IBM

System i

Database
IBM DB2 Web Query for System i

IBM

System i

Database
IBM DB2 Web Query for System i

> **Note**
>
> Before using this information and the product it supports, read the information in "Notices" on page 775.

Contents

Chapter 5. Analyzing Data in an OLAP Report 167

Preface

This documentation describes the functions that are available for IBM® DB2® Web Query for System i®.

Documentation Conventions

The following table lists and describes the conventions that apply in this manual.

Convention	Description
THIS TYPEFACE or this typeface	Denotes syntax that you must enter exactly as shown.
this typeface	Represents a placeholder (or variable), a cross-reference, or an important term.
underscore	Indicates a default setting.
this typeface	Highlights a file name or command. It may also indicate a button, menu item, or dialog box option you can click or select.
Key + Key	Indicates keys that you must press simultaneously.
{ }	Indicates two or three choices; type one of them, not the braces.
[ ]	Indicates a group of optional parameters. None is required, but you may select one of them. Type only the parameter in the brackets, not the brackets.
|	Separates mutually exclusive choices in syntax. Type one of them, not the symbol.
...	Indicates that you can enter a parameter multiple times. Type only the parameter, not the ellipsis points (...).
. . .	Indicates that there are (or could be) intervening or additional commands.

Chapter 1. Introducing IBM DB2 Web Query for System i

Provides an overview of IBM DB2 Web Query for System i and describes the end-user components and tools you use to run, view, create, and edit reports.

This documentation provides an overview of IBM DB2 Web Query for System i. It also describes the components and tools you use to run, view, create, and edit reports. Use this documentation to learn about the structure and the capabilities of each component and tool.

IBM DB2 Web Query for System i Overview

IBM DB2 Web Query for System i is a complete, Web-ready data access and reporting system, which takes advantage of the low-cost, low-maintenance, and wide distribution capabilities of the World Wide Web and internal corporate Web sites. This system also provides a streamlined reporting environment that virtually eliminates the complexities of today's corporate data.

Reporting Concepts

The reporting environment includes Domains which are the highest level of organization. Domains provide data on a particular topic (such as sales, inventory, or finance). The data is stored in different forms in the following domain tree components: Reports and Other Files. The Other Files folder, available to Administrators and Developers, can contain images (.gif, .jpg), HTML files, and other files that you can use to customize and enhance reports.

Reporting Features

A selection of reporting tools is available for you to create and edit reports, manipulate data in an existing report, and view a report.

Power Painter. Power Painter is a Web layout and report creation tool that enables you to create output and page layout formats. It combines reporting, graphs, and page layout design in a single tool.

Report Assistant. Report Assistant is an HTML-based tool that you use to create tabular reports. From Report Assistant you select the data that constitute your report, create new data from existing data, apply screening conditions to the data, as well as format and style your report.

Graph Assistant. The Graph Assistant is an HTML-based tool that guides you step-by-step through the creation of a graph. The Graph Assistant enables you to create and style your graphs.

Reporting Interface

The reporting interface is ideal for users who quickly need to run reports. Check your installation manual for information about browser compatibility. This interface offers you the ability to access reporting tools such as Power Painter, Report Assistant, and Graph Assistant.

The reporting interface is an HTML-based front-end that allows you to:
- Run reports.
- Create reports and graphs using Power Painter, Report Assistant, or Graph Assistant.
- Save reports and graphs.
- Share reports with other users.
- Edit reports.

Development Environment

The DB2 Web Query Developer Workbench product is an optional add-on Windows-based development environment for creating applications. Developer Workbench provides intuitive GUI tools that free developers from the hassles of coding, thereby allowing them to concentrate on interface design, business logic, and data manipulation. Using Developer Workbench, developers can build powerful Web page interfaces that allow users to run customized reports.

Chapter 2. Using the Reporting Interface

Describes the reporting interface for IBM DB2 Web Query for System i, which you can use to select a domain, use items (reports, graphs, URLs) in the Domain Tree, search domains, access reporting tools such as Power Painter, Report Assistant, and Graph Assistant, and much more.

When you enter the reporting interface you are automatically connected to IBM DB2 Web Query for System i.

From the reporting interface, you can:
- Select a domain.
- Use the items (reports, graphs, or URLs) in the Domain Tree.
- Access reporting tools such as Power Painter, Report Assistant, and Graph Assistant.

Note:

The browser Back and Forward buttons cannot be used to navigate between pages in the interface, or from an interface page to a page viewed before connecting to the interface.

Opening the Reporting Interface

Before you can access the IBM DB2 Web Query for System i environment, you must log on to the reporting interface.

Log On to the Reporting Interface

1. From the Login page of the reporting interface, click **Login** or enter the following URL in your Web browser.

   `http://hostname:[port]/webquery`

   where:

   hostname:[port]

   Is the host name and optional port number (specified only if you are not using the default port number) where the application is deployed.

2. Enter a valid i5 user ID and password.
3. Click **Submit**. The reporting interface opens.

Recommended Browser Setup

The following browser setup is recommended for use with the interface:
- Do not cache page content.
- Disable pop-ups for reporting environment.

For information on browser setting options, see the browser Help system.

Reporting Interface Layout

The reporting interface contains the following areas:

- **Domain Tree.** Contains the list of reports that you can access. You can expand the Domain Tree to view the entire name of an item by dragging the control bar that separates the Domain Tree from the content area.
- **Content area.** Contains content when reports are displayed.

In the following image of the interface, the:
- Domain Tree is on the left side of the window.
- Content area is on the right side of the window.
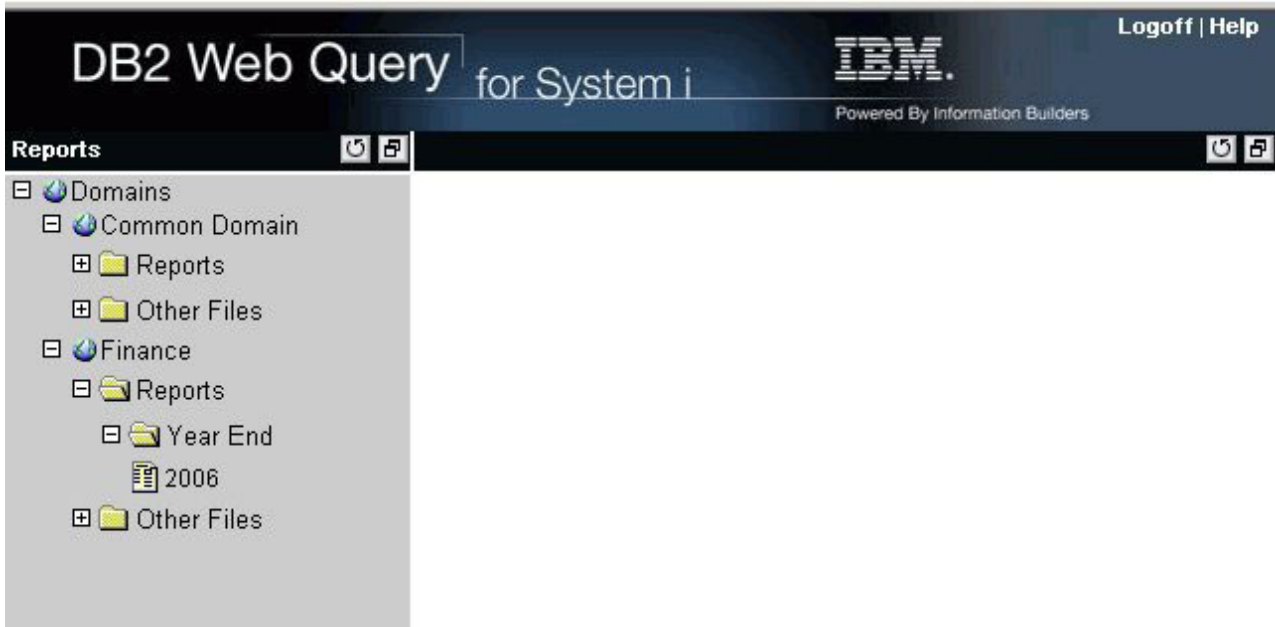
Selecting a Domain

All of the domains that a user is authorized to access are displayed in the Domain Tree. Depending on how your user profile is set up, you may not have access to all domains.

The icon located next to an item in the Domain Tree represents the type of item. In the following table, the first column lists the icon, and the second column describes what the icon represents.

Icon	Identifies...
- Domain Tree	Highest level of the report organization containing folders and reports you can access.
- Reports Folders	Location of created reports.
- Reports	Reports in a folder.
- Internet Links	Web pages and reports run from launch pages.

Using Domain Tree Items

A domain can contain reports, which are located in the Reports folders.

When you select an item from one of the Reports folders, a menu appears; it allows you to select one of the available options. The option(s) for Reports (identified by the Report icon) include Run, Power Painter, Report Assistant, Graph Assistant, Delete, and Properties.

When you use the Delete option to delete a report, the Domain Tree refreshes and collapses. If the report is the only item in the folder, the folder is also deleted when you delete the report. You also have the option of deleting a folder including all of its contents.

Note:

- If you are running very large reports, you may need to increase the virtual memory on your machine. See your System Administrator for details.
- A menu does not appear if you are not an Administrator or a Developer.

Loading Domain Tree Folders

When a user opens one of the default folders in the Domain Tree, a request is sent to retrieve subfolders directly under the top level folder. As each node of the tree is expanded, only that section of the tree is populated. This reduces the time needed to create the tree when a Domain is selected.

When reports from a folder are loaded into the Domain Tree, only a maximum of 25 report items are loaded each time. A link labeled View More is displayed in the tree to enable users to retrieve up to an additional 25 items.

Creating Reports

In addition, you can create new reports from scratch in the Reports folder. You can create your own reports using Report Assistant, Graph Assistant, or Power Painter (if you have the required privilege). Depending on which tool you use to create your report, you can edit your report using the same tool (Power Painter, Report Assistant, or Graph Assistant). You can change the name of your report from the Properties window.

Note: You may not be able to create reports if you do not have the privileges to do so.
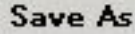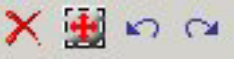
Create a Report in the Interface

1. In the Domain Tree, expand the **Reports** folder.
2. Right-click a subfolder and select **Power Painter**, **Report Assistant**, or **Graph Assistant**.
3. Create and save your report. The report is saved in the Reports folder with the name you provide.

Text Editor

Administrators and Developers only can use the text editor to troubleshoot source code for reports in the reporting interface.

The following table describes all of the functions available in the text editor. The first column lists the buttons, and the second column lists the actions they produce.

Button	Action...
Save - Save button	Saves the report. Only appears if the Text Editor is opened from the Reports subfolder.
Save As - Save As button	Saves the report in the Reports folder with a name you specify.
Run - Run button	Runs the current report.
Quit - Quit button	Exits the Editor window. If you made changes to the original report, a window prompts you to save or cancel the changes.
Help - Help button	Opens the online help.
- Cut, Copy and Paste buttons	Cut, copy, or paste, respectively, the highlighted text.
- Delete, Select All, Undo, and Redo buttons	Delete, select all, undo, and redo, respectively.
- Find and Replace button	Find and replace text.
- Go to line button	Enables you to go to a particular line number in the report.
AB - Uppercase button	Converts highlighted text to uppercase.
ab - Lowercase button	Converts highlighted text to lowercase.
- Text color button	Sets the text color in the editor.

Button	Action...
- Background color button	Sets the background color of the editor. **Note:** The text color and background colors are for the current session only.
Times New Roman - Font drop-down menu	Enables you to change the font of the editor.
12 - Font size drop-down menu	Enables you to change the font size of the text in the editor.

Note:

- If you create a report using the Power Painter, Report Assistant, or Graph Assistant, and then edit for troubleshooting with the text editor, you can only open and edit the report using the text editor.
- If you use a Firefox browser, the editing toolbar, which begins with the cut, copy, paste features listed in the table, does not appear. Use the standard control keys (Ctrl+X, Ctrl+C, Ctrl+V) to cut, copy, and paste.
- When you click the X in the upper right corner of the text editor, the procedure is saved, however the Domain Tree does not refresh. To refresh the Domain Tree contents, click the refresh button (circle with arrow) in the control bar.

Using Banner Hyperlinks

The hyperlinks in the banner provide a way to easily navigate the reporting interface:

- **Logoff.** Allows you to logoff the interface.
- **Help.** Opens a menu where you can select:
  - **Contents and Index**, which opens the online help.
  - **About DB2 Web Query**, which provides release information.

Chapter 3. Using Power Painter

Describes how to use the Power Painter to create reports, graphs, and page layouts.

Power Painter is a Web layout and report creation tool (available with DB2 Web Query), built using AJAX technology, that enables you to create output formats and page layout formats. It combines reporting, graphs, and page layout design in a single tool.

Functionality of Power Painter

You can accomplish the following tasks and functions using the Power Painter tool:

- Create a Compound Report, which is a reporting document with multiple reports and charts.
- Create a Coordinated Compound Report, which is a compound document where the report is collectively sorted and collated by a common sort group.
- Create exact positioning requirements by adjusting the layout properties in the canvas.
- Position components and report elements relative to each other and the page.
- Style reporting elements individually by applying a Style Sheet, theme, or conditional styling rules.
- Access and create Compound Reports using multiple data sources to build reports and perform ad hoc analysis.
- Develop reports using real-time data or sample data and limit the number of records for the entire report, individual reports, or individual graphs.

Selecting and Adding a Data Source

You can use Power Painter to access a Data Source to build reports and perform ad hoc analysis. You can open and work with Master Files by using the Data Sources palette in Power Painter.

Open a Data Source

1. Upon opening the Power Painter tool, you are prompted to select a Data Source. Select a Master File and click **OK**.
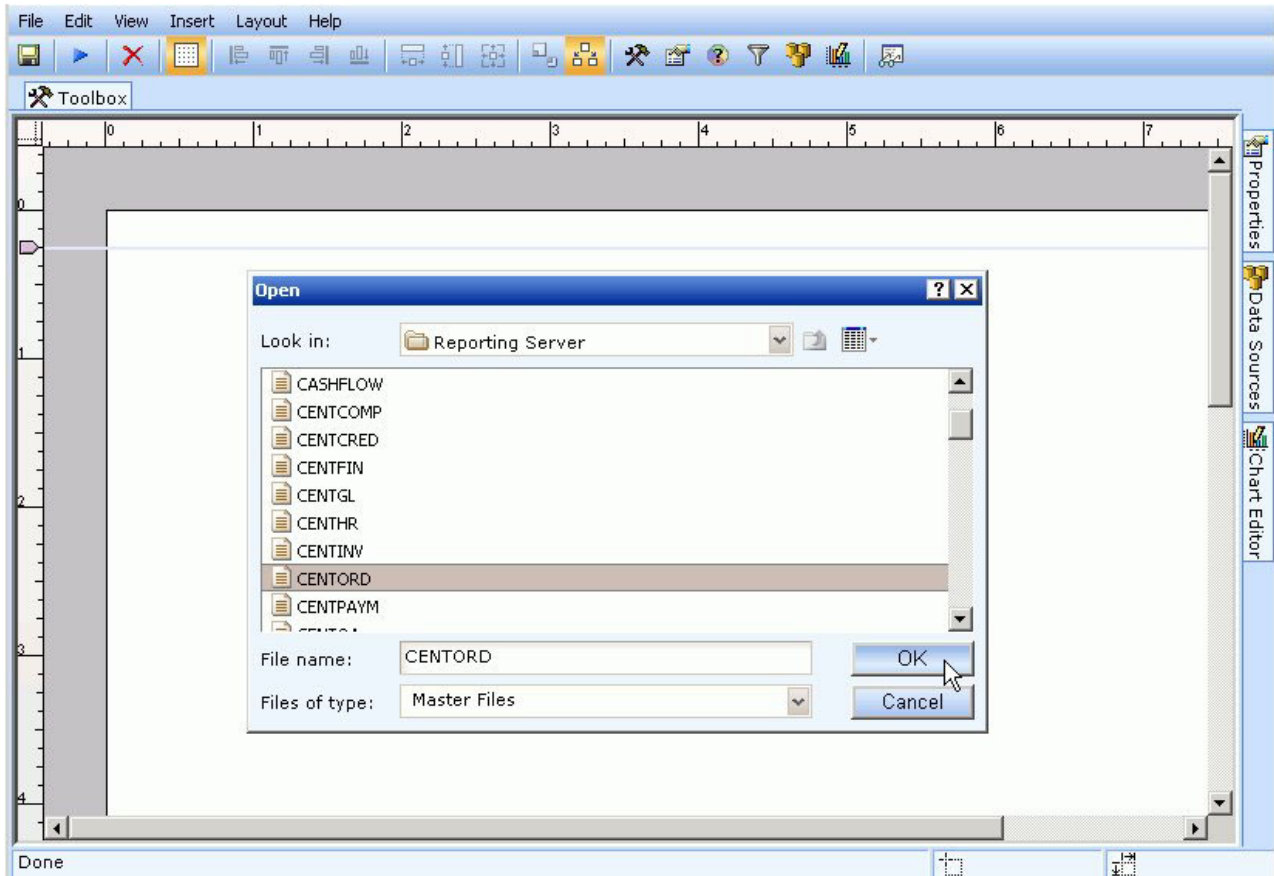
The fields from the Data Source are available in the Data Sources palette.

Note: This step is optional. You can open or close Data Sources in Power Painter at any time.

2. To open a new Data Source, click the **Open a new data source** button from the Data Sources palette toolbar.

The Open dialog box appears prompting you to select a Data Source.

Close a Data Source

Note: If there is a component on the canvas that is referencing data from an open Data Source, you will not be able to close the Data Source unless you delete the associated component.

To close an open Data Source, click the **Close current data source** button from the Data Sources palette toolbar.

The fields are removed from the Data Source palette.

Using Power Painter Options and Palettes

The following image is an example of the Power Painter tool with multiple components. Several options and palettes enable you to customize the layout for the Power Painter tool.
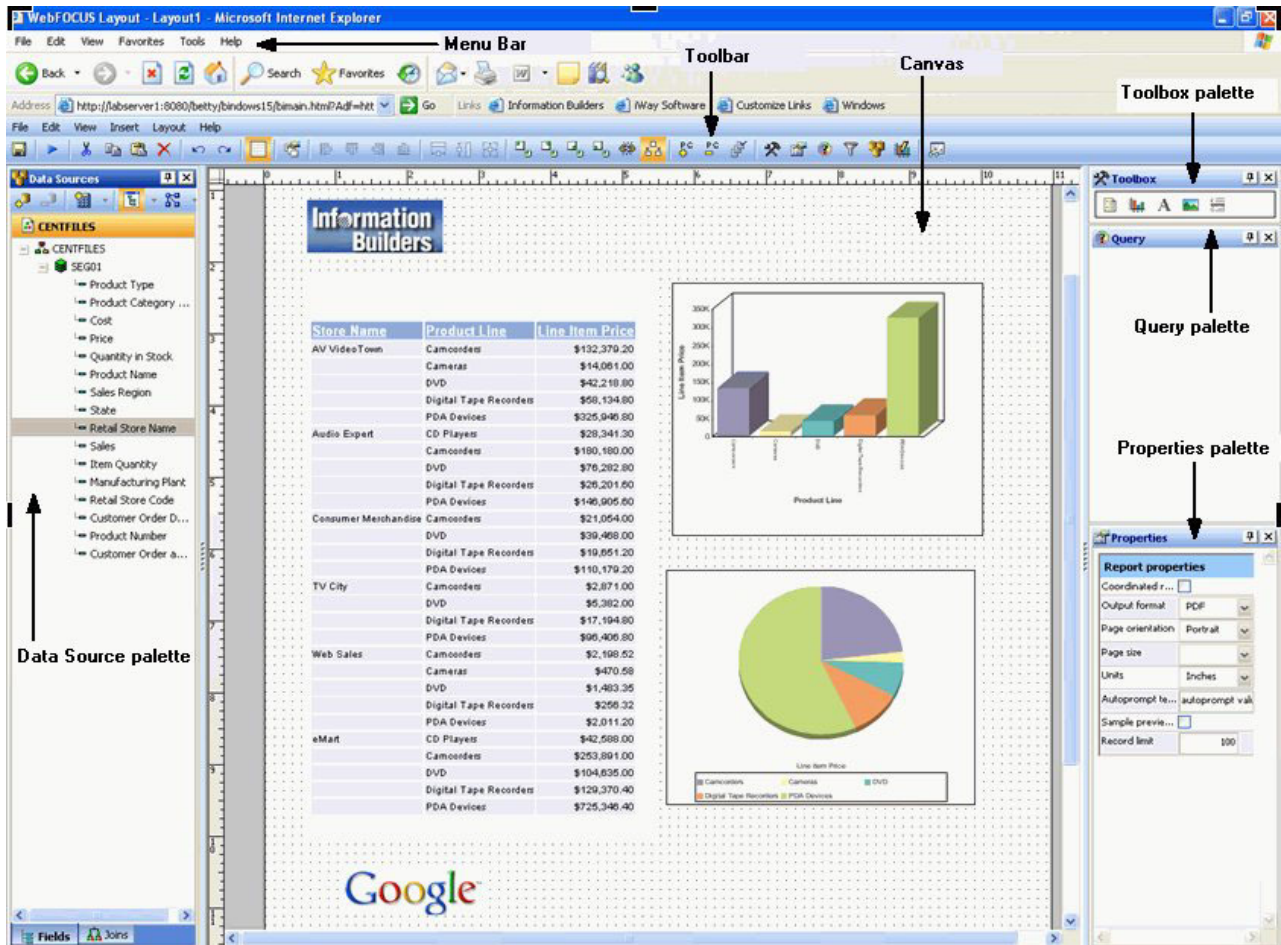
The following options are available in Power Painter:

Menu bar

> Standard browser menu bar options such as File, View, Tools, and so on.

Toolbar

> Power Painter toolbar options such as Layout, Run, View, Alignment, Palettes, and so on.

Canvas

> The canvas where you drop objects and paint the reporting document. The reporting document can be made up of text elements, images, tabular report objects, charts, and lines.

The following palettes are available in Power Painter:

Toolbox palette

Contains the tabular report, chart, text, image, and line objects for painting your document.

Query palette

Enables you to build and customize queries for tabular reports and charts.

Data Sources palette

Contains your Data Source information.

Properties palette

Contains the property settings for the components in the canvas.

Using Palettes in the Canvas

Upon opening the Power Painter tool, the canvas appears with all of the palettes minimized and pinned in the Power Painter margins. You can show, hide, or move these palettes as needed. Power Painter retains these settings.

Power Painter is able to store information so that the options you set in a single session are remembered and applied the next time you use the tool. Examples of the settings that can be saved are how the palettes are displayed, the theme selected, field display options, and so on. The information for these settings is stored under each user's directory in the DB2 Web Query repository.

Note: Reusing stored settings is only available if you access the same DB2 Web Query environment.

Palettes in Power Painter:

- **Resizable.** You can change the height and width by dragging the sides of the palette.
- **Dockable.** You can dock the palettes to the sides of the screen (top, bottom, left, right) or float them anywhere in the work area.
- **Stackable.** If multiple palettes are docked on the same side of the canvas, they become stacked.
- **Pinnable.** You can minimize the palettes and pin them in the Power Painter margins.

Dock and Undock a Palette

1. Required: Double-click a palette to dock it on the canvas.

   or

   Drag a palette within 10 pixels of the edge (top, bottom, left, right) of the canvas.

   Note: A palette is docked when both the pin and close button are available from the palette toolbar.

2. To undock a palette (float it), click the palette toolbar, left-click and hold your mouse, and drag the palette to a new location on the canvas.

Stack Palettes

Drag and dock multiple palettes to one side of the canvas to create vertical or stacked palettes. You can adjust the height/width of each palette in the docking area.

Note: Resizing any palette automatically resizes the rest of the palettes so that they can fit into the docking area.

The following image is an example of vertically stacked palettes in the Power Painter tool.
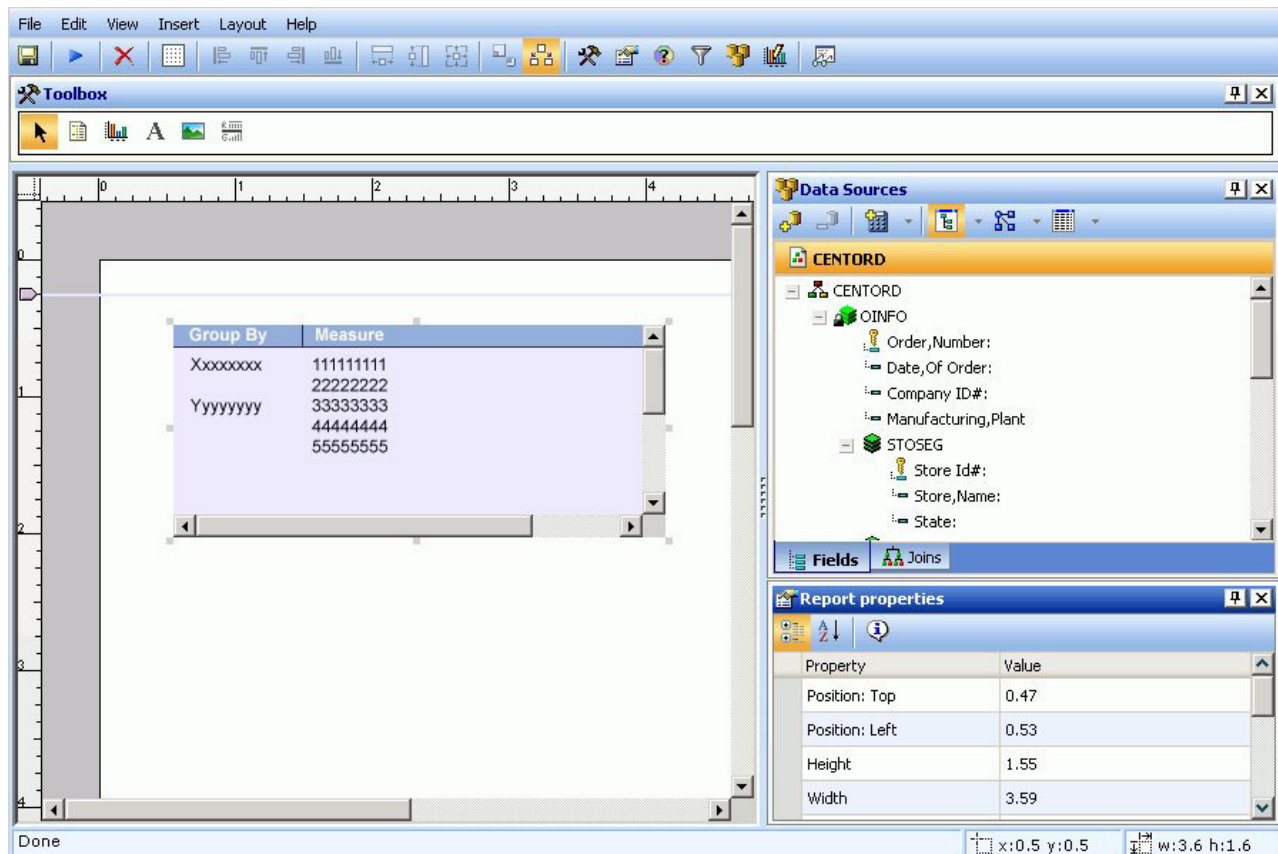
Pin a Palette

1. Required: Click the **pin** button located on the toolbar of an open palette. The palette is pinned and minimized.
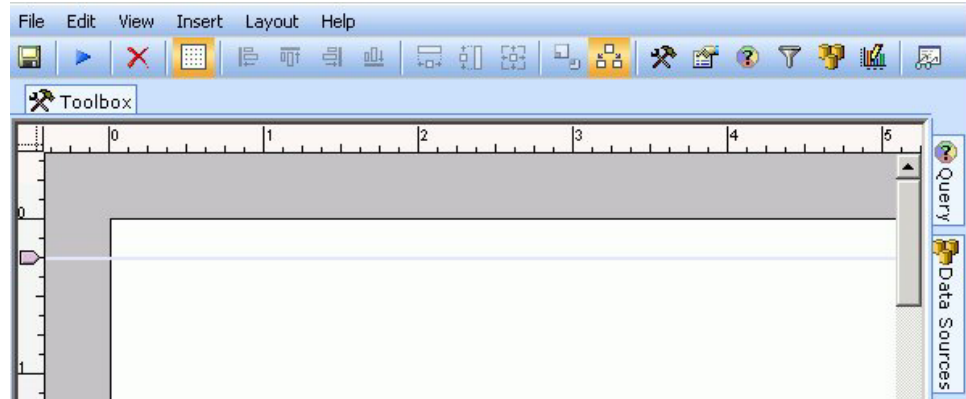
2. You can perform the following actions with pinned palettes:
   - Hover your mouse over the palette to show its contents.
   - Hover your mouse out of the palette to minimize it again.
   - Double-click a palette to dock it on the canvas.

Show a Palette

1. Select **Palettes** from the View menu.

   Palettes that are checked are shown in the canvas.

2. Select an unchecked palette name.

   The palette is shown maximized on the canvas. You can move or resize the palette as needed.

Close a Palette

1. Required: Select **Palettes** from the View menu.
2. Required: Select a checked palette name from the list.

   The palette is closed and removed from the canvas.

Note: You can also close an open palette by clicking the **Close** button from the palette toolbar.

Reset the Power Painter Layout

From the View menu, select **Reset Layout** from the Palettes submenu.
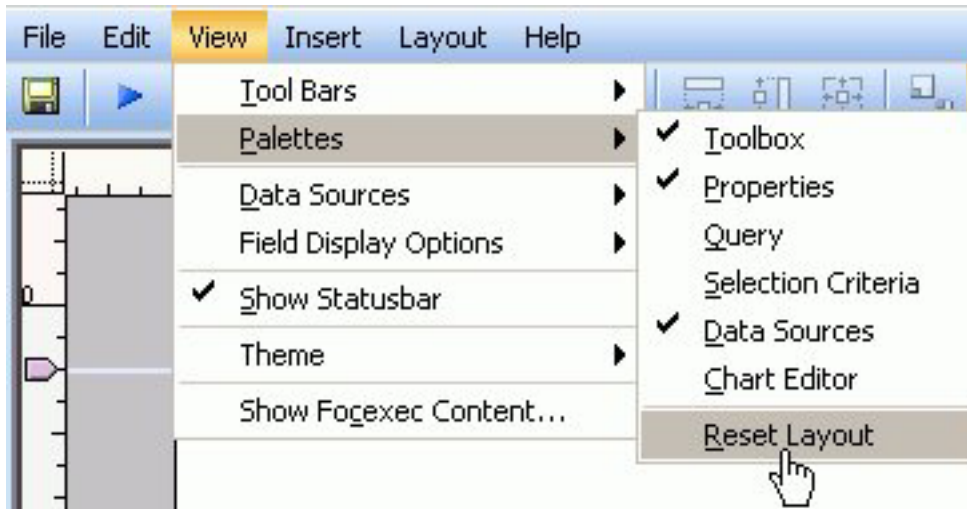
The Power Painter layout resets with the default palettes shown (Toolbox, Properties, Data Sources) in the canvas.

Using Power Painter Components

The available Power Painter components are Report, Graph, Image, Text, and Line. You can select these components from the Toolbox palette or from the Main toolbar. The component objects are associated with corresponding properties that you can set using the properties palette.

The following image is an example of the Toolbox palette in Power Painter.

Insert a Report Component in Power Painter

1. Required: Select the **Report** button from the Toolbox palette and drag it onto the canvas.

   or

   Select **Report** from the Insert menu. The component is automatically inserted into the canvas.

   The report appears on the canvas.

2. From the Data Sources palette, select a field and drag it to the report object.

   When you drag a field into the report, drop the field on the By, Across, Sum, or Coordinated folder.

   You can also double-click a field to automatically add it to the report. Numeric fields, such as decimal and floating point, are added to the Sum folder. Non-numeric fields, such as alphanumeric, and date fields are added to the By folder.

   Note: By and Across folders are for sorting, Sum folders are for aggregating and displaying, and Coordinated folders contain a common sort field for

creating multiple reports and graphs that are burst into separate page layouts. Each value for a coordinated sort field displays on a separate page.
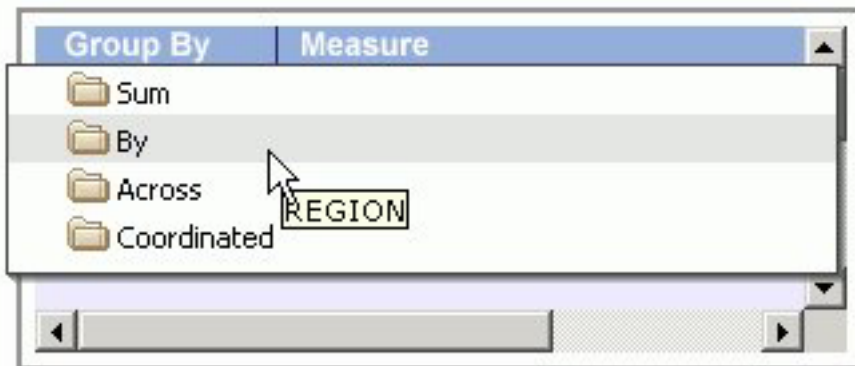
3. Required: To edit or delete these field selections, open the Query palette or the Query Popup.
   - From the View menu, select **Query** from the Palettes submenu.
   - Select a report, right-click and select **Query Popup** from the context menu.
     or
     Double-click the report to bring up the Query Popup.
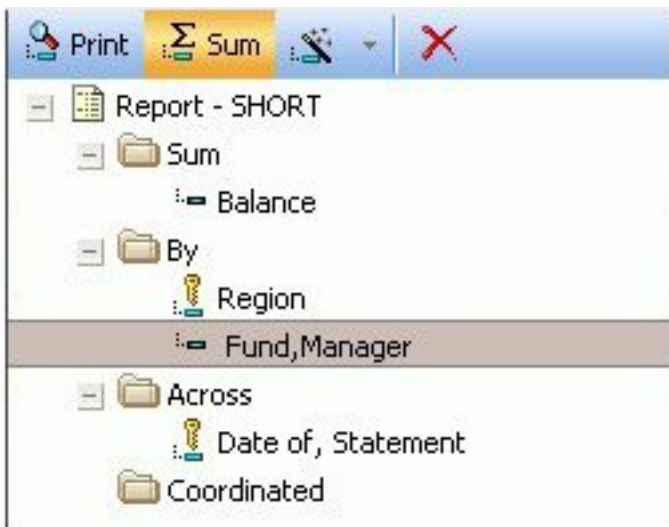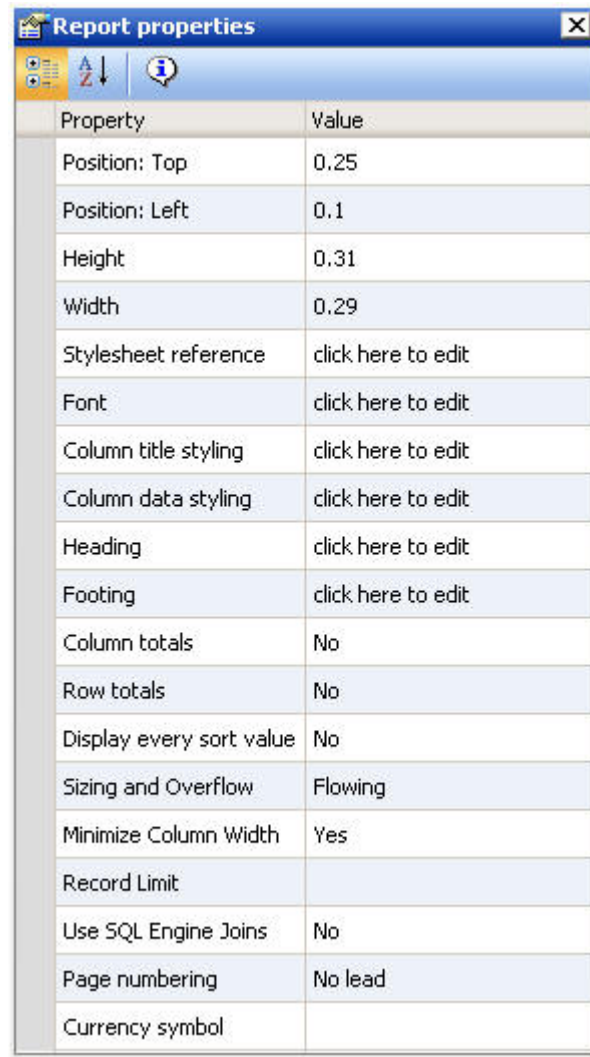
The By, Sum, Across, and Coordinated folders appear with the selected fields in the report. You can drag and drop fields into different folders, delete fields, and add and edit compute fields from this dialog box. You can also select **Print** to display all of the values found in the data source for a selected field, or select **Sum** to sum the values of a selected numeric field.

4. You can select a field from the Query palette to further customize the field properties in the report.
5. You can select the report to view or edit the properties for the report in the Report properties palette.
6. Select **Save** from the File menu to save the report.

Report Properties

When a report object is selected, the following properties are available from the Report properties palette.

Property	Value
Position: Top	0.25
Position: Left	0.1
Height	0.31
Width	0.29
Stylesheet reference	click here to edit
Font	click here to edit
Column title styling	click here to edit
Column data styling	click here to edit
Heading	click here to edit
Footing	click here to edit
Column totals	No
Row totals	No
Display every sort value	No
Sizing and Overflow	Flowing
Minimize Column Width	Yes
Record Limit	
Use SQL Engine Joins	No
Page numbering	No lead
Currency symbol	

Position: Top

Indicates the starting position for the top edge of the report. You can type in a different value to change the position or manually drag the report on the canvas.

Position: Left

Indicates the starting position for the left edge of the report. You can type in a different value to change the position or manually drag the report on the canvas.

Height

Indicates the height of the report object. You can type in a different value to change the height or manually drag the report object on the canvas.

Width

Indicates the width of the report object. You can type in a different value to change the width or manually drag the report object on the canvas.

Stylesheet: reference

Click the browse (...) button to select an external stylesheet. The report references the selected stylesheet which controls the global level styling of the report. You can choose from a predefined set of stylesheets, or a stylesheet from the Import directory.

Font

Shows the default global font for the report.

Column title styling

Click the browse (...) button to open the font dialog box. Select the font, style, size, text, and background color for the column titles in the report.

Column data styling

Click the browse (...) button to open the font dialog box. Select the font, style, size, text, and multiple background colors for the data in the report.

Heading

Adds a heading to the report. Click the browse (...) button to open the Heading dialog box.
- You can use the Headings toolbar to customize and modify the font, style, size, text color, background color, and so on.
- You can use the Data Sources area to add fields from the Master File as your heading.

Footing

Adds a footing to the report. Click the browse (...) button to open the Footing dialog box.
- You can use the Footings toolbar to customize and modify the font, style, size, text color, background color, and so on.
- You can use the Data Sources area to add fields from the Master File as your footing.

Column totals

Shows columns totals in the report. No is selected by default. To show column totals, use the drop-down list to select **Yes**.

Row totals

Shows row totals in the report. No is selected by default. To show row totals, use the drop-down list to select **Yes**.

Display every sort value

Determines when all instances of a vertical sort (By) field are displayed in the report. No is selected by default to omit duplicates. To display every instance of a vertical sort field value, use the drop-down list to select **Yes**.

Sizing and Overflow

Determines the vertical sizing of the report by setting fixed or flowing overflow. For more information about sizing and overflow, see Setting Overflow Options in Power Painter.

Minimize Column Width

Shrinks the column width to fit the widest value in the report. Yes is selected by default. Select **No** from the drop-down list to expand all of the columns.

Record Limit

Limits the number of records used to retrieve data at run time.

Use SQL Engine Joins

Enables the database to use its internal optimization techniques which reduce the volume of database-to-server communications and improve response time.

Page numbering

Select On, Off, or No lead from the list box.

Currency symbol

Overrides the default currency symbol for the reporting server. Use the drop-down list to select from Default, USD, GBP, JPY, EUR, or $. You can also type a one-character custom symbol directly in the value column adjacent to the Currency symbol property.

Right-click Report Properties

When you right-click a report object, the following options are available from the dialog box that appears:

Query

Opens the query dialog box.

Size to Fit

Resizes the report as fields are added and deleted.

Delete

Deletes the report.

Properties

Displays the Report properties dialog box.

Field Properties

When a report is selected, the following properties are available from the Field properties palette.

Note: The field properties available vary depending on the type of sort field selected.

Field properties: REGION

Property	Value
Visible	Yes
Title	
Sorting	
Total	No
Page break	No
Subtotal	click here to edit
Underline	No
Include missing instan…	No
Drill down	click here to edit
Conditional styling	Click here to add a rule
Subheading	click here to edit
Subfooting	click here to edit
Ranking	No
Ranking limit	
Ranking title	
Ranking title font	click here to edit
Ranking data font	click here to edit
Title font	click here to edit
Data font	click here to edit

Visible

Controls whether a field appears in the report graph, or is hidden. If Visible is set to No, the column is hidden, but is still used in calculations.

Title

Enables you to enter a title for the field in the report graph.

Sorting

Sorts the column data in ascending or descending order.

Total Applies aggregation and sorting to numeric columns in one pass of the data and is available for Sort by and Sort across fields. You must have an aggregating display field, such as Sum, in the report.

Page break

Starts a new page when the value of a selected sort field changes.

Subtotal

Enables you to display subtotals for numeric data whenever the value of the selected sort (By) field changes. By default, numeric field values are summed and displayed as subtotals. Additionally, you can display other

summary values by applying prefix operators to numeric fields, including computed fields, by clicking the browse (...) button to open the Sub Total Options dialog box. For more information, see Display Subtotals and Other Summary Values Using Prefix Operators.

Underline

Includes a horizontal line across the width of a report after the value of the selected sort field changes.

Include missing instances

Displays missing instances of data for related fields in the report.

Drill down

Enables you to supply information to another report by using a hyperlink, procedure, or parameter.

Conditional Styling

Enables you to assign a conditional style to a report element. Conditional styling enables you to define conditions that determine when to apply particular fonts, text style, and so on. For more information about conditional styling, see Create Conditional Styling Rules.

Subheading

Adds a subheading to the report at sort breaks. Click the browse (...) button to open the subheading dialog box.

- You can use the subheadings toolbar to customize and modify the font, style, size, text color, background color, and so on.
- You can use the Data Sources area to add fields from the Master File as your subheading.

Subfooting

Adds a subfooting to the report at sort breaks. Click the browse (...) button to open the subfooting dialog box.

- You can use the Footings toolbar to customize and modify the font, style, size, text color, background color, and so on.
- You can use the Data Sources area to add fields from the Master File as your footing.

Ranking

Includes a column with a numeric rank for each row in a vertically sorted report.

Ranking limit

If specified, this value limits the number of ranked values to be retrieved.

Ranking title

Displays a default column title as RANK with the sort field. You can type in a unique title name for the ranked field.

Ranking title font

Click the browse (...) button to open the font dialog box. Select the font, style, and size for the ranking title.

Ranking data font

Click the browse (...) button to open the font dialog box. Select the font, style, and size for the ranking data.

Title font

Click the browse (...) button to open the font dialog box. Select the font, style, size, text and multiple background colors for the column title.

Data font

Click the browse (...) button to open the font dialog box. Select the font, style, size, text and multiple background colors for the data column.

Table of Contents

Enables you to generate a Table of Contents (TOC) page which shows a summary of the document's contents, along with page numbers, and can be printed with the document. The entries in the Table of Contents enable you to easily navigate to a particular section while viewing the document.

Note: The Table of Contents only appears as a field option if you are using a coordinated report.

Insert a Graph Component in Power Painter

1. Select the **Graph** button from the Toolbox palette and drag it onto the canvas.

   or

   Select **Graph** from the Insert menu.

   The New Chart dialog box appears.

Vertical Clustered Bars: Side by side groups of bars. The standard type of two-dimensional bar chart. Any series can optionally be displayed as line or area rather than a bar.

☐ CrossTab   < Back   Finish   Cancel
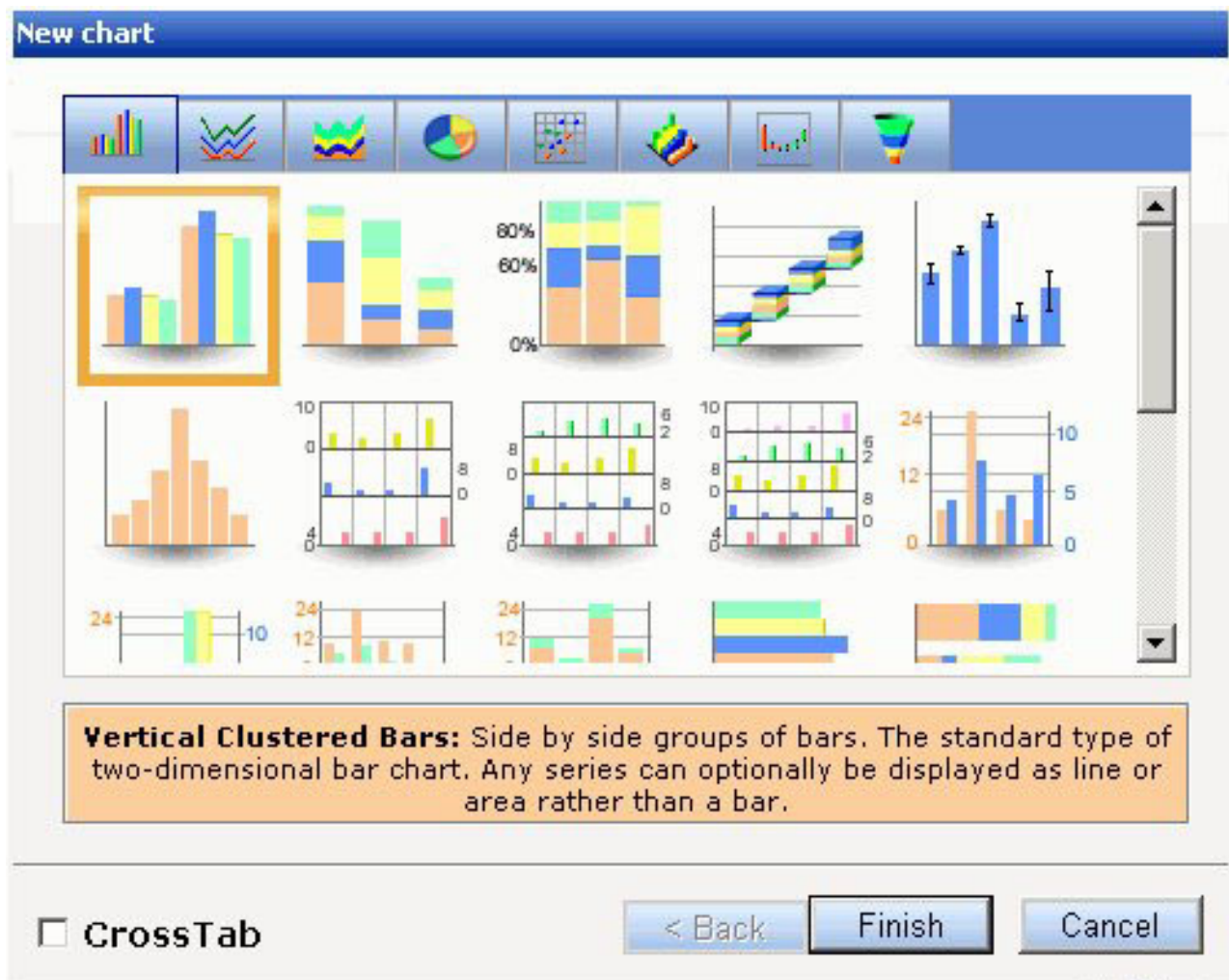
2. Select a chart type and click **Finish**.

   The graph appears on the canvas.

   Tip: To view the New Chart dialog box after the graph has been added to the canvas, select **Show Chart Types** from the right-click context menu of the graph.

3. From the Data Sources palette, select a field and drag it to the graph object.

   When you drag a field into the graph, drop the field on the Y axis, X group axis, Coordinated, or other specialized field container.

   You can also double-click a field to automatically add it to the graph. Numeric fields, such as decimal and floating point, are added to the Y axis (or other specialized) container. Non-numeric fields, such as alphanumeric, and date fields are added to the X group axis (or other specialized) container.

   Note: The Coordinated field container contains a common sort field for creating multiple reports and graphs that are burst into separate page layouts. Each value for a coordinated sort field displays on a separate page.
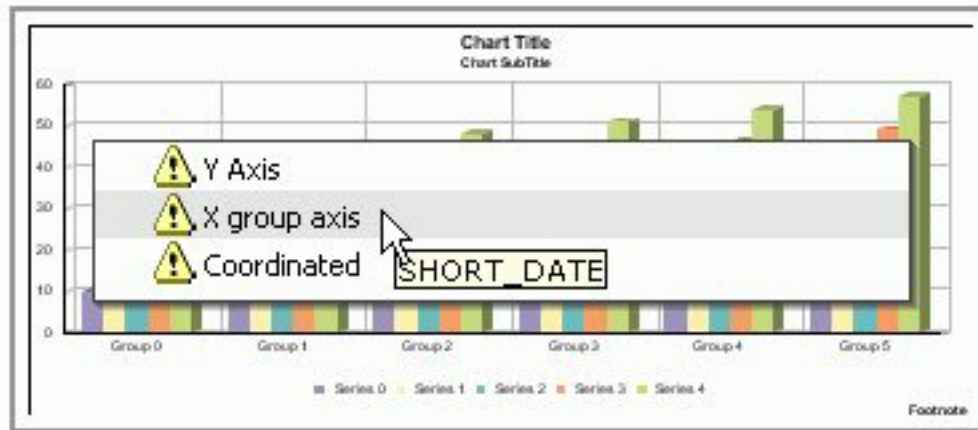
4. Required: To edit or delete these field selections, open the Query palette or the Query Popup.
   - From the View menu, select **Query** from the Palettes submenu.
   - Select a graph, right-click and select **Query Popup** from the context menu.
     or
     Double-click the graph to bring up the Query Popup.
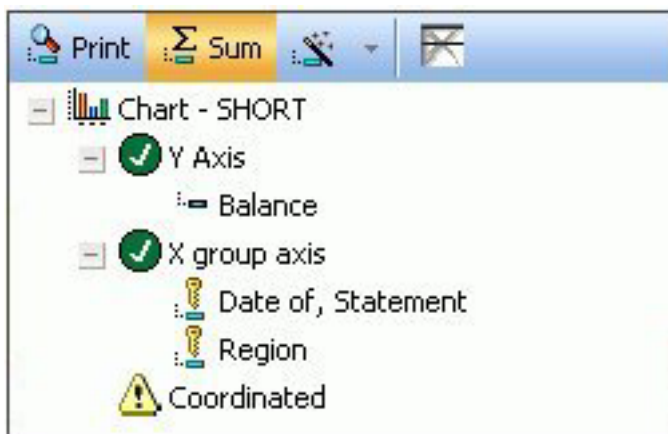
The Y axis, X group axis, and Coordinated field containers appear with the selected fields in the graph. Note that some specialized graphs have other field containers. You can drag and drop fields into different containers, delete fields, and add and edit compute fields from this dialog box. You can also select **Print** to display all of the values found in the data source for a selected field, or select **Sum** to sum the values of a selected numeric field.

5. You can select a field from the Query palette to further customize the field properties in the graph.
6. You can select the graph to view or edit the properties for the graph in the Chart properties palette.
7. From the View menu, select **Chart Editor** from the Palettes submenu to further customize the chart titles, legends, axis, and so on.
8. Select **Save** from the File menu to save the graph.

Chart Properties

For most chart types, when a graph object is selected, the following properties are available from the Chart properties palette. **Note:** Some chart types with unique properties, such as pie and three dimensional charts, have specialized properties not shown below.

Property	Value
Crosstab Mode	No
Record Limit	0
Use SQL Engine Joins	No
Top	1.0
Left	1.0
Height	4.0
Width	3.0
Apply template	SouthWestern
Chart title	Chart Title
Chart subtitle	Chart Subtitle
Chart footnote	Footnote
Use 3D effect	Yes
Legend position	Auto
Show data values	No
Y-axis Major grid	Yes
X-axis Major grid	Yes
Fit to Chart area	Yes

Crosstab Mode

Select from Yes or No. Displays the Series-legend icon and enables secondary X-axis functionality.

Record limit

Type a number to edit the record limit value.

Use SQL Engine Joins

Enables the database to use its internal optimization techniques which reduce the volume of database-to-server communications and improve response time.

Top

Indicates the starting position for the top edge of the graph. You can type in a different value to change the position or manually drag the graph on the canvas.

Left

Indicates the starting position for the left edge of the graph. You can type in a different value to change the position or manually drag the graph on the canvas.

Height

Indicates the height of the graph object. You can type in a different value to change the height or manually drag the graph object on the canvas.

Width

Indicates the width of the graph object. You can type in a different value to change the width or manually drag the graph object on the canvas.

Apply template

Select from any of the available templates to apply a predefined color scheme to change the appearance of your chart.

Chart title

Type the desired title text.

Chart subtitle

Type the desired subtitle text.

Chart footnote

Type the desired footnote text.

Use 3D effect

Select from Yes or No to apply or not apply a three dimensional effect on the graph.

Legend position

Select the desired option for positioning the chart's legend.

Show data values

Select from Yes or No to display or not display data values on the graph.

X-axis Major grid

Select from Yes or No to display or not display X-axis major gridlines on the graph.

Y-axis Major grid

Select from Yes or No to display or not display Y-axis major gridlines on the graph.

Fit to Chart area

Select from Yes or No to turn auto placement on or off.

Chart Editor Dialog Box

When the Chart Editor dialog box is open in Power Painter, properties are available for each object in the chart editor. The following image is an example of the Chart Editor and the properties palette in Power Painter.
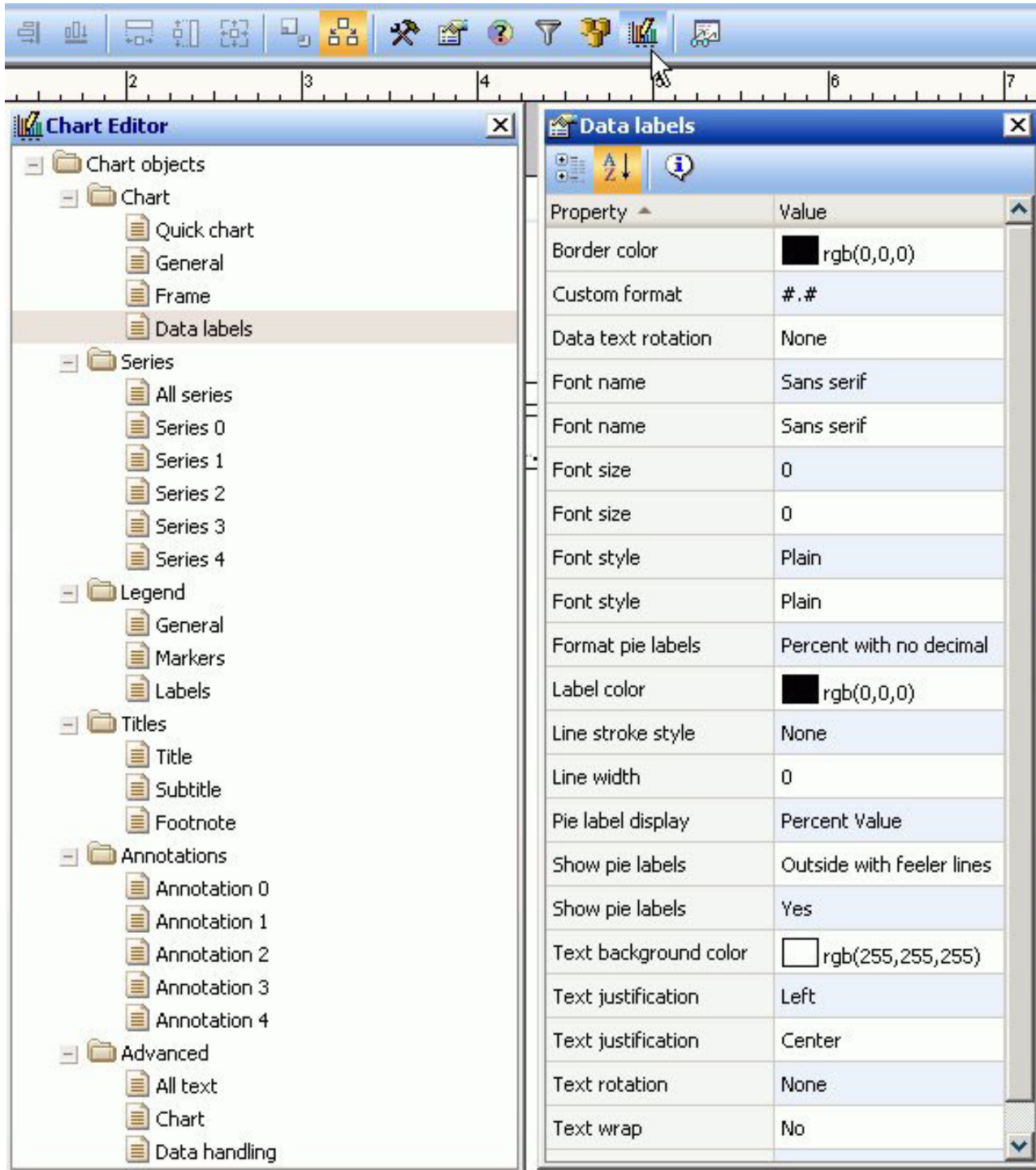
Insert a Text Component in Power Painter

1. Required: Select the **Text** button from the Toolbox palette and drag it onto the canvas.

   or

   Select **Text** from the Insert menu.

   The Text Component dialog box appears on the canvas.
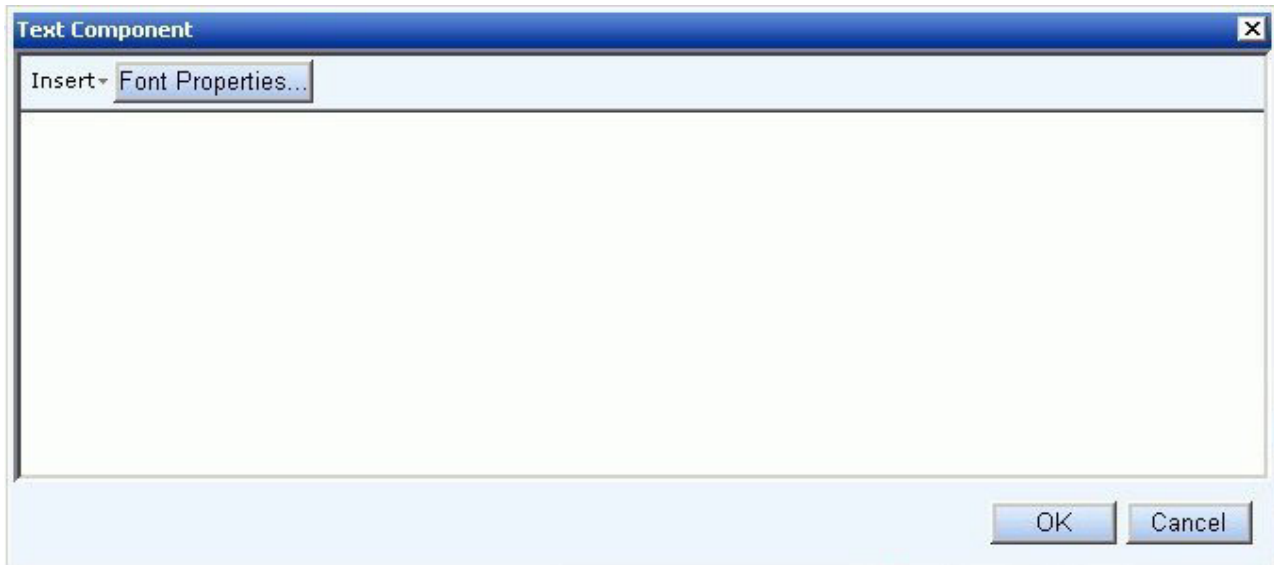
2. Manually type text into the text box area to use the default font.
3. Click the **Font Properties** button to open the Font dialog box.

Font ? ✕

Font...	Style:	Size
ARIAL	Normal	10

ARIAL ▲	Normal ▲	8 ▲
ARIAL UNICODE MS	Bold	9
AVANT GARDE GOTHIC	Italic	10
BATANG	Underline	12
BATANGCHE	Bold and italic	14
BOOKMAN ▼	Bold and und ▼	24
◄ ►	◄ ►	36 ▼

Options

Text Color ■

Sample Text

Sample Text Line 1

OK   Cancel
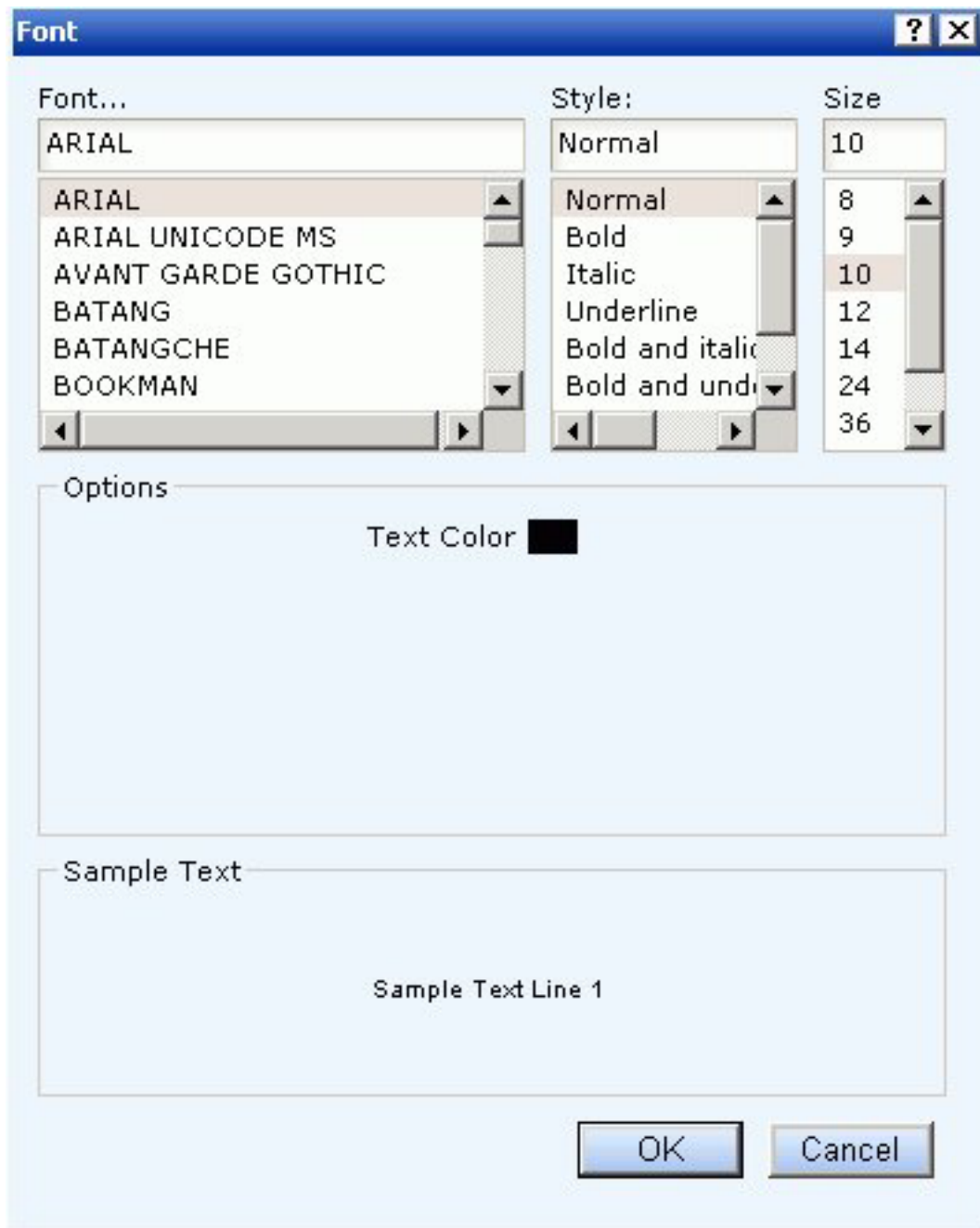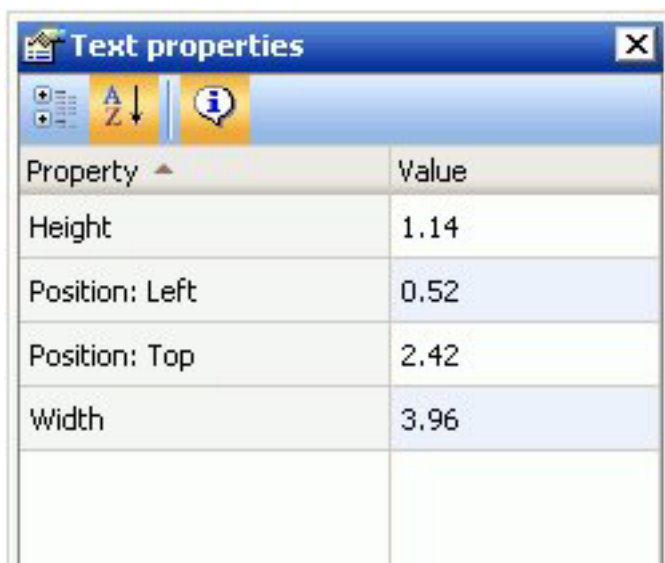
You can customize the font, style, size, and text color of the text.

4. Click **OK** to close the Font dialog box.
5. Required: Click **OK** to close the Text component dialog box.

The text is inserted into the canvas.

6. You can select the text component to view or edit the properties for the text in the Text properties palette.

7. Select **Save** from the File menu to save the text.

Text Properties

When a text object is selected, the following properties are available from the Text properties palette.

Height

Indicates the height of the text object. You can type in a different value to change the height or manually drag the text object on the canvas.

Position: Left

Indicates the starting position for the left edge of the text.

Position: Top

Indicates the starting position for the top edge of the text.

Width

Indicates the width of the text object. You can type in a different value to change the width or manually drag the text object on the canvas.

Insert an Image Component in Power Painter

1. Select the **Image** button from the Toolbox palette and drag it onto the canvas.

   or

   Select **Image** from the Insert menu.

   The Open dialog box appears on the canvas.

2. Select from the list of image files available.

   Note: The Reporting Server is the default location for all stored images.

   Images need to be copied to the appropriate directories by your administrator.

3. Required: Click **OK** to insert the image. The image is added to the canvas, and the size defaults to the size of the image.

4. You can select the image component to view or edit the properties for the image in the Image properties palette.

5. Select **Save** from the File menu to save the image.

Right-click Image Properties

When you right-click an image, the following options are available from the dialog box that appears:

Reset to original size

Resets the image to the original size.

Deletes

Deletes the image.

Properties

Opens the Image properties dialog box.

Note: Hover over an image to view the file name.

Image Properties

When an image object is selected, the following properties are available from the Image properties palette.
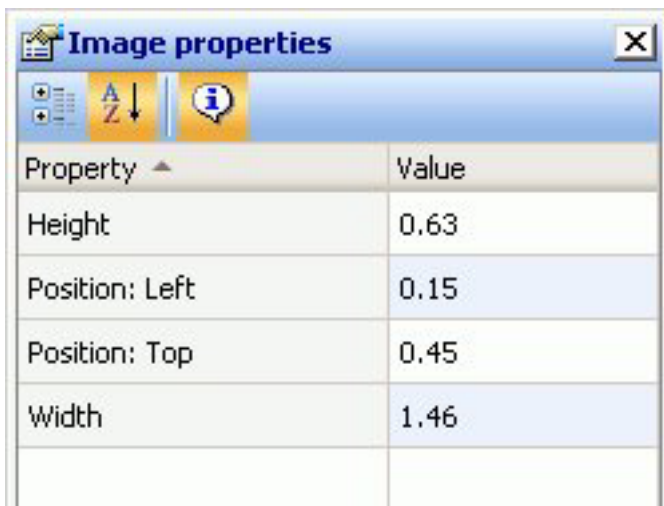
Height

Indicates the height of the image object. You can type in a different value to change the height or manually drag the image object on the canvas.

Position: Left

Indicates the starting position for the left edge of the image.

Position: Top

Indicates the starting position for the top edge of the image.

Width

Indicates the width of the image object. You can type in a different value to change the width or manually drag the image object on the canvas.

Insert a Line Component in Power Painter

1. Required: Select the **Line** button from the Toolbox palette and drag it onto the canvas.

   or

   Select **Line** from the Insert menu.

The line object is added to the canvas as a positioned horizontal line.
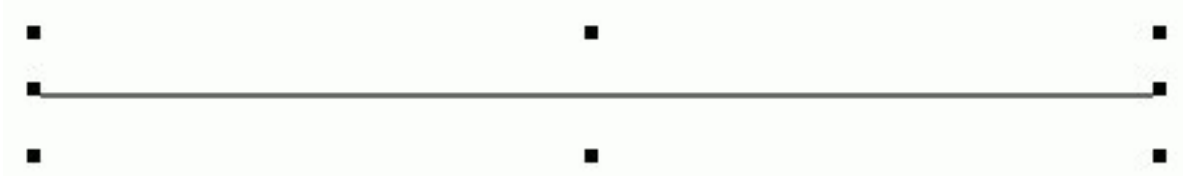
> **Note:** To insert a vertical line, use the properties palette to change the orientation to vertical.

2. You can select the line component to view or edit the properties for the line in the Line properties palette.
3. Select **Save** from the File menu to save the line.

Line Properties

When a line object is selected, the following properties are available from the Line properties palette.

Color

> Sets the color of the line. Click in the color value field to open the color dialog box.

Height

> Indicates the height of the line object. You can type in a different value to change the height or manually drag the line object on the canvas.

Orientation

> Indicates the horizontal or vertical orientation of the line on the canvas.

Position: Left

> Indicates the starting position for the left edge of the line.

Style

> Sets the style of the line. Use the drop-down list to select from solid, dotted, dashed, and so on.

Weight

> Sets the thickness of the line to light, medium, or heavy.

Width

> Indicates the width of the line object. You can type in a different value to change the width or manually drag the line object on the canvas.

Display Subtotals and Other Summary Values Using Prefix Operators

The Subtotal property in the Field Properties palette enables you to display subtotals, and other summary values using prefix operators, for any or all of the numeric fields, including computed fields, in a report query. From the Field Properties dialog, when you click the browse (...) button to the right of the Subtotals property, the Sub Total Options dialog box opens.

1. Select the **Sub Totals** option to replace the default setting of None.

   If you are using computed fields, select the **Recomputed Sub Totals** option instead.

2. Numeric field values are summed and displayed as subtotals in the report by default. To display summary values other than the default subtotals, select prefix operators using the drop-down lists in the Prefix area that are adjacent to each of the selections in the Columns area. Note that the Sum prefix produces the same subtotal results as not selecting a prefix operator.

   All of the available prefix operators are listed and described in the following table.

Prefix	Description	Displayed in Prefix List
SUM.	Compute the sum of values	Sum
ASQ.	Compute the average sum of squares	Average Square
AVE.	Compute the average value	Average
CNT.	Count the items	Count
FST.	Select the first value only	Show first in group
LST.	Select the last value only	Show last in group
MAX.	Select the maximum value only	Maximum
MIN.	Select the minimum value only	Minimum

3. To display all numeric field values as subtotals or other summary values in the report, select the **Apply totals to all columns** check box in the Columns area.

4. Required: To display subtotals or other summary values for a subset of the numeric fields in the report, make sure the **Apply totals to all columns** check box is de-selected, then select check boxes for the individual numeric fields in the Columns area of the Subtotal tab.

   When **Apply totals to all columns** is selected, options for all individual fields in the Columns area are inactive by default.

5. To modify the default subtotal title, which is set to *TOTAL, type the desired text you want to display as the title for all subtotals and any other selected summary values in the Current Subtotal Text field.

6. To hide subtotals for sort fields that have only a single value, select the **Suppress for single lines** check box.

The following image shows the Sub Total Options dialog box.
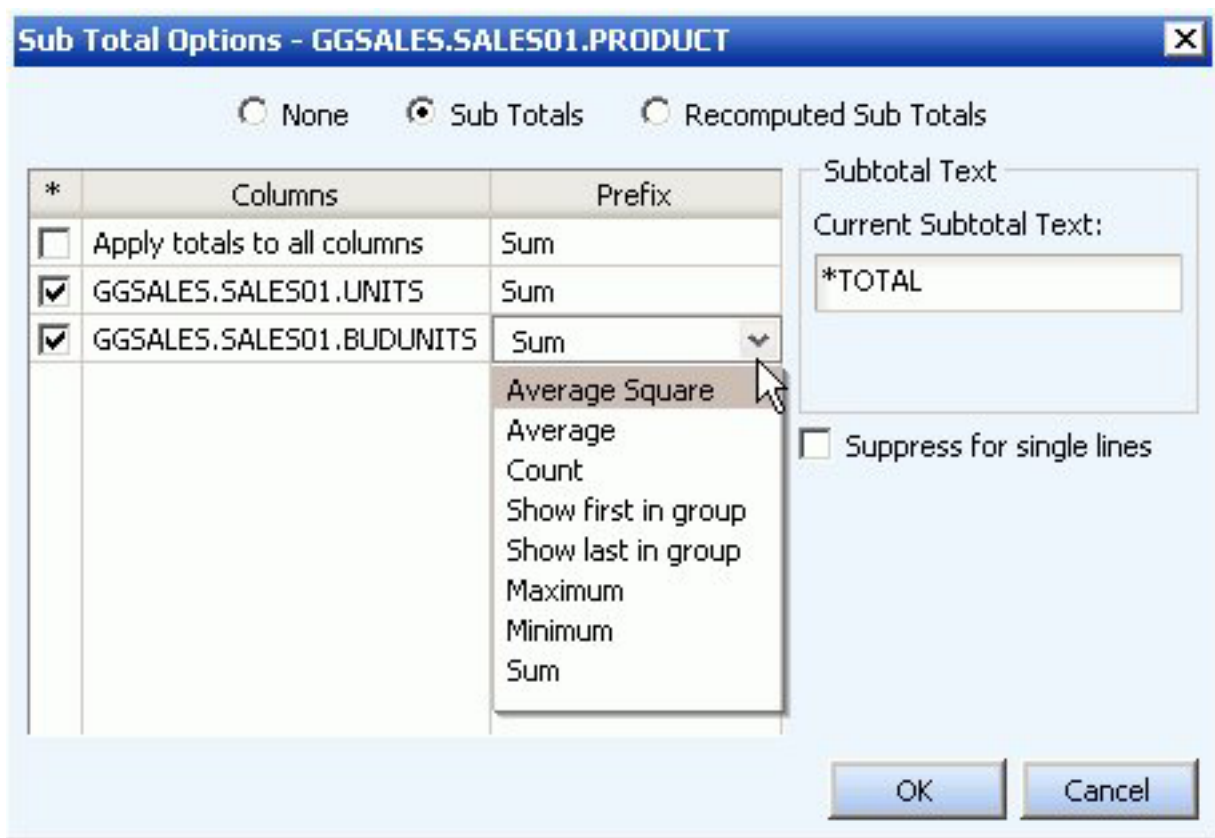
Using Reporting Functions

The following reporting functions are available within the Data Sources palette of the Power Painter tool:

- Create a virtual field (DEFINE) for your report.
- Create a report level calculation (COMPUTE) for your report.
- Add a JOIN relationship by using the Joins tab in the Data Sources palette.
- Create conditional styling rules by using the Create Conditional Style Rule dialog box.
- Create filters (WHERE/WHERE TOTAL statements) using the Selection Criteria palette.

* Use the Auto Prompting Facility to select parameters for your report.

Create a DEFINE Statement

1. Required: Click the **Define** button from the Data Sources palette.
   The Field creator dialog box opens.
2. Required: Type the name of the virtual field in the Field input box.
3. Required: Enter an expression in the expressions box.
4. Required: Click **OK** to close the Define field creator dialog box.
   The DEFINE field is added to the report field list.
5. Drag and drop the DEFINE field into the report.

Create a COMPUTE Statement

1. Click the **Compute** button from the Query palette.
   The Field creator dialog box opens.
2. Type the name of the computes field in the Field input box.
3. Enter an expression in the expressions box.
4. Click **OK** to close the Compute field creator dialog box and apply the COMPUTE statement.
   The COMPUTE field is added to the report.

Create a JOIN Between Data Sources

1. Click the **Joins** tab in the Data Sources palette.

2. Click the **Add Join** button from the Data Sources palette toolbar.
3. Required: Select a target Master File and click **OK**.
   The Create Join dialog box opens.
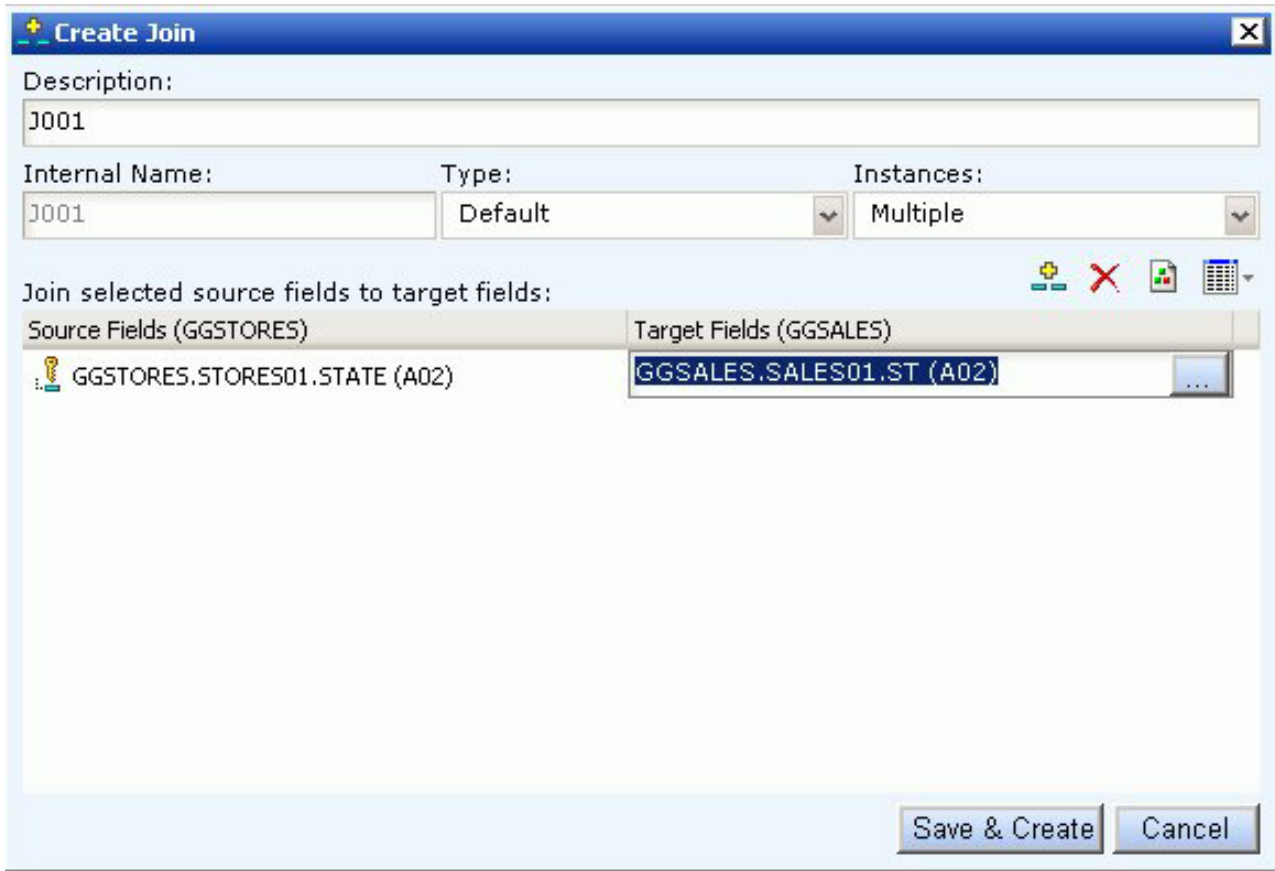
> **Note:** Joins can also be created on defined fields. For more information about defined fields, see Create a DEFINE Statement.

4. Type a name for the Join in the Description field.

5. Select the Type and Instances for the Join or use the default selections.

6. Click the **Select a source field** browse (...) button to select the Join source field.

7. Click the **Select a target field** browse (...) button to select the target fields.

> **Note:** If no target fields are available, you are notified that there are no fields found.

8. Click **Save & Create** to save the Join relationship.

   The joined field is added to the Joins tab.

9. Click the **Fields** tab in the Data Source palette to view the joined data.
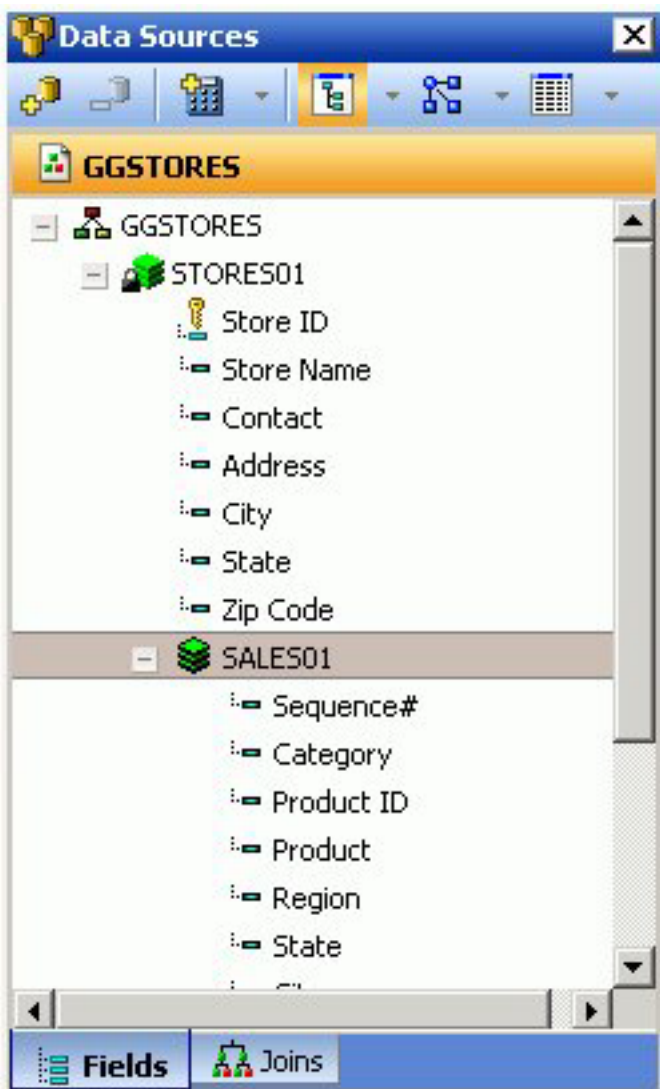
10. Drag and drop joined fields onto the canvas as you would any other field.

Note: To edit a join, double-click the join.

Create Conditional Styling Rules

1. Ensure that the Query palette or popup is open on the canvas:
   - From the View menu, select **Query** from the Palettes submenu.
   - Select a report or graph component, right-click and select **Query Popup** from the context menu.

     or
   - Double-click the report or graph component to bring up the Query Popup.
2. Select the field name in the Query palette.

   When fields are selected, properties for each field are available in the Field properties palette.
3. Click the Field properties palette and scroll down until you see Conditional styling.
4. Click in the Conditional styling value field and select the **New styling rule** button.

The Create Conditional Style Rule dialog box opens.

Create Conditional Style Rule: SHORT.SHORT.BALANCE

Existing style conditions: New Condition

Condition name: Condition_1

Field: Balance

Relationship: EQUAL to

Value: SHORT.SHORT.BALANCE

25
110
222
237
340
417
483
495

Compare type
- Another field
- Values

Set font when condition is met
☑ Default font
Font... | Default Font

Set color when condition is met
☑ Default Color

Standard colors

Gray Scale

Additional

None

Selected Color: | Custom

Done | Cancel

5. Required: Create the style conditions for the field.
   a. Required: Type a name for the condition or accept the default.
   b. Required: Select a relationship for the condition from the Relationship drop-down list.
   c. Specify a value by doing one of the following:
      - Type a literal value in the Value input field.
        or
      - Click **Values** from the Compare type section to display existing data source values.
   d. Click **Done** to save and apply the conditional style rule.
      The condition is added to the Field properties palette and the condition is applied to the data on the Power Painter canvas.

The following image is an example of the Field properties palette with a Low Balance condition rule. The report object shows red text where the conditional styling rule was applied.
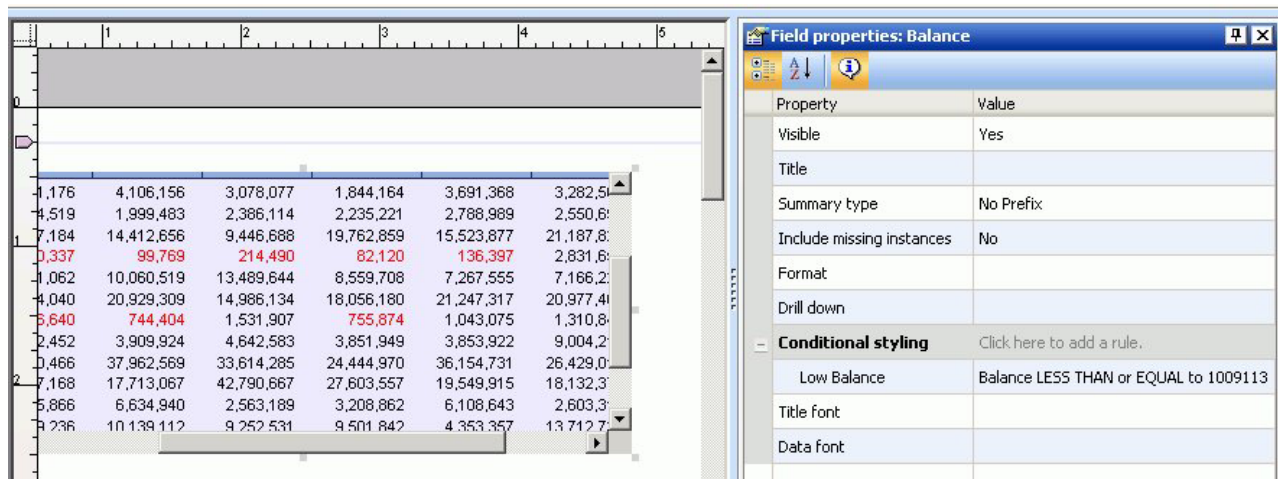
Create a WHERE Statement

1. Required: Select a report or graph object in the Power Painter canvas.
2. Required: From the View menu, select **Selection Criteria** from the Palettes submenu.

   The Selection Criteria palette appears on the bottom of the canvas.
3. Required: Drag a field name from the Data Sources palette to the Selection Criteria palette.

   The WHERE statement is added to the Selection Criteria palette.
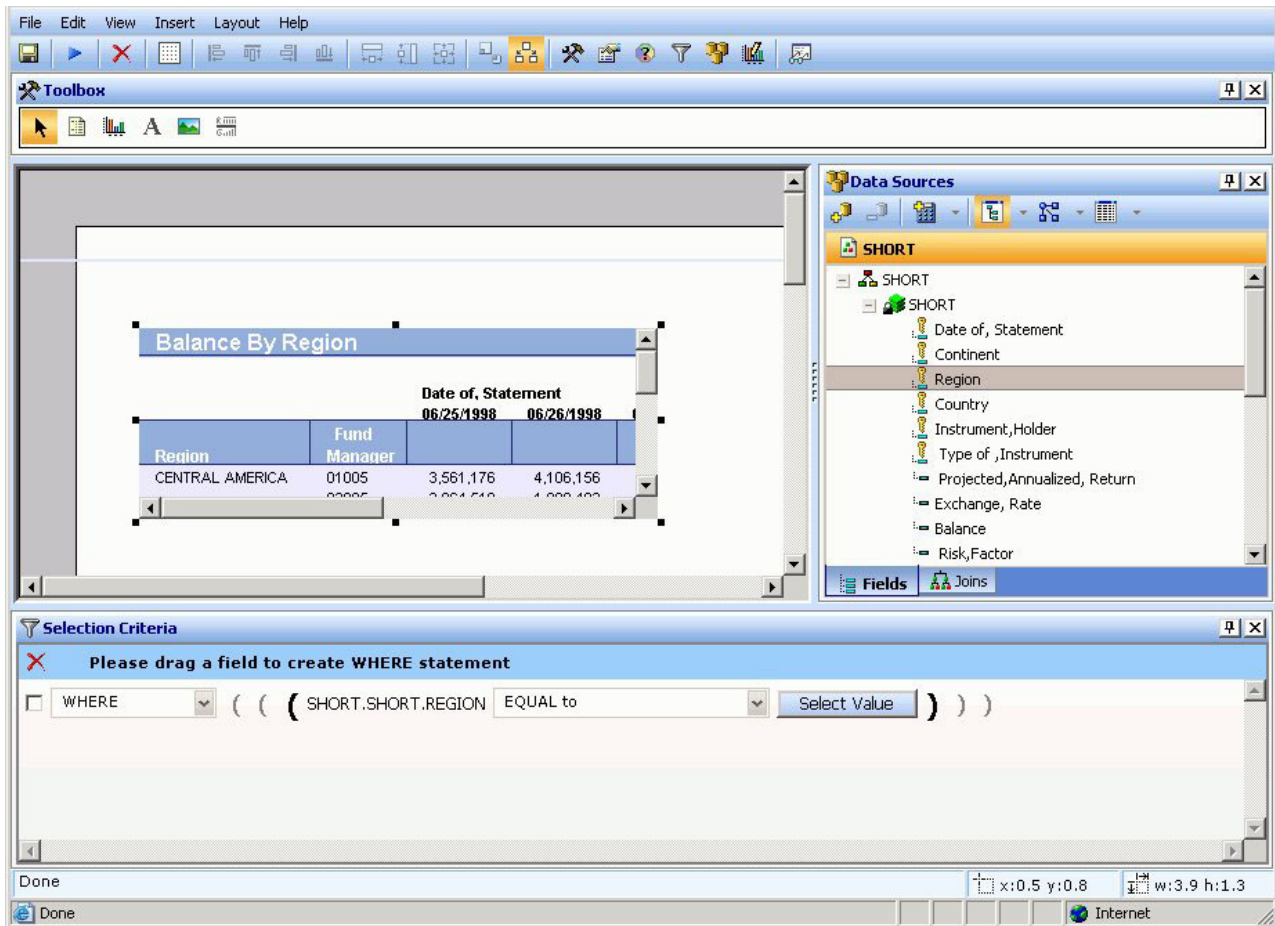
4. Required: Specify the condition by using the drop-down list.
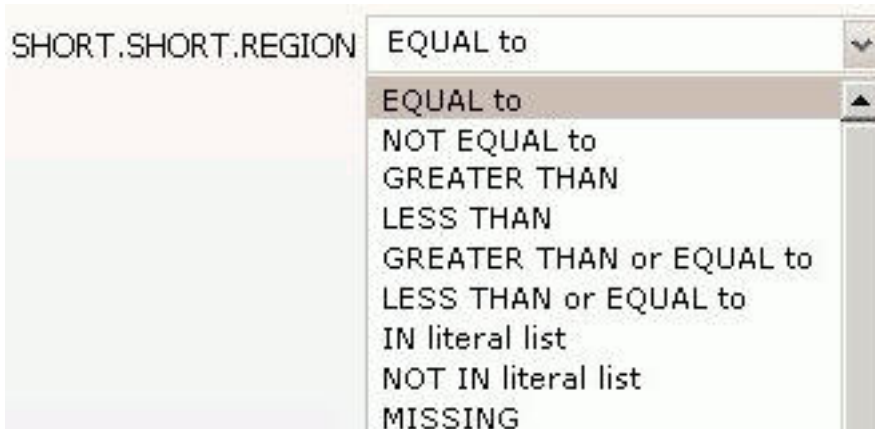
5. Click **Select Value** to open the Select Value dialog box.
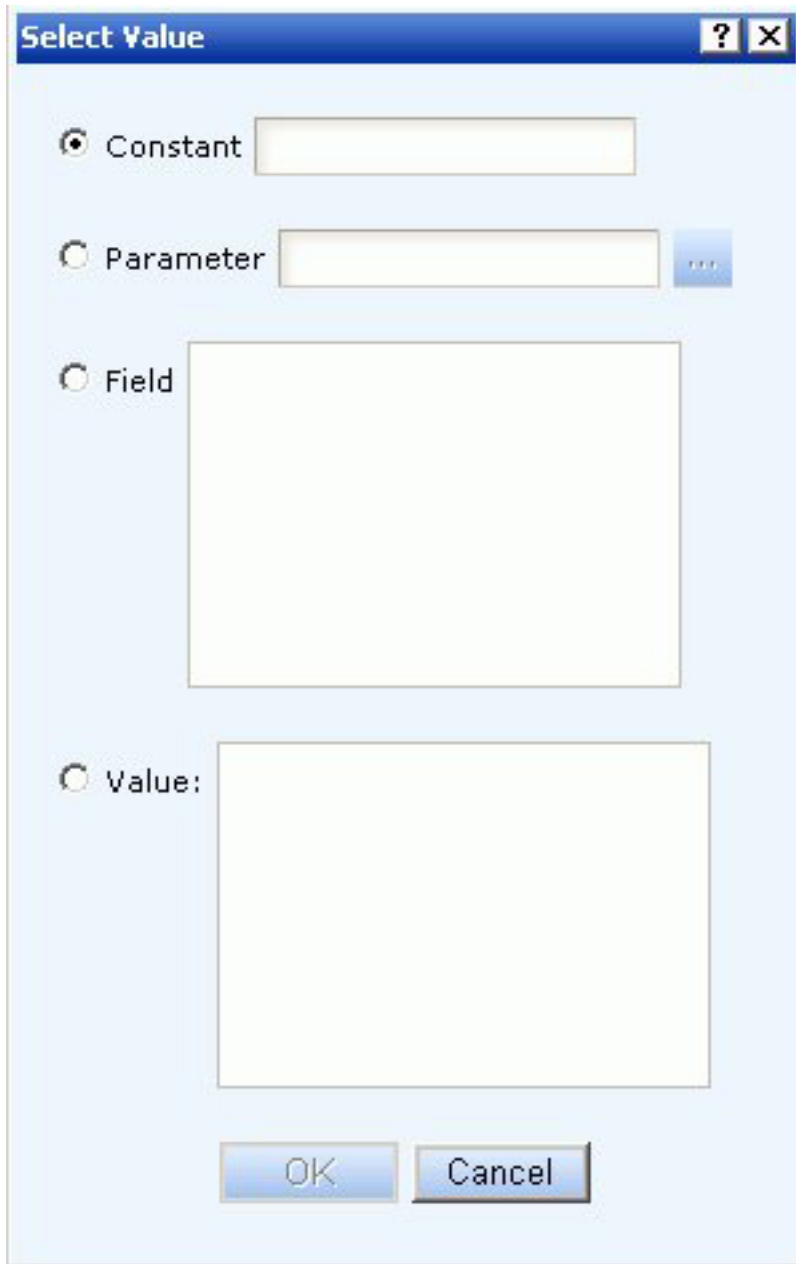
6. Required: Select a **Constant**, **Parameter**, **Field** or **Value** selection button.
   - If creating a constant, manually type in a value for the WHERE statement.
   - If creating a parameter, click the browse (...) button to open the Variable Editor dialog box. For details, see Create Parameters in a Standard Report. Select the name, selection, a value options for the WHERE statement.
   - If creating a Field or Value, select a value from the list supplied.
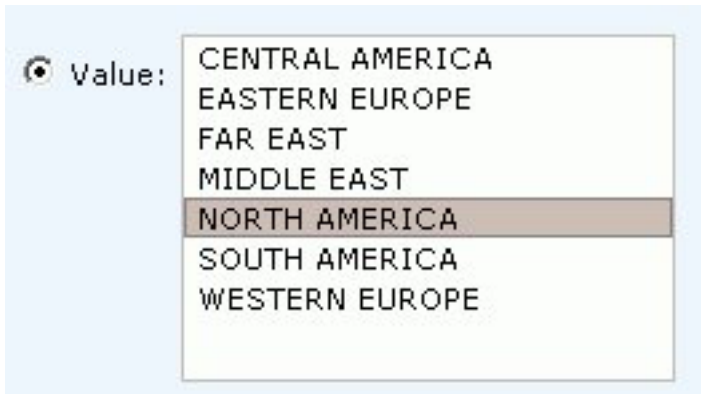
7. Required: Click **OK** to close the Select Value dialog box.

The value is added to the Selection Criteria palette and the report or graph reflects the WHERE statement as soon as the value is selected.
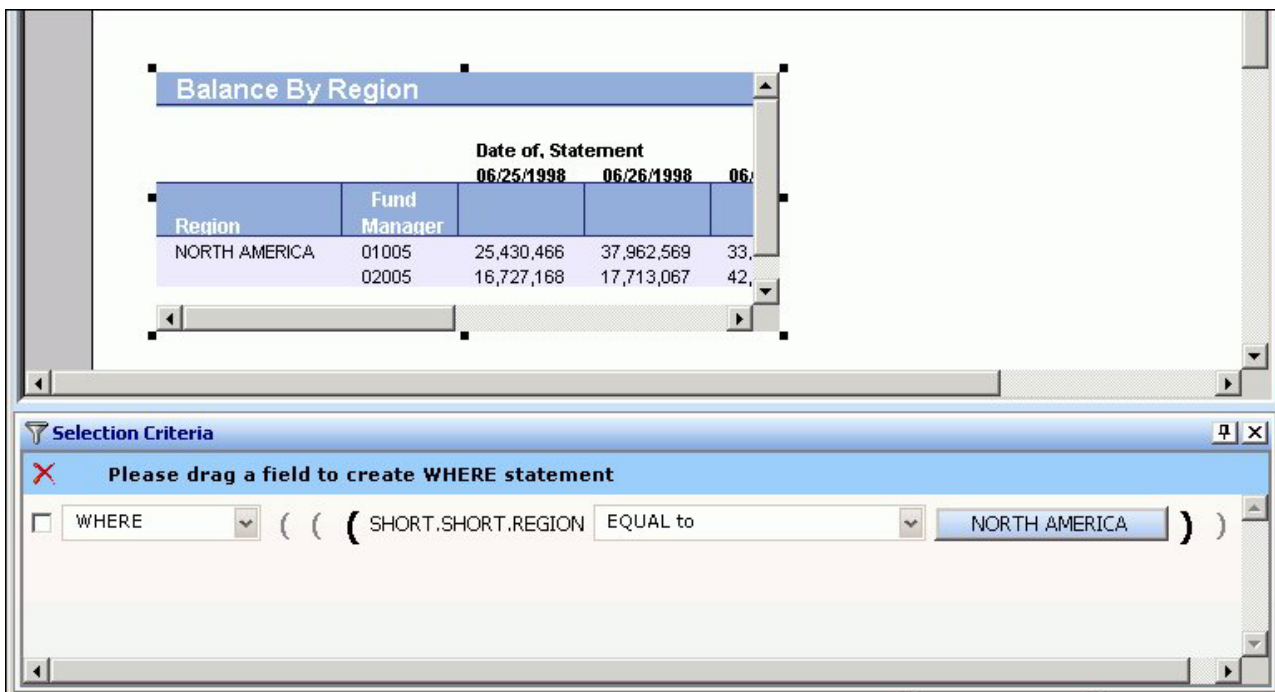
Note: To delete the WHERE statement, select the WHERE statement check box and click the red X on the blue highlighted line in the Selection Criteria palette.

Importing Values From External Files for WHERE Statements

When developing a query, you can use a locally saved, external file as selection criteria to limit query results. This enables you to quickly build a query containing a large number of WHERE statement values without having to manually enter repetitive or readily available data.

Create a Value List From an External File

1. Required: Navigate to the **Selection criteria** area.
2. Required: Begin creating a WHERE statement by selecting a field from the list of available fields.

3. Required: Select a data comparison option that can accept multiple values. Valid options include EQUAL to, NOT EQUAL to, IN literal list, NOT IN literal list, EXCLUDES literal list, and INCLUDES literal list.

4. Click **Select values**.

   The Values dialog opens.

5. Click the **import** button as shown in the following image.

A dialog box opens as shown in the following image.

6. Select either the Flat file or Excel Spreadsheet (XLS) File Format option.

7. Click the **Browse** button.

   A standard file selection dialog opens.

8. Select an external file to import from your local machine or network.

   Note: The external file must only contain text with new line delimiters.

9. Click **OK**.

   The values contained in the file are displayed in the Multiple values entered list in the right pane of the dialog.

10. Optionally, you can remove specific values from the Multiple values entered list or move values up or down within the list.

11. Click **OK**.

    You can save the query for future use and reopen the query to append new or remove existing values.

Creating Parameters for Use in the Auto Prompting Facility

When a parameter selection criteria is added in Power Painter, the auto prompting facility appears at run time.

Create Parameters and Run in the Auto Prompting Facility

1. From the Selection Criteria palette, create a WHERE statement.

2. Click the **Select Value** button.

   The Select Value dialog box opens.

3. From the Select Value dialog box, select **Parameters** and click the **Edit Parameter Properties** button.

   The Variable Editor dialog box opens.

4. In the Variable Editor dialog box, verify the Name and Description fields, which are automatically populated based on the field you selected to create the WHERE statement.

   You can edit the populated values as needed.

5. Required: Choose from the following Selection options:

   • **Dynamic.** This is the default option. The Data Source and Field values are selected automatically based on the field used to create the WHERE statement and can be changed as needed.

- **Static.** Select the Constant or Value option. For Constant, enter one or more values. For the Value option, all of the values for the selected field appear in the Value list box. Move the values you want to the Multiple Values Entered list box using the plus button.

  For either the Dynamic or Static option, you can select the **Select multiple values at runtime** check box to provide more than one value to the report from the auto prompt page. In addition, the user will be able to select all values from the values list.

6. Click **OK** to exit the Variable Editor.

   The parameter is added to the Select Value dialog box.
7. Click **OK** to close the Select Value dialog box.
8. Click **Run** from the File menu in Power Painter.

   The Auto Prompting facility appears.
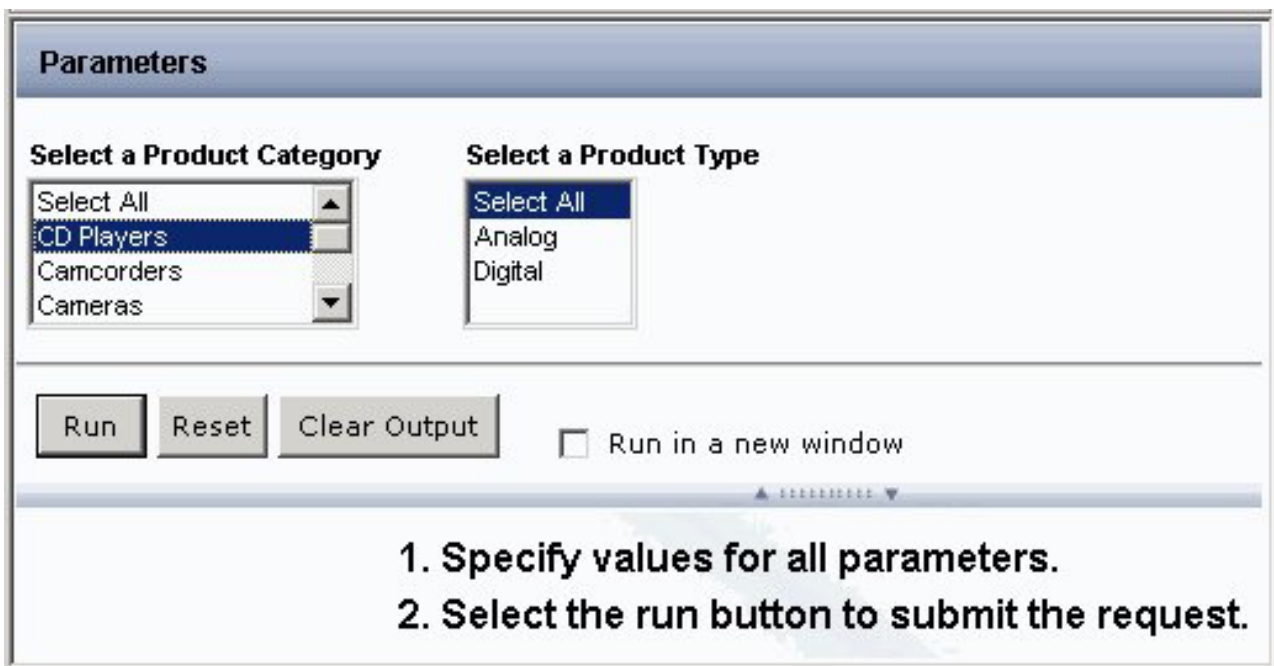
Controlling the Overflow and Relative Positioning of Objects in Power Painter

In Power Painter, you can control the overflow of reports, and relative vertical positioning of reports and graphs, so they fill an entire page and maintain their relative positions in the document. The Sizing and Overflow property in the Report Properties palette is used to control overflow and relative positioning.

Overflow is defined as the area of a report that exceeds its defined space in the page layout. For example, a multi-page report and a graph can both be added to a single page layout with the graph positioned beneath the report. The report can flow to fill the page and the graph can be relatively positioned to the report so it is placed at the end of the document following the complete rendering of the report.

Note: There is currently no method of controlling horizontal overflow.

Setting Overflow Options in Power Painter

When executing a report, the area that flows outside of the allocated space for the report is considered the overflow. The following image is an example of the canvas in Power Painter.
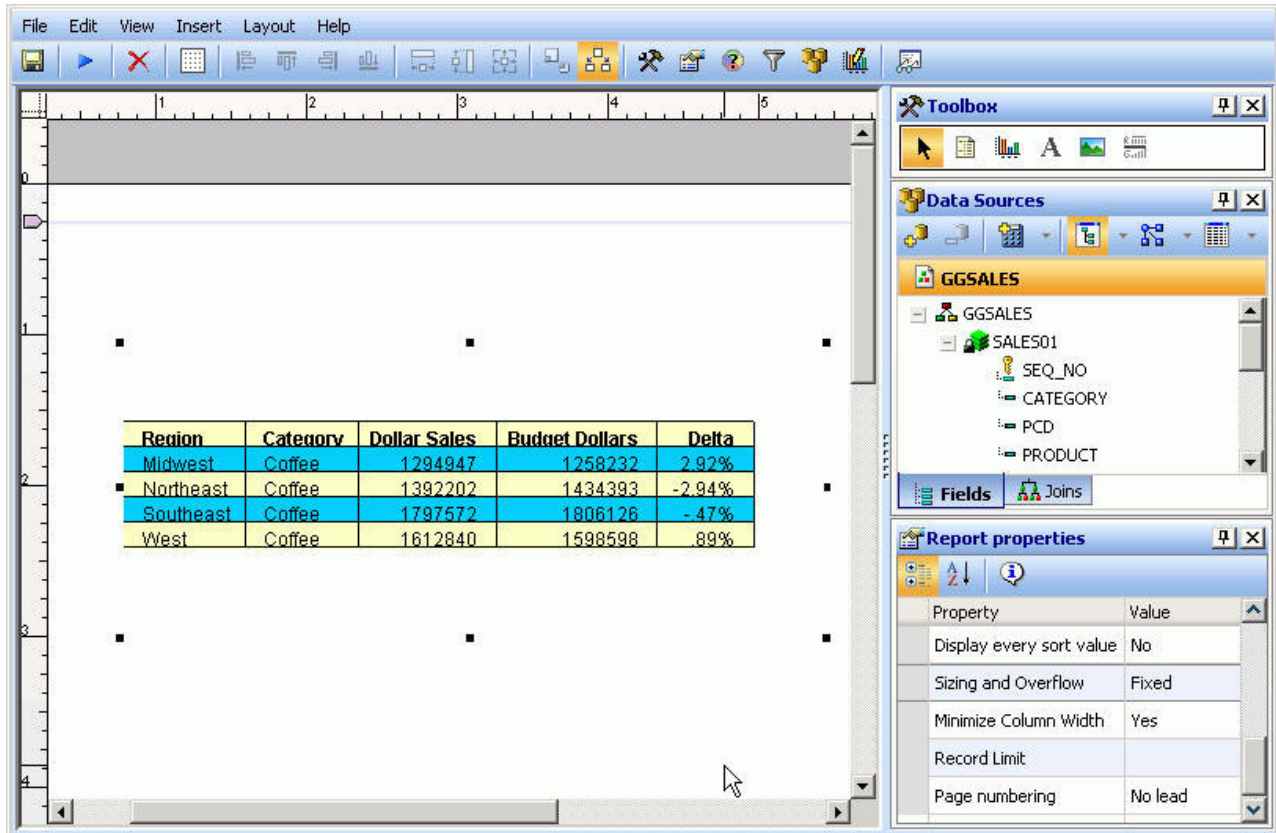
You can set the size and overflow of a report object to control the overflow at run time. Sizing and overflow options are available through the Report Properties palette.

- **Fixed.** When Sizing and Overflow is set to Fixed, the report is confined to the dimensions of the canvas at run time and continues to output on multiple pages within the same fixed dimensions until all of the data is shown.

- **Flowing.** When Sizing and Overflow is set to Flowing, the report fills the page at run time and automatically pushes surrounding objects down the page until the report is finished. This enables other objects such as repeating images and text (headers and footers) to be shown and not overlapped by the flowing report.

Note: These options only apply to report objects. There are no size and overflow properties for graphs, images, text, or lines.

The following image is an example of a report with Sizing and Overflow set to Fixed. Note how the report output maintains the size of the report object in the Power Painter canvas, spreading the report across 6 pages.
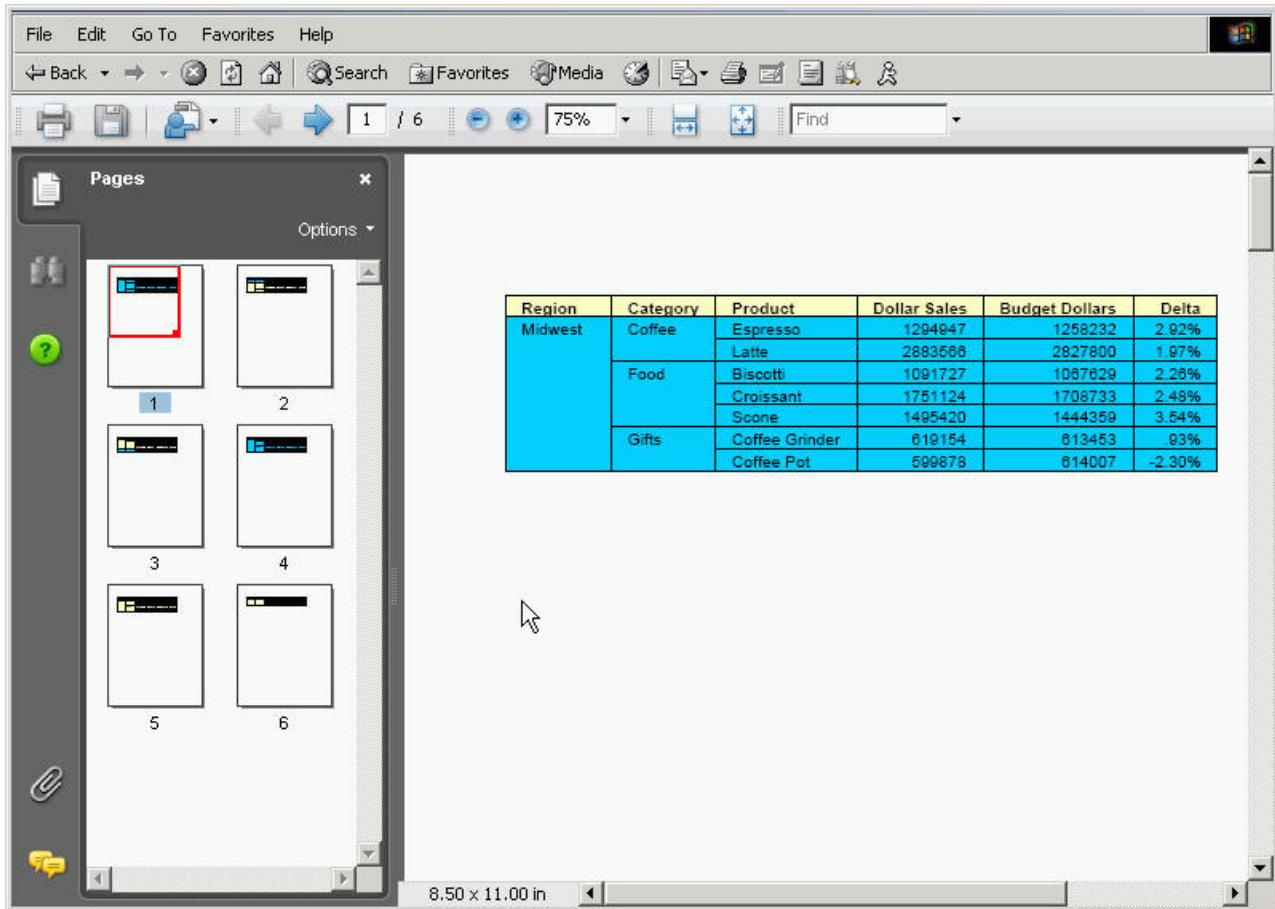
The following image is the same report with Sizing and Overflow set to Flowing. Note how the report expands to fill the page at run time.

Region	Category	Product	Dollar Sales	Budget Dollars	Delta
Midwest	Coffee	Espresso	1294947	1258232	2.92%
		Latte	2883566	2827800	1.97%
	Food	Biscotti	1091727	1067629	2.26%
		Croissant	1751124	1708733	2.48%
		Scone	1495420	1444359	3.54%
	Gifts	Coffee Grinder	619154	613453	.93%
		Coffee Pot	599878	614007	-2.30%
		Mug	1086943	1096150	-.84%
		Thermos	577906	564010	2.46%
Northeast	Coffee	Capuccino	542095	561491	-3.45%
		Espresso	850107	872902	-2.61%
		Latte	2771815	2818069	-1.64%
	Food	Biscotti	1802005	1848682	-2.52%
		Croissant	1670818	1739522	-3.95%
		Scone	907171	865703	4.79%
	Gifts	Coffee Grinder	509200	511642	-.48%
		Coffee Pot	590780	573349	3.04%
		Mug	1144211	1170314	-2.23%
		Thermos	604098	615247	-1.81%
Southeast	Coffee	Capuccino	944000	956661	-1.32%
		Espresso	853572	849465	.48%
		Latte	2617836	2625303	-.28%
	Food	Biscotti	1505717	1512019	-.42%
		Croissant	1902359	1969906	-3.43%
		Scone	900655	927363	-2.88%
	Gifts	Coffee Grinder	605777	569585	6.35%
		Coffee Pot	645303	654579	-1.42%
		Mug	1102703	1124345	-1.92%
		Thermos	632457	618745	2.22%
West	Coffee	Capuccino	895495	877304	2.07%
		Espresso	907617	923941	-1.77%
		Latte	2670405	2722718	-1.92%
	Food	Biscotti	863868	861804	.24%
		Croissant	2425601	2406554	.79%

Using Fixed Overflow in Power Painter

Reports using fixed overflow maintain the size and position of the report object from the Power Painter canvas in the report output. You can customize and maintain the fixed position of the report output by adjusting the position and size options from the Report properties palette.

Customize the Fixed Position and Size of an Object

1. Required: Select an object from the Power Painter canvas.

   The Report properties palette shows the available properties for the selected object.
2. Select **Fixed** from the Sizing and Overflow drop-down list.
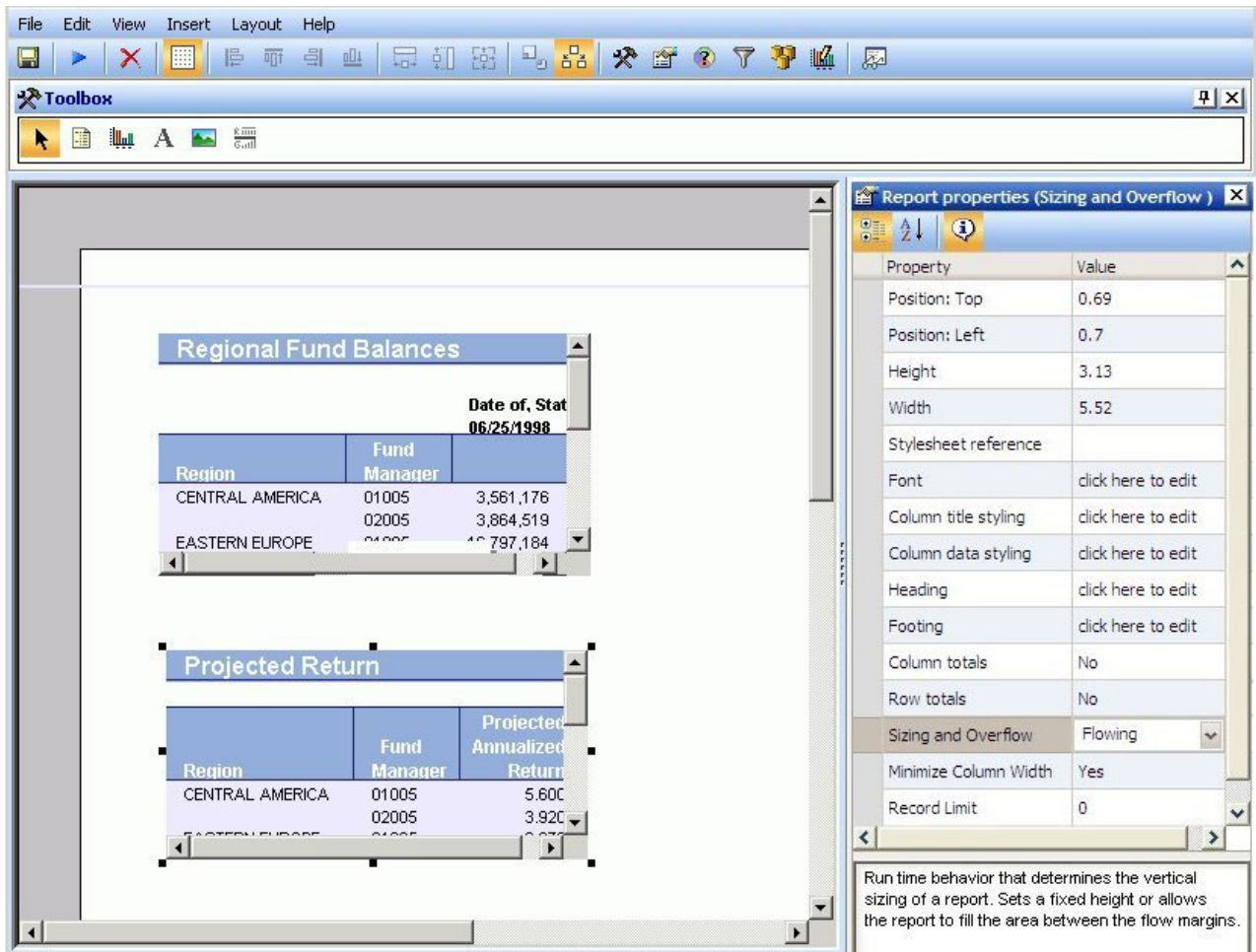
3. To adjust the position and size of the object at run time, edit the Position Left, Position Top, Height, and Width properties in the Report properties palette.

   or

   To adjust the position, hover the mouse pointer over the object until a four-way arrow appears, then click and drag the object in any direction. To adjust the size, hover the mouse pointer over any of the square corners or midpoints along the border of the object until a two-way arrow appears, then click and drag the square in any direction.

4. Run the report.

Note: If the document contains multiple fixed reports, the report output maintains the same size and position as the report objects in the Power Painter canvas, as shown in the image below.
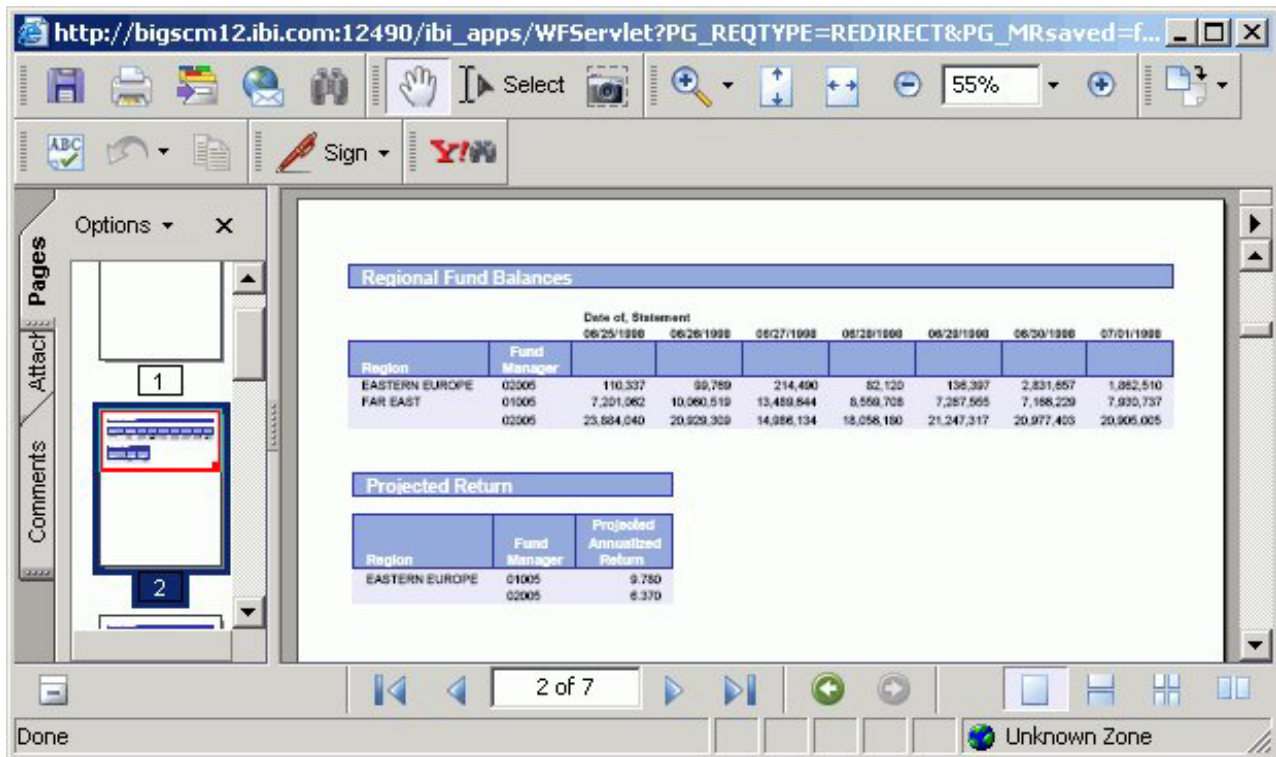
Note: If the document contains a fixed report and a graph, the report may be broken up around the graph object. You can set the relationship between the graph and the report to have the graph display after the report. For more information about setting relationships, see Relate Surrounding Objects to a Report.
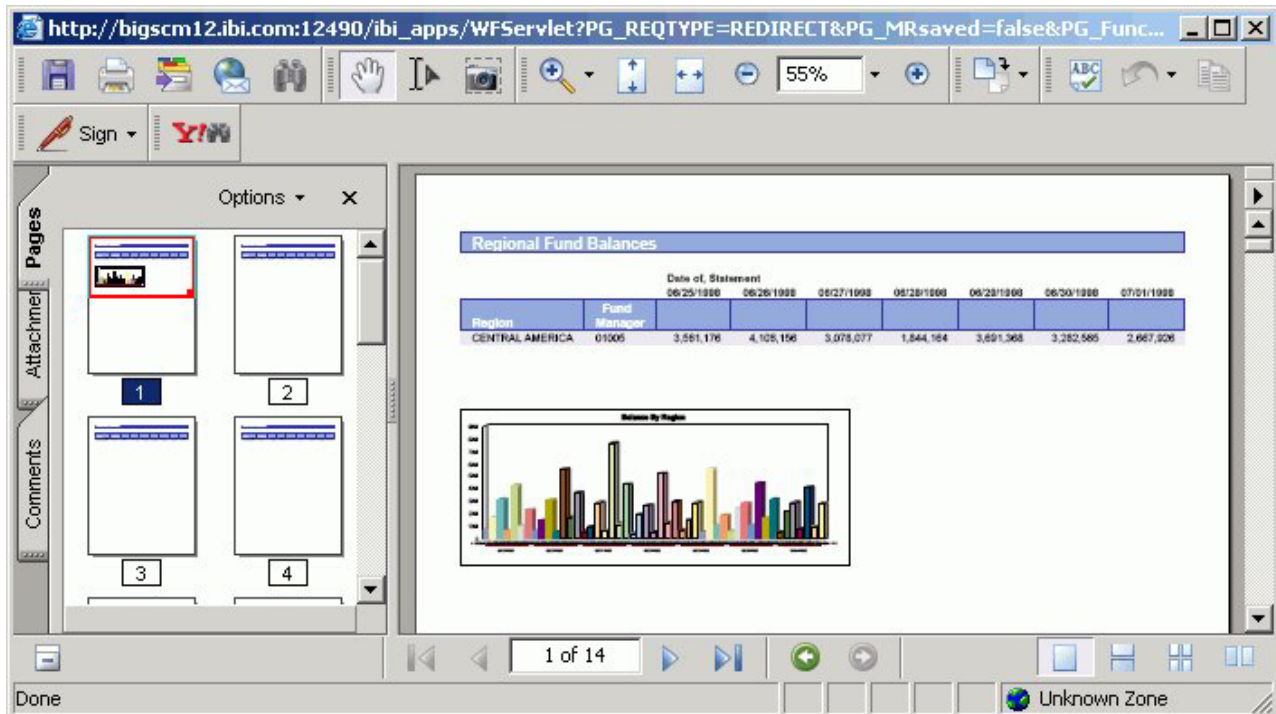
Using Flowing Overflow in Power Painter

Reports using flowing overflow begin the report output at the upper left corner of the report object and fill the page with the report results. Flow margins are available which enable you to adjust the canvas so that flowing objects fill the page.

Note: Flowing overflow is not available for graphs, images, text, or lines. For example, if a graph is included in the document and the graph overlaps the report at run time, you must set a relationship between the graph and the report so the graph flows after the report at run time.

Set the Flowing Report Property

1. Required: Select an object from the Power Painter canvas.

   The Properties palette shows the available properties for the selected object.
2. Required: Select **Flowing** from the Sizing and Overflow drop-down list.

Report properties (Sizing and Overflow) ☒

Property ▲	Value
Column data styling	click here to edit
Column title styling	click here to edit
Column totals	No
Display every sort value	No
Font	click here to edit
Footing	click here to edit
Heading	click here to edit
Height	2.09
Minimize Column Width	Yes
Page numbering	No lead
Position: Left	2.02
Position: Top	0.41
Record Limit	
Row totals	No
Sizing and Overflow	Flowing
Stylesheet reference	click here to edit
Width	3.09

Tip: Flow margins are available for the canvas which further enable you to adjust the page so that the report fills each page between the flow margins. For more information about flow margins, see Set Flow Margins for the Power Painter Canvas.
3. Run the document.

Note: If the document contains multiple flowing reports and the reports overlap at run time, you must set the relationship between the reports. For more information about setting relationships, see Relate Surrounding Objects to a Report.
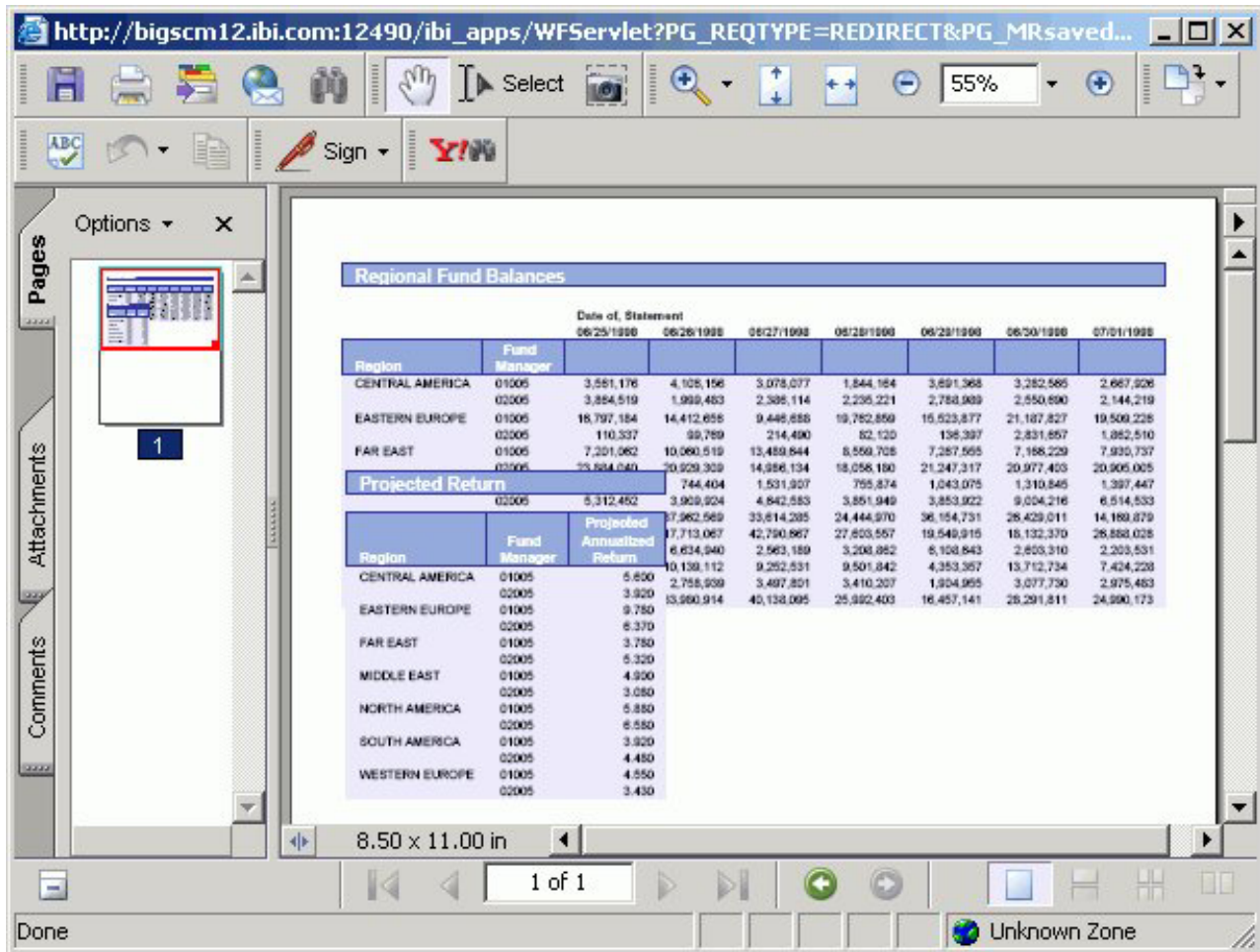
Note: If the document contains a flowing report and a graph, and the graph overlaps the report at run time, you must set the relationship between the graph and the report. For more information, see Relate Surrounding Objects to a Report.
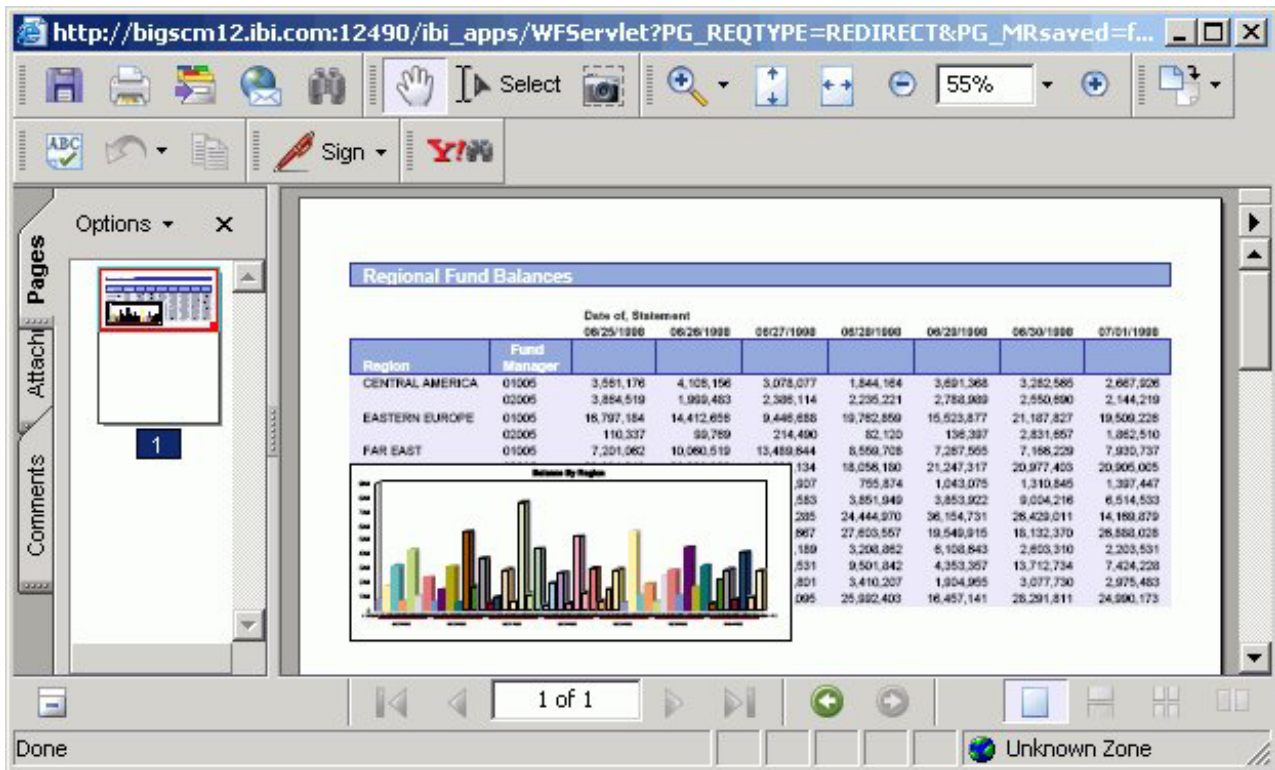
Set Flow Margins for the Power Painter Canvas

You can set the flow margins for the document, which become page margins in the report output when Sizing and Overflow is set to Flowing for the report. Flow margins are set through the top and bottom margins, and flowing reports utilize flow margins to ensure that header and footer information is preserved in the output.

Note: There is currently no method of controlling horizontal overflow.

1. Select the canvas from Power Painter.
2. From the Document properties palette, manually type in the **Flow margin: bottom** value.

   The flow margin bottom value sets the ending vertical coordinate where a flowing report ends on each page.
3. From the Properties window, manually type in the **Flow margin: top** value.

   The flow margin top value sets the beginning vertical coordinate where a flowing report starts on each new page.

   or

   You can also set flow margins on the canvas. To display flow margins, select Layout from the toolbar menu and select **Show Flow Margins**.

   Note: Flow margins are indicated by blue lines that appear on the top and bottom of the Power Painter canvas.

Property	Value
Page orientation	Portrait
Page size	Letter
Flow margin: top	0.708333
Flow margin: bottom	0.25
Output format	PDF
Table of contents	No
Data preview	No
Design Record Limit	500
Units	Inches
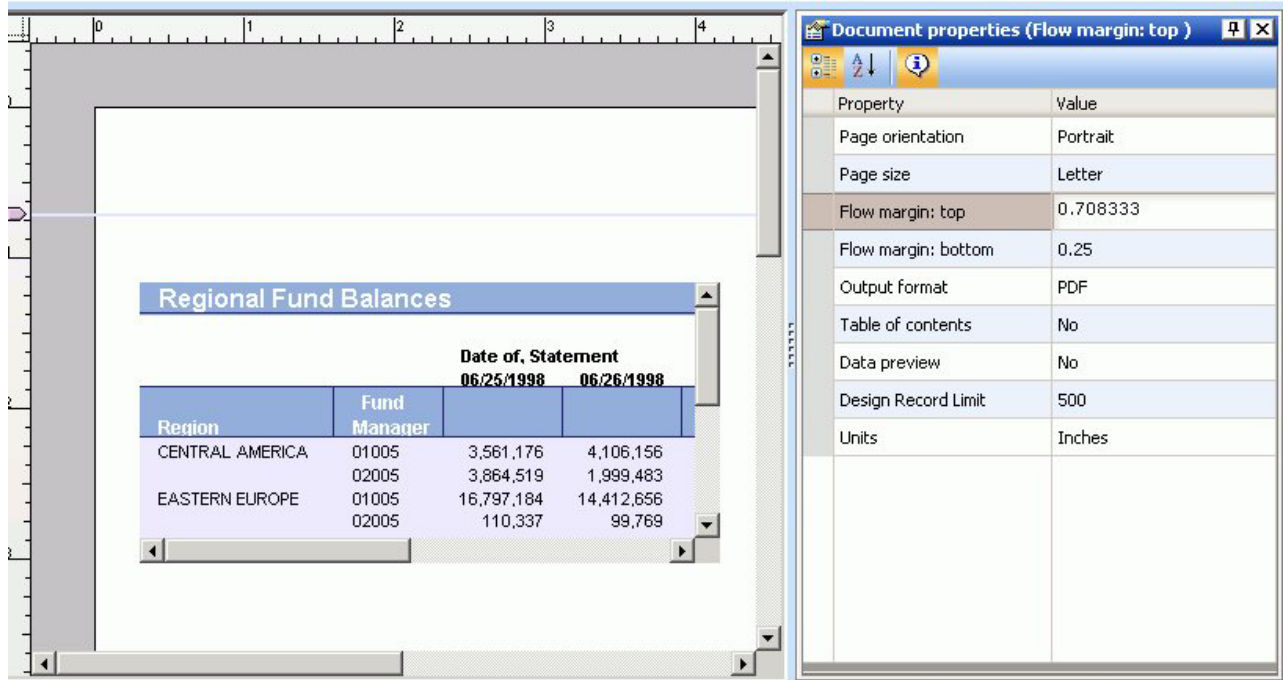
4. Adjust the objects on the canvas between the flow margins and run the report.

The report fills each page between the flow margins until it reaches the end. For example, in the image below the flow margin was adjusted so that an image could be used as a header.

http://bigscm12.ibi.com:12490/ibi_apps/WFServlet?PG_REQTYPE=REDIREC...

Save a Copy | Search | Select

55%

Information Builders

POWERED BY
Information
Builders

Regional Fund Balances

| | | Date of, Statement | | | | | | |
Region	Fund Manager	06/25/1998	06/26/1998	06/27/1998	06/28/1998	06/29/1998	06/30/1998	07/01/1998
CENTRAL AMERICA	01005	3,561,176	4,105,156	3,078,077	1,844,164	3,691,368	3,282,565	2,667,926
	02005	3,854,519	1,999,453	2,386,114	2,235,221	2,788,989	2,550,690	2,144,219
EASTERN EUROPE	01005	16,797,184	14,412,656	9,446,658	19,762,859	15,523,877	21,187,827	19,509,228
	02005	110,337	99,769	214,490	82,120	136,397	2,831,857	1,862,510
FAR EAST	01005	7,201,062	10,060,519	13,489,644	8,569,708	7,288,229	7,188,229	7,930,737
	02005	23,684,040	20,929,309	14,966,134	18,058,180	21,247,317	20,977,403	20,905,005
MIDDLE EAST	01005	796,640	744,404	1,531,907	755,874	1,043,075	1,310,845	1,397,447
	02005	5,312,462	3,909,924	4,642,583	3,851,949	3,853,922	9,004,216	6,514,533
NORTH AMERICA	01005	25,430,458	37,962,569	33,614,285	24,444,970	36,154,731	26,429,011	14,169,879
	02005	16,727,188	17,713,067	42,790,687	27,803,557	19,549,915	18,132,370	26,588,028
SOUTH AMERICA	01005	3,155,866	6,634,940	2,563,189	3,208,862	6,108,643	2,603,310	2,203,531
	02005	8,799,238	10,139,112	9,252,531	9,501,842	4,353,357	13,712,734	7,424,228
WESTERN EUROPE	01005	2,021,971	2,758,939	3,497,801	3,410,207	1,904,955	3,077,730	2,975,463
	02005	21,052,751	33,980,914	40,138,095	25,992,403	16,457,141	28,291,811	24,990,173

|◀ ◀ 1 of 1 ▶ ▶|

Done | Unknown Zone

Setting a Relative Position Between Objects in Power Painter

You can set a relative (vertical) relationship between objects. Similar to the flowing report option, you can select multiple report and graph objects and set a relationship. The following relationship options are available from the Main toolbar, or from the Layout menu in Power Painter:

- **Relate objects.** Sets the relative relationship or distance between two objects; a surrounding object (below) and the primary object (above) that precedes it.
- **Show relationships.** Shows or hides all relationships on the canvas.

Relative positioning is determined by setting where a surrounding object's upper left corner is positioned relative to the lower left corner of the preceding primary object.

Note: There is currently no method of setting horizontal relationships.

Relate Surrounding Objects to a Report

A surrounding object can only be related to one flowing report. However, a flowing report can have multiple surrounding objects related to itself.

1. Select a report or graph (surrounding) object from the Power Painter canvas.
2. Hold down the Ctrl key while selecting a flowing report (primary) object that precedes or is above the first selected object.
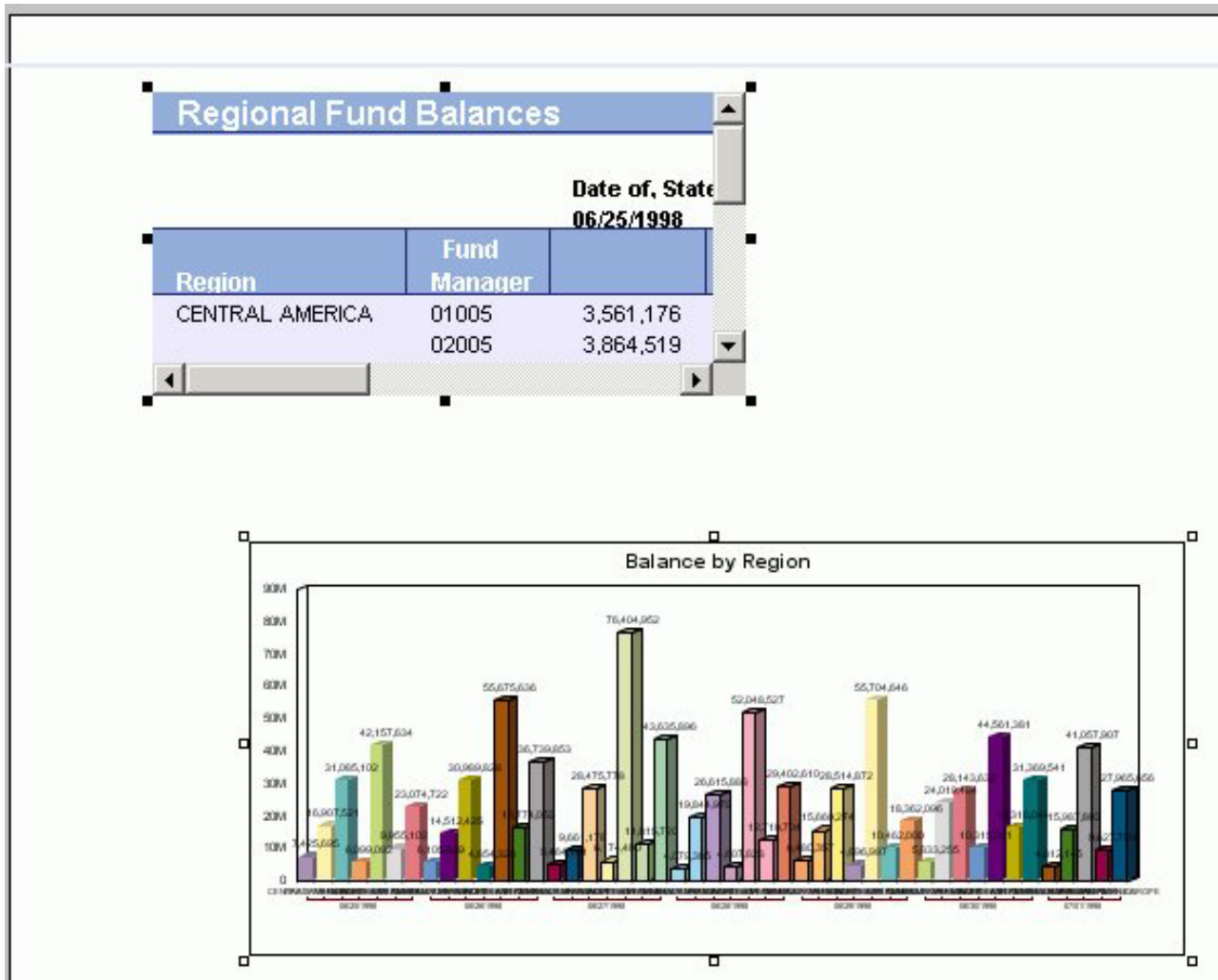
Note: The primary object is indicated by solid black squares around the edges, when selected. The surrounding object is indicated by clear squares around the edges, when selected.

3. Set the relationship between the objects by selecting the **Relate objects**

 button from the Main toolbar.

Note: The relationship buttons are only available from the Main toolbar when multiple objects are selected.

An arrow appears on the canvas, indicating that the relationship has been set.

In the following image, the graph is the surrounding object to the report.
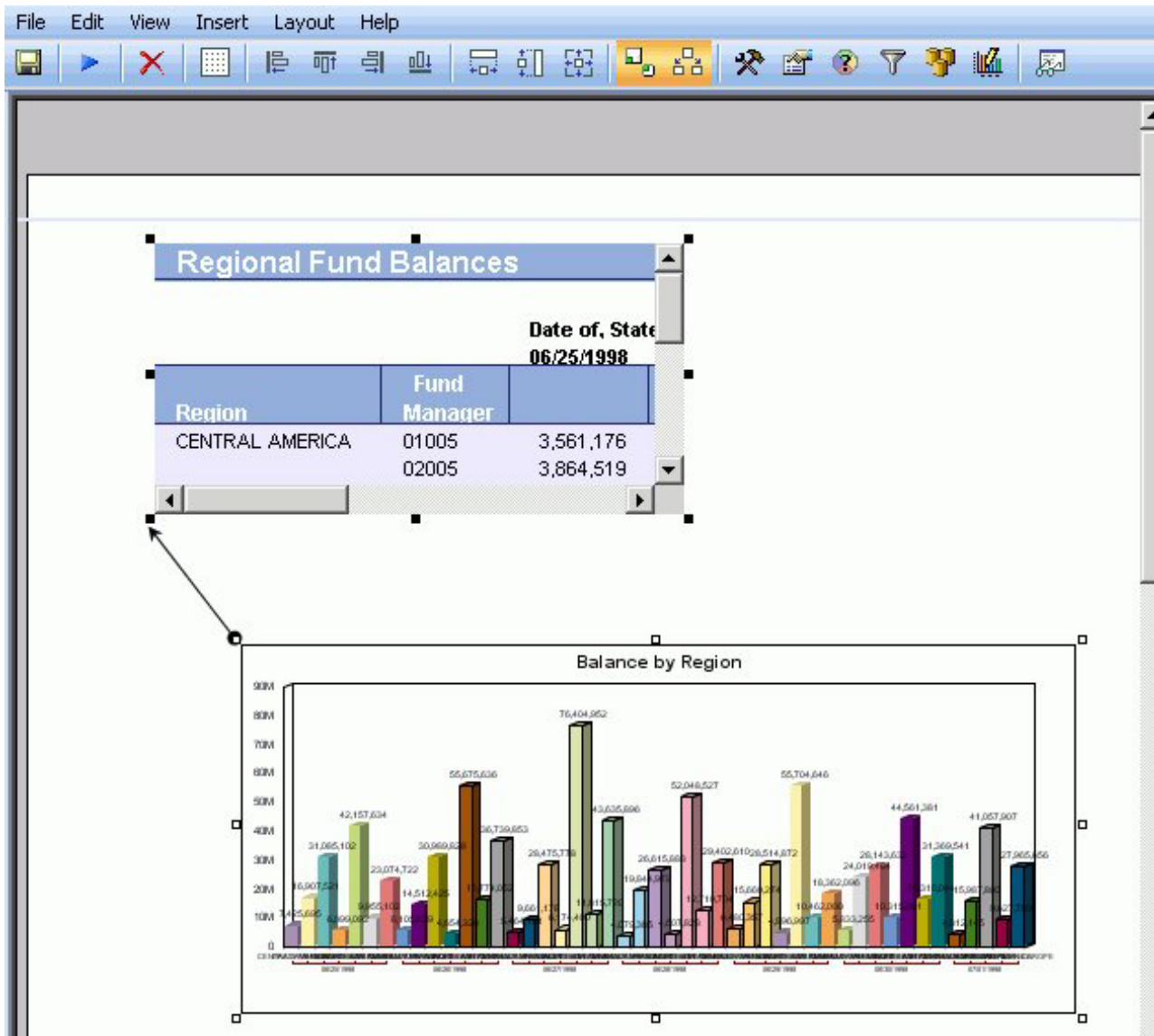
> **Tip:** To break the relationship, click the same surrounding objects using the Shift key and click the **Relate objects** button from the Main toolbar. The arrow is removed and the relationship is broken.

4. Run the report.

Note: If the document contains multiple reports, and the relationships have been set, the first report runs until it is complete, followed by the surrounding reports, at run time.

| 55% |

Sign ▾

Regional Fund Balances

Region	Fund Manager	Date of Statement 06/25/1998	06/26/1998	06/27/1998	06/28/1998	06/29/1998	06/30/1998	07/01/1998
CENTRAL AMERICA	01005	3,561,178	4,106,155	3,078,077	1,844,164	3,691,383	3,282,585	2,887,928
	02005	3,864,519	1,999,483	2,386,114	2,235,221	2,788,989	2,560,690	2,144,219
EASTERN EUROPE	01005	18,797,184	14,412,856	9,446,838	19,762,859	15,523,877	21,187,827	19,509,226
	02005	110,337	99,769	214,480	82,120	138,397	2,831,657	1,882,510
FAR EAST	01005	7,201,052	10,080,519	13,489,644	8,559,708	7,287,555	7,166,229	7,930,737
	02005	23,884,040	20,929,309	14,986,134	18,056,180	21,247,317	20,977,403	20,905,006
MIDDLE EAST	01005	788,540	744,404	1,531,907	756,874	1,043,075	1,310,845	1,397,447
	02005	5,312,452	3,909,924	4,642,583	3,851,949	3,853,922	9,004,218	8,514,533
NORTH AMERICA	01005	25,430,466	37,982,589	33,614,285	24,444,970	38,154,731	28,429,011	14,169,879
	02005	16,727,168	17,713,087	42,790,887	27,803,557	19,549,915	18,132,370	26,888,028
SOUTH AMERICA	01005	3,165,888	8,634,940	2,583,169	3,206,882	6,108,643	2,803,310	2,203,531
	02005	8,799,238	10,139,112	9,252,531	9,501,842	4,353,357	13,712,734	7,424,228
WESTERN EUROPE	01005	2,021,971	2,758,939	3,497,801	3,410,207	1,904,955	3,077,730	2,975,483
	02005	21,052,751	33,980,914	40,138,095	25,992,403	18,457,141	28,291,811	24,990,173

Risk Class By Country

Country	Continent	RISK_CLASS
ARGENTINA	AMERICAS	Medium
BRAZIL	AMERICAS	Medium
CANADA	AMERICAS	Medium
CZECH. REP	EUROPE	High
ENGLAND	EUROPE	Medium
FRANCE	EUROPE	Medium
GUATEMALA	AMERICAS	Medium
HONDURAS	AMERICAS	Medium
HONG KONG	ASIA	Medium
ISRAEL	ASIA	Medium
JAPAN	ASIA	Medium
MEXICO	AMERICAS	Medium
SAUDI ARABIA	ASIA	Medium
SKLARVIA	EUROPE	High
UNITED STATES	AMERICAS	Medium

Projected Return

Region	Fund Manager	Projected Annualized Return
CENTRAL AMERICA	01005	5.800
	02005	3.920
EASTERN EUROPE	01005	9.780
	02005	8.370
FAR EAST	01005	3.780
	02005	5.320
MIDDLE EAST	01005	4.900
	02005	3.080
NORTH AMERICA	01005	5.580
	02005	8.580
SOUTH AMERICA	01005	3.920
	02005	4.480
WESTERN EUROPE	01005	4.550
	02005	3.430

Pages

Attachments

Comments

1 of 1

Done — Unknown Zone

Note: If the document contains a report and a graph, and the relationship has been set, the report runs until complete, followed by the graph, at run time.
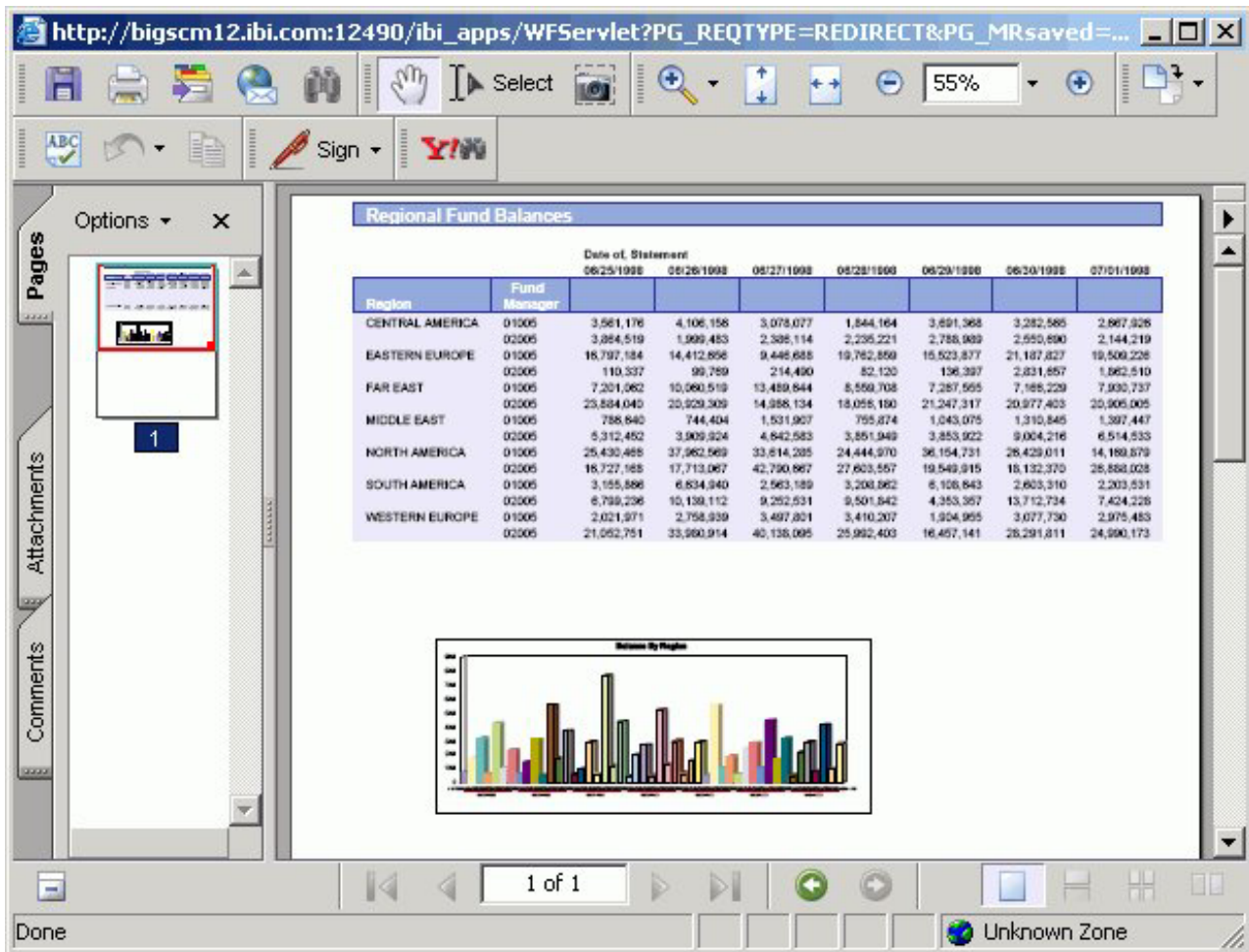
Rules for Setting Relationships Between Objects

The following rules apply when setting relationships between objects:

Setting Relationships

>This sets a relative distance between the flowing report and a surrounding object. The surrounding object is rendered immediately following the end of a flowing report with the related distance between the end of the flowing report and the beginning of the surrounding object.

Fixed objects

>>A fixed object cannot be broken across a page break, it needs enough vertical space to fit on the last page of the flowing report.

Flowing object (reports)

>>If a surrounding object is another flowing report, then the required space does not need to be set. A flowing report can be broken across pages. The reporting server determines if there is enough room to begin rendering the surrounding flowing report.

Draw object

Relationships do not apply to draw objects such as images, lines, and text. Draw objects can be placed above flowing reports in a page layout, but draw objects placed below a flowing report should be placed outside of the bottom flow margin.

Setting a relationship between a report and a graph

If the overflow property of the report is set to flowing and there is no set relationship between the graph and report, then the report will overlap and display on top of the graph at run time. There is no way for Power Painter to automatically set relationships, so you must set the relationship between the report and the graph.

Note: A surrounding object can only be related to one flowing report; however, a flowing report can have multiple surrounding objects related to itself.

Customizing the Power Painter Environment

After the components are in the canvas, you can customize the layout properties, alignment options, and page to configure the canvas for precise output. You can also adjust the work area and change the theme of the Power Painter environment.

Setting Preferences

When you access the Edit menu and select Preferences, the following dialog box appears where you can set preferences for various defaults.
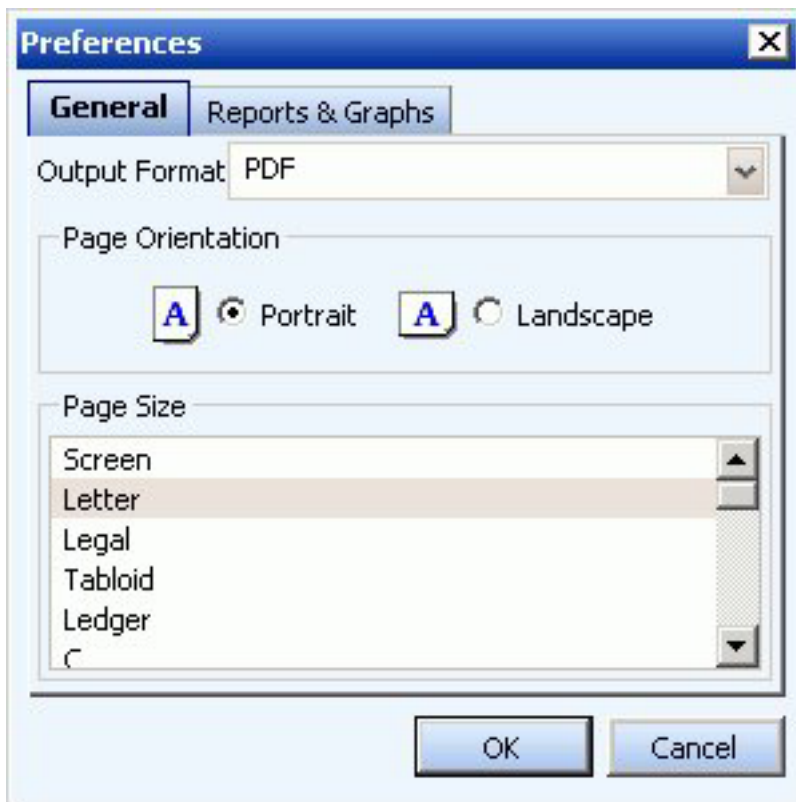
The Preferences dialog contains a General tab and a Reports and Graphs tab. The General tab enables you to select Output Format, Page Orientation, and Page Size options. The Reports and Graphs tab enables you to select a Preview option and

set a Record Limit for both reports and graphs, select a Default Stylesheet for reports, and select a Default Template for graphs.

Note: These settings do not affect the current Power Painter session. The new settings are applied the next time Power Painter is opened.

Setting Field Display Options

When you access the View menu and select Field Display Options, another menu opens providing you with options to display fields as a Name, Alias, Title, or Description. You can also choose the Prefix with option to prefix fields with a filename or segment name.

Adjusting the Layout of Objects in the Canvas

You can use the Main toolbar and the Layout menu to adjust the layout of objects in the Power Painter canvas. The following layout options are available:

* Show grids, rulers, and guides in the canvas to fulfill pixel-perfect positioning requirements.
* Align and size objects in the canvas.

Showing Grids, Rulers, and Guides in the Canvas

* To **show grids** in the canvas, select **Show Grid** from the Layout menu or click the Toggle Grid icon from the Main Toolbar.

  The following image is an example of how the canvas appears with grids showing.
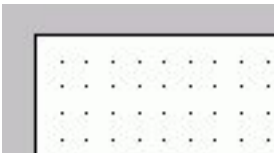
* To **show rulers** in the canvas, select **Show Rulers** from the Layout menu.

  The following image is an example of how the canvas appears with rulers showing.
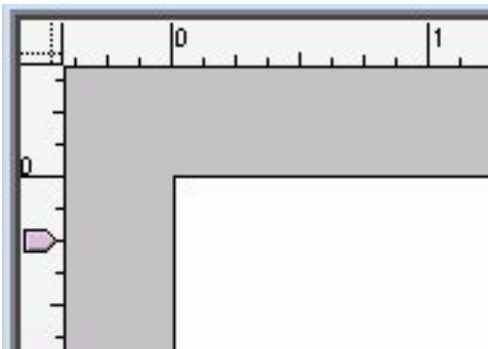
* To **show guides** in the canvas, select **Show Guides** from the Layout menu.

  The following image is an example of how the canvas appears with guides showing.
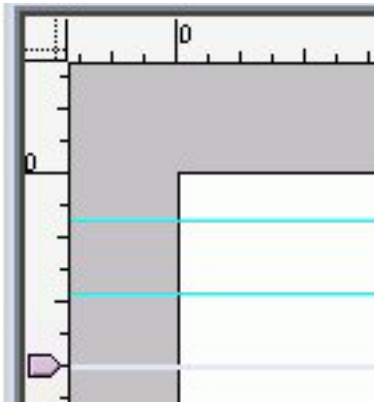
Tip: Use the ruler to click and drag guides onto the canvas. Guides are indicated by the aqua line.

- To **show flow margins** in canvas, select **Show Flow Margins** from the Layout menu.

  The following image is an example of how the canvas appears with flow margins showing. Flow margins are indicated by the light blue line.
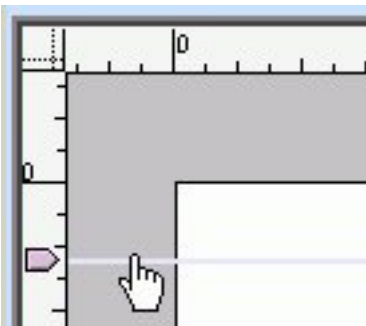
Tip: You can left-click on the horizontal ruler or the vertical ruler and drag the flow margins up and down the canvas.

The following image shows the Layout menu in Power Painter.

Aligning and Sizing Objects in the Canvas

You can align objects relative to one another or to the canvas. Alignment options are available from the Layout menu or from the Main Toolbar.

- Align objects to the page by using the align options from the Main toolbar.
- Align multiple objects to each other using the Align Items option from the Layout menu.
- Size objects to be the same size by using the Size Items option from the Layout menu.
- Use the Relative Positioning options to maintain the relative position of the objects to each other at run time. The Relative Positioning options are Relate Objects, and Show Relationships-available from the Main toolbar or Layout menu.

The following image shows the Main toolbar in Power Painter.
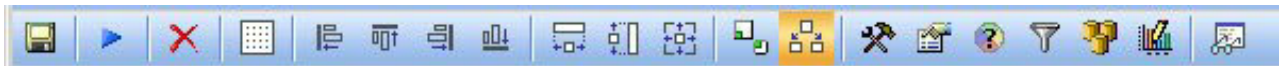
Adjusting the Power Painter Work Area

Click anywhere on the canvas and select the Properties palette to adjust the document properties.

Data preview

> Shows live data or sample data for reports and graphs.
>
> **Note:** By default, sample data is shown in the Power Painter canvas.

Design Record Limit

> Sets the number or records to be shown when gathering data for reports and graphs.
>
> **Note:** The default record limit value for the report is 500.

Flow margin: bottom

> Sets the bottom margin for flowing reports.

Flow margin: top

> Sets the top margin for flowing reports.

Output format

> Enables you to select the output format from the following options: PDF, DHTML, Power Point, Active Report, or Excel.

Page orientation

> Sets the page orientation of the document. Select either Portrait or Landscape.
>
> **Note:** Portrait is the default page orientation.

Page size

> Sets the page size of the document. Use the drop-down list to select a page orientation option.
>
> **Note:** Letter is the default page orientation.

Units

Specifies the unit of measurements for the tool. Select Inches, Centimeters, or Points.

Note: Inches is the default unit of measurement.

Changing the Theme of the Power Painter Environment

Power Painter enables you to customize your environment for seamless integration with your organization's color scheme.

From the View menu, select **Theme** to access the available skins to choose from.
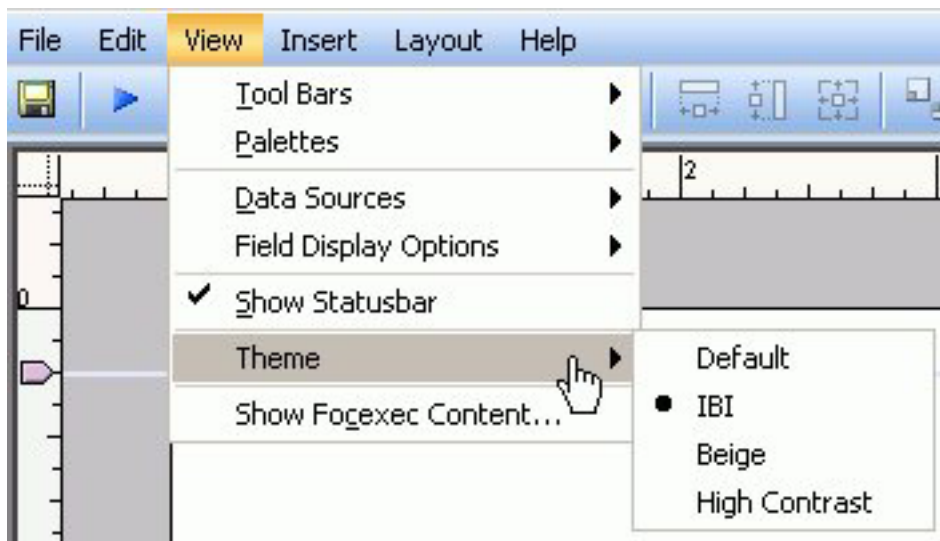
The Power Painter environment changes to the theme of the environment that you have selected.

Accessing Power Painter

You can access Power Painter from a report in the Reports folder.

Users who have the capability to create reports can access Power Painter to create a new report in the Domains tree. From a Reports folder, right-click a folder and select Power Painter from the context menu. Users can then save a report for editing or running at a later time.

To edit a report, click a report and select Power Painter from the pop-up menu that opens. The report you previously saved opens in Power Painter.

Chapter 4. Creating a Report With Report Assistant

Describes how to use the HTML Report Assistant to create tabular reports.

DB2 Web Query Report Assistant is an HTML-based graphical tool that enables you to select a data source, specify any sorting or grouping information, and display the report in your browser or another desktop application. When you use Report Assistant, DB2 Web Query creates a styled report that you can deploy on the Web without the necessity of learning the complexities of any reporting language. You can then take full advantage of the capabilities of the reporting language to efficiently display your company's data.

At the end of the chapter, reference topics provide details for every field in every tab and sub-tab.

Note: If you are using the HTML Report Assistant with a Firefox browser, the display may be slightly different than Internet Explorer. These differences do not effect functionality.

Accessing HTML Report Assistant

HTML Report Assistant is accessible when creating or editing a Report.

Selecting the Master File in Report Assistant

To choose the Master File to work with, select the file and click **OK**. To search for a specific Master File, use the scroll bar.

The following image shows the Browse Data dialog box containing available data descriptions for selection.
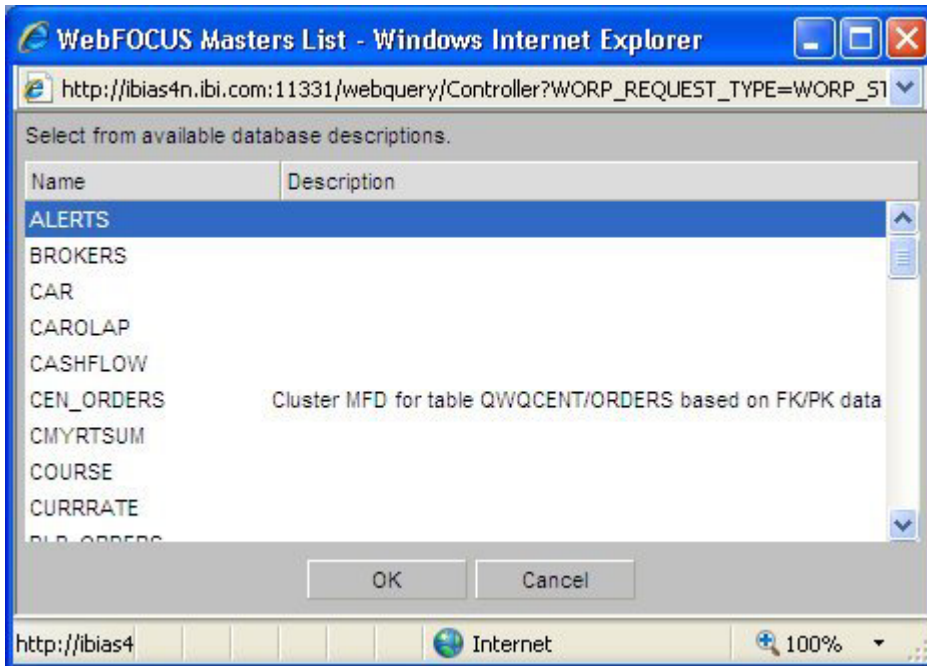
You can also type the first character of the name of the Master File anywhere in the list to jump to the name of that Master File.

Layout of Report Assistant

Report Assistant builds your report as you select graphical options by internally constructing a procedure according to the rules of the reporting language.

The following image shows the Report Assistant which is divided into these sections or tabs:
- Field selection
- Report headings
- Selection criteria
- Join options
- Report options

Each tab contains explanatory text designed to guide you on using the specific features accessed within the tab.
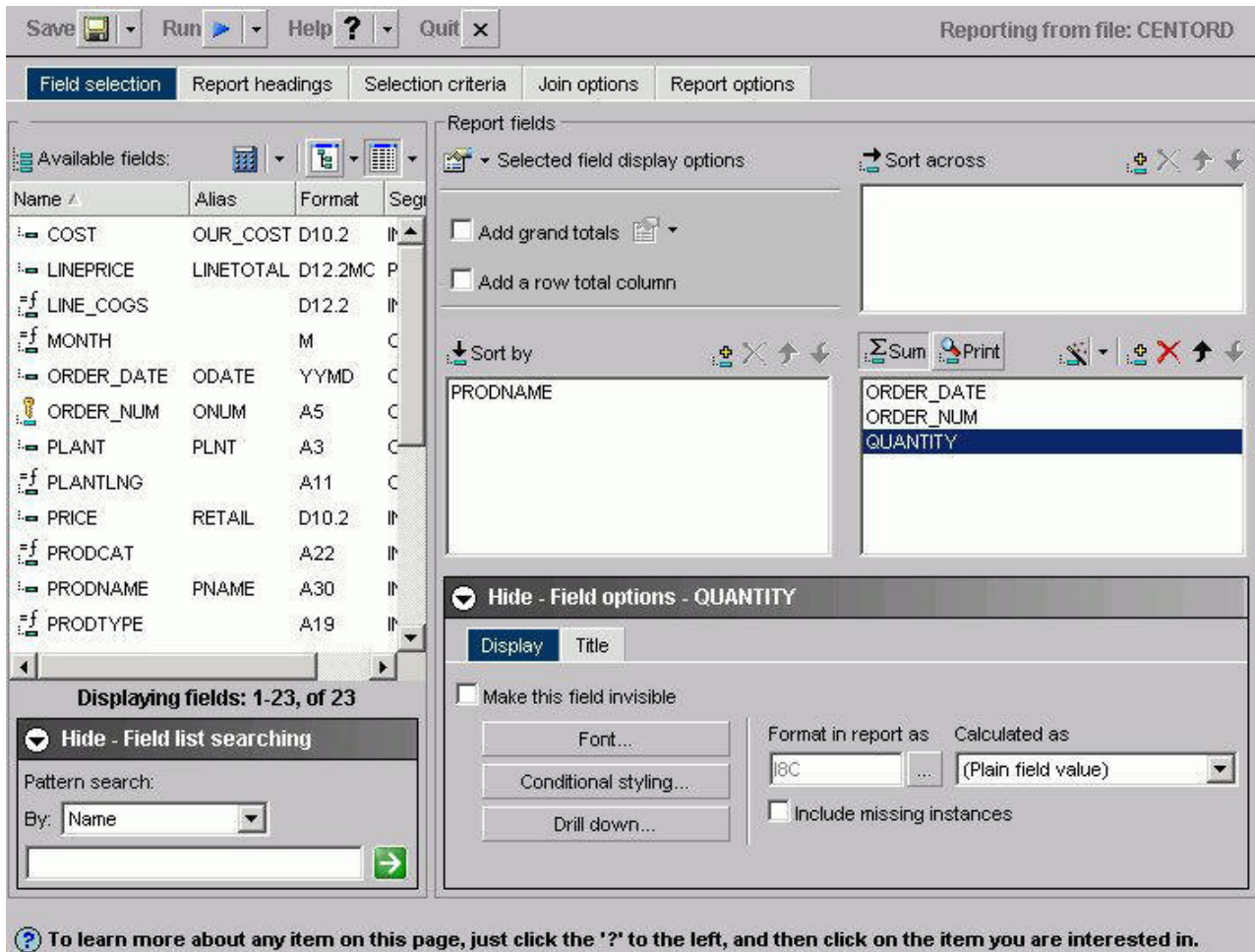
Each tab contains a set of options that enable you to customize both the content and appearance of your report. To toggle between tabs, select each tab at the top of the window. Click **About** under the Help button to display the version and release number of Report Assistant.

Above the tabs there are buttons for:

Save/Save As

> Click **Save** to save your report.

> Click **Save As** to open the Save New Report dialog box.

Run

> Click **Run** to run your report.

Help/About

> Click the icon to access help about the area of the tab in which you are working.

> Click **About** to open a window describing the version of Report Assistant in which you are working.

Quit

> Click to quit the application.

Building Your Report With Report Assistant

The most basic report you can construct is a tabular report. Other report types build on the basic design of the tabular report. A tabular report is a report whose information is arranged vertically in columns. Internally, DB2 Web Query constructs your report with a series of simple commands that mark the beginning of the request, identify the data source, and mark the end of the request. Other aspects of the request, such as styling information, are not necessary to produce a valid report request.

Use the following basic steps in constructing your report:

- **Design** your presentation. How would you like to organize the information in your report?

  See Selecting Report Fields for more information about establishing basic organizational criteria for your report.

- **Identify** the information you would like to present. Is it a full data set, or do you want to construct a subset of some larger set?

  See Selecting Records for a Report for more information about establishing selection criteria for your data.

- **Style** your presentation. Do you want to apply customized headings or footings to your report to make it more dynamic?

Selecting Report Fields

Begin building your report in the Field selection tab by selecting fields to include. A field is the smallest meaningful element of data in a file. DB2 Web Query lists available fields in several formats:

- As a list of fields shown in the following image.

Available fields:			
Name △	Alias	Format	Segi
COST	OUR_COST	D10.2	IN
LINEPRICE	LINETOTAL	D12.2MC	P
LINE_COGS		D12.2	IN
MONTH		M	C
ORDER_DATE	ODATE	YYMD	C
ORDER_NUM	ONUM	A5	C
PLANT	PLNT	A3	C
PLANTLNG		A11	C
PRICE	RETAIL	D10.2	IN
PRODCAT		A22	IN
PRODNAME	PNAME	A30	IN
PRODTYPE		A19	IN
PROD_NUM	PNUM	A4	IN
PROD_NUM	PNUM	A4	P
QTY_IN_STOCK	QIS	IT	IN

Displaying fields: 1-23, of 23
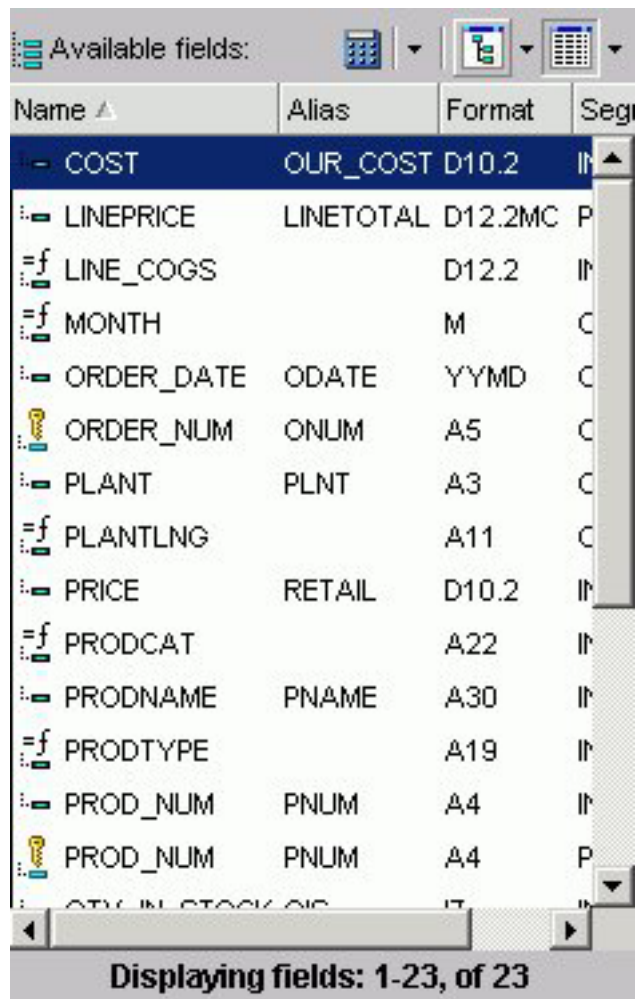
List is the default. To enable List mode, you click the List icon in the Available Fields box. You can view one or more options by clicking the Options arrow and selecting: Name, Alias, Title, Format, Description, Segment, and Filename. In List mode, you can sort any list of variables alphabetically from a to z or from z to a by clicking the attribute, for example, Name or Alias.

Note: If you choose an option, such as alias, it does not appear with a DEFINE, COMPUTE, or drill-down.

• As a tree separated by segments (the Field Tree) when expanded, as shown in the following image.

Available fields:

CAR
ORIGIN
   COUNTRY

Segment:	BODY
Alias:	TYPE
Title:	
Description:	
Format:	A12

   BODYTYPE
   SEATS
   DEALER_COST
   RETAIL_COST
   SALES
   SPECS
EQUIP
   STANDARD
WARANT
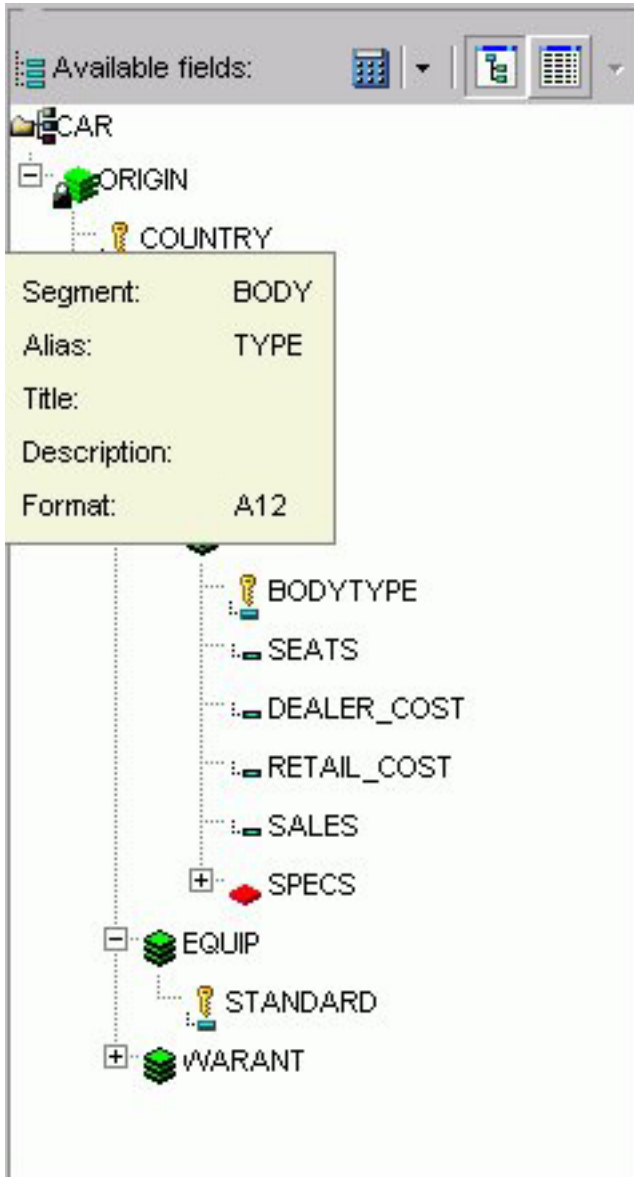
- As a tree separated by Dimensions (the Dimension Tree), as shown in the following image. This is applicable only when the OLAP license option is activated.
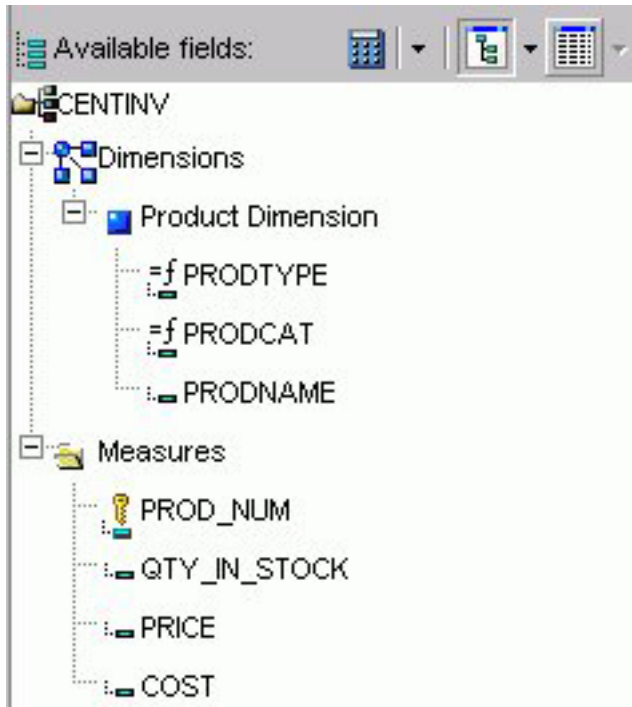
> **Note:** If you are viewing a Fields List in the Dimension Tree mode and you create a recursive join (joining a file to itself), you will not be able to view the Dimensions in the Fields List after the join is created. This is due to duplication of field names in a recursive join. If you delete the join, you will be able to view dimensions again.

To enable Tree mode, you click the Tree icon in the Available Fields box. Click the arrow next to the Tree icon to select the Field Tree or the Dimension Tree. The following options appear in a pop-up as you mouse over each field: Segment, Name, Alias, Title, Description, and Format. You can change the field attributes that appear in the tree by selecting from the list shown when clicking on the tree icon. Select from: Name, Alias, Title or Description.

The Report fields section enables you to use the prefix with the segment name or filename. You can also show the field's alias instead of the name. You can show the field's title or description instead of name or alias, but title and description cannot be prefixed with the segment name or filename. The qualified field identification cannot exceed 66 characters.

Your selections in the Report Fields menu remain in effect for your current browser session and are automatically saved with your request.

> **Note:** Field names that contain a hyphen "-" cannot be used in the HTML Report Assistant.

Notes on the Fields List

- Key fields are represented with a key icon in all field lists (tree and list displays) in the HTML Report and Graph Assistant.

The number of fields that are currently displayed shows at the bottom of the fields list. If a field list contains more than 200 fields, the first 200 fields display initially. Additional fields are retrieved and displayed when you select the next set of fields.

- To select the next set of fields, scroll to the bottom of the field list and select the Next option that appears. The Next option describes which fields will be returned, for example in the following image it shows that the next 201-400 fields of 417 will be retrieved:

- To go back, scroll to the top of the field list and select the previous set of fields. For example, the following image shows the Previous option to select the previous 1-200 of 417 fields:

Fully Qualified Field Names

The HTML Report Assistant always saves a procedure with its fully qualified field name. If you want to view this information, click the Prefix with the segment name and Prefix with the filename options in the Report Fields section of the Field selection tab.

Viewing Fully Qualified Field Names

Prefix with the segment name and Prefix with the filename are checked. Note that the fields under Sort by demonstrate the field, segment, and file names.

The following image shows the result after selecting both the prefix with segment name and prefix with the filename.
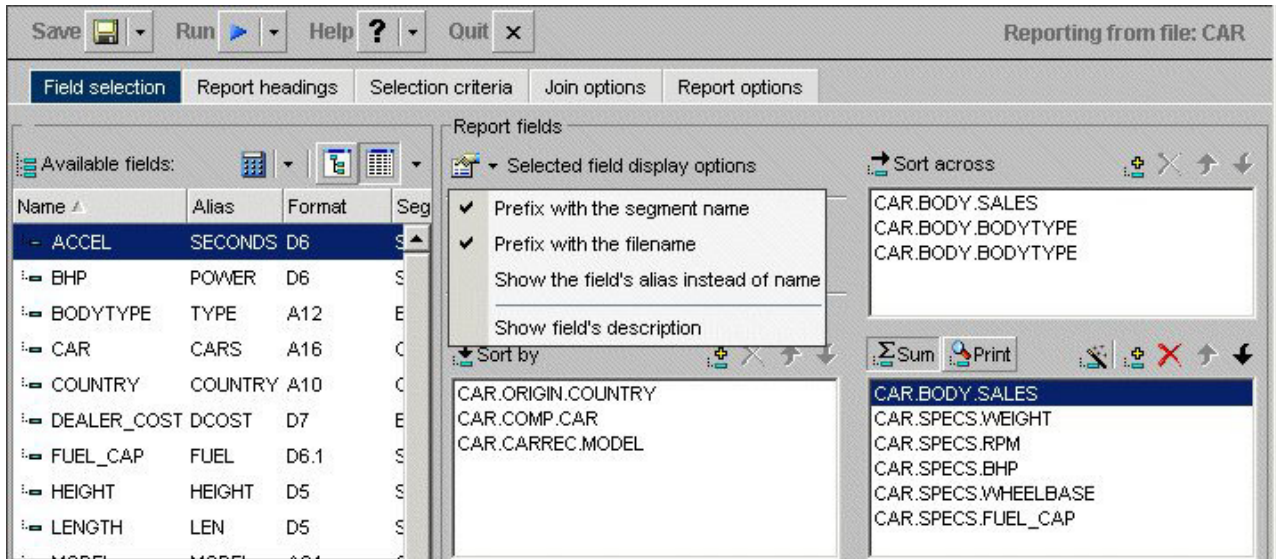
If you open a report procedure in the Editor, the fields appear with their qualifiers prefixed to the field name or alias.

The information for the highlighted field appears. If multiple fields are selected, the information that appears applies to the last field chosen.

Adding Totals and Grand Totals to a Report

You can add grand total values to the end of your report and add a row total column by checking the appropriate boxes under Report fields as shown in the following image. The Add grand totals drop-down list provides two options. The Totals option displays grand total values for each column in your report. The Recompute Totals option recalculates computed fields before displaying the grand total values. Selecting the Add a row total column check box adds a column to display the totals for each row.
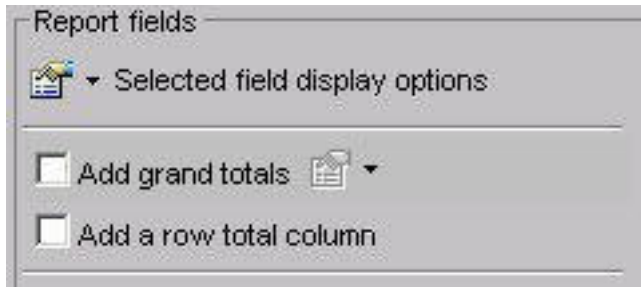
Using the Sort By, Sort Across, and Column Phrases to Organize Data

Select fields for a report by highlighting a field in the Available fields list and clicking the Add icon above the Sort by, Sort across, or Sum/Print field box. From either the List or Tree view, you can also drag and drop a field on the desired field box, or double-click a field to add it automatically.

Note: When you double-click fields, numeric fields are added to the Sum/Print field box, and non-numeric and date fields are added to the Sort by field box.

Numeric fields include decimal, floating point, and other formats, and non-numeric fields are usually in alphanumeric format.

The following image shows a field selected in each of the data field boxes, the COUNTRY field in the Sort across data field box, the CAR in the Sort by data field box, and the SALES field in the Sum/Print display field list.
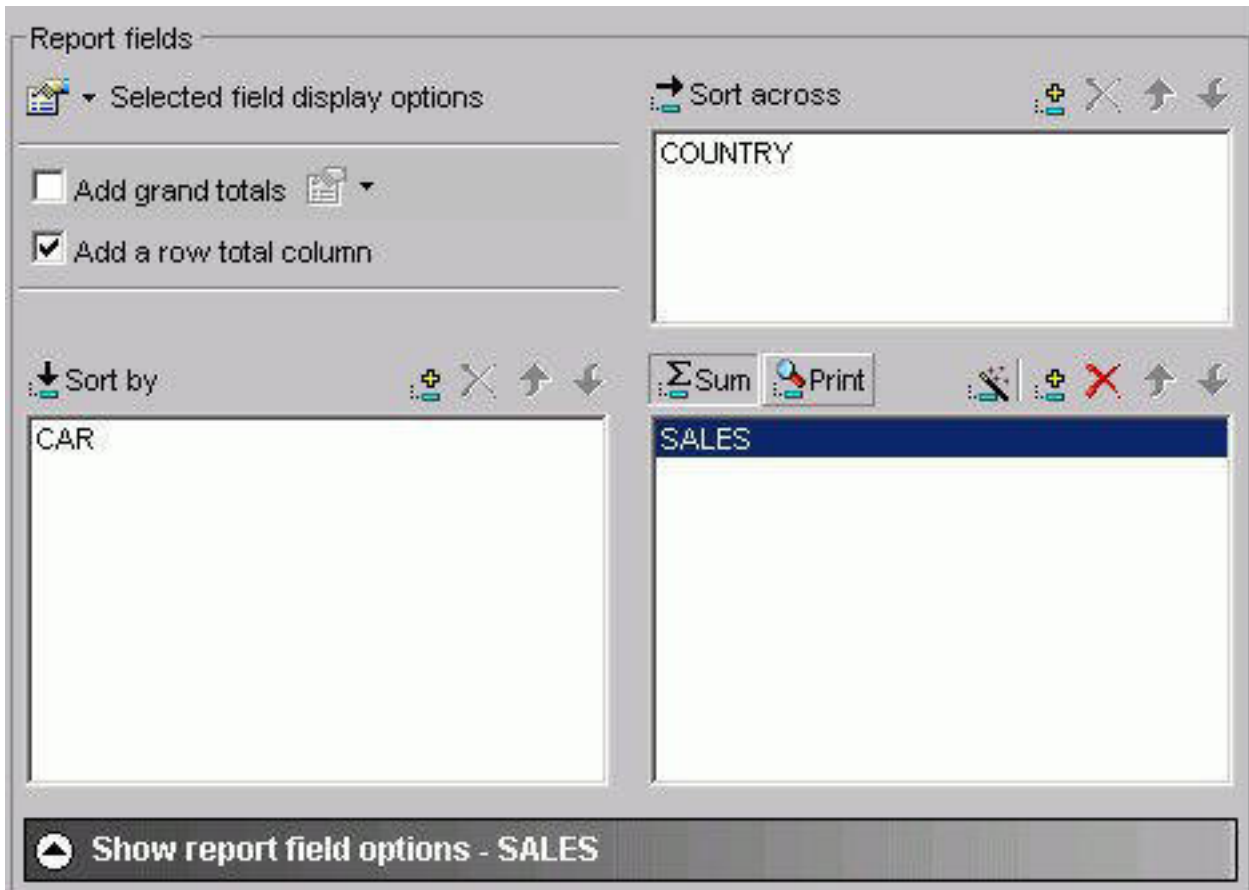
Specifying sort phrases enables you to organize the presentation of information in a desired sequence. Adding fields to the Sort across fields box establishes the field you selected as a column in your report. You can create a matrix report by combining sort and across phrases. You can include a maximum of five ACROSS fields in each report. If you try to enter a sixth ACROSS field, the following message appears:

```
The Maximum number of horizontal dimensions is 5.
```

Adding fields to the Sort by fields box establishes your field as a sort field in your report. Sort fields (also known as BY fields) are rows in your report. Any field can be a sort field and you can include up to 32 sort fields in a report.

The Column data fields box designates the selected field as Sum or Print (Detail). After selecting the field, you must also decide if you want DB2 Web Query to add the values together (Sum) or print the values individually (Print). DB2 Web Query displays the field you select in the last column of your report and performs the requested operation on the data (Sum or Print). The default selection is Sum.

You must select at least one field for the Sort or Sum/Print field lists to create a report.

Add Report Fields in Tree or List View

1. Click the **Tree** or **List** icon.
2. Select a field from the Available fields list and click the **Add** icon above the Sort by, Sort across, or Sum/Print field box to add the chosen field.

   or

   Click and hold the left mouse button, and drag the field to the desired field box.

   or

   Double-click a:
   - Numeric field to add it to the Sum/Print field box.
   - Non-numeric or date field to add it to the Sort by field box.
3. Optionally, you can use shortcut keys to quickly go to one of the field boxes.

   In the following table, the first column (Field Box) describes where you go when using the shortcut keys and the second column (Shortcut Keys) defines the shortcut key combination.

Field Box	Shortcut Keys
Sort across	Alt+A
Sort by	Alt+S
Column	Alt+C

   Press **Alt+M** to go to the Available Fields list, tab to the field and press **Enter** to add the chosen field.

Multi-Select Fields in List View

1. Click the **List** icon.
2. Required: If the fields are:
   - Adjacent to each other, click the first field you want to select and hold down the Shift key while clicking the last field you want to include in the report.
   - Not adjacent to each other, hold down the Ctrl key while clicking the fields you want to include in the report.
3. Click the **Add** icon to the Sort across, Sort by, or Column fields box to add the chosen fields.

Note: Multi-select of fields is only possible when in List mode. You must drag and drop or use the Add icon to add individual fields when in Tree mode.

Select a Report Type

To select a report type, click one of the following buttons in the Sum/Print display field list:
- **Sum.** A summary report shows the summarized values of the selected data source fields.
- **Print.** A detailed report shows all selected records from a data source.

Delete Report Fields

1. Select the field in the Sort Across, Sort by, or Column fields box.

   or

Use the shortcut keys to quickly go to one of the field boxes. In the following table the first column (Field Box) describes where you go when using the shortcut keys and the second column (Shortcut Keys) defines the shortcut key combination.

Field Box	Shortcut Keys
Sort across	Alt+A
Sort by	Alt+S
Column	Alt+C

Use the arrow keys to move to the field you wish to delete.

2. Click the **Delete** button or press the Delete key.

Searching Field Lists

You can search the field list on the Field selection tab, Report Headings tab, Selection criteria tab, and Join options tab (for the host Master File) in the HTML Report Assistant and HTML Graph Assistant. You can search by the following Master File attributes: Name, Alias, Title, Format, Description, Segment, and Filename.

When you apply a search to a field list, your search results appear in all other areas of the tool, until you reset the field list. Your search results are saved when you exit the tool, so when you open the procedure again, you only see the results of your search in the field list.

Note:
- The field list that appears for conditional styling is already restricted to the fields selected for the report and is not further restricted by the search string.
- Search options are available when viewing fields in List mode. It is not available when viewing fields in Tree mode.

Search a Field List

1. In the Field selection tab, Headings tab, or Selection criteria tab in the HTML Report Assistant or HTML Graph Assistant, click **Show - Field list searching**.

   If you are searching for a field name from the Join options tab, click the **Field list searching** icon.

   The Field list searching dialog box opens.

   The following image shows the Field list searching dialog box. It contains a drop-down list for what pattern you want to search by, a text box, and a submit button.

2. Select the Master File attribute for which you want to search. The following table lists the pattern attributes and a description of each.

Attribute	Description	DSPFFD Equivalent
Filename	Name of the data source.	File
Name	Name of the field.	Field
Alias	Alias for the field.	Alternative Name
Title	Display title for the field, as defined in the Master File.	Column Heading
Format	Field format, for example A4.	Data Type
Description	Field description, as defined in the Master File.	N/A
Segment	A segment of fields in the Master File.	N/A

3. Type your search string. You can use an asterisk (*) as a wildcard. For example, type c* to find all fields that begin with the letter "c". The character string is not case sensitive.

4. Click the right arrow ![arrow] and results appear in the field list.

Reset the Field List After a Search

1. In the Field selection tab, Headings tab, or Selection criteria tab in the HTML Report Assistant or HTML Graph Assistant, click **Show - Field list searching**.

   If you are searching for a field name from the Join options tab, click the **Field list searching** ![icon] icon.

   The Field list searching dialog box opens.

2. Enter a blank value (space bar) or an asterisk (*) and click the arrow ![arrow].

   The field list is reset to the default display, which is to show all fields.

Creating Temporary Fields

A temporary field is a field whose value is not stored in the data source, but can be calculated from the data that is there, or assigned an absolute value. When you create a temporary field, you determine its value by writing an expression. You can combine fields, constants, and operators in an expression to produce a single value. For example, if your data contains salary and deduction amounts, you can calculate the ratio of deductions to salaries using the following expression: deduction / salary.

You can specify the expression yourself, or you can use one of the many supplied functions that perform specific calculations or manipulations. In addition, you can use expressions and functions as building blocks for more complex expressions, as well as use one temporary field to evaluate another.

There are two types of temporary fields—a virtual field and a calculated value—which differ in how they are evaluated:

* A virtual field (Define) is evaluated as each record that meets the selection criteria is retrieved from the data source. The result of the expression is treated as though it were a real field stored in the data source.

Virtual fields (Defines) are not applicable to queries that originated in IBM Query/400. They can be used only with new queries created in the Web Query product.

- A calculated value (Compute) is evaluated after all the data that meets the selection criteria is retrieved, sorted, and summed. Therefore, the calculation is performed using the aggregated values of the fields.

You can create temporary fields from the Field selection tab.

Create a Temporary Field

1. Required: On the Field selection tab, click one of the following:

   - **New define field** ![icon] icon (located above the fields list)

   - **New computed field** ![icon] icon (located with the Sum/Print fields).

   The Field Creator dialog box opens.

2. Enter the name for the field in the Field box. Field names for temporary fields cannot exceed 66 characters.

3. Enter the format for the field in the Format box. Click the **Format** button for assistance in selecting a format. See Change Format Dialog Box.

4. Click in the expression box to enter an expression.

   If you are familiar with coding Compute and Define expressions, you can type the expression in the text box. Otherwise, you can use the key pad to assist you in creating the expression. To use the key pad:

   a. Click the **Tree** or **List** button to open the fields list or click the **Functions** button to open the DB2 Web Query functions list. If you hover your cursor over a function, a brief description appears.

   b. Double-click a field or function to add it to the expression box.

   c. If you select a field, complete the expression using the key pad and additional fields, as necessary. See Field Selection Tab: Field Creator Dialog Box for details on key pad operations.

5. Click **OK**. New calculated values (Compute) are added to your report's column (Sum/Print) list. New virtual (Define) fields are added to the fields list, where you can select them as if they were real fields.

Note: When you create a defined field in Report Assistant, if you want that field value to be blank, the blank must be contained in quotation marks.

Edit a Temporary Field

1. Required: Select the temporary field.

2. Required: Click the down arrow next to the Define icon and select **Edit define field**.

   or

   Above the Sum/Print display field list, click the **Edit computed field** menu item.

   The Field Creator dialog box opens.

3. Make your desired edits and click **OK**.

Delete a Temporary Field

Select a field in the Sum/Print display field list and click the **Delete** icon (or press the **Delete** key).

or

Perform the following steps:

1. Select a virtual field in the Available fields box.
2. Click the down arrow next to the Define icon.
3. Click **Delete define field**.

Examples of Creating A Virtual Field and Creating a Calculated Value

The following are examples of creating a virtual field and creating a calculated value.

Creating a Virtual (Defined) Field Using a Function

The following image shows the Define field creator dialog box with a defined field that uses a function (ARGLEN). This field uses the EMPLOYEE sample data source.
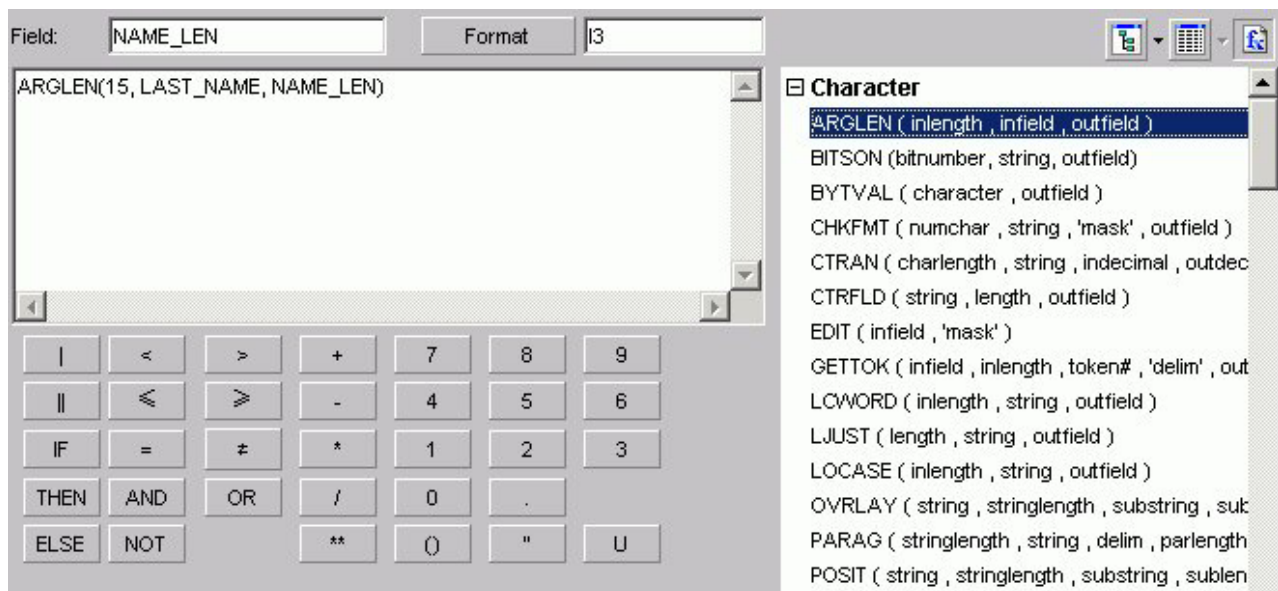
Creating a Calculated (Computed) Value

The following image shows a computed field created using the SALES sample data source. The expression creates the computed field REVENUE based on the product of the existing fields UNIT_SOLD and RETAIL_PRICE.

Field:	REVENUE		Format	D12.2			

UNIT_SOLD * RETAIL_PRICE

	<	>	+	7	8	9
‖	≤	≥	-	4	5	6
IF	=	≠	*	1	2	3
THEN	AND	OR	/	0	.	
ELSE	NOT		**	()	"	U

Name	Alias	Format	Segment
AREA	LOC	A1	STOR_SEG
CITY	CTY	A15	STOR_SEG
DAMAGED	BAD	I3	PRODUCT
DATE	DTE	A4MD	DATE_SEG
DELIVER_AMT	SHIP	I5	PRODUCT
OPENING_AMT	INV	I5	PRODUCT
PROD_CODE	PCODE	A3	PRODUCT
RETAIL_PRICE	RP	D5.2M	PRODUCT
RETURNS	RTN	I3	PRODUCT
REVENUE		D12.2	
STORE_CODE	SNO	A3	STOR_SEG
UNIT_SOLD	SOLD	I5	PRODUCT

Changing a Field Format

The following image shows the Change Format Dialog box, which you use to assign a format to a virtual or column field. Refer to the following procedures for the step-by-step instructions.
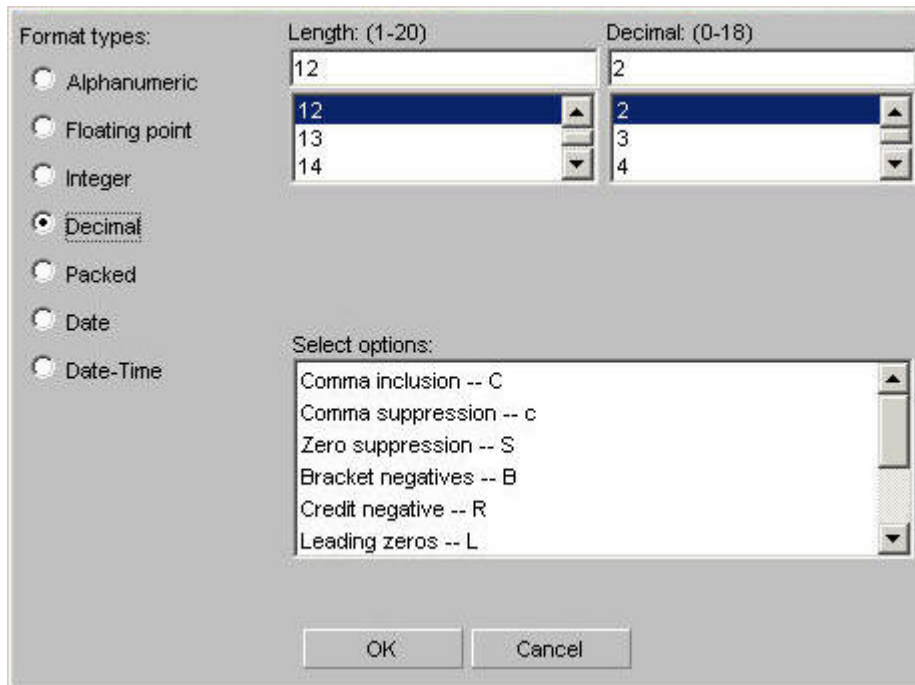
Assign an Alphanumeric Format

1. Required: Select the virtual or column field.
2. Required: Click the down arrow next to the Define icon and select **Edit define field**.

or

Click the **New computed field** icon.

The Field Creator dialog box opens.

3. Click **Format**. The Change Format dialog box opens.

4. Select the **Alphanumeric** option button in the Format Types option button group.

5. To assign a different length, specify a number between 1 and 256 in the Length spin box.

6. Click **OK** twice to close the Change Format dialog box and return to the Field selection window. The new format appears in the Format column in the Available fields list.

Assign a Numeric Format

1. Select the virtual or column field.

2. Click the down arrow next to the Define icon and select **Edit define field**.

or

Click the **New computed field** icon.

The Field Creator dialog box opens.

3. Click **Format**. The Change Format dialog box opens.

4. Select one of the following option buttons in the Format Types option button group:

   - **Floating Point** (default length 7.2)
   - **Integer** (default length 5)
   - **Decimal** (default length 12.2)
   - **Packed** (default length 12.2)

   If the selected field matches the selected format type, its current length appears in the Length spin box. Otherwise, the default length appears in the Length spin box. The Decimal spin box shows the number of decimal places for Floating Point, Decimal, and Packed.

5. To assign a different length, specify numbers in the Length spin box for format types as follows: 1- 9 for Floating Point, 1- 11 for Integer, 1- 20 for Decimal, and 1- 33 for Packed.

6. To assign a different number of decimal places for Floating Point, Decimal, or Packed, specify the number in the Decimal spin box.

7. Click **OK** twice to close the Change Format dialog box and return to the Field selection window. The new format appears in the Format column in the Available fields list.

Assign a Date Format

1. Select the virtual or column field.

2. Click the down arrow next to the Define icon and select **Edit define field**.

or

Click the **New computed field** icon.

The Field Creator dialog box opens.

3. Click **Format**. The Change Format dialog box opens.

4. Select the **Date** option button in the Format Types option button group. The default date display format is MDY.

5. To assign a different date display format, click the down arrow to the right of the Date Format list box.

6. Choose a date format.
7. Click **OK** twice to close the Change Format dialog box and return to the Field selection window. The new format appears in the Format column in the Available fields list.

Add a Percent Sign to a Numeric Field

You can add a percent sign to the end of a numeric value (Decimal, Integer, Floating Point format types). This numeric display option includes a percent sign along with the numeric data, but does not calculate the percent.

1. Select the virtual or column field.
2. Click the down arrow next to the Define icon and select **Edit define field**.

   or

   Click the **New computed field** icon.

   The Field Creator dialog box opens.
3. Click **Format**. The Change Format dialog box opens.
4. Click the down arrow in the Select options list box to choose **Percent sign - %**.
5. Click **OK** twice to close the Change Format dialog box and return to the Field selection window. The new format appears in the Format column in the Available fields list.

Using the Percent Sign Edit Option

The following table consists of a Format column for each numeric format, a Data column for its actual data value, and a Display column for how it appears.

Format	Data	Display
I2%	21	21%
D7%	97	97%
F3.2%	48	48.00%

Using Calculated Values From the Master File

In the HTML Report and Graph Assistants, calculated values (COMPUTEs) that exist in the Master File appear in the fields list with all other temporary virtual fields and calculated values. The following image shows the fields list in the HTML Report Assistant with a calculated value from the Master File (PROFIT).
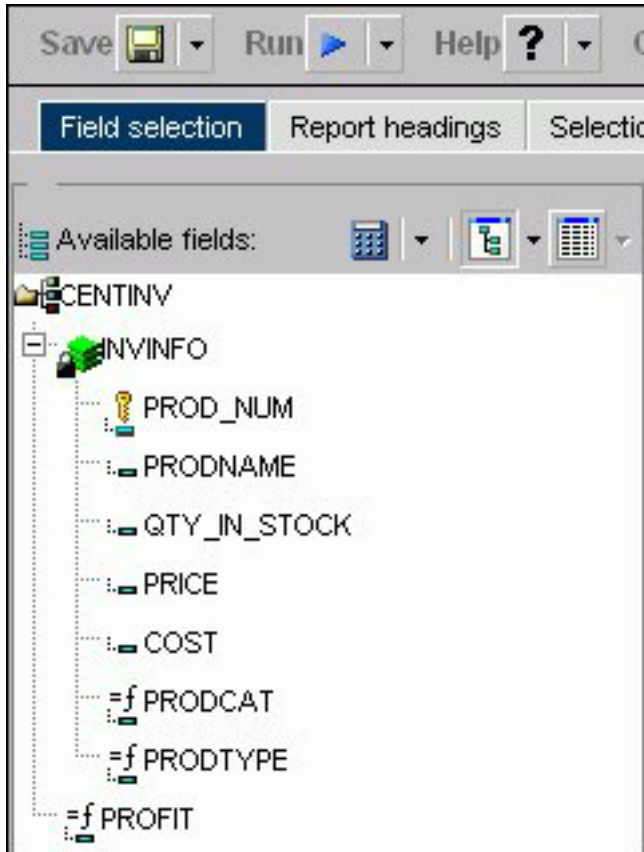
Calculated values from the Master File are available in all fields lists in the HTML Report and Graph Assistant except for the fields lists that appear for Define fields and Joins.

Support for Calculated Values From the Master File

You can use a calculated value from the Master File:

- As a Sum/Print field.
- As a Sort by field. You can also use a temporary calculated value as a Sort by field.

  Note: The By sort is automatically converted to a Total field since calculated values are calculated after data has been sorted and aggregated, but before the report is displayed. Since sorting (BY) has already been done, the phrase BY TOTAL is used to indicate that this sort is performed after any BY sorting. This applies to calculated values from the Master File and temporary calculated values.

- For conditional styling.
- In a temporary calculated value.
- In a page heading or page footing (Report headings tab).
- In selection criteria statements. You can also use a temporary calculated value in selection criteria.

  Note: When you use calculated values for selection criteria, the Where is automatically converted to Where Total. This occurs because the Where Total is added after the data is retrieved and sorted, but before the report is finalized. This applies to selection criteria created with calculated values from the Master File and temporary calculated values.

Calculated fields from the Master File are not supported:

- As Across sort fields.
- As the X-axis in a graph. This means that you won't be able to add a calculated value to the **Field value plotted on X axis** or **Create a separate graph for each value of this field** fields on the Field selection tab of the HTML Graph Assistant.
- In a Join statement or Defined field.
- With prefix operators, for example MAX.

Calculated Value Dependencies

These dependencies apply to calculated values from the Master File and to temporary calculated values.

When you include a calculated value in selection criteria or in a page heading/footing, the field must exist in the report as either a By sort field or a Sum/Print field. If you create a selection criteria statement with a calculated value or add a calculated value to a page heading/footing and the field does not already exist in the report, then it is automatically added to the Sum/Print display field list. The **Make this field invisible** option is automatically selected in the Field options dialog box so that the calculated value does not appear in the output. When this occurs, a message appears stating:

"The field has been added to the list of fields in the report."

Note: You do not see this message for temporary calculated values (those that do not come from the Master File) that are added to a page heading/footing since these already appear in the Report headings fields list.

You can change the attributes of the calculated value from the Field options dialog box on the Field selection tab. If you attempt to delete the calculated value from the Sum/Print display list and the same field exists in either selection criteria or in a page heading/footing, you receive a one of the following messages:

> ⚠ This field is linked to the Page heading or Page footing. Remove the field from the Heading or Footing before removing it from the Selected fields.

or

> ⚠ This field is linked to the Selection Criteria. Remove the field from the Selection criteria before removing it from the Selected fields.

You must remove the field from the page heading/footing or selection criteria before you can remove it from the Sum/Print display list.

Defining Field Characteristics

To choose field options, select the field in the Sort across, Sort by, or Column (Sum/Print) box in the Field selection tab. Click **Show field options** to style each field in your report.

The following table lists and describes the available Field options in the Display, Sorting, Ranking, Subhead, and Subfoot tabs that appear for each selected field type. The ability to create a title in the Title tab is available for all field types. If a Field option tab is not available for a selected field type, "n/a" appears in the table.

Field Type	Display	Sorting	Ranking	Subhead & Subfoot
Sort across	Make this field invisible check box Font button that opens styles dialog Drill down button that opens Drill down dialog	Ascending/ Descending options Total check box	n/a	n/a
Sort by	Make this field invisible check box Font button that opens styles dialog Conditional styling button that opens conditional styling dialog Drill down button that opens Drill down dialog Include missing instances check box	Ascending/ Descending options Total check box Create a page break check box (activates the 'And reset page number to 1' check box) Separate with underline check box Subtotal numeric sum/print fields check box (activates the Subtotal tab, and the 'And higher level sort fields' and 'Recalculate computed fields with subtotals' check boxes)	Add Ranking column check box (activates the 'Limit the number of ranked values to' check box, and the Font button that opens styles dialog)	Select Font type, size, and style. Select text alignment. Set text and background colors.

Field Type	Display	Sorting	Ranking	Subhead & Subfoot
Column (Sum/ Print)	Make this field invisible check box Font button that opens styles dialog Conditional styling button that opens conditional styling dialog Drill down button that opens Drill down dialog Format in report as button opens change format dialog 'Calculated as' drop-down list adds prefix to column Include missing instances check box	n/a	n/a	n/a

You can use this table to determine which of the following options are available for a selected field in the Field selection tab:

- Suppress the display of a field (make a field invisible).
- Apply font options with the styles dialog box.
- Apply conditional styling with the conditional styling dialog box.
- Create drill down parameters with the Drill down dialog box.
- Select format options in the report with the change format dialog box.
- Specify how the field will be calculated by choosing an option from a drop-down list box.
- Include references to missing instances in reports.
- Specify field display titles, subheads, subfoots, and subtotals.
- Specify ascending or descending field sorting.
- Include the Sort by options for page breaks.
- Include the Sort by options for subtotals, sub-totals, and recalculated subtotals.

Note:
- **Subtotal numeric sum/print fields** is not checked by default. If checked, numeric values are summed and a subtotal is displayed when the selected Sort by field value changes.

- If you select **And higher level sort fields**, SUB-TOTAL syntax is used to display subtotals for numeric values when the selected Sort by field value changes and when any higher-level Sort by field value changes.
- If you are using computed fields, select **Recalculate computed fields using subtotal values**.

Change the Field's Display Title

1. Select the field whose title you want to change.
2. If hidden, click **Show report field options** for the selected field.
3. Click the **Title** tab.
4. Enter the title you want to display in the report output.

   Note: When using HTML output, leading spaces in the field title do not display.

Suppress the Display of a Field

The Field selections tab enables you to conceal the data of a selected field in a report. To suppress the display of a field:

1. If hidden, click **Show report field options** for the selected field.
2. Check the **Make this field invisible** check box.

Include References to Missing Instances in Reports: The ALL. Prefix

In a report, you can include parent segment instances that lack descendants by attaching the ALL. prefix to one or more column fields, as follows:

1. If hidden, click **Show report field options** for the selected field.
2. Check the **Include missing instances** check box.

Note:

- The ALL. prefix and an additional prefix operator from the Column Options drop-down list may be applied to the same field.
- The ALL. prefix can be applied to multiple fields.
- The ALL. prefix can be applied to both column fields and sort fields.

Arrange Fields in Ascending or Descending Order

To list sort field data in the report from greatest to smallest (descending) or from smallest to greatest (ascending):

1. If hidden, click **Show report field options** for the selected field.
2. Select the **Sorting** tab.
3. Select the **Ascending** or **Descending** option button.
4. Click **Save**.

If you have multiple sort fields, you can specify a different order for each one.

Include Page Breaks

To start a new report page when the value of a selected sort field changes:

1. If hidden, click **Show report field options** for the selected field.
2. Select the **Sorting** tab.
3. Check the Create a **page break** check box.
4. Click **Save**.

Include Subtotals

To display a subtotal for numeric data when a selected sort field changes:

1. If hidden, click **Show report field options** for the selected field.
2. Select the **Sorting** tab.
3. Check the **Subtotal numeric sum/print fields** check box.

   Subtotals are activated for this report and the Subtotal tab appears to the right of the Subfoot tab. Note that the default *TOTAL title is displayed next to each subtotal value. For more information, see Include Display Titles for Subtotals.

   Additionally, the following options are activated:

   - And subtotal all higher level sort fields. Displays subtotals for numeric values when the Sort by field value changes and when any higher-level Sort by field value changes.
   - Recalculate computed fields with subtotal values.

4. Required: Optionally, select the **Subtotal** tab to:

   - Modify the default subtotal title text in the Current Subtotal Text field.
   - Suppress subtotals for sort fields that have only a single value.
   - Apply subtotals to a subset of the numeric fields in a report.
   - Apply prefix operators, which enables you to display the sum, average, average square, count, first in group, last in group, minimum, or maximum value for any or all of the numeric fields, including Computes, in a report.

   For more information, see Displaying Subtotals and Other Summary Values Using Prefix Operators.

5. Click **Save**.

Include Display Titles for Subtotals

When including subtotals, you can also specify display titles for them. These titles appear next to each subtotal in the report. The default subtotal display title is *TOTAL. To edit the default display title:

1. If hidden, click **Show report field options** for the selected field.
2. Click the **Sorting** tab.
3. Check the **Subtotal numeric sum/print fields** check box. The Subtotal tab appears to the right of the Subfoot tab.
4. Click the **Subtotal** tab.
5. Type the desired display title in the Current Subtotal Text field in the Subtotal tab.
6. Click **Save**.

Note: If the Subtotal numeric sum/print fields check box is not selected, subtotals are not displayed in the report.

Include Subheads/Subfoots for Sort Fields

The Fields Options tab enables you to include subheads (which appear above) and subfoots (which appear below) for each of the sort field's records in the report.

1. If hidden, click **Show report field options** for the selected field.
2. Click the **Subhead** or **Subfoot** tab.
3. Click in the text box or press **Alt+T** to enter a subhead or subfoot title.
4. Style the subhead/subfoot using the formatting toolbar.
5. Click **Save**.

Editing a Field Format

Use the report field options section on the Field selection tab to change the field format for the Column field. The following image shows the Field selection tab with a field selected and the Field Options pane open.
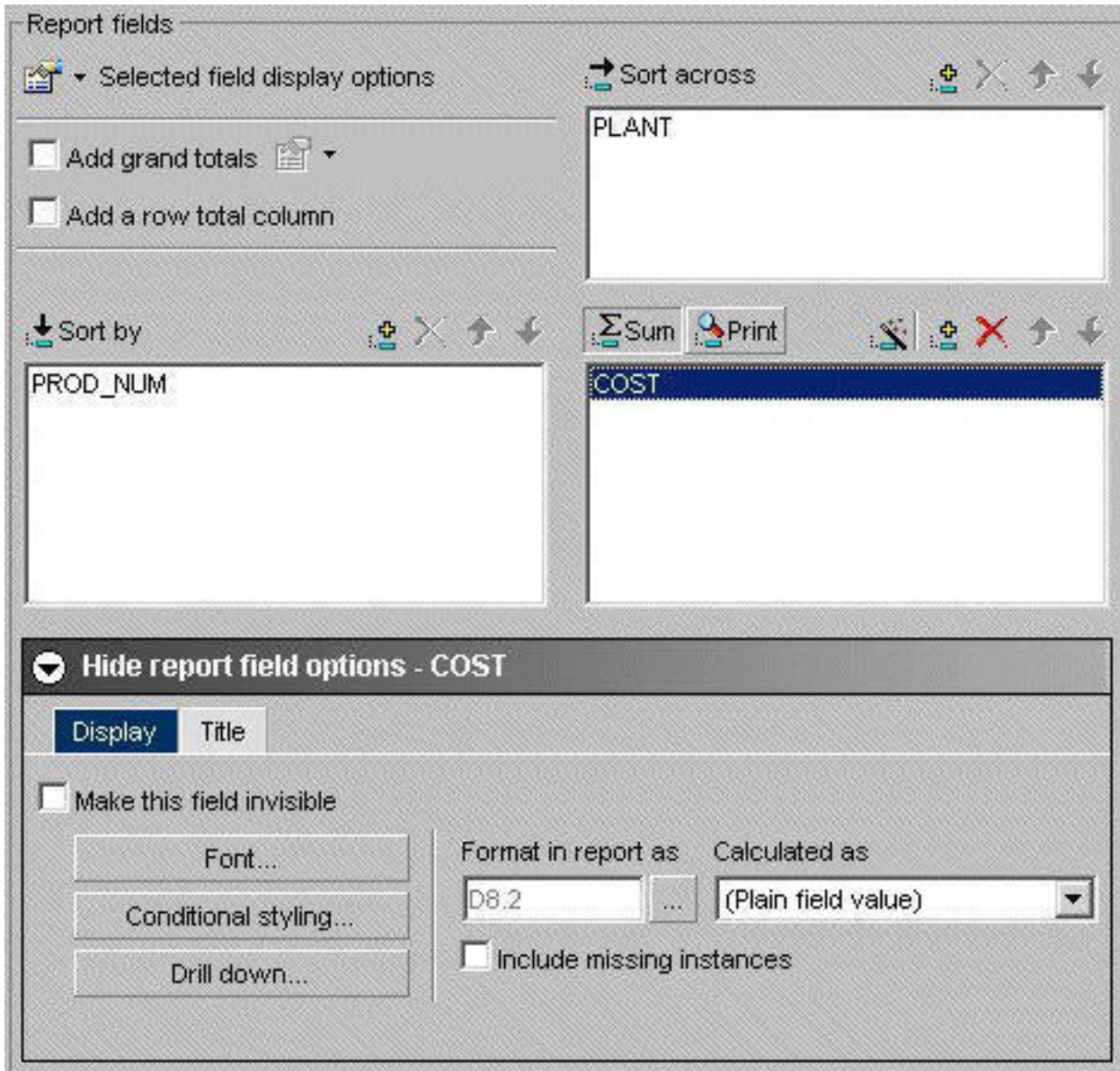
Change the Field Format for a Column Field

1. Required: If hidden, click **Show report field options**.
2. Required: Select a field in the Sum/Print display field list.
3. Required: Click the browse (...) button next to the **Format in report as** field to open the Change Format dialog box.

   Note: You can also edit the format by typing directly in the **Format in report as** field. This is not recommended unless you know the exact format because validation is not performed until run time.
4. Select a Format type and the desired options for the field, then click **OK**.

For more information, see Changing a Field Format.

Styling Fields

Use the styling options on the Field selection tab to style report fields.

Style Sort by, Sort Across, or Column Fields

1. If hidden, click **Show report field options**.
2. Select a field and click the **Display** tab in the report field options section.

   The following image shows the Display tab of the Field Options pane.
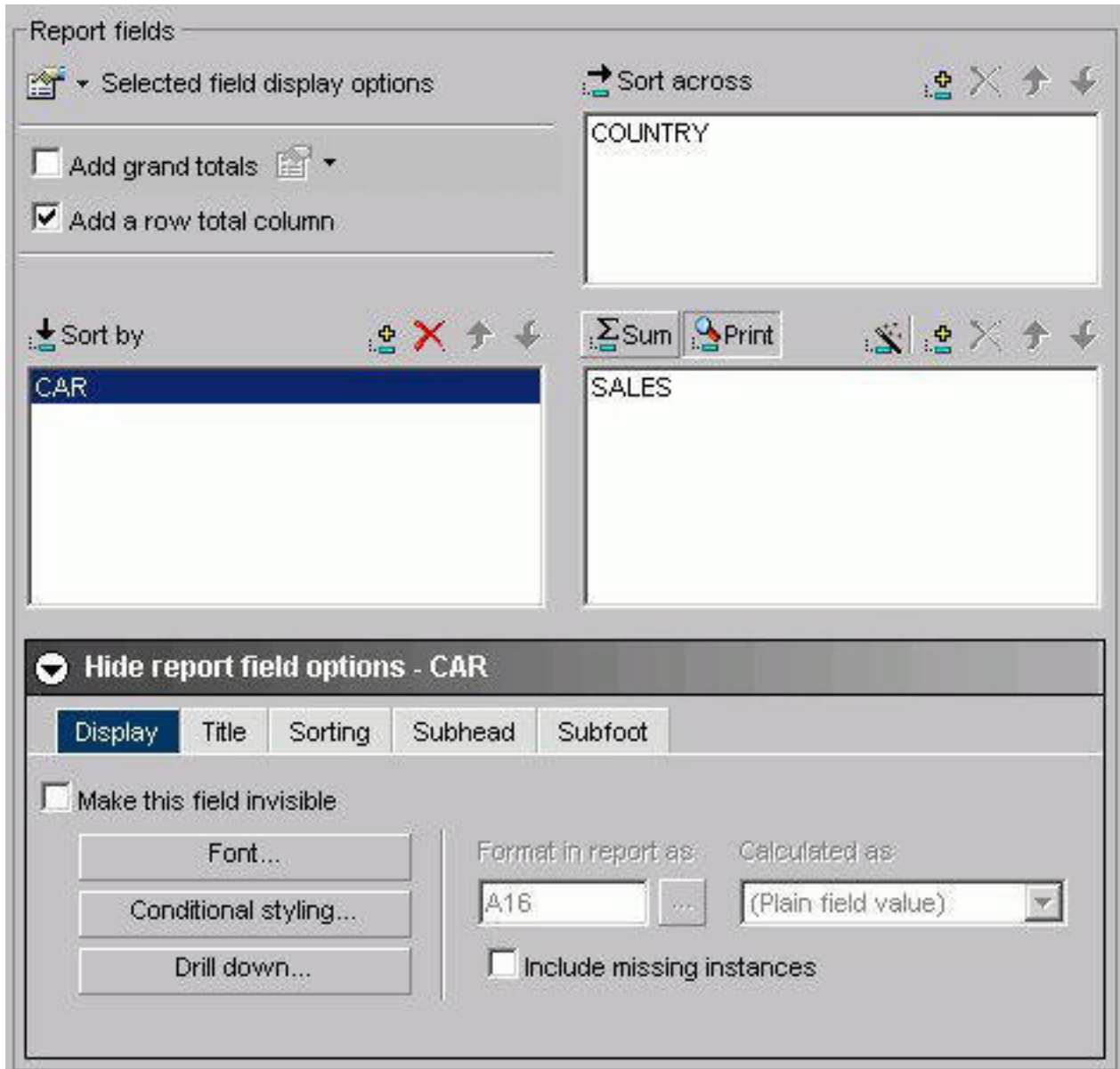
3. Click the **Font** button.
4. Select the desired styling options from the Font, Style, and Size boxes.
5. In the Set Styles for box, select the **Title & Data**, **Title Only**, or the **Data Only** option button, depending on the report element you wish to style.

   Note: If you select a font property (font, style, size, text color, background color, or justification) and select the **Title & Data** option button, the changes will be reflected in both the Title Style and Data Style boxes.

However, if you then select a different parameter for a given property using the **Title Only** option button, the Title Style will change while the Data Style will not reflect this change. To change the Data Style, you must click the **Data Only** option button and then make your change. Likewise, if you select the **Data Only** option button and make a change, this change will not be reflected in the Title Style. You must click the **Title Only** option button to make your change.

6. Choose a text color from the list box.
7. Choose a background color from the list box.
8. Choose a justification (left, center, or right) from the list box.
9. Click **OK**.

Note: Clicking Reset returns the styling options to the following default settings:
- Font: Times New Roman
- Style: Normal
- Size: 13
- Text Color: Black
- Background Color: Transparent
- Justification: Left

Drilling Down to New Procedures

When you click Drill Down in the report fields options section on the Field selection tab, the No action option button is the default. When you select Execute Procedure, a list of procedures contained within the Reports subfolder displays. You can also select procedures from different domains.

For more information, see Drill Down Dialog Box.

End User Drill-down Capability

Along with Administrators and Developers, end users can drill down to other reports from within the currently displayed report.

Styling Specified Field Values

Use the Conditional Styling dialog box to style records based on specified field values. Conditional styling, also referred to as stoplighting, enables you to define conditions that determine when to apply particular fonts, point size, text style, foreground and background color, and drill-down procedures to your report's data when the report is run.

You can style specified values for one or several column and sort fields. However, you cannot style values for sort across fields.

Style Records Based on Specified Field Values

1. In the HTML Report or Graph Assistant, if hidden, click **Graph Field Options**.
2. Click the **Display** tab in the report field options section.
3. Click the **Conditional styling** button.
   The Conditional Styling dialog box opens.
4. Click the **Add** button and create a new condition.
   The following image shows the edit condition dialog box that opens where you begin to set the conditional styling.
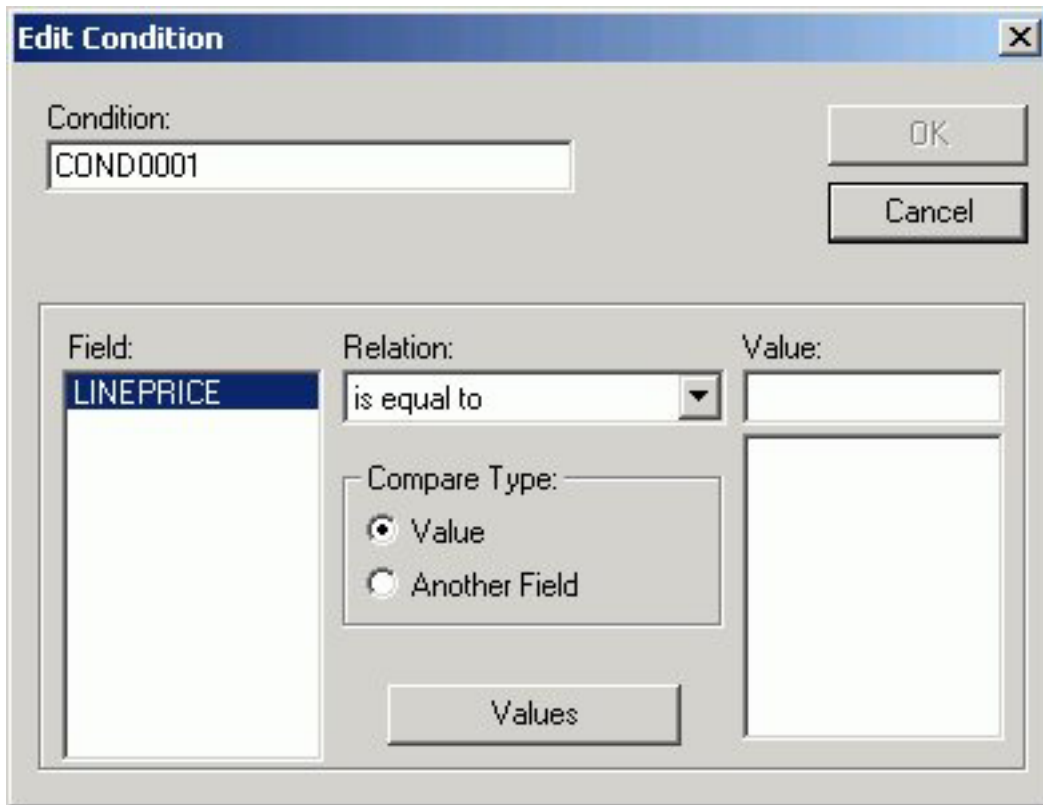
5. Provide a meaningful and unique file name in the Condition input box.
6. Select a field name from the Field list box and a relation from the Relations list box.
7. To specify a value that completes the relation, first select one of the following in the Compare Type option button group:
   - **Value**, to compare the selected report field to a data source value or literal value.
   - **Another Field**, to compare the selected report field to the value of another field. The list of report fields appears in the possible values area. Select one of the report fields to compare to.
8. If you select Value under Compare Type, perform one of the following to complete the relation:
   - Click **Values** to display existing data source values in the Value list box, and select a value.
   - Type a literal value in the Value input box.
9. Click **OK**.

   You return to the Conditional Styling dialog box.

Creating Parameters

Parameters enable you to specify criteria and conditions for drill-down reports. By defining parameters, you can control the amount and type of information to retrieve when you click a hotspot.

Create a Parameter in the Main and Drill Down Procedures

1. Select the **Selection criteria** tab.

2. Create an expression (WHERE statement) that defines a parameter. For more information, see Creating a Where or Where Total Statement.
3. If hidden, click the **Show report field options** section.
4. Select the field that you want to drill down on.
5. Click **Drill Down**. The DB2 Web Query Drill Down Dialog opens.
6. Click **Execute Procedure (FOCEXEC)** or **Execute URL**.

   The following image shows the Execute Procedure option button selected and the location and procedure name of the drill-down report (app/css.fex) in the Execute procedure field.
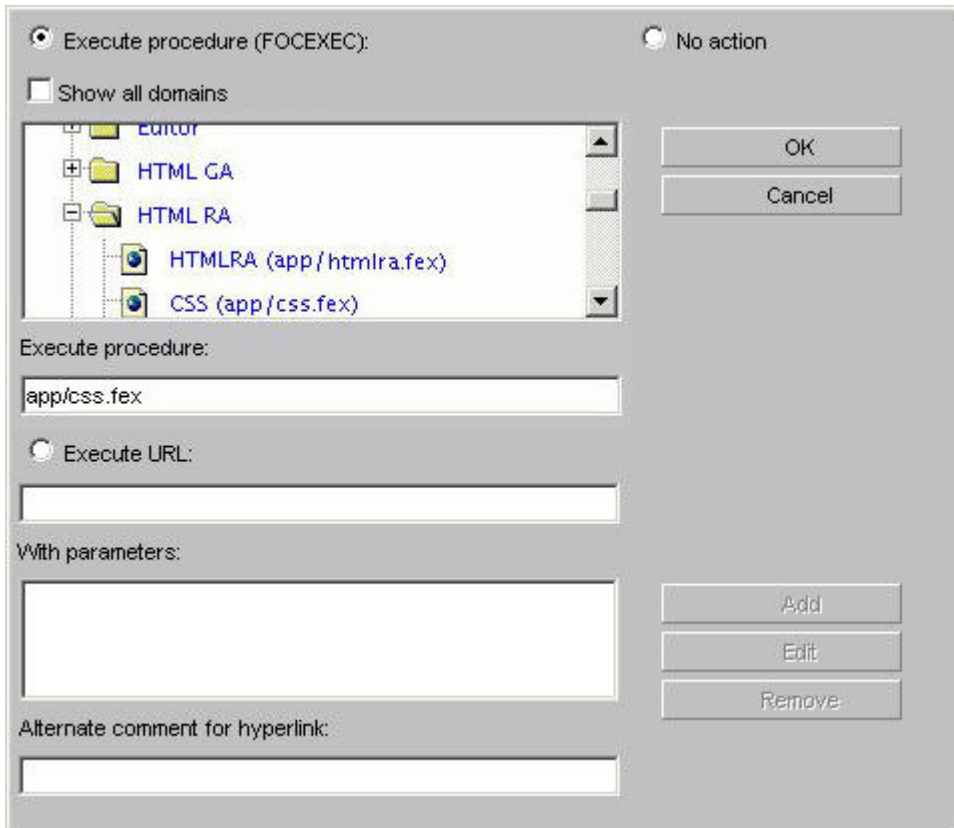
7. Click **Add**. The Drill Down Parameter Dialog box opens where you create the drill down parameter.

   The following image shows the Drill Down Parameter dialog box, which contains a text box where you enter the name of the parameter, and option buttons to select the parameter as a field or a constant value.

8. Enter the name of the parameter you created in the drill down procedure in the Parameter Name text box, for example, **Classes**.

   Note: When passing parameters to a drill-down procedure, you must use local amper variables (&variables). Global amper variables (&&variables) cannot be used as drill-down parameters. Also, when entering your parameter name it is not necessary to type an ampersand (&) before the parameter name. This will create a global amper variable that cannot be used as a drill-down parameter.

9. When you pass the parameter to the drill down procedure, you must set a value for it in the Drill Down Parameter dialog box. If you select:

   • **Field.** The parameter will be set to the corresponding value of the object the user drills down on in the specified field.

   • **Constant value.** The parameter is set to the specified value.

   Note: If the drill-down report contains a -DEFAULTS statement that sets a default value to the same amper variable passed from the main report, the amper variable value passed down overwrites the -DEFAULTS statement in the target procedure.

10. Once a value has been supplied, click **OK** to return to the Drill Down Dialog box.

    The parameter is added to the With Parameters list box as shown in the following image.
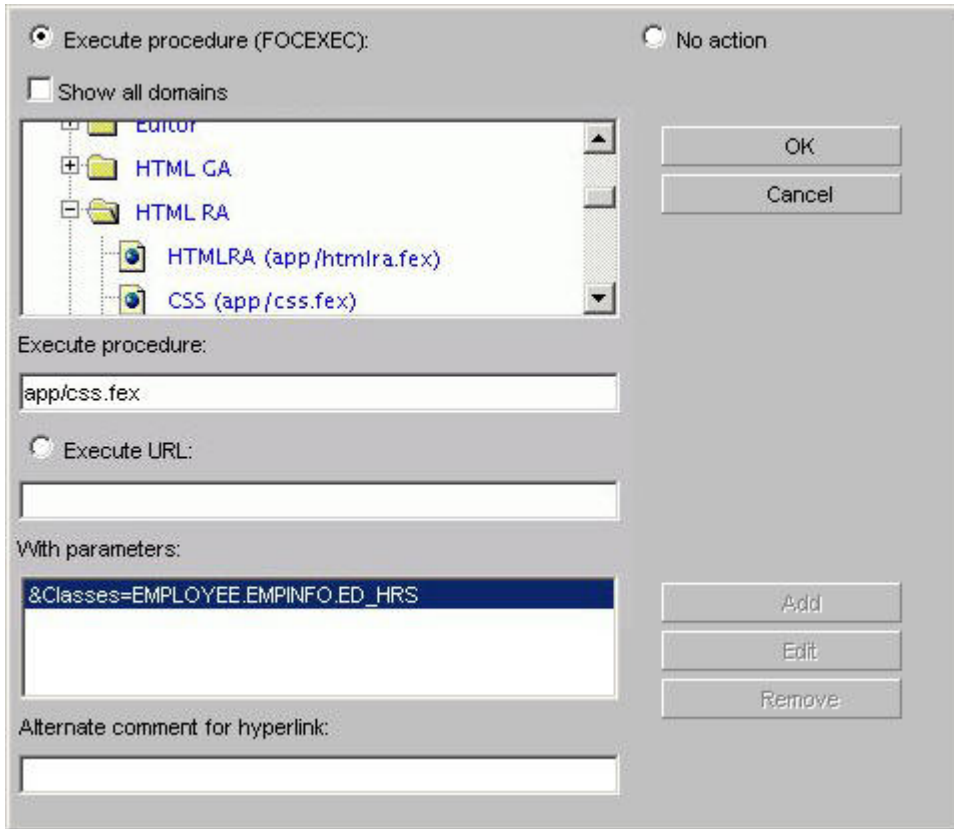
Aggregating and Sorting Fields

You can apply aggregation and sorting simultaneously to numeric columns in your report in one pass of the data using the Total sort option. The Total option is available for Sort by and Sort across fields. For the Total sort option to work correctly, you must have an aggregating display field, such as Sum, in the report. A non-aggregating display field, such as Print, simply retrieves the data without aggregating it.

Note: When you add a computed field to the Sort by field, it is automatically given the Total property.

The following image shows the PRICE field with the Total option selected in the Field options.
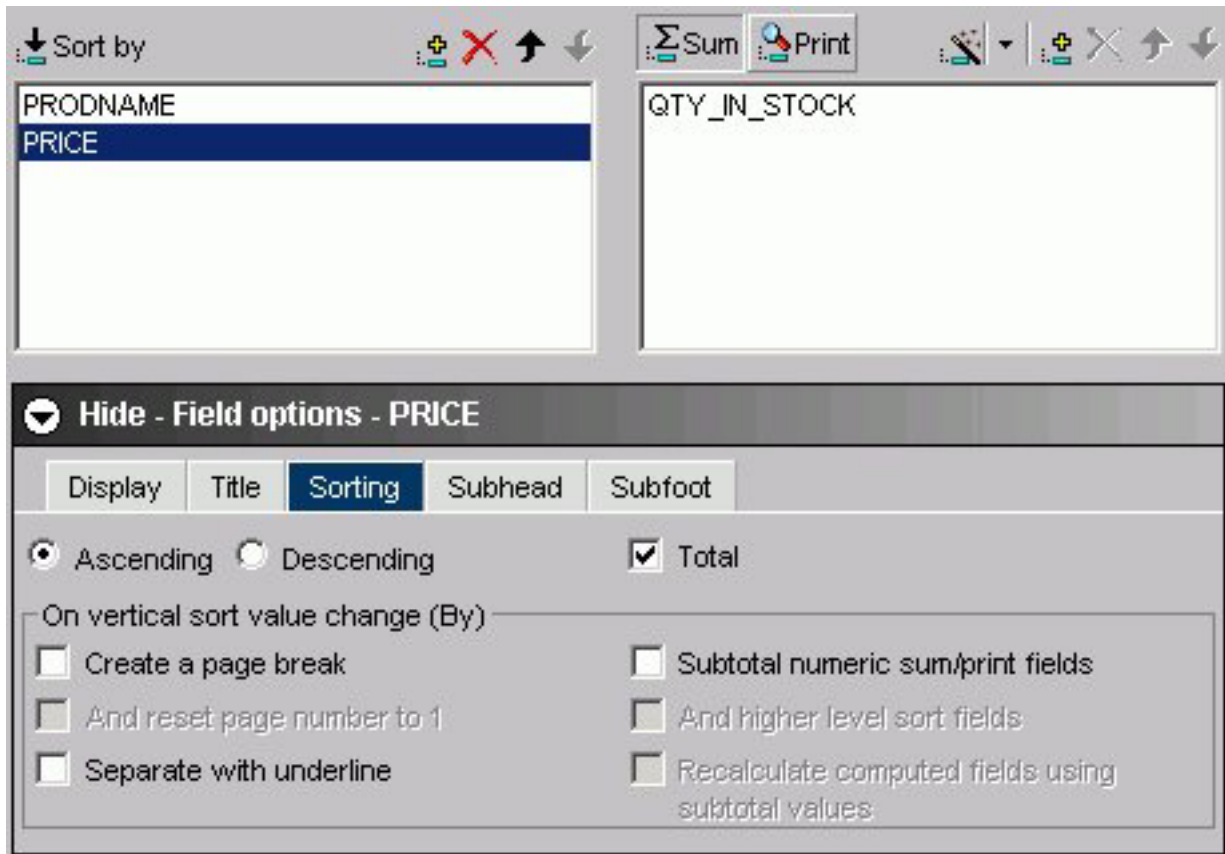
Rank Data With the Total Sort Option

To rank report data using the Total option on the Sorting tab, perform the following steps:

1. Add multiple fields to the Sort by data field box, and highlight the Sort by field you want to rank.
2. Add any other desired fields to the Sum/Print field box.
3. If hidden, click **Show field options** for the selected field.
4. Click the **Sorting** tab.
5. Select the **Total** check box.

   Tip: Select the Descending option to make the largest number the first row of ranked data in the report. The default sorting option is Ascending, which ranks the smallest number first.

6. Click the **Ranking** tab.
7. Select the **Adding Ranking column** check box.

   Optionally, in the text box, type a custom title for the ranking column. The default title is RANK.

8. Run the report.

Note: For the Total sort option to work correctly, you must have an aggregating (numeric) display field, such as Sum.

Displaying Subtotals and Other Summary Values Using Prefix Operators

The Subtotal tab enables you to display subtotals, and other summary values using prefix operators, for any or all of the numeric fields, including computed fields, for a selected Sort by field in a report query. You can choose to display subtotals for all numeric fields whenever a selected Sort by field value changes, or you can select to display subtotal and other summary values for a subset of the numeric fields.

In the Subtotal tab, as shown in the following image, you can customize the title text used to display subtotals and other summary values, and suppress subtotals for sort fields that have only a single value, in addition to working with prefix operators.

Display Subtotals and Other Summary Values Using Prefix Operators

To access the Subtotal tab to apply subtotals, prefix operators, and modify other subtotal text and settings, perform the following steps:

1. If hidden, click **Show report field options** for the selected field.
2. Select the **Sorting** tab.
3. Select the **Subtotal numeric sum/print fields** check box. The Subtotal tab appears to the right of the Subfoot tab.
4. Select the **Subtotal** tab.
5. Numeric field values are summed and displayed as subtotals in the report by default. To display summary values other than the default subtotals, select prefix operators using the drop-down lists in the Prefix area that are adjacent to each of the selections in the Columns area. Note that the Sum prefix produces the same subtotal results as not selecting a prefix operator.

   All of the available prefix operators are listed and described in the following table:

Prefix	Description	Displayed in Prefix List
SUM.	Compute the sum of values	Sum

Prefix	Description	Displayed in Prefix List
ASQ.	Compute the average sum of squares	Average Square
AVE.	Compute the average value	Average
CNT.	Count the items	Count
FST.	Select the first value only	Show first in group
LST.	Select the last value only	Show last in group
MAX.	Select the maximum value only	Maximum
MIN.	Select the minimum value only	Minimum

6. To display all numeric field values as subtotals or other summary values in the report, select the **Apply totals to all columns** check box in the Columns area.

7. To display subtotals or other summary values for a subset of the numeric fields in the report, make sure the **Apply totals to all columns** check box is de-selected, then select check boxes for the individual numeric fields in the Columns area of the Subtotal tab.

   When **Apply totals to all columns** is selected, options for all individual fields in the Columns area are inactive by default.

8. To modify the default subtotal title, which is set to *TOTAL, type the desired text you want to display as the title for all subtotals and any other selected summary values in the Current Subtotal Text field.

9. To hide subtotals for sort fields that have only a single value, select the **Suppress for single lines** check box.

10. Click **Save** to save your settings and selections in the report query in which you are working.

Applying Predefined and Cascading StyleSheets

You can select a predefined StyleSheet from Report Assistant, instead of applying custom styling. A predefined StyleSheet is a StyleSheet that a DB2 Web Query Administrator places in a domain using Developer Workbench. Any user who then accesses Report Assistant from the domain can apply the predefined StyleSheet to their report.

When a new procedure is created, Report Assistant sets a default StyleSheet. You can apply a different StyleSheet to a report using the StyleSheet drop-down menus in the Global report styling area of the Report options tab.

The following image shows the report styling options in the Report options tab.
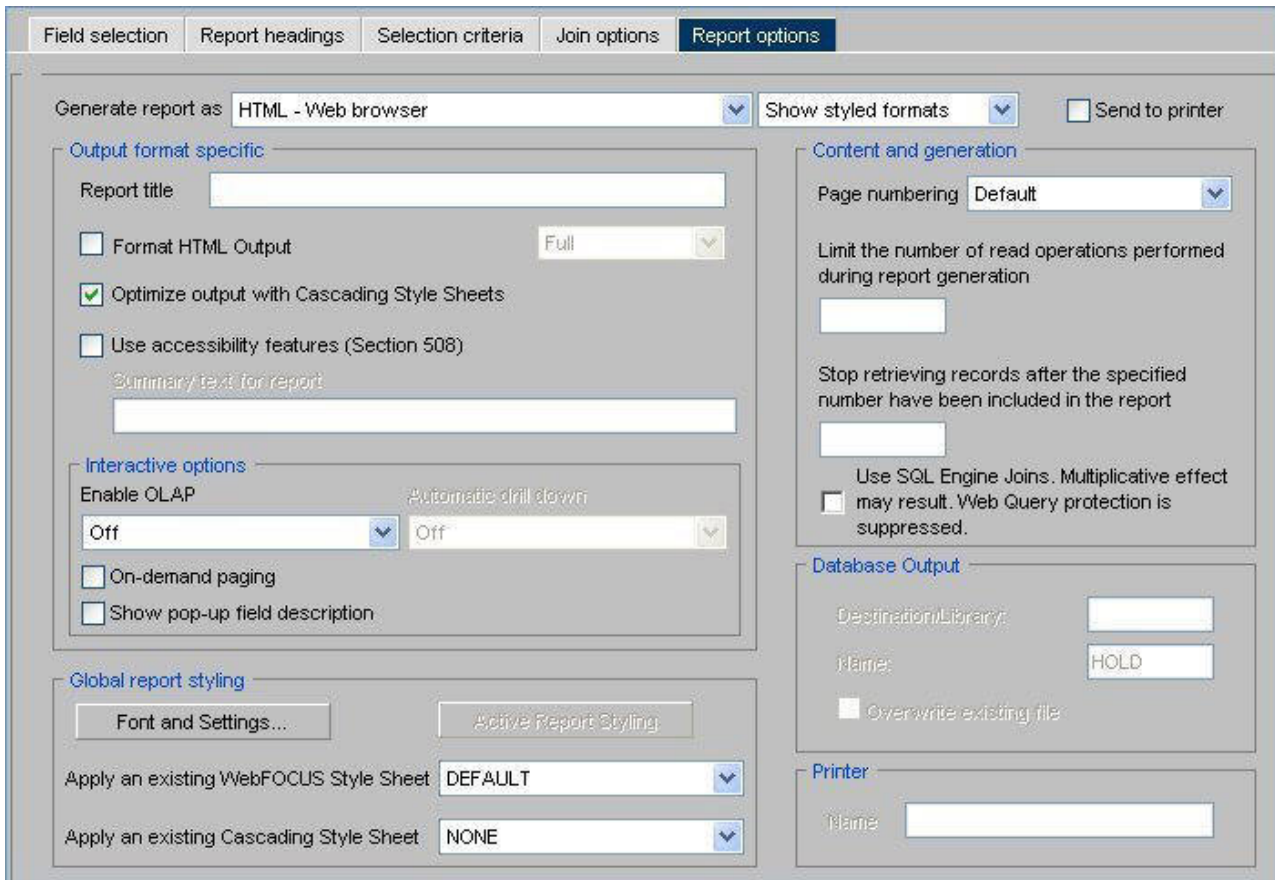
The Global report styling section includes the Apply an existing DB2 Web Query Style Sheet and Apply an existing Cascading Style Sheet list boxes. To apply a StyleSheet to a report, select a StyleSheet from the appropriate list box. The list box for DB2 Web Query StyleSheets includes all StyleSheet files stored in the domain.

When a predefined StyleSheet is applied to a report, Report Assