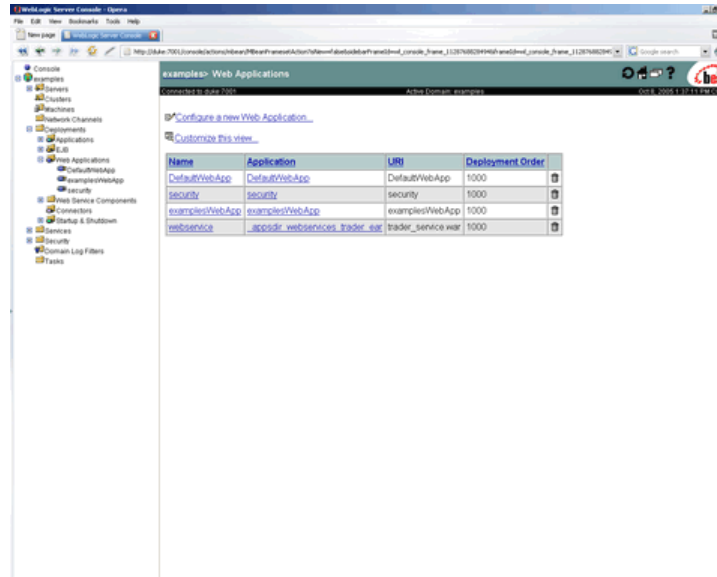


6. This will show up the following panel that will allow you to specify optional settings of the configuration procedure. From this panel, enter the information for EDAB and then press the *Finish* button.

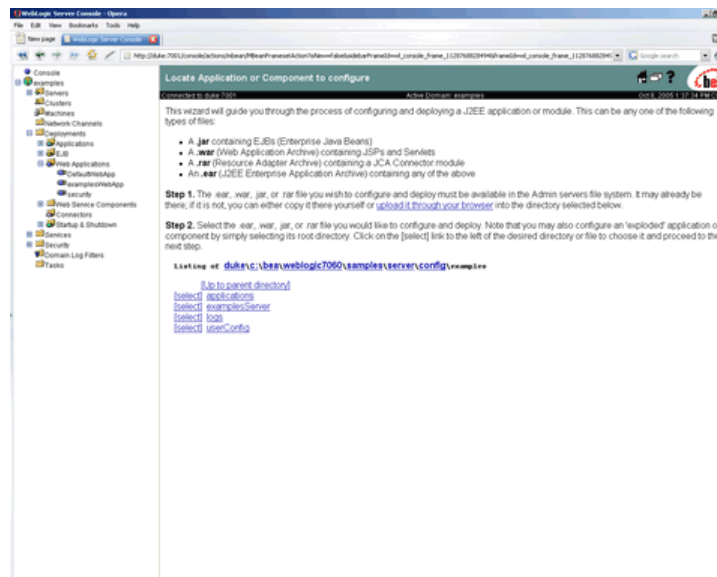


7. If the deployment has been installed and added to the list of pending changes successfully, press the *Activate Changes* button from the Change Center dialog to activate the pending changes.
8. Activate the EDAB application by checking the checkbox to the left of the EDAB link and press the "Start" button.

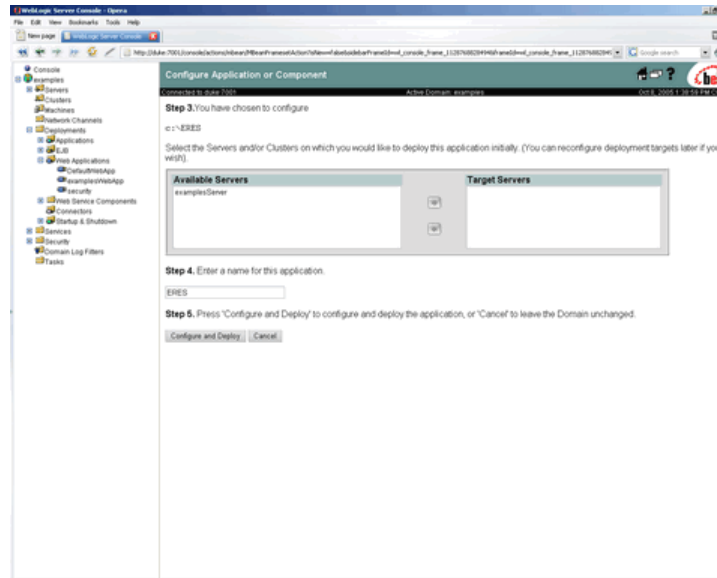
1. Start WebLogic example server by executing `<WL_INSTALL_DIR>/samples/server/config/examples/startExamplesServer.cmd`
2. When Example Server web page is loaded, click the *Administration console* link in the home page.
3. Login to the *administration console* page and click examples → Deployment → Web Applications from the left panel. This will bring up the Web Applications panel on the right.



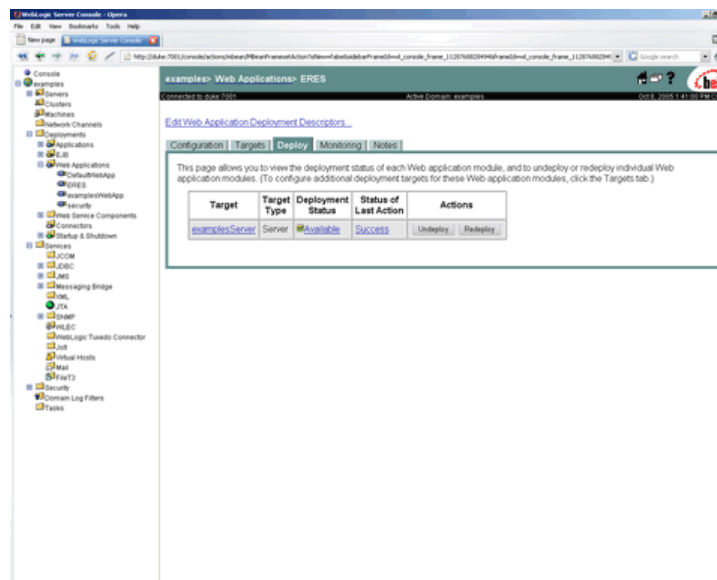
4. Click the *Configure a new web application* link from the above page. This will direct you to install and configure the EDAB web application.



5. From the right panel, browse to the expanded application directory of EDAB, which in this case will be the installation directory of EDAB `<EDAB_INSTALL_DIR>`, and then click the *Select* link to the left of the EDAB directory. This will bring you to the following panel to continue with Steps 3, 4 and 5 of the configuration procedure. From this panel, enter the information for EDAB and choose the targeted server which in this case will be the *exampleServer*.



6. Then press the *Configure and Deploy* button to deploy EDAB.



7. If hot deployment is not enabled in your WebLogic application server, stop and restart the exampleServer.
8. Now go to the EDAB Start page `http://<server>:<port>/<EDAB_CONTEXT>/index.jsp`. From the start page, start the EDAB server.
9. From the EDAB Start page, login as administrator and launch the *Admin Console*.
10. From Admin Console page, go to *Setting Info* tab and change EDAB Servlet Context entry to `/<EDAB_CONTEXT>/servlet`. Please substitute `<EDAB_CONTEXT>` with your actual context for EDAB.
11. Stop and restart the EDAB Server.

9.3.3.1.3. WebLogic Server 7.0 - Quick Deployment

These instructions will accomplish the same effect - deploying EspressoDashboard under the WebLogic *example-Server* - as the previous section. This section explains how to deploy EDAB without using the WebLogic administration console. Users who are not very familiar with WebLogic should use the instructions in the previous section.

1. Modify the <WL_INSTALL_DIR>/samples/domains/examples/config.xml file to add the following entry:

```
<Application Deployed="true" Name="EDAB" Path="<PARENT of
EDAB_INSTALL_DIR>" TwoPhase="true">

    <WebAppComponent Name="EDAB" Targets="examplesServer" URI="EDAB"/>

</Application>
```

Path is the parent directory of <EDAB_INSTALL_DIR>. Application name is the EDAB installation directory name. For example, if you install EDAB under C:/EDAB, this should be "EDAB". WebAppComponent Name is the name of this component. These instructions show how to deploy EDAB with exampleServer. This is declared by the Target attribute in the config.xml file. URI is how to access this web component. This is the context path that will be used to access this web application. "EDAB" is used as the default context path.

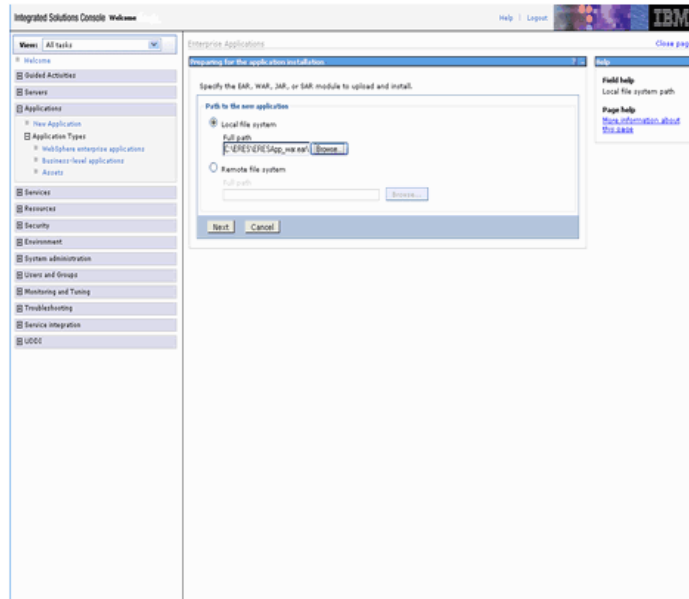
2. Start the WebLogic 7.0 exampleServer by executing the following command: <WL_INSTALL_DIR>/samples/server/config/examples/startExamplesServer.cmd
3. Now go to the EDAB Start page http://<server>:<port>/<EDAB_CONTEXT>/index.jsp. From the start page, start the EDAB server.
4. From the EDAB Start page, login as administrator and launch the *Admin Console*.
5. From Admin Console page, go to *Setting Info* tab and change the EDAB Servlet Context entry to /<EDAB_CONTEXT>/servlet. Please substitute <EDAB_CONTEXT> with your actual context for EDAB.
6. Stop the EDAB Server.
7. Stop and restart Weblogic if necessary.
8. Restart the EDAB Server.

9.3.4. WebSphere

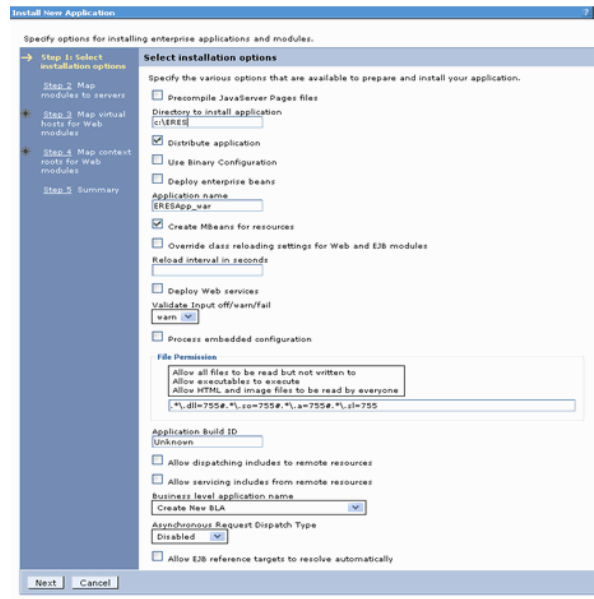
9.3.4.1. WebSphere 7.1

While EspressoDashboard can be deployed in WebSphere 7.1 using several ways, the following instructions show how to do so with minimal changes to WebSphere and EDAB. The instructions assume WebSphere has been installed, started, and running on the same machine as EDAB.

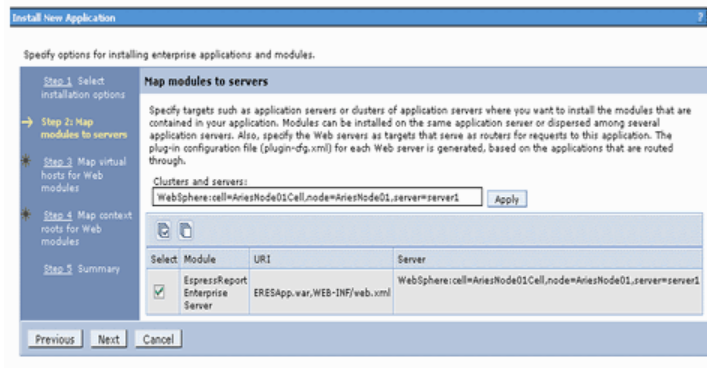
1. Reinstall EDAB under the following directory EDAB/EDABApp_war.ear/EDABApp.war (for example, C:\EDAB\EDABApp_war.ear\EDABApp.war), then enter the host name/IP of the WebSphere machine and the port it is running on when prompted in the EDAB installer. Leave the default EDAB as the servlet context.
2. In order to deploy EDAB under WebSphere, you will need to convert the EDAB web application into a WAR file, called EDABApp.war, using the Java's JAR tool. To do this, make sure that the java/bin directory is in your path and navigate to <EDAB_INSTALL_DIR> in a command window. Then execute the following command:
jar -cvf EDABApp.war *
3. Start the administration console for WebSphere. From the left panel of the administration console, select the *Applications* node and then click the *Install New Applications* link.
4. Select *New Enterprise Application* and then browse to the created war file (which should be EDAB/EDABApp_war.ear/EDABApp.war/EDABApp.war). After selecting EDABApp.war, click on *Next*.



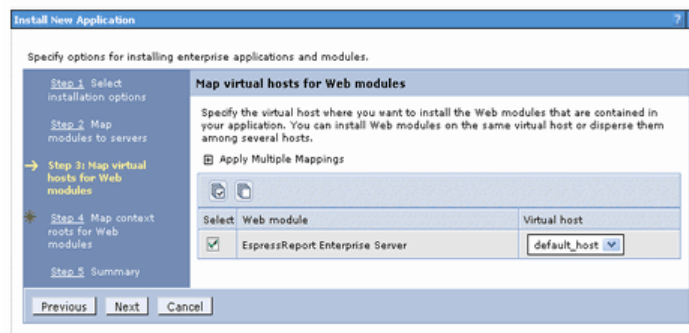
5. When you get asked about how to install the application, select *Fast Path* and click the *Next* button.
6. Under *Directory to install application*, enter the path to /EDAB. For example, if you installed EDAB under c : \EDAB\EDABApp_war.ear\EDABApp.war, enter c : \EDAB. Click *Next*.



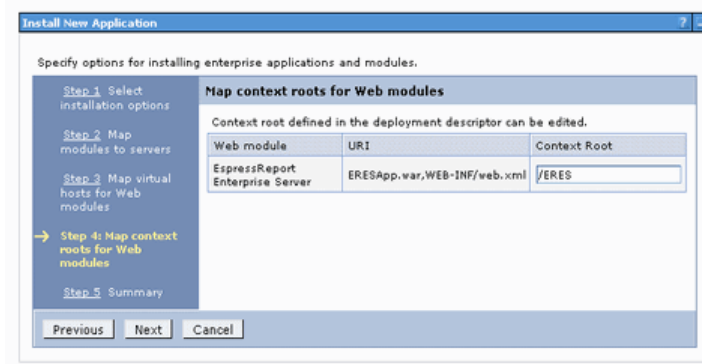
7. Under *Map modules to application servers*, select *Module* and click on *Next*.



8. Under *Map virtual hosts for Web modules*, select *Web module* and click on *Next*.



9. For *Map context root for Web modules*, enter */EDAB* under *Context Root* and click on *Next*.



10. A summary of the installation options is then shown. Click on *Finish* and the WebSphere will now deploy the EDAB web application.

11. If the deployment was successful, finish it by clicking the *save* link on this page.

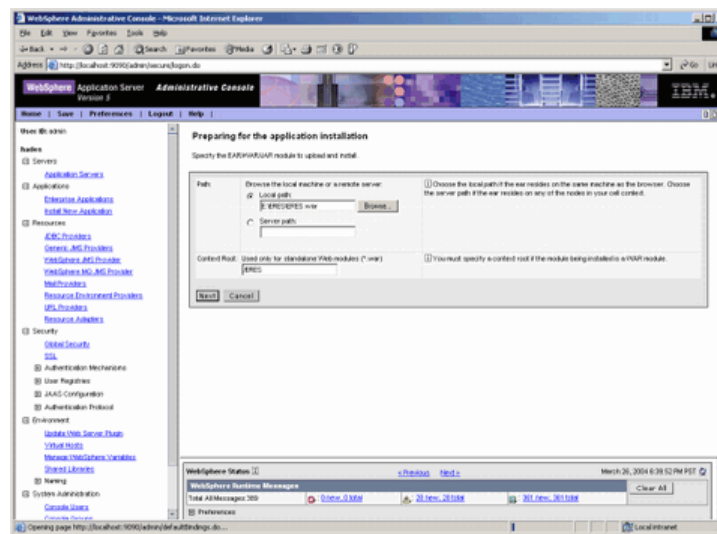
12. Expand *Applications* on the right side and select *WebSphere enterprise applications*. Then select *EDABApp-war* and click on *Start*.

13. Now go to the EDAB Start page `http://<server>:<port>/EDAB/index.jsp`. Note that port is 9080 by default. From the start page, start the EDAB server.

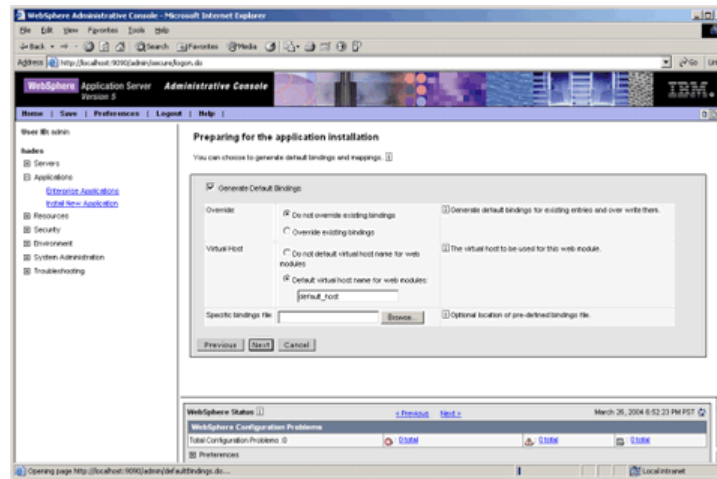
9.3.4.2. WebSphere 5.0/6.1

EspressDashboard can be deployed in WebSphere in a number of ways. The following instructions show how to deploy EDAB under WebSphere Application Server 5.0 or 6.0/6.1 with minimal configuration changes. The instructions assume that WebSphere Application Server has been installed on the system. The location of the WebSphere installation is referenced as <WS_INSTALL_DIR>. WebSphere is also assumed to be started and running.

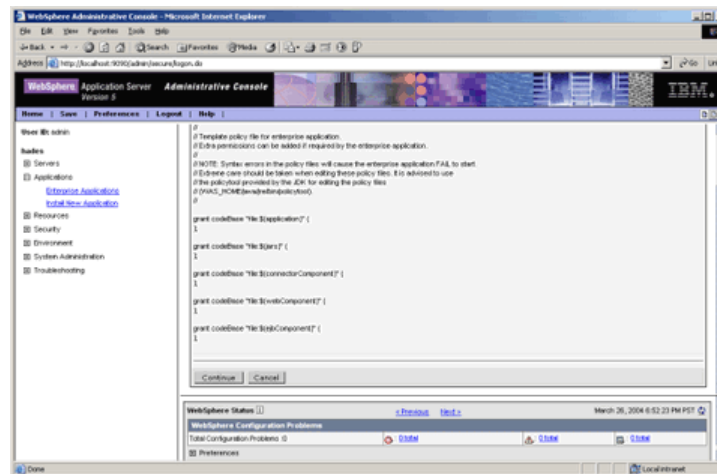
1. Reinstall EDAB under the following directory EDAB/EDABApp_war.ear/EDABApp.war (for example, c:\EDAB\EDABApp_war.ear\EDABApp.war), then enter the host name/IP of the WebSphere machine and the port it is running on when prompted in the EDAB installer. Leave the default EDAB as the servlet context.
2. In order to deploy EDAB under WebSphere, you will need to convert the EDAB web application into a WAR file, called EDABApp.war, using the Java's JAR tool. To do this, make sure that the java/bin directory is in your path and navigate to <EDAB_INSTALL_DIR> in a command window. Next, execute the following command: `jar -cvf EDABApp.war *`
3. Start the administration console for WebSphere. From the left panel of the administration console, select the *Applications* node and then click the *Install New Applications* link.
4. You will be redirected to the following window for preparing the application installation. From the right panel, provide the war file path (which should be EDAB/EDABApp_war.ear/EDABApp.war/EDABApp.war), enter **EDAB** for the context path and then click the *Next* button. Please note that if you are deploying EDAB under WebSphere 6.1, you can skip steps 5 and 6 and continue with step 7.



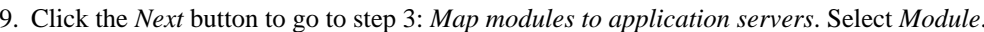
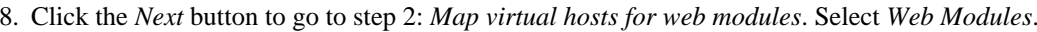
5. In case you do not have any binding specified, select *Generate Default Bindings* in the next page.



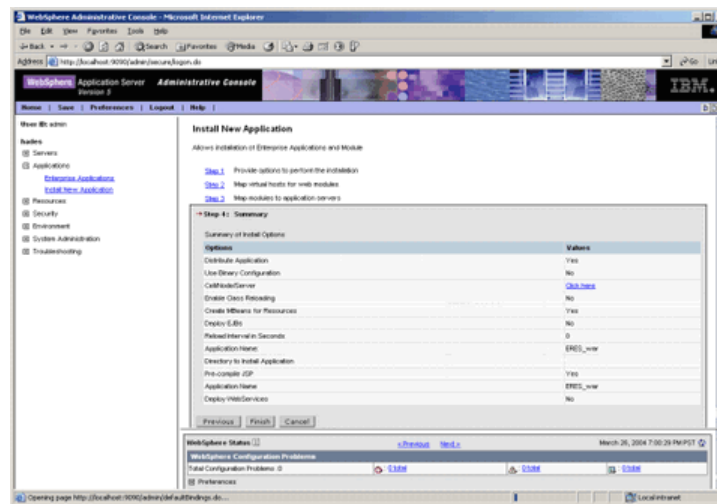
6. Click the *Next* button.



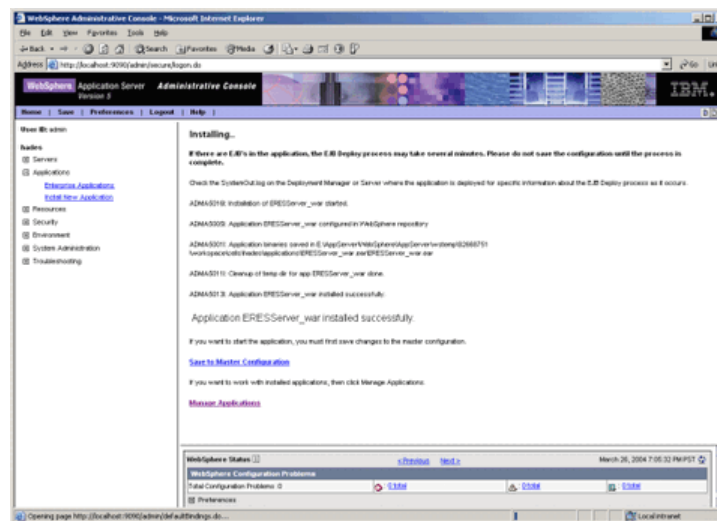
7. Click the *Continue* button. Now you are redirected to the step 1 of the installation: *Provide options to perform the installation*. Under *Directory to install application*, enter the path to /EDAB. For example, if you installed EDAB under `c:\EDAB\EDABApp_war.ear\EDABApp.war`, enter **c:\EDAB**.



10. Click the *Next* button to go to step 4: *Summary of this web application*. Please make sure that all information is correct. If it's not, go back to the previous pages to edit the information.



11. After verifying all informations, click the *Finish* button. WebSphere will now try to deploy the EDAB Web application.



12. If the deployment was successful as shown above, save the changes either by clicking the *save* link in this page, or clicking on the *save* link on top of the administration console. Note that you may have to go under *Applications* in the right frame, expand *Enterprise Applications* and start *EDABApp_war*.

13. Now go to the EDAB Start page <http://<server>:<port>/EDAB/index.jsp>. Note that port is 9080 by default. From the start page, start the EDAB server.

9.3.4.3. WebSphere on z/OS

Deploying ExpressDashboard to WebSphere for z/OS has a few additional steps:

1. Download *EDAB.war* and deploy it on WebSphere (see above section for general WebSphere deployment). You will need to use *viacii* to configure text files that are mentioned in the instructions. Also, since the required jar files are in the *WEB-INF/lib* directory inside the war file, you can skip the steps for adding them as classpath to your servlet container.

2. EDAB requires Java graphics support to run. If you are using JDK 1.5 or higher, the solution is to set it to run in headless mode by specifying the `-Djava.awt.headless=true` flag for the java command that starts your WebSphere. For JDK 1.3 or lower, you can use PJA graphics emulation.
3. Stop and restart WebSphere if necessary. EDAB is now set up and accessible using URL: `http://host:port/EDAB/index.jsp`.

9.3.5. JBoss 3.2.3 (with Tomcat 5.0) / JBoss 4.0.5/6.0.0

The following instructions show how to deploy EspressoDashboard under JBoss 3.2.3 with Tomcat 5.0 / JBoss 4.0.5 / 6.0.0. The instructions assume you have JBoss installed on the system. The location of the JBoss installation is referenced as `<JBOSS_INSTALL_DIR>`.

For reference, the default port is 8080 and the default working directory is `<JBOSS_INSTALL_DIR>/bin`.

1. You need a `jboss-web.xml` deployment descriptor to deploy EDAB. This file should be added to the `<EDAB_INSTALL_DIR>/WEB-INF` directory. This is the same location as `web.xml` file. This deployment descriptor is to specify the EDAB context. The content should look like this:

```
<?xml version="1.0" encoding="UTF-8"?>
<!DOCTYPE jboss-web PUBLIC "-//JBoss//DTD Web Application 2.3//EN"
    "http://www.jboss.org/j2ee/dtds/jboss-web_3_0.dtd">

<jboss-web>

    <context-root>/EDAB</context-root>

</jboss-web>
```

2. Rename the EDAB installation directory `<EDAB_INSTALL_DIR>` to `EDAB.war`, then move the entire `EDAB.war` directory to `<JBOSS_INSTALL_DIR>/server/default/deploy`. This new directory is referenced as `<EDAB_DEPLOY_DIR>`. Next time when JBoss starts, server will automatically deploy the EDAB web application.
3. Open `<EDAB_DEPLOY_DIR>/WEB-INF/classes/QB.properties` file and change the `AutoStart-ServerInstallDir` entry to point to `<EDAB_DEPLOY_DIR>`. Example:
Original entry

```
AutoStartServerInstallDir=<EDAB_INSTALL_DIR>
```

New entry

```
AutoStartServerInstallDir=<EDAB_DEPLOY_DIR>
```

Now you have to change Database URL. If the `DatabaseUrl` entry already exists in your `QB.properties` file, just change it to point to the hsql database under `<EDAB_DEPLOY_DIR>`. Example:

Original entry

```
DatabaseUrl=jdbc\:hsqldb\:<EDAB_INSTALL_DIR>/data/quadbasedb
```

New entry

```
DatabaseUrl=jdbc\:hsqldb\:<EDAB_DEPLOY_DIR>/data/quadbasedb
```

If the entry does not exist in your file, save the changes and open <EDAB_DEPLOY_DIR>/WEB-INF/orguserdb/config.txt file. In the file, modify [database url] entry to point to the hsql database under <EDAB_DEPLOY_DIR>. Example:

Original entry

```
[database url] jdbc:hsqldb:<EDAB_INSTALL_DIR>/data/quadbasedb
```

New entry

```
[database url] jdbc:hsqldb:<EDAB_DEPLOY_DIR>/data/quadbasedb
```

4. Execute run.bat/.sh in the <JBoss_INSTALL_DIR>/bin directory to start the JBoss Server. The EDAB application will be automatically deployed. The deployment information should be printed out on the JBoss console.
5. Go to the EDAB Start page http://<server>:<port>/<EDAB_CONTEXT>/index.jsp to start the EDAB server. EDAB_CONTEXT is defined in your jboss-web.xml file. In this case, it will be EDAB.
6. From the EDAB Start page, login as administrator and launch the *Admin Console*.
7. From Admin Console page, go to *Setting Info* tab and change EDAB Servlet Context entry to /<EDAB_CONTEXT>/servlet. Please substitute <EDAB_CONTEXT> with your actual context for EDAB.
8. Stop and restart the EDAB server.

9.3.6. Oracle Containers for J2EE (OC4J) 10g (9.0.4.0/10.1.3.5)

The following instructions show how to deploy EspressoDashboard under OC4J 10g (9.0.4.0/10.1.3.5). The instructions assume that you have OC4J installed on the system. The location of the OC4J installation is referenced as <OC4J_INSTALL_DIR>

For reference, the default port is 8888 and the default working directory is <OC4J_INSTALL_DIR>/j2ee/home.

1. The easiest way to deploy EDAB is to put EDAB under default-web-app. Copy the entire EDAB directory <EDAB_INSTALL_DIR> to <OC4J_INSTALL_DIR>/j2ee/home/default-web-app. This new directory is referenced as <EDAB_DEPLOY_DIR>.
2. You will need to copy the tools.jar file from your Java development kit to the <OC4J_INSTALL_DIR>/j2ee/home/applib directory (for JSP compilation), or add a library path entry for the file. Check your <OC4J_INSTALL_DIR>/j2ee/home/config/application.xml file. The following entry should exist in the file: <library path=" ../applib"/>
3. Open <EDAB_DEPLOY_DIR>/WEB-INF/classes/QB.properties file and change the AutoStart-ServerInstallDir entry to point to <EDAB_DEPLOY_DIR>. Example:

Original entry

```
AutoStartServerInstallDir=<EDAB_INSTALL_DIR>
```

New entry

AutoStartServerInstallDir=<EDAB_DEPLOY_DIR>

Now you have to change Database URL. If the DatabaseUrl entry already exists in your QB.properties file, just change it to point to the hsql database under <EDAB_DEPLOY_DIR>. Example:

Original entry

DatabaseUrl=jdbc\:hsqldb\:<EDAB_INSTALL_DIR>/data/quadbasedb

New entry

DatabaseUrl=jdbc\:hsqldb\:<EDAB_DEPLOY_DIR>/data/quadbasedb

If the entry does not exist in your file, save the changes and open <EDAB_DEPLOY_DIR>/WEB-INF/orguserdb/config.txt file. In the file, modify [database url] entry to point to the hsql database under <EDAB_DEPLOY_DIR>. Example:

Original entry

[database url] jdbc:hsqldb:<EDAB_INSTALL_DIR>/data/quadbasedb

New entry

[database url] jdbc:hsqldb:<EDAB_DEPLOY_DIR>/data/quadbasedb

4. Backup the web.xml file under <OC4J_INSTALL_DIR>/j2ee/home/default-web-app/WEB-INF, then move all directories and files (except for the /orguserdb/ directory) under the <EDAB_DEPLOY_DIR>/WEB-INF to the <OC4J_INSTALL_DIR>/j2ee/home/default-web-app/WEB-INF directory. This will make all the class files and servlets accessible from OC4J. If necessary, merge the old web.xml file with the web.xml file for EDAB to create a new web.xml file.
5. Start the Oracle server by executing the command `java -jar oc4j.jar` in the <OC4J_INSTALL_DIR>/j2ee/home directory.
6. Go to the EDAB Start page `http://<server>:<port>/<EDAB_CONTEXT>/index.jsp` to start the EDAB server. The EDAB_CONTEXT will be the directory name of <EDAB_DEPLOY_DIR>.
7. From the EDAB Start page, login as administrator and launch the *Admin Console*.
8. From Admin Console page, go to *Setting Info* tab and change the EDAB Servlet Context entry to `/servlet/`.
9. Stop and restart the EDAB server.

9.3.7. Oracle Application Server 10g R3 (10.1.3)

The following instructions show how to deploy EspressDashboard under Oracle Application Server 10g R3. The instructions assume that you have Oracle AS installed on the system with standard installation. The location of the Oracle AS installation is referenced as <O10g_INSTALL_DIR>

1. Install EDAB to the <O10g_INSTALL_DIR>/j2ee/home/default-web-app/ or
2. Copy the EDAB directory to <O10g_INSTALL_DIR>/j2ee/home/default-web-app/.

3. The new location for the EDAB directory is referenced as <EDAB_HOME>.
4. If the database is set up with direct path, open <EDAB_HOME>/WEB-INF/classes/QB.properties and modify the database info as shown below.

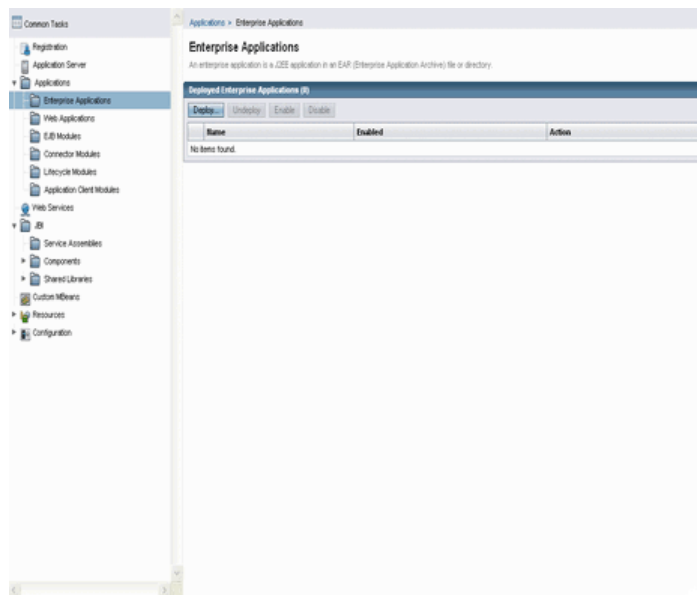
```
DatabaseUrl=jdbc:hsqldb:<EDAB_HOME>/data/quadbasedb
```

5. Move all directories and files (except for the /orguserdb/ directory) from <EDAB_HOME>/WEB-INF to <O10g_INSTALL_DIR>/j2ee/home/default-web-app/WEB-INF/ directory.
6. Add <EDAB_HOME>/lib/EDABServer.jar to the CLASSPATH user variable.
7. Go to <http://<machine>:80/j2ee/EDAB> and start EDAB server.
8. In the admin console, change the servlet context to /j2ee/servlet.
9. Restart EDAB Server

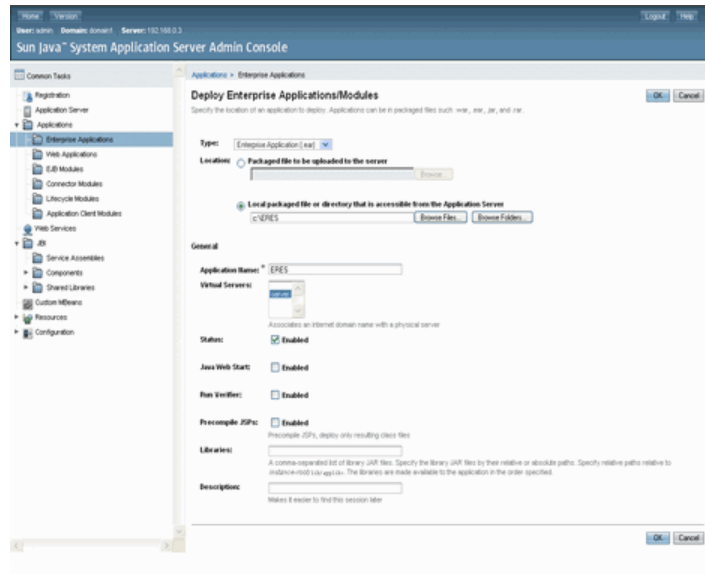
9.3.8. GlassFish Server 3.0.1

The following instructions show how to deploy EspressoDashboard under GlassFish Server 3.0.1. The instructions assume that the application server and EDAB are installed on the same machine and that GlassFish is up and running.

1. Log in to the GlassFish Admin Console. Click on *Applications* or (in Glasfish v2) expand *Applications* on the left and then click on *Enterprise Applications*.



2. Click on *Deploy* to deploy a new enterprise application.
3. Select *Browse Folders* and select <EDAB_INSTALL_DIR>.
4. Enter **EDAB** under *Application Name* and select the server under *Virtual Server*.

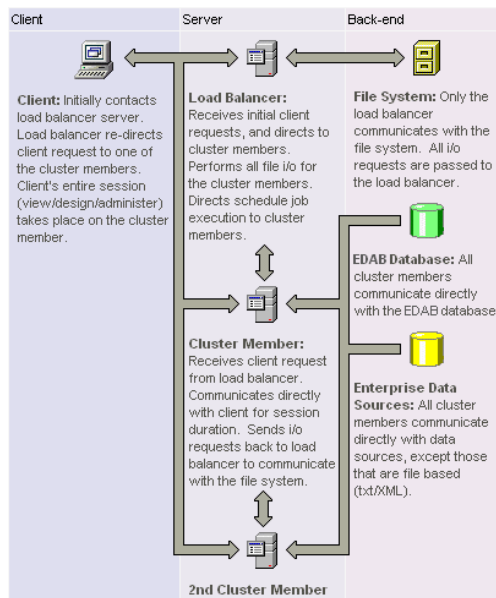


5. Click on the *OK* button to finish deployment.
6. In the Enterprise Application window, select the EDAB application you have created and then select *Launch*. This will enable the enterprise application (i.e. start it) and launch it in a separate browser window. You can now start using EDAB.

9.4. Clustering/Load Balancing

Scalability is an important concern for enterprise reporting. One of the best ways to easily scale an application while eliminating server down-time is to deploy the application in a clustered environment. To provide for maximum scalability, EDAB can be deployed in a clustered environment.

When running EDAB in a cluster, one server acts as the load balancing machine, receiving incoming requests and routing them to other servers in the cluster where they are processed. This following diagram illustrates EDAB running in a clustered environment.



EDAB Clustering Diagram

All incoming requests are handled by the load balancing server, which will redirect the requests to one of the cluster members. The client's entire session will then take place on the cluster member. Users have an option to either make the load balancer one of the cluster members, or use it solely as the balancer. Each of the cluster members will directly access data sources and read/write information from the EDAB Server, but only the load balancer will access the file system. All i/o requests from the cluster members are routed through the load balancer. This ensures that users are working with the same set of deployed dashboards, regardless of which cluster member they are currently using.

The following chapter details how to deploy EDAB cluster in different application server environments.

9.4.1. Tomcat 5.0

In this section, you will be modifying the following:

- <TomcatInstallDir>/conf/server.xml
- Admin Console
- <TomcatInstallDir>/webapps/balancer/WEB-INF/web.xml
- <TomcatInstallDir>/webapps/balancer/WEB-INF/config/rules.xml
- <EDABInstallDir>/index.html
- <EDABInstallDir>/WEB-INF/web.xml

The first step in setting up EDAB cluster is to deploy EDAB to all the servers in the cluster. You will need to install EDAB on all the machines and deploy it in the Tomcat instance. You can follow the instructions in Section 9.3.1 - Tomcat 4.1/5.x/6.0.x/7.0.x.

Next, modify the server.xml under <TomcatInstallDir>/conf on the load balancing machine. Add the following configuration to this file to enable clustering.

```
<Cluster className="org.apache.catalina.cluster.tcp.SimpleTcpCluster"

    managerClassName="org.apache.catalina.cluster.session.SimpleTcpReplicationManager"
        expireSessionsOnShutdown="false"
        useDirtyFlag="true">

    <Membership
        className="org.apache.catalina.cluster.mcast.McastService"
        mcastAddr="228.0.0.4"
        mcastPort="45564"
        mcastFrequency="500"
        mcastDropTime="3000"/>

    <Receiver
        className="org.apache.catalina.cluster.tcp.ReplicationListener"
        tcpListenAddress="auto"
        tcpListenPort="4000"
        tcpSelectorTimeout="100"
        tcpThreadCount="6"/>

    <Sender
        className="org.apache.catalina.cluster.tcp.ReplicationTransmitter"
        replicationMode="pooled"/>
```

```
<Valve className="org.apache.catalina.cluster.tcp.ReplicationValve"
      filter=".*\.(gif|.*\.(js|.*\.(jpg|.*\.(htm|.*\.(html|.*\.(txt|"/>

</Cluster>
```

There is a default clustering implementation in the `server.xml` file, but it is commented out. For the load balancer server, you will need to use this slightly different implementation instead.

For all the other servers in the cluster, you can use the default cluster configuration in the `server.xml` file. You just need to uncomment the default configuration. In the default configuration, the default `tcpListenPort` is 4001.

If any of the cluster members have more than one IP address, it may be necessary to add `mcastBindAddress` parameter to the `<Membership>` tag. This argument allows you to specifically set an IP for the cluster service.

If you are using HSQL as the EDAB database, you will need to change the configuration of this database on the load balancer machine. HSQL usually runs as application process and cannot be accessed from other machines. In order to make it available to other servers in the cluster, it needs to be configured to run in client-server mode. For setup instructions, please see Section 1.3.2.1.2 - Running HSQL in Client-Server Mode.

Next, you will need to log in as **Admin** and enter the *Admin Console*. Go to Setting Info → Clustering & Load Balancer Settings and add the entries for `Server Host`, `Server Port Number` and `Cluster Member List`.

`Server Host: 192.168.0.8`

`Server Port Number: 8080`

`Cluster Member List: 192.168.0.8:8080`
`192.168.0.10:8080`
`192.168.0.16:8080`

The `Server Host` and `Server Port Number` entries should be the address and port number for the load balancing server. The first entry under the `Cluster Member List` heading should also be the load balancer server. It needs to be listed under this argument even if the load balancer is not acting as a cluster member. The order of other servers should be same in the Admin Console for each EDAB instance.

Next, you will need to deploy the balancer web application to the load balancer server. This is a standalone Web application for Tomcat that is included in the EDAB installation under `<EDABInstallDir>/Clustering/Tomcat/balancer`. To deploy the Web application, copy the contents of the `/balancer/` directory to the load balancer server and place it under `<TomcatInstallDir>/webapps/`. The `web.xml` file for the balancer application, now under `<TomcatInstallDir>/webapps/balancer/WEB-INF`, maps `LoadBalancer` to a `BalancerFilter` and gives the definition of the `BalancerFilter` with its class file and `rules.xml` file.

```
<web-app>

    <!-- BalancerFilter definition -->

    <filter>

        <filter-name>BalancerFilter</filter-name>
        <filter-class>org.apache.webapp.balancer.BalancerFilter</filter-class>
        <init-param>
            <param-name>configUrl</param-name>
            <param-value>/WEB-INF/config/rules.xml</param-value> </init-
param>
```



```
</filter>

<!-- BalancerFilter mapping -->

<filter-mapping>

    <filter-name>BalancerFilter</filter-name>
    <url-pattern>/LoadBalancer</url-pattern> </filter-mapping>

</web-app>
```

Rules.xml defines how the BalancerFilter should redirect requests to server instance. The following is an example of ruler.xml file and it is based on RoundRobinRule. RoundRobinRule.class and several other rule classes are included in the balancer package.

```
<rules>

    <!-- Redirect to server instance based on RoundRobinRule -->
    <rule className="org.apache.webapp.balancer.rules.RoundRobinRule"
        serverInstance="1"
        maxServerInstances="3"
        tcpListenAddress="127.0.0.1"
        tcpListenPort="4000"
        redirectUrl="http://192.168.0.8:8080/EDAB/index.jsp" />

    <rule className="org.apache.webapp.balancer.rules.RoundRobinRule"
        serverInstance="2"
        maxServerInstances="3"
        tcpListenAddress="127.0.0.1"
        tcpListenPort="4001"
        redirectUrl="http://192.168.0.10:8080/EDAB/index.jsp" />

    <rule className="org.apache.webapp.balancer.rules.RoundRobinRule"
        serverInstance="3"
        maxServerInstances="3"
        tcpListenAddress="127.0.0.1"
        tcpListenPort="4001"
        redirectUrl="http://192.168.0.16:8080/EDAB/index.jsp" />

</rules>
```

Each defined server instance in the rules.xml file should point to one of the EDAB cluster members. If you want the load balancer server to also function as a cluster member, include it as a server instance here. You need to make sure that the tcpListenPort in the rules.xml file matches the port specified in the server.xml file for that server. The RoundRobinRule, as depicted above, routes each incoming request to the next cluster member in sequence using the default tcpListenPort. The balancer application also includes RandomRedirectRule option. This will randomly choose a cluster member for each incoming request.

Finally, you need to create a new index.html (instead of the current index.jsp) page in the <EDABInstallDir> directory on the load balancer server. The page can be blank, however, it needs to have the following meta tag at the top of the page:

```
<meta http-equiv="refresh" content="0; URL=http://servername:8080/balancer/
LoadBalancer" />
```

This will redirect all incoming requests to the balancer application, where it will be routed to one of the cluster members following the logic defined in the `rules.xml` file. Replace `servername` with the domain or IP address of your load balancer server in the URL. In order to make the new `index.html` page the default start page for the load balancer machine, you have to edit the `web.xml` file under `<EDABInstallDir>/WEB-INF` and change the following entry:

```
<welcome-file-list>

    <welcome-file>index.jsp</welcome-file>

</welcome-file-list>
```

to

```
<welcome-file-list>

    <welcome-file>index.html</welcome-file>

</welcome-file-list>
```

9.4.1.1. Running EDAB in a Tomcat Cluster

Once you complete the deployment, start Tomcat on your load balancing server and then do the same on all other servers in the cluster. To get to the EDAB home page, call the `index.html` file that you created on the load balancer server. You will then be redirected to the Start page on one of the cluster machines. When you start the EDAB server on this cluster machine, it will automatically start the EDAB server on all other machines in the cluster.

Once you have been redirected to a particular server through the load balancer, your entire session will take place on that machine. However, all file I/O will take place on the load balancing machine. This ensures that all users will interact with the same set of reports, charts and dashboards which are stored in the file system. All other system information is stored in the EDAB database, which can be accessed by any server in the cluster.

9.4.1.2. Set up EDAB clustering using Apache Web Server and Tomcat

The following instructions will help you setup EDAB using Apache Web Server as the load balancer and Tomcat Clusters. In this section, you will be modifying the following files:

- `<ApacheInstallDir>/conf/httpd.conf`
- `<ApacheInstallDir>/conf/workers.properties`
- `<TomcatInstallDir>/conf/server.xml`

1. Setup EDAB cluster with Tomcat using the instructions in Section 9.4.1 - Tomcat 5.0.

2. Apache HTTP server:

- a. Download Apache HTTP server 2.2.4 from The Apache HTTP Server Project (<https://httpd.apache.org/> [<https://httpd.apache.org/>])
- b. Run the installer and select standard installation.



Note

Do not install to the default Program Files directory as space in directory name could cause problems later. Install directory name must not have space in it, for example "Apache2.2" is fine.

- c. Open the Apache Server Monitor and start the web server if it's not already running.
 - d. Point your browser to `http://localhost/` to verify that Apache is running on port 80.
 - e. Stop Apache.
3. Apache-Tomcat connector
- a. Download Mod JK Tomcat connector from <https://tomcat.apache.org/download-connectors.cgi>.

**Note**

You want to download the binary - click on JK 1.2 Binary Releases → win32 → jk-1.2.21 → `mod_jk-apache-2.2.4.so`

- b. Copy the `mod_jk-apache-2.2.4.so` to the modules directory in your Apache installation.
- c. Rename it to `mod_jk.so`
- d. Open `httpd.conf` in the `conf` directory of your Apache installation in a text edit and add the following line at the end of the LoadModule statements: `LoadModule jk_module modules/mod_jk.so`
- e. Create a file called `workers.properties` in the `conf` directory. Add these lines to it:

```
workers.tomcat_home=<tomcat_install_directory>
workers.java_home=<jdk_install_directory>
worker.list=balancer

worker.worker0.port=8009
worker.worker0.host=192.168.0.8
worker.worker0.type=ajp13
worker.worker0.lbfactor=1

worker.worker1.port=9009
worker.worker1.host=192.168.0.8
worker.worker1.type=ajp13
worker.worker1.lbfactor=1

worker.worker2.port=8009
worker.worker2.host=192.168.0.10
worker.worker2.type=ajp13
worker.worker2.lbfactor=1

worker.worker3.port=9009
worker.worker3.host=192.168.0.10
worker.worker3.type=ajp13
worker.worker3.lbfactor=1

worker.balancer.type=lb
worker.balancer.balance_workers=worker0,worker1,worker2,worker3
worker.balancer.method=B
worker.balancer.sticky_session=True
```

We have specified the worker list to a single worker called balancer and specified that the worker type of balancer is 'lb' or load balancer. The workers it manages are `worker0`, `worker1`, `worker2` and `worker3`. And we set the balance method to 'B' or balance by busy factor. Apache will delegate the next

request to the Tomcat instance which is the least busy. Please note that there are several options for the balance method - consult the Apache/Tomcat documentation, which lists the options for workers properties, to help you decide the best method for your type of application.

**Note**

Port numbers must match the Connector port (with protocol="AJP/1.3") defined in Tomcat's `server.xml`. 8009 is the original port and 9009 for the second instance of Tomcat on the same machine.

- f. Edit the `server.xml` for each instance of Tomcat and add a `jvmRoute` attribute to the Engine element:

```
<Engine name="Catalina" defaultHost="localhost" jvmRoute="worker0">
```

for the first instance and

```
<Engine name="Catalina" defaultHost="localhost" jvmRoute="worker1">
```

for the second.

```
<Engine name="Catalina" defaultHost="localhost" jvmRoute="worker2">
```

for the third.

```
<Engine name="Catalina" defaultHost="localhost" jvmRoute="worker3">
```

for the last.

- g. Specify the worker properties in `httpd.conf`:

Add these lines just after the LoadModule definitions:

```
JkWorkersFile c:/apache2.2/conf/workers.properties
JkLogFile c:/apache2.2/mod_jk.log
JkLogLevel info
JkLogStampFormat "[%a %b %d %H:%M:%S %Y] "
JkOptions +ForwardKeySize +ForwardURISCompat -ForwardDirectories
JkRequestLogFormat "%w %V %T"
#use apache balancer
JkMount /EDAB balancer
JkMount /EDAB/* balancer
```

4. Start up

- a. Startup Tomcat50, Tomcat51, Tomcat52 and Tomcat53

- b. Start Apache Server

- c. Go to `http://192.168.0.8:8080/EDAB/index.html` and start the EDAB server

- d. Go to `http://192.168.0.8/EDAB/index.jsp`



Note

The internal Tomcat server (<http://192.168.0.8:8080/EDAB>) is for Administrator to start and stop EDAB server and should not be available to other users. Other users must go through Apache web server (<http://192.168.0.8/EDAB/>) and their requests will be forwarded to Tomcat cluster which is invisible to them. Apache HTTP server will serve loadbalancer.

The above example creates four instances of EDAB on two separate machines (192.168.0.8 and 192.168.0.10). To change the number of clusters and machines, modify the worker configuration in step 3.

9.4.2. JBoss 3.2.5 (with Tomcat 5.0)

In this section, you will be modifying the following files:

- Admin Console
- `<JBossInstallDir>/server/default/deploy/loadbalancer.sar/META-INF/jboss-service.xml`
- `<EDABInstallDir>/index.html`

The first step in setting up an EDAB cluster is to deploy EDAB to all servers in the cluster. You will need to install EDAB on all machines and deploy it in the JBoss instance. You can follow the instructions in Section 9.3.7 - Oracle Application Server 10g R3 (10.1.3).

If you are using HSQL as the EDAB database, you will need to change the configuration of this database on the load balancer machine. HSQL usually runs as an application process and cannot be accessed from other machines. In order to make it available to other servers in the cluster, it needs to be configured to run in client-server mode. For setup instructions, please see Section 1.3.2.1.2 - Running HSQL in Client-Server Mode.

Next, you will need to log in as **Admin** and enter the *Admin Console*. Go to Setting Info → Clustering & Load Balancer Settings and add the entries for Server Host, Server Port Number and Cluster Member List.

Server Host: 192.168.0.8

Server Port Number: 8080

Cluster Member List: 192.168.0.8:8080
192.168.0.10:8080

The Server Host and Server Port Number entries should be the address and port number for the load balancing server. The first entry under the Cluster Member List heading should also be the load balancer server. It needs to be listed under this argument even if the load balancer is not acting as a cluster member. The order of other servers should be same in the *Admin Console* for each EDAB instance.

Next, you will need to deploy the balancer web application to the load balancer server. This is a standalone web application for JBoss that is included in the JBoss installation under `<JBOSS_INSTALL_DIR>\docs\examples\varia\loadbalancer`. To deploy the web application, copy the contents of `/loadbalancer.sar/` directory to the load balancer server and place it under JBoss deploy directory (`<JBOSS_INSTALL_DIR>\server\default\deploy` if you follow the instructions from this manual).

Edit `loadbalancer.sar/META-INF/jboss-service.xml` to match your configuration. Modify `<host-url>` and add new `<host>` so the configuration file contains all cluster members. The example below shows the configuration for two clusters. Value of `<lb-factor>` isn't important because it is not used in this JBoss version.

```
<hosts>

    <host>
        <host-url>http://192.168.0.8:8080/EDAB</host-url>
        <lb-factor>1</lb-factor> </host>
    <host>
        <host-url>http://192.168.0.10:8080/EDAB</host-url>
        <lb-factor>2</lb-factor> </host>

</hosts>
```

Uncomment the first Monitor service in the `jboss-service.xml`.

```
<!-- Monitor Services -->
<!-- A monitor that only checks that a given path is reachable -->
<mbean code="org.jboss.web.loadbalancer.monitor.SimpleMonitorService"
    name="jboss.web.loadbalancer:service=Monitor">

    <depends optional-attribute-
name="Scheduler">jboss.web.loadbalancer:service=Scheduler</depends>
    <attribute name="Interval">15000</attribute>
    <attribute name="Timeout">20000</attribute>
    <attribute name="Path">/</attribute>

</mbean>
```

You can find a sample `jboss-service.xml` in `<EDAB_DEPLOY_DIR>/Clustering/JBoss`. Or you can take a look at `<JBOSS_INSTALL_DIR>\docs\examples\varia\loadbalancer\loadbalancer.-doc` for more detailed description of the load balancer application and its configuration files.

If you use Tomcat 5.0 plugin, you must remove the `<JBOSS_DEPLOY_DIR>/jbossweb-tomcat50.sar/ROOT.war` directory because the load balancer also wants to be root.

Finally, you need to modify `index.html` page in `<EDAB_DEPLOY_DIR>` directory on all cluster members. The page can be blank, but it needs to have the following meta tag at the top:

```
<meta http-equiv="Refresh" content="0; url=http://machine:port/EDAB/
index.jsp">
```

Replace `machine/port` with IP address/port of the machine the cluster member is running on. EDAB is EDAB context on the cluster member. Make the `index.html` page the default start page for all cluster members. You can do this by editing the `web.xml` file under `<EDAB_DEPLOY_DIR>/WEB-INF` on all cluster members. Change the following entry:

```
<welcome-file-list>

    <welcome-file>index.jsp</welcome-file>

</welcome-file-list>
```

to

```
<welcome-file-list>
```

```
<welcome-file>index.html</welcome-file>

</welcome-file-list>
```

9.4.2.1. Running EDAB in JBoss Cluster

Once you complete the deployment, start JBoss on all servers in the cluster and then start JBoss on your load balancing server. Do not forget to start the HSQL database server first (if you are using it). To get to the EDAB home page, open `http://machine:port`, where machine is your load balancing server and port is the port JBoss is running on (8080 by default). You will then be redirected to the Start page on one of the cluster machines. When you start the EDAB server on this cluster machine, it will automatically start the EDAB server on all other machines in the cluster.

Once you have been redirected to a particular server through the load balancer, your entire session will take place on that machine. However, all file I/O will take place on the load balancing machine. This ensures that all users will interact with the same set of reports, charts and dashboards which are stored in the file system. All other system information is stored in the EDAB database, which can be accessed by any server in the cluster.

9.4.3. JRun 4 (with Apache Web server)

In this section, you will be modifying the following files:

- Admin Console
- `<JRUNInstallDir>\lib\wsconfig\1\jrunserver.store`
- `<EDABInstallDir>/index.html`
- `<EDABInstallDir>/WEB-INF/web.xml`

The first step in setting up an EDAB cluster is to deploy EDAB to all servers in the cluster. You will need to install EDAB on all machines and deploy it in the JRun instance. You can follow the instructions in Section 9.3.4 - WebSphere. EDAB needs to be installed in the same context on all cluster members.

If you're using HSQL as the EDAB database, you will need to change the configuration of this database on the load balancing machine. HSQL usually runs as application process and cannot be accessed from other machines. In order to make it available to the other servers in the cluster, it needs to be configured to run in client-server mode. For setup instructions, please see Section 1.3.2.1.2 - Running HSQL in Client-Server Mode.

Next, you will need to log in as **Admin** and enter the *Admin Console*. Then go to Setting Info → Clustering & Load Balancer Settings and add the entries for Server Host, Server Port Number, and Cluster Member List.

Server Host: 192.168.0.8

Server Port Number: 8080

Cluster Member List: 192.168.0.8:8080
 192.168.0.10:8080
 192.168.0.16:8080

The Server Host and Server Port Number entries should be the address and port number for the load balancing server. The first entry under the Cluster Member List heading should also be the load balancing server. It needs to be listed under this argument even if the load balancer is not acting as a cluster member. The order of other servers should be same in the Admin Console for each EDAB instance.

Use the following steps to complete the JRun deployment:

1. This configuration uses Apache web server as a load balancing server. You will need to install it on the load balancing machine.
2. Next, go to the JRun admin console on the load balancing server and log in. The default port of admin console is 8000.
3. Click *Register Remote Server* at the top of the page. Enter the IP address, server, and JNDI port of the cluster member which is hosted on a machine other than the load balancer. Repeat this step for all cluster members which are hosted on other machines.
4. Then click *Create New Cluster* at the top of the page. Enter the name of the cluster and click *next*. Select all cluster members on the next page. Then click *next* and *done*.
5. Now run all cluster members and run the JRun Web Server Configuration Tool. Here click *Add*. In the following dialog, select the previously created cluster in the *JRun server* field. Select Apache web server and insert Apache configuration directory (<APACHE_INSTALL_DIR>\conf). Then click *OK* and close the Web Server Configuration.
6. Open the <JRUN_INSTALL_DIR>\lib\wsconfig\1\jrunserver.store file. There will be a similar entry to the one below:

```
proxyservers=192.168.0.8:51000
```

Add all cluster members to this file (separate them with semicolon “;”). The port is the cluster member proxy port (51000 by default). You can find this port in the admin console.

```
proxyservers=192.168.0.8:51000;192.168.0.10:51000
```

Finally, you need to modify `index.html` page in <EDAB_INSTALL_DIR> directory on all cluster members. The page can be blank, but it needs to have the following meta tag at the top of the page:

```
<meta http-equiv="Refresh" content="0; url=http://machine:port/EDAB/index.jsp">
```

Replace machine/port with IP address/port of the machine the cluster member is running on (port is 8100 by default). EDAB is the EDAB context on the cluster member. Make the `index.html` page the default start page for all cluster members. You can do this by editing the `web.xml` file under <EDAB_INSTALL_DIR>/WEB-INF on all cluster members. Change the following entry:

```
<welcome-file-list>
    <welcome-file>index.jsp</welcome-file>
</welcome-file-list>
```

to

```
<welcome-file-list>
    <welcome-file>index.html</welcome-file>
</welcome-file-list>
```


9.4.3.1. Running EDAB in a JRun Cluster

Once you complete the deployment, start JRun on all servers in the cluster and then start Apache on your load balancing server. Do not forget to start the HSQL database server first (if you are using it). To get to the EDAB home page, open `http://machine:port`, where `machine` is your load balancing server and `port` is the port Apache is running on (80 by default). You will then be redirected to the Start page on one of the cluster machines. When you start the EDAB server on this cluster machine, it will automatically start the EDAB server on all other machines in the cluster.

Once you have been redirected to a particular server through the load balancer, your entire session will take place on that machine. However, all file I/O will take place on the load balancing machine. This ensures that all users will interact with the same set of reports, charts and dashboards which are stored in the file system. All other system information is stored in the EDAB database, which can be accessed by any server in the cluster.

9.4.4. WebSphere 6.0

In this section, you will be modifying the following files:

- Admin Console

To deploy EDAB in a WebSphere cluster, you will first need to have a WebSphere cluster setup (using HTTP Server). We will call your deployed WebSphere cluster `<Cluster_Name>`

Next, you will need to deploy EDAB to each of the WebSphere cluster members. To do this, you will need to WAR the `/EDAB/` directory and deploy it using the WebSphere administration console. or more information about deploying EDAB in WebSphere, see Section 9.3.6 - Oracle Containers for J2EE (OC4J) 10g (9.0.4.0/10.1.3.5). Follow the steps in this section until you get to *Map modules to servers* step. In this step, map the EDAB Web application to the `<Cluster_Name>` and your HTTP server.

Make sure that you deploy EDAB on all cluster members using the same context.

Next, you will need to log in as **Admin** and enter the *Admin Console*. Go to Setting Info → Clustering & Load Balancer Settings and add the entries for `Server Host`, `Server Port Number`, `Cluster Member List`, `Member Host` and `Member Port Number`.

`Server Host: 172.26.35.12`

`Server Port Number: 9080`

`Cluster Member List: 172.26.35.12:9080`
`172.26.35.13:9080`
`172.26.35.14:9080`
`172.26.35.15:9080`

`Member Host: 172.26.35.15`

`Member Port Number: 9080`

The `Server Host` and `Server Port Number` entries should be the address and port number for the load balancing server. The first entry under the `Cluster Member List` should also be the load balancer server. It needs to be listed under this argument even if the load balancer is not acting as a cluster member. The order of the other servers should be the same in the Admin Console for each EDAB instance. The `Member Host` entry should be the IP address of the current machine, and the `Member Port Number` entry should be the server port number for the current machine. These entries will change for each cluster member.

In the WebSphere administration console, click on Servers → WebServers → Generate Plug-in and Propagate Plug-in. The re-start the HTTP Server.

Next, start the WebSphere cluster. Navigate to the `index.jsp` page on the load balancer machine, and start the EDAB Server. Note that because the IBM HTTP server does the balancing, you do not need to change the start page

for EDAB. Users can simply start their session by pointing to the `index.jsp` page on the load balancer machine. The HTTP server on that machine will re-direct their requests to the cluster members.

Chapter 10. Internationalization

10.1. Internationalizing EDAB

EDAB provides many different features that allows you to generate reports for just about any locale and language. Different internationalization features can require different system or setting configurations depending on your specific requirements.

10.1.1. Specifying Locales

EDAB allows different time zones and locales reports and charts. They are not limited to the locale on the machine where they are created. The locale can be set through API right before the report or chart is run.

10.1.1.1. Locale-Specific Formatting

For data formatting (date and numeric), EDAB allows you to set locale-specific formatting for report elements. Locale-specific formatting allows data elements to be displayed in the correct format for the particular locale that is being used for the report. Locale-specific options are available in the data formatting dialog.

The locale for report and charts, as well as the time zone, can be set at run-time through the API. Note that this only effects the date, the time, and the data formatting.

10.1.1.2. Setting Time Zones and Locales Using the API

To set the time zone or locale, you can use the methods in QbReport/QbChart (applies to the entire report/chart) or use `LocaleDateTimeFormat` and `LocaleNumericFormat` objects (applies to a specific object in the report/chart). The following code fragments shows how to do this with two different approaches.

```
report.setLocale(Locale.UK);  
report.setTime zone(time zone.getTime zone("GMT"));
```

Here, the formats are applied to the entire report. Alternatively, you can apply the format to specific data columns as follows:

```
LocaleNumericFormat currencyFormat =  
    LocaleNumericFormat.getCurrencyInstance();  
currencyFormat.setLocale(Locale.UK);  
int columnIndex = 2; //column 2 is the "Unit Price" column  
report.getTable().getColumn(columnIndex).setDataFormat(currencyFormat);  
  
LocaleDateTimeFormat dateFormat = LocaleDateTimeFormat.getDateInstance();  
dateFormat.setTime zone(time zone.getTime zone("GMT"));  
report.getPageFooter().getData(1).setDataFormat(dateFormat);
```

10.1.2. Language and Encoding

EDAB includes a set of features that removes limitations caused by language differences. By utilizing a simple to use interface, it is possible to translate all text, buttons, and menus within EDAB to a different language. Through some basic configuration, you will be able to import, view, and type characters in a foreign language, as well as save and export reports in that language.

10.1.2.1. EDAB Language Translation

Internationalization is supported through the use of an `.xml` file, `EDAB_Language.xml` (located in the <EDAB installation directory>). A GUI is provided to work with the `EDAB_Language.xml` file and replace the lines of English with those of another language.

The `EDAB_Language.xml` file has the following structure:

```
<Product name="REPORTDESIGNER" dir="quadbase/reportdesigner/designer">

    <File name="ReportMenubar.java">
        <CODE>File</CODE>
        <TEXT>File</TEXT>
        <CODE>New</CODE>
        <TEXT>New</TEXT>
        ... </File>

</Product>
```

The file has the following tree structure: Product (Directory) → File → text. You can easily search the product, file, or text you want to translate and simply replace the Text between `<Text>` and `</Text>` with the translation. EDAB will replace `<Code>` with `<Text>` in the Java Swing UI and/or in the EDAB web application. The translation GUI will list all products and each product will list codes that need translation. A same code (for example, File) may be listed under several products to maintain the completeness of each product. However, the translation for a code is duplicated across all products. The same code cannot have two different translations for different products.

The `EDAB_Language.xml` file is located in the install directory. It is recommended that you make a copy (of `EDAB_Language.xml`) and then use the GUI provided (on the copy) to put in the translation without touching the xml file directly. To utilize the user interface, navigate to the EDAB root directory and enter the following command:

```
java -classpath "./lib/EDABOrganizer.jar;."
quadbase.internationalization.TranslateWizard -file:<fileName> -
enc:<encoding>
```

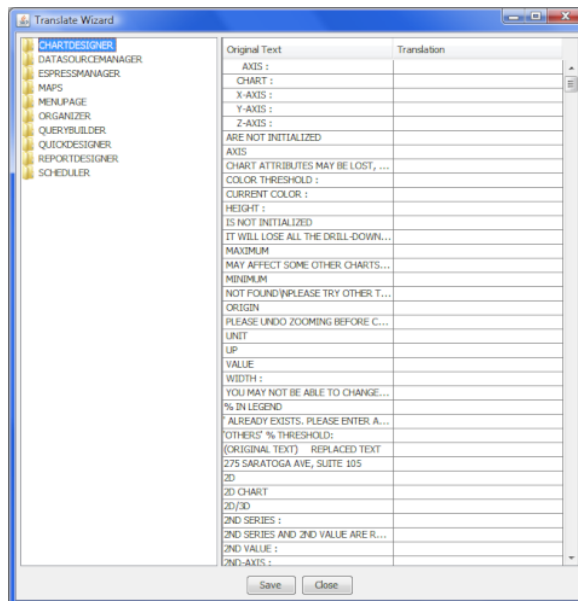
where `<fileName>` represents the xml file that will store the translation and `<encoding>` represents the encoding for the translation. For correct results, the proper encoding must be specified. If the file name and encoding are not provided, the default file name (`EDAB_Language.xml`) and default encoding (Cp1252) will be used.

An example of the above command is given below:

```
java -classpath "./lib/EDABOrganizer.jar;."
quadbase.internationalization.TranslateWizard -file:Chinese.xml -enc:gbk
```

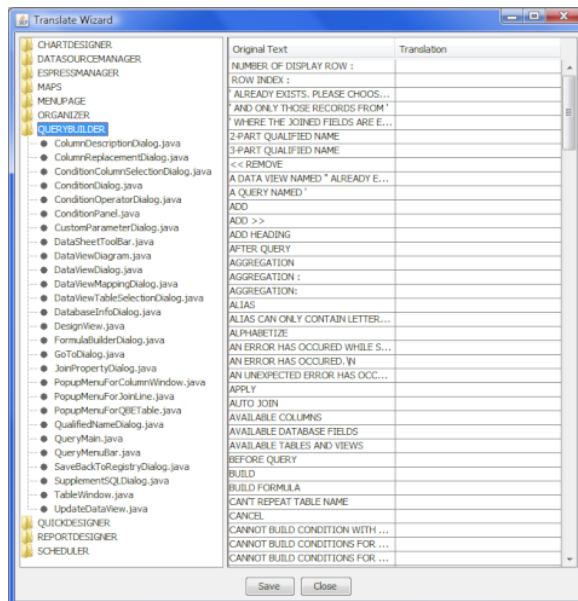
Where gbk is the encoding used to save Chinese.xml file.

When the Translate wizard is started, it appears as below:



Translate Wizard

You can see the various codes (available for translation) by selecting appropriate Product. Products are shown in a tree on the left that contains all the files available for translation. For example, on the image below you can see expanded *QUERYBUILDER* product. You can either translate entire product by selecting the product node or just select a file that you want to translate.



Insert Translation

The English texts are listed on the left and you can enter the translation on the right. The translation can be entered by clicking on the cell and typing in the text. The translation can be saved (to the file specified by the command that started the wizard) by clicking on the *SAVE* button. You can navigate to the previous screen by clicking on the *PREVIOUS* button. Note that if a translation for a code has already been set, it will appear for every instance of the code.

You can use `EDAB_Language.xml` (or the copy you modified) in EDAB Organizer by adding the `-file` and `-enc` arguments (e.g. `-file:ERES_Language.xml` and `-enc:UTF-8`) to the java web start (jnlp) file that starts Organizer (in `ERESOrganizer.jnlp`).

For the web application, edit `index.jsp` page as follows:

- Add an import for the `LanguageEncoder` class:

```
<%@page
  language="java" import="quadbase.common.util.internal.LanguageEncoder"
  errorPage = "MenuError.jsp" contentType="text/html; charset=UTF-8" %>
<jsp:useBean id="index" scope="session" class="quadbase.auth.bean.Index" /
>
<jsp:setProperty name="index" property="*" />
```

- Add a few lines near the beginning of the jsp to load the language file.

```
<%
    quadbase.common.util.internal.LanguageEncoder.load("<language
file>", "<encoding>");
    index.setRequest(request, application);
%>
```

where `<language file>` is the language file (for example, `Chinese.xml`) and `<encoding>` is the encoding (for example, `gbk`).

10.1.2.1.1. Upgrading Language File

When the user upgrades to a newer version, the `EDAB_Language.xml` file needs to be upgraded as well. The language upgrade program will copy over the translations from the previously customized `EDAB_Language.xml` file and append the additional entries in the new version. To use the language upgrade program, navigate to the EDAB directory from a console window and use the following command:

```
java -classpath ".;\lib\EDABOrganizer.jar"
quadbase.internationalization.UpgradeLanguageXMLFile -from:oldfile -
to:newfile -enc:encoding
```

It may be necessary to replace the semicolon with colon and backslash with slash on non-Windows environments.

Oldfile refers to the customized `EDAB_Language.xml` from a previous version of EDAB, newfile refers to the `EDAB_Language.xml` included in the upgraded EDAB installation, and the `enc` is the language encoding used in the translation. After running the program, the resulting file will be named `EDAB_Language.xml` and you can open it with Translate Wizard to further translate any new entries.

10.1.2.2. Displaying Foreign Characters

Foreign characters can be easily displayed in Designer and various Viewers. To display foreign characters in a report/chart, you will need to have fonts for that language installed in your system. Then, in the report/chart, you can set the font for the object that contains foreign characters to the appropriate system font.

Another option is to modify the `font.properties` file in the JVM so that foreign characters are supported in the default JVM fonts. For Sun JVMs, the `font.properties` file is located under the `jre/lib` directory. The different language files have names like `font.properties.ru` for Russian, `font.properties.zh` for Chinese, etc. To change the language settings for the JVM, rename the current `font.properties` file to back it up and change the name of the desired language file to `font.properties`. With the language settings changed in the JVM, the default fonts in EDAB (Dialog, Serif, Monospaced, etc.) will display with foreign characters.

10.1.2.3. Entering Foreign Characters

In order to enter foreign characters into any EDAB interface (for example, Organizer and Designer), the following changes to the system settings are required:

- The *default locale* of your system must be set to the region for the language you want to use.

- The *input locale* for the system must also be set to the region for the language you want to use

Note that for Windows the settings can be accessed through *Regional Options* in the *Control Panel*.

In addition, the font settings in the JVM must be adjusted to the desired language following the instructions discussed in Section 10.1.2.2 - Displaying Foreign Characters.

Once these settings have been applied, foreign characters can be entered in labels, queries, formulas, and parameters in reports/charts.

10.1.2.4. XML Encoding

By default, EDAB use UTF-8 character set for encoding when writing to XML. This includes data registry files, XML report templates, XML exports, and XML representations of global format information and font mapping. Please note that for most of users it is not necessary to change character set encoding, because UTF-8 fully supports all languages.

This encoding can be changed for other languages by adding a run-time parameter to Organizer and EDAB Server. For Organizer, you can do this by modifying the JSP source of the applet page used to launch the Organizer.

To change the XML encoding, you will need to add the following parameter to the `Organizer.jsp` page: `<PARAM NAME="xml_encoding" VALUE="ISO-2022-JP">`.

This parameter also needs to be set for the server. For more information about server settings, please see Section 1.4.1.3.1 - Secured Parameter.

10.1.2.5. Exporting With Foreign Characters

Most of the EDAB export formats will be automatically generated with the UTF-8 character encoding. In order to view the exported reports/charts, the client will need to have the appropriate system fonts installed. The only exception to this is the PDF export. PDF format does not depend on client fonts for viewing. Therefore, foreign language fonts will need to be embedded into the PDF document in order for characters to display properly. To do this, you will need to use the PDF Font Mapping feature. The Font Mapping can be set in Designer and in API.

If you wish to export the report/chart to a locale different from the one that is set on the machine, you can specify a different encoding before doing the export. This is done using the following method in `QbReport`:

```
public void setExportEncoding(String encoding);
```

where `encoding` is a character encoding supported by the Java 2 platform. For example, the character encoding for English (Windows Latin-1) is `Cp1252`. Please note that you have to use the canonical name used by the `java.io` and `java.lang` APIs.

You can also specify the character set used in the DHTML exports using the following method in `QbReport`:

```
public void setHTMLCharSet(String charset);
```

where `charset` is a valid character set. For example, a character set for English used in DHTML documents is `UTF-8`.

10.1.2.6. Specifying Encoding for Text Data Files

By default, EDAB use system encoding for any text data files. The encoding for input text data files can be set via API using the following block of code:

```
report.getInputData().setDataFile("sample.dat", false, "ASCII");
```

The above block use the following method found in `quadbase.reportdesigner.util.IInputData`:

```
public void setDataFile(String fileName, boolean isDataSorted, String  
    encoding);
```

Chapter 11. Alerts

11.1. What is an Alert

An alert is basically an exception handling mechanism that notifies user when certain data is out of a pre-defined range. Typically, the user is looking at some KPIs (key performance indicators) that are important to his/her organization. Some example KPIs on which a user may want to set alerts are inventory level, profit margin, sales growth rate, percentage of dropped calls in a mobile network, number of intrusion attempts on a company network, response time on customer support/service center etc..

11.2. Specifying alerts

Alerts are specified in a displayable object, namely maps, charts and reports.

11.2.1. Online Maps

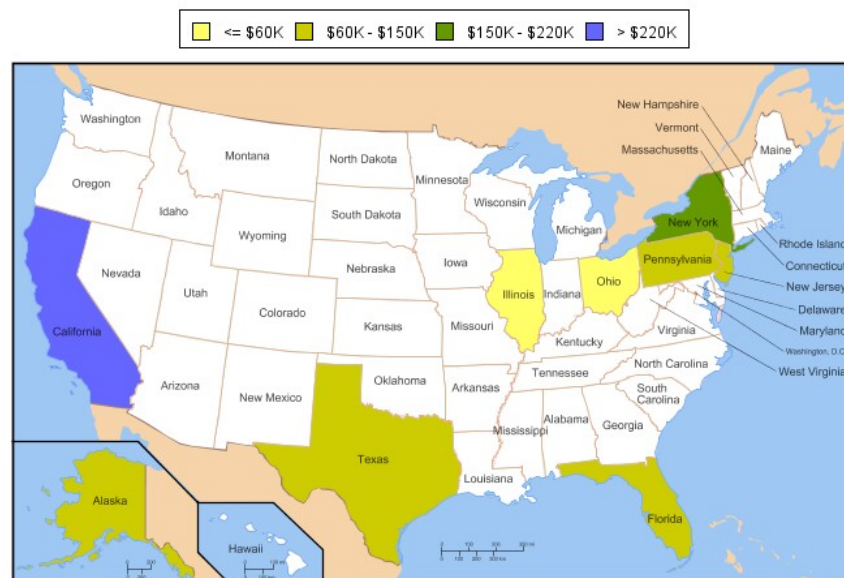
There is currently no support for alerts in Online Maps.

11.2.2. SVG Maps

For SVG Maps, alerts can be specified on all thresholds in the map. You can select ranges that are critical for you. If any of the areas on the map would be in that range, alert will trigger.

If the map doesn't have any thresholds, alerts will not be available for the map.

Example:



SVG map with thresholds

You can for example say that ranges <= \$90000 and \$90000 - \$330000 are critical and you will get alert if there are any yellow or light green areas in the map.

11.2.3. Charts

For Charts, you can specify alerts on control areas. Similarly to SVG maps, you can choose the ranges that are critical for you. If there are no control ranges, alerts will not be available for the chart.

Example:

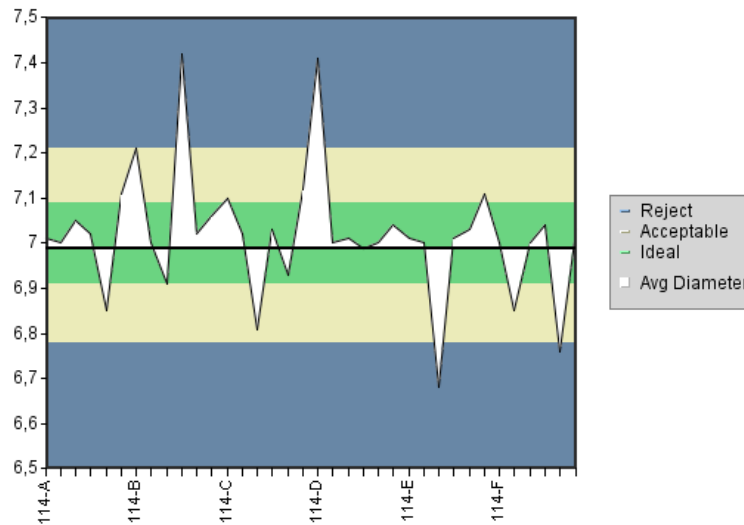


Chart with control areas

You can say that the Reject control area is critical and you will get alert whenever there is at least one data point in this (blue) control area.

Control area ranges are evaluated as left-closed, right-open intervals [start, end). In other words, the alert will trigger if the following condition is true: $\text{start} \leq \text{ActualValue} < \text{end}$.

11.3. Alert types

EspressDashboard supports two types of alerts, dashboard alerts and monitoring alerts. Dashboard alerts are evaluated only on currently opened dashboards. They are not triggered when the dashboard is not open. Monitoring alerts are alerts in scheduled objects (reports, charts, maps, dashboards). Monitored alert tasks are run at certain times. You can set a time pattern or specify custom times (timing possibilities will be described later). At the time when alert monitoring task is run, alert conditions are evaluated and the monitoring task actions (i.e. Send email, upload to FTP) are run only if an alert was triggered. If no alerts in the alert monitoring task are triggered, no action is performed.

11.3.1. Dashboard alerts

Dashboard alerts are set up for individual templates (reports, charts, maps) in the dashboard using the user interface in the dashboard builder. Alert (exception) conditions are evaluated when the templates are exported, i.e. during initial loading, auto refresh, manual refresh of the dashboard, changing parameters and resizing the template. Auto refresh is enabled every time you preview or view a dashboard. Default auto refresh interval is 60 seconds.

11.3.2. Visual alert display

If an alert was triggered, color of template border will change and the border will start blinking. For templates in folders, tab header color will change and blink.

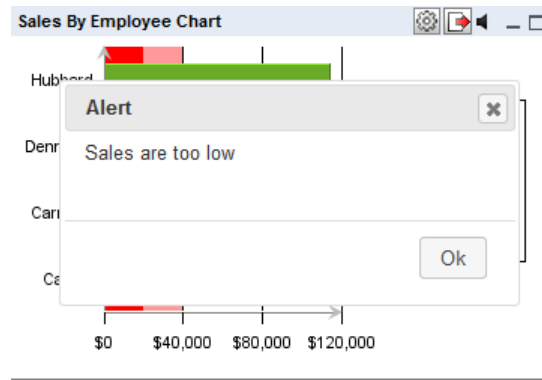
In addition, the template will have its own “native” highlighting of the values:

- coloring of areas in the SVG maps
- control ranges in charts

Please note that alert highlighting, blinking and alert messages are visible only in *Preview* of the Dashboard Builder.

11.3.3. Alert messages

If you also set up an alert message, a message with your text will pop-up.




Triggered alert message

11.3.4. Sound alert

In some cases, sound alerts may be preferred over the visual alerts. When an alert is triggered, the template will continuously make beeping sound. If the alert is muted and you change a parameter in the template and the watched alert is triggered again, the alert sound will be active again. However, if the same watched alert is triggered after auto refresh, the alert sound will not reset, but if another watched alert is triggered, the alert sound will be active again. If the alert is disabled during auto refresh, it will reset back to active state.

11.3.5. User interface

Alerts dialog can be accessed using this button  in the header of every template in the dashboard. This button is only visible for templates that can have some alerts. It will not be displayed for templates that cannot have alerts (i.e. all Google maps, SVG maps without thresholds, charts without control ranges and reports without scripts with ALERT variable).

After clicking the button, you will get a dialog with all possible alerts that can be watched (i.e. names of all control ranges for charts or all ranges for SVG maps).

watched	Alert Name	Sound	Alert Message
<input type="checkbox"/>	<1000	<input type="checkbox"/>	
<input type="checkbox"/>	1000-10000	<input type="checkbox"/>	
<input type="checkbox"/>	10000-20000	<input type="checkbox"/>	
<input type="checkbox"/>	20000-30000	<input type="checkbox"/>	
<input type="checkbox"/>	30000-40000	<input type="checkbox"/>	
<input type="checkbox"/>	>40000	<input type="checkbox"/>	

Watch alerts dialog

All the *watched* checkboxes will be unchecked by default, which means that no alert is watched. After checking a *watched* checkbox, you enable *Sound* checkbox and *Alert message* textbox, which will allow you to use sound alerts and alert messages. The text you fill in alert message textbox will then display as a pop-up message when the watched alert is triggered. If you leave it empty, no message will appear. You can then check the checkboxes and start watching the alerts. The alert name cannot be changed. It will always be read from the template.

All alerts will be joined using logical OR. If any of these alerts trigger, the dashboard alert will trigger.

11.4. Monitoring

Monitoring is supported for dashboards and also for individual reports, charts and maps (let's call them **monitored objects**). The monitored objects are checked in given time intervals and notifications are sent if alert occurs in some

of the monitored objects. This is done in background, so the notifications are sent even if the monitored object is not viewed.

The user interface to set up monitoring is a simple wizard dialog. If the monitored object has parameters, user can set up several parameter sets. Monitoring has many options for testing intervals (e.g. one time, time interval, fixed days, etc.). Monitoring is not supported for drilldowns.

Administrators and designers are the only users who can set up monitoring (only for objects they have read access to). They can also select recipients of notification emails. Viewers are not allowed to set up monitoring.

11.4.1. Alert notifications

Alert notifications are sent via emails. Exported templates can also be uploaded to FTP. Alert notification can be sent every time an alert occurs or it can be send only when an alert starts (i.e. if alert wasn't triggered during last check, but it is triggered during current test).

You can set email subject and body. They have to be the same for the entire monitoring (for all templates, alerts and parameter sets). Body of the notification email can contain some variables that will be substituted during runtime. List of these variables will be detailed later (Section 11.4.3.1.1 - Email Delivery Options).

You can set different recipients for different templates and different parameter sets. It is not possible to set different recipients for different alerts in one template. If you want that, you have to create another alert monitoring task (you can add as many tasks as you want).

11.4.2. Failed emails

If some of the monitored objects are not available, or their data sources are not available, or if there are any other problems with exporting the monitored objects or alert evaluation (e.g. watched control area was removed), the user can be notified with failed emails.

You can set failed email subject and body. Recipients are the same as for the notifications. Error log (with stack traces) can be sent to a selected user (typically admin). Failed emails dialog will be described later (Section 11.4.3.1.1 - Email Delivery Options).

11.4.3. User interface

11.4.3.1. Create/Edit monitoring dialog

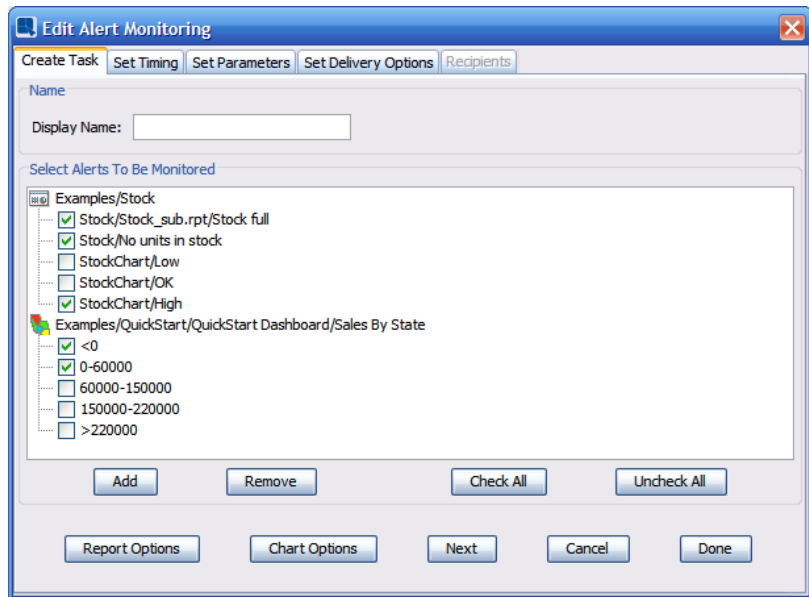
To set up new alert monitoring task, please use the following steps:

Launch the *Create alert monitoring wizard* -

first of all, launch *Organizer* and select at least one file you want to monitor. You can also select multiple files from single directory, but you don't have to select all files you want to monitor at once because you will be able to modify the file list later (and maybe add more files from different organizer folders if needed). Then you have two options: either right-click on one of the selected files and then choose the *Set alert monitoring* option from the pop-up menu, or you can click *Monitoring* (in the main menu) and then select *Set Alert Monitoring*. Both methods will open the first step of the *Setup alert monitoring wizard*.

Select alerts -

Select templates you want to monitor in the task. You can select any dashboard, stand-alone report, chart or map.



Create monitoring task dialog

The monitored template list is a tree.

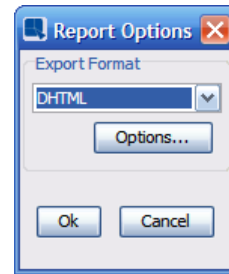
The first level of the tree is the selected templates (before the dialog was opened) or templates added using the *Add* button. The leaf nodes are available alerts (for their parent node). You can check alerts that should be monitored. You have to select at least one alert for each template. If you don't want to monitor any alert of a monitored object, remove the object by selecting it and clicking the *Remove* button.

If the selected template is dashboard that already have some dashboard alerts defined, the checkboxes are initialized according to the dashboard alerts (the alerts that are watched in the dashboard are checked). If the selected template is not dashboard, all the checkboxes are unchecked by default.

Following two buttons allow you to set export format. Monitored templates will be exported to this format in case an alert was triggered. Also, if you select additional delivery options like email or FTP delivery (will be described later), templates will be delivered in this format.

Report options -

shows dialog allowing you to set report export format. For reports, you can select DHTML, PDF, CSV, Excel (XLS), Excel 2007 (XLSX), text, rich text, or XML. Additional options allows you to select single page or multi-page exporting for DHTML and XML. The *Options...* button allows you to set format-specific options for the exported file. For DHTML, you can set single or multi-page exporting and you can also specify which CSS options to use. For PDF, you can set encryption, and for text you can specify the delimiter.



Report Options Dialog

Chart options -

shows dialog allowing you to set chart export format. For charts, you can select JPG, GIF, PNG, PDF, SVG, or FLASH. Other options allow you to specify PNG compression, set JPEG quality, and specify background transparency for GIFs. You can also enable and configure Flash animation if the image depiction should be animated. You can also specify the size of the exported image.

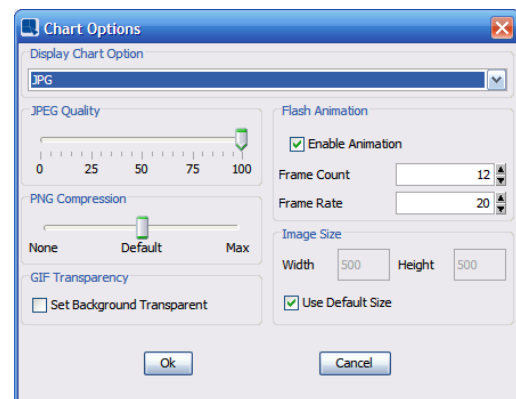


Chart Options Dialog

Set timing –

allows you to set a time pattern of the alert monitoring task.

Set timing dialog

You have following timing options:

One time only -

task is run one time only at pre-defined time. You can choose when the task will run using the *Begin on* option (will be described later).

At regular time interval -

alert monitoring task will run at fixed intervals for a pre-defined period of time. For example, you can set the task to run every X hours (or minutes, days, months...) from *Begin on* to *End on* time and date. When you select this option, the *Set time interval* dialog pops up. This dialog will be described later.

On fixed days/dates every week/month -

If you need to run your task irregularly, this option should do the trick. It allows you to run the task on certain week day(s) (for example: every Monday and Friday), or certain days of the month (for example: first day of a month). This is set in the *Set fixed days/dates Interval* dialog which pops up when you select the option. Also, when you select this option, the *Daily frequency* section is enabled. These two dialogs will be described later.

You also need to set when the task should start running and when it should stop. You can do that using the following options:

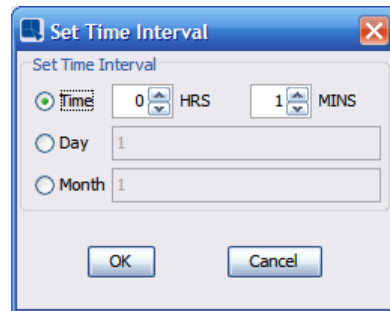
Begin on -

allows you to set when the task will start running. By default, the *Start now* option is enabled. It means that the task start time and date will be set few minutes in the

future, thus the task will start running immediately after you're done configuring it. If you don't want the task to start immediately, uncheck this option to pick the start time and date from a calendar. You can access the calendar by clicking the *Change* button (which is available only when the *Start now* option is disabled).

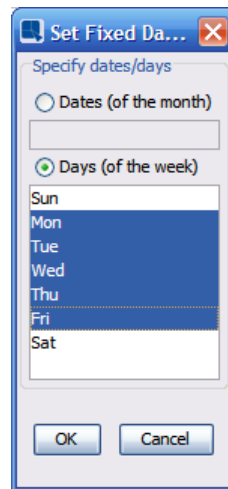
End on -

controls when the task will end. This option is very similar to the *Begin on* option. The only difference is that the *Start now* option is replaced by *Run indefinitely* option. If you choose this option, the task will never stop unless you stop it manually. Or you can unselect this option and pick the *End on* time and date manually from a calendar.



Set Time Interval Dialog

This dialog is enabled only if the *At regular time interval* option is selected. First of all, choose a time unit (namely: hours and minutes, days or months) that suits your needs. Then enter a number of the time units that should be waited for between each task runs.



Set Fixed Days/Dates Dialog

This dialog is enabled only if the *On fixed days/dates every week/month* option is selected. First choose whether you want to run the task on a weekly basis or monthly basis. Then select days of the week or month the task should run on. After you're done setting this dialog, click the *OK* button and proceed to the *Daily frequency* settings.

Daily frequency settings

After you have set the days of the month/week you want the task to run on, you have to set when exactly it should run during those days. That can be done in the *Daily frequency* section. You have two basic options:

Set run times -

allows you to manually enter the run times. You can add as many values as you want. You can use 12-hour or 24-hour format. Individual values must be separated by comma. **For example:** 08:05AM, 11:15PM, 22:35

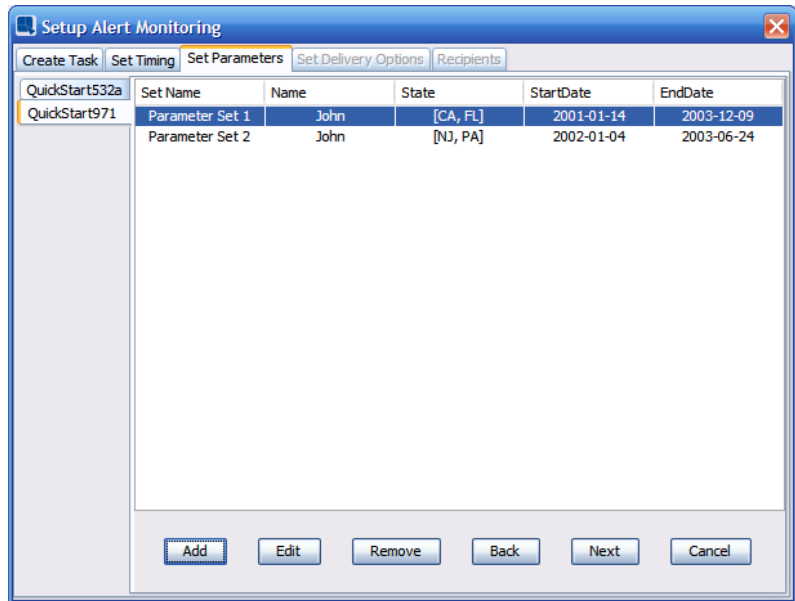
Set fixed interval -

use this option to run the task periodically from the *From:* time to the *To:* time every X hours/minutes. To set the times, you can use up and down arrows or you can enter the value manually (click in the field and enter a number). If you enter an incorrect time (for example: 11:65), the value will be marked with red color. Correct values are marked with green color.

Once you finish specifying the periodicity, click the *Next* button to continue with the schedule wizard.

Set parameters –

this step is skipped if there are no parameters in the selected templates. If you have selected to monitor a report, chart, map or dashboard that contains parameters, the next tab will appear allowing you to set the parameter values you want to use.



Set parameters dialog

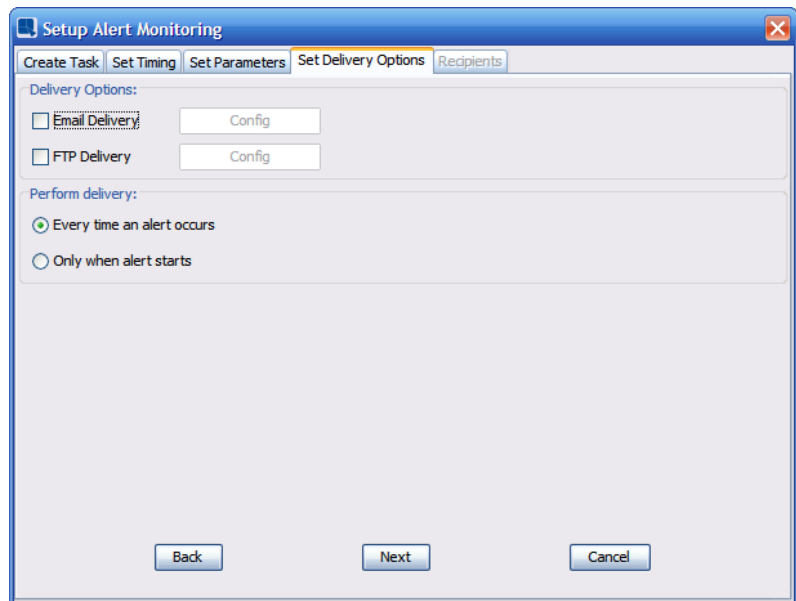
To add a set of parameters, select the *Add* button. This will bring up the parameter prompt dialog allowing you to select the set of parameter values you want to use. Once you have selected a parameter set, your choices will appear in the dialog. You can add as many different combinations of parameter sets as you like. A separate file will be generated for each set of parameters you specify. Alerts will be evaluated separately for each parameter sets. Only the templates that triggered some alerts will be sent in the notification email or uploaded to FTP.

You can also specify a name for each parameter set by double clicking on the first column. The name specified here will be used in later dialogs to help you organize your recipients.

Once you specify all parameters, click the *Next* button to continue.

Set delivery options -

This dialog allows you to configure the delivery options for the alert notifications.



Delivery options dialog

You have two delivery options:

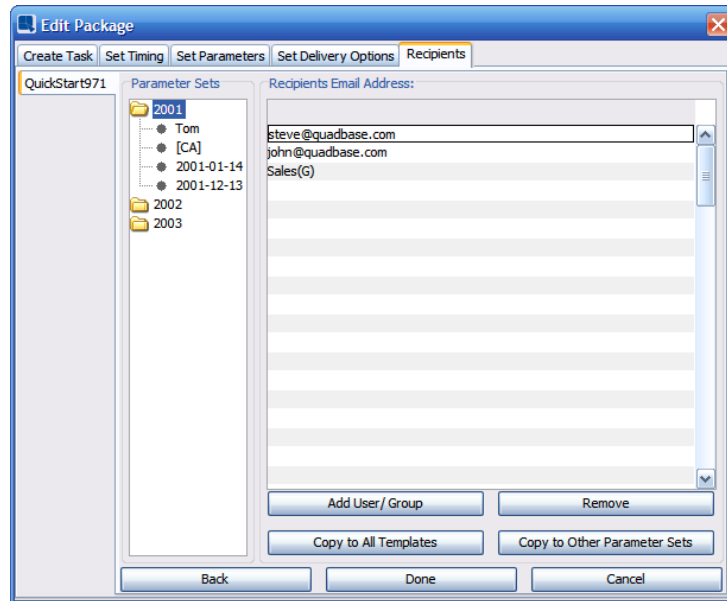
Email Delivery - allows you to send an email when some alert was triggered during task runtime. How to configure email settings will be described later (Section 11.4.3.1.1 - Email Delivery Options).

FTP Delivery - uploads exported report(s) to a FTP server if a monitored alert was triggered. FTP delivery settings will be described later (Section 11.4.3.1.2 - FTP Delivery Settings).

You can select if you want to perform the delivery (send emails or upload files to FTP) every time an alert occurs or if you only want to perform it when alert starts (i.e. if alert wasn't triggered during last check, but it is triggered during current test).

Email Recipients -

If you selected email delivery, the next tab allows you to specify the recipient list. The recipient list is set up differently depending on the type of monitored object you are monitoring. If the monitored object contains parameters, you will see the following dialog. This allows you to send different parameter sets to different users or groups.



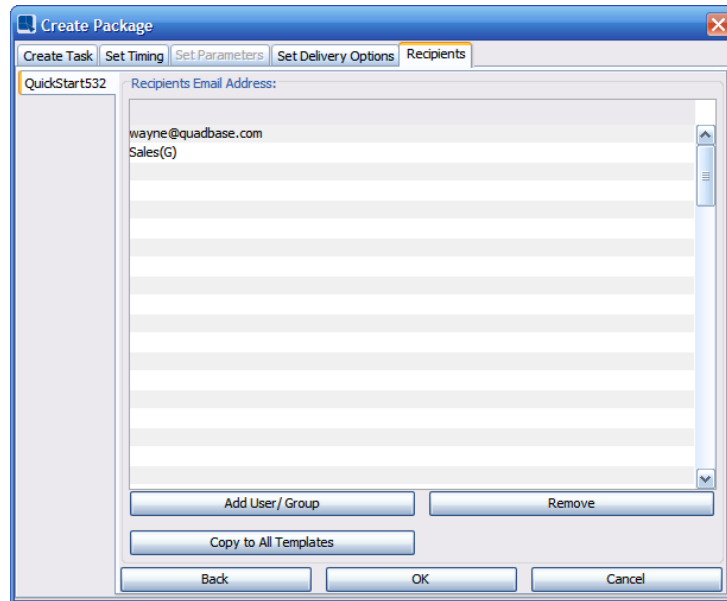
Set recipients dialog for parameterized reports

Entering recipients into the list is simple, click on the *Add User/Group* icon to select users and groups from the EDAB database. You can also add regular email addresses by double clicking any row in the recipients list and typing it in manually. You can also click on *Add User/Group* and view the *Prev. List* tab for previously listed recipients.

If you want to send all parameter sets or all templates to the same recipients, you don't have to add the recipients manually into all parameter sets and templates. Just add the recipients into one recipients list and then use one of the following buttons:

- | | |
|---------------------------------------|--|
| Copy to all templates - | click this button to copy current recipient list into all scheduled templates recipient lists. |
| Copy to other parameter sets - | allows you to copy current recipient list to the rest of the parameter sets. |

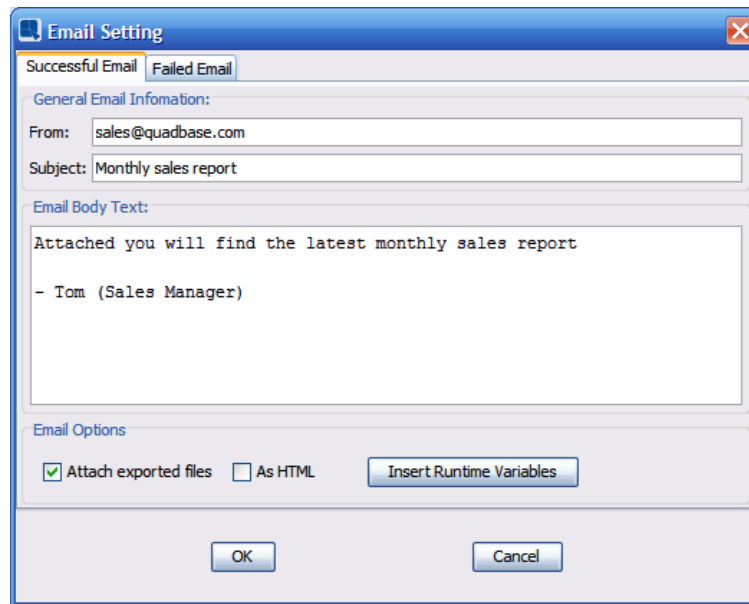
For all other cases, you will see the following dialog containing only one recipients list.



Set recipients dialog

11.4.3.1.1. Email Delivery Options

When you click the *Config* button for email delivery, the following dialog will open allowing you to set an email message.



Email settings dialog

The first tab contains information for the email sent when the report or chart has successfully exported. The *General Email Information* portion of this tab allows you to specify the from addresses for the email as well as the subject.

The first tab is called *Successful email* and it allows you to set an email message which will be sent in case the scheduled task finished with no problems.

There are following elements:

From: - enter an email address into this field. This address will be used as an email sender's address. Please make sure that your SMTP server won't reject the address (depends on your SMTP server security settings).

Subject: -	email subject. Enter any convenient text in here.
Email body text -	enter a message that will be sent on every task run. You can enter any text and you can also use <i>Runtime variables</i> (will be described later).
Attach exported files -	allows you to attach all exported reports and charts to the email.
as HTML -	if this option is enabled, email body text will be replaced by the scheduled report or chart in HTML format. This option is available only if the <i>Attach exported files</i> is enabled and there is only one scheduled report/chart in the scheduled task. Also, PDF, SVG and Flash charts can't be added into HTML emails and only HTML and DHTML reports can be added into HTML emails. You can set the export format on the <i>Create task</i> tab - described in Section 11.4.3.1 - Create/Edit monitoring dialog.
Insert runtime variables -	allows you to insert certain variables into the email body text. Variables are replaced by real values at the task run time. All variables also have a “default value”. If no convenient data can be inserted into the variable, the default value is used (for example: when user hasn't provided his name). You can choose among following variables:
<USERNAME> -	recipient's user name. For example: you can use this variable in a salutation like Hello <USERNAME>. This variable works only for email recipients that were chosen from the EDAB database (adding email recipients was described in previous chapter). If you add an email address to recipients list manually, no username is displayed because no username was provided.
<FULLNAME> -	recipient's full name. Just like the USERNAME variable, this variable also doesn't work for manually entered recipients email addresses.
<EMAIL> -	recipient's email address.
<TIMESTAMP> -	current time and date. Shows when the email was sent.
<PARAMETERS> -	list of all parameter sets and parameter values.
<SAVED_LINKS> -	links of all files in which at least one alert was triggered. Users can access the exported files through these links.
<LIVE_LINKS> -	links of all files in which an alert was triggered. Unlike the SAVED LINKS, these links lead to DHTML Viewer which allows users to access current version of the triggered files.
<ALERTS> -	names of all alerts that were triggered during the export.
<ALERT_DETAILS> -	For reports, alert details are replaced by whole row of data in a format like (COLUMN1_NAME) VALUE1 (COLUMN2_NAME) VALUE2 . . . For charts, it shows values of the tooltips that would be displayed on the data points that triggered alerts. For maps, it shows

all values from the data source rows associated to the areas that triggered alert.



Note

In order to send email correctly, the SMTP host needs to be set up. How to configure a SMTP server is described in Section 1.4.1.2 - Setting Info.

The *Failed Email* tab allows you to send a different email (subject and body) to your recipients if an error has occurred during the task running (for example: one of the scheduled reports has been removed).

Failed email settings dialog

This tab also allows you to send error logs to a specific user. Typically, this would be the admin or technical user who is able to diagnose and troubleshoot the problem.

11.4.3.1.2. FTP Delivery Settings

When you click the *Config* button for FTP delivery, the following dialog will open allowing you to set options for FTP schedule delivery.

FTP Delivery Options Dialog

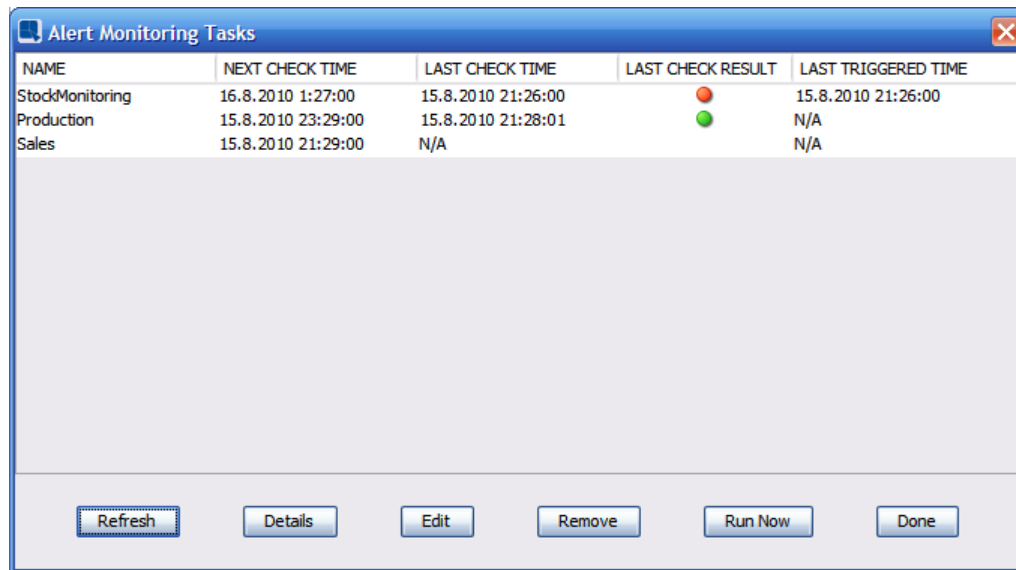
The following information is required for FTP schedule delivery:

FTP Host Name: This is the hostname of the FTP server you want to use to upload the reports. If you have a Secure FTP Server (SSH FTP), enter **sftp://** in front of the hostname.

- FTP Upload Path:** This is the path or directory (relative to the FTP root) where you would like to write the exported report files.
- FTP User Name:** This is the username for the FTP server.
- FTP Password:** This is the password for the FTP server.

11.4.3.2. Monitoring list

You can view monitoring list from pull-down menu Schedule/Archive → View Alert Monitoring Tasks. The dialog allows users to view, edit and remove current monitoring tasks.



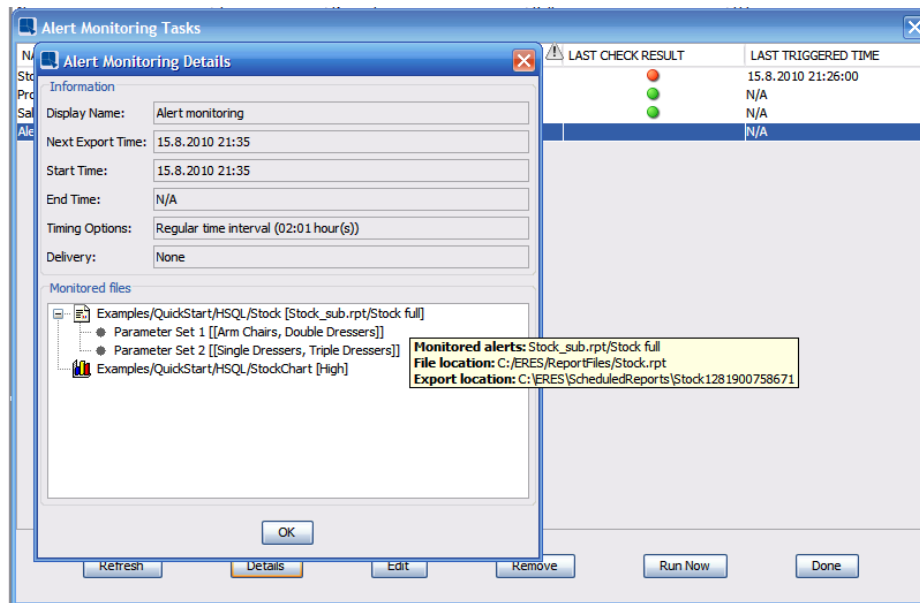
View alert monitoring tasks dialog

It also displays the following info about each monitoring task.

- name
- next check time
- last check time
- last check result (alert was triggered or not) – red light means that an alert was triggered, green light means no alerts
- last triggered time

The *Check Now* button is similar to the *Run Now* button in the Schedule list dialog. It checks selected alerts immediately (and sends notifications if necessary).

You can use the *Details* button to view more details about selected monitoring.

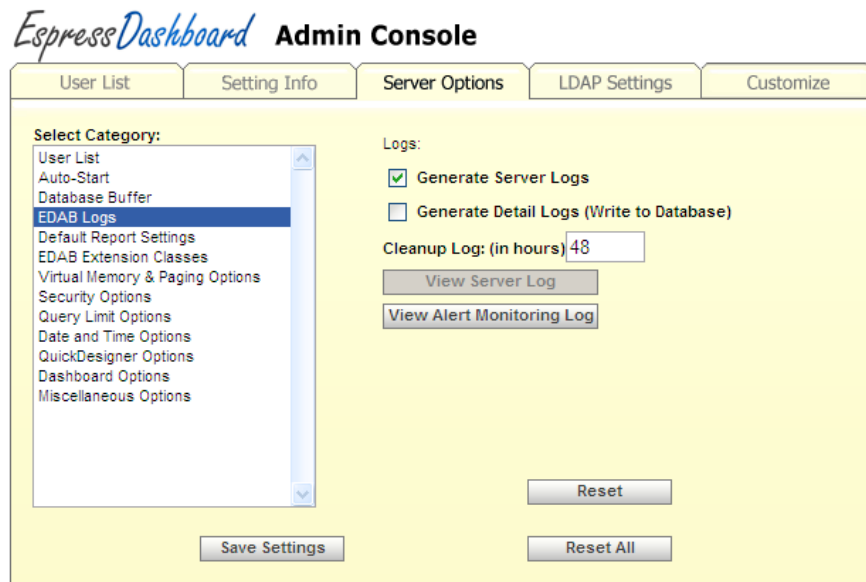


Alert monitoring details dialog

The Monitored files list contains list of all monitored files with all monitored alerts (in the square brackets). It also contains all parameter sets if the template is parameterized. If you hold cursor over a template or parameter set in the Monitored files list, you will get a tooltip with more details.

11.4.4. Monitoring log

To open the monitoring log, log in as **admin** and click *Admin Console*. Then go to the *Server option* tab, select *EDAB logs* category and click the *View Alert Monitoring Log* button.



View alert monitoring log button

The monitoring log will pop up in a new window. The log contains monitoring tasks that have been checked at least once in the past.

Alert monitoring log

29.11.2010 10:18:04

Name	Created by	Last checked	Next check	Last triggered	Email del.	FTP del.	Success
QuickStart2	admin	11/25/10 16:33:00	12/25/10 16:33:00	At last check!	<input type="checkbox"/>	<input type="checkbox"/>	OK
Template name		Parameter set name		Parameter values		Alert name	
Examples/Dashboards/Alerts Components/Sales By Employee Chart		Parameter Set 1		2002.Q2		No sales	
QuickStart3	admin	11/25/10 16:38:00	12/09/10 16:38:00	Never	<input type="checkbox"/>	<input type="checkbox"/>	OK
Template name		Parameter set name		Parameter values		Alert name	
Examples/Dashboards/Alerts Components/Sales By Employee Chart		Parameter Set 1		2002.Q2		Low sales	
QuickStart4	admin	11/25/10 17:19:00	Finished!	At last check!	<input type="checkbox"/>	<input type="checkbox"/>	NO
Template name		Parameter set name		Parameter values		Alert name	
Examples/Dashboards/Alerts Components/Sales By Employee Chart		Parameter Set 1		2001.Q1		Low sales	
		Parameter Set 2		2002.Q3		Low sales	
		Parameter Set 3		2001.Q1		Low sales	
Examples/Dashboards/Alerts Components/Sales Overview Report						Low revenue	

Alert monitoring log

The most important indicator is whether an alert was triggered at the *last check*. Basically, triggered alerts are highlighted by red color.

There are several possibilities:

Task that doesn't contain any triggered alerts looks like this:

Name	Created by	Last checked	Next check	Last triggered	Email del.	FTP del.	Success
Stock	admin	11/25/10 16:44:00	Finished!	11/25/10 16:43:00	<input type="checkbox"/>	<input type="checkbox"/>	OK
Template name		Parameter set name		Parameter values		Alert name	
New Project/Stock						Low stock	

Task with no triggered alerts

Task that contains at least one triggered alert looks like this:

Name	Created by	Last checked	Next check	Last triggered	Email del.	FTP del.	Success
QuickStart4	admin	11/25/10 17:19:00	Finished!	At last check!	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	NO
Template name		Parameter set name		Parameter values		Alert name	
Examples/Dashboards/Alerts Components/Sales By Employee Chart		Parameter Set 1		2001.Q1		Low sales	
		Parameter Set 2		2002.Q3		Low sales	
		Parameter Set 3		2001.Q1		Low sales	
Examples/Dashboards/Alerts Components/Sales Overview Report						Low revenue	

Task with one triggered alert

Alert that wasn't triggered at the last check:

Name	Created by	Last checked	Next check	Last triggered	Email del.	FTP del.	Success
QuickStart4	admin	11/25/10 17:19:00	Finished!	At last check!	<input type="checkbox"/>	<input type="checkbox"/>	NO
Template name		Parameter set name		Parameter values		Alert name	
Examples/Dashboards/Alerts Components/Sales By Employee Chart		Parameter Set 1		2001.Q1		Low sales	
Examples/Dashboards/Alerts Components/Sales Overview Report						Low revenue	

Untriggered alert

Alert that was triggered at the last check:

Name	Created by	Last checked	Next check	Last triggered	Email del.	FTP del.	Success
QuickStart5	admin	11/29/10 10:17:00	11/29/10 10:18:00	At last check!	<input type="checkbox"/>	<input checked="" type="checkbox"/>	OK - no notif.
Template name		Parameter set name		Parameter values	Alert name		
Examples/Dashboards/Alerts Components/Sales By Employee Chart		Parameter Set 1		2001,Q1	Low sales		
					No sales		
		Parameter Set 2		2001,Q3	Low sales		


Triggered alert

Each task has a *Success* indicator which is not to be confused with triggered/not triggered alerts. The indicator tells whether the last check was performed successfully. There are a few things that can go wrong during checks - monitored templates or alerts could have been renamed or removed, one of the delivery options (FTP or Email delivery) may have failed because of network problem... If a check has failed, the *Success* indicator will look like this: **NO**. You can get more details about failed checks in a drilldown, which will be described later.

A template can have multiple parameter sets and/or multiple monitored alerts. In such situations, the template will take several rows in the template list, but its name will be displayed only once (see the following picture).

Name	Created by	Last checked	Next check	Last triggered	Email del.	FTP del.	Success
QuickStart5	admin	11/29/10 17:25:34	11/29/10 17:26:00	At last check!	<input type="checkbox"/>	<input checked="" type="checkbox"/>	OK - no notif.
Template name		Parameter set name		Parameter values	Alert name		
Examples/Dashboards/Alerts Components/Sales By Employee Chart		Parameter Set 1		2001,Q1	Low sales		
					No sales		
		Parameter Set 2		2001,Q3	Low sales		
					No sales		

Parameterized chart with two monitored alerts

It is possible to exclude tasks without any triggered alerts from the log. To do that, click on the  *Filter* icon and set the *Show only tasks with triggered alerts?* parameter to *true*.

Filter...

Show only tasks with triggered alerts?

true

Ok

Cancel

Log filter

To display detailed report about a task, click on its name. A second level of the monitoring log will open.



1 / 1

Triggered alert Not triggered alert Task with no triggered alerts

Stock log

Created by:	admin
Next check:	This task has finished
Last triggered:	11/25/10 16:44:00
Delivery settings:	Email delivery: No
	FTP delivery: No
	Send notifications: Always

Check time: 11/25/10 16:44:00				Total success:	OK
Name	Parameter set name	Parameter values	Alert name	Success	
New Project/Stock			Low stock	OK	
Check time: 11/25/10 16:43:00				Total success:	OK
Name	Parameter set name	Parameter values	Alert name	Success	
New Project/Stock			Low stock	OK	
Check time: 11/25/10 16:42:00				Total success:	OK
Name	Parameter set name	Parameter values	Alert name	Success	
New Project/Stock			Low stock	OK	
Check time: 11/25/10 16:41:00				Total success:	OK
Name	Parameter set name	Parameter values	Alert name	Success	
New Project/Stock			Low stock	OK	

Task details

The second level log contains results of all alert checks (unlike the root level log which only contains result of the last check). Triggered alerts are also highlighted by red color. The *Success* indicator is displayed for all templates.

If you are wondering why notifications were not sent, always check the *Send notifications* indicator. The indicator has two possible states:

Always -

notifications will be sent every time an alert occurs

First alert occurrence -

notifications will be sent only when an alert wasn't triggered during the previous check, but was triggered at the current check. So if an alert was triggered in two consecutive checks, notifications would be sent only once. If notifications were not sent because of this feature, the task's *Success* indicator will look like this

OK - no notif.

To go back to the root level log, click on the  *back* icon.

If the task uses some additional delivery method(s) (FTP or Email), you can click on the *Success* indicator. A third level of the monitoring log will open showing the delivery method's settings and results. If there is no additional delivery method, the *Success* field will not be clickable.

The third log level's appearance may vary. If a notification was performed, its detailed log will open. If a notification wasn't performed (for example: the alert wasn't triggered), only delivery settings will be shown.



Note

There is a difference between not performed delivery and failed delivery. A delivery isn't performed when the alert wasn't triggered or, in some cases, when the *Send notifications on first alert occurrence only* feature is enabled (as described above). In that case, the EDAB server doesn't even try to perform the delivery. If a delivery failed, it means that the EDAB server tried to perform the delivery (because the alert was triggered), but it wasn't able to finish (for example: because of a network problem, a human error...).

Delivery settings

FTP delivery settings

Name:	Examples/Dashboards/Alerts Components/Sales By Employee Chart
FTP Host:	gamma.quadbase.com
FTP Path:	/reports
FTP User name:	user
Parameter set name:	Parameter Set 1
Parameter values:	2001,Q1
Alert name:	Low sales

FTP delivery settings

Email delivery settings

Name:	Examples/Dashboards/Alerts Components/Sales By Employee Chart
Parameter set name:	Parameter Set 1
Parameter values:	2001,Q1
Alert name:	Low sales
Subject:	Monthly report
From:	clint@quadbase.com
Recipients	
	peter@quadbase.com, john@quadbase.com

Email delivery settings

Delivery logs

FTP delivery log shows FTP delivery settings + results. If the delivery has failed, an error message is also displayed.

FTP Delivery log

Run time:	11/25/10 17:19:00
Name:	Examples/Dashboards/Alerts Components/Sales Overview Report
FTP Host:	gamma.quadbase.com
FTP Path:	/reports
FTP User name:	user
Parameter set name:	
Parameter values:	
Alert name:	Low revenue
Success:	NO
Error message	
gamma.quadbase.com	

FTP delivery log

Email delivery log shows basic email settings and results for all recipients. In the following picture, you can see that emails from `clint@quadbase.com` to `john@quadbase.com` and `peter@quadbase.com` (not actual email addresses) couldn't be sent because the EDAB server wasn't able to connect to the SMTP server at `localhost:25`. It could be caused by a network problem or by incorrect SMTP settings (as described in Section 1.4.1.2 - Setting Info). This is probably the most common problem with email delivery.

Email delivery log

Name:	QuickStart4
Check time:	11/25/10 17:19:00
From:	clint@quadbase.com
Subject:	Monthly report

To: john@quadbase.com				
Name	Parameter set name	Parameter values	Alert name	Success
Examples/Dashboards/Alerts Components/Sales Overview Report			Low revenue	NO
Error message				
Sending failed; nested exception is: class javax.mail.MessagingException: Could not connect to SMTP host: localhost, port: 25; nested exception is: java.net.ConnectException: Connection refused: connect				

To: peter@quadbase.com				
Name	Parameter set name	Parameter values	Alert name	Success
Examples/Dashboards/Alerts Components/Sales Overview Report			Low revenue	NO
Error message				
Sending failed; nested exception is: class javax.mail.MessagingException: Could not connect to SMTP host: localhost, port: 25; nested exception is: java.net.ConnectException: Connection refused: connect				

Email delivery log

11.5. Handling special situations

11.5.1. Change in script/control area/range

If for some reason an alert is removed or renamed, the monitoring job or dashboard alert cannot find the missing alert. The following approaches will be taken to resolve the situation.

- For monitoring – An error email is sent. An explanation is included, e.g. alert <ALERT_NAME> not found in the template <TEMPLATE_NAME>.
- For dashboard alert – An error icon is displayed in the template header. If you mouse over the error icon, you will see error message explaining the problem. If you are using the Dashboard Builder at that time, you can fix the problem immediately.

11.5.2. Missed monitoring jobs

If a monitor check is missed, because server was not running, it will run immediately after the server is started. Each missed task will only run once, even when more checks were missed. After the missed monitoring jobs were run, checks will be performed as before according to their schedule.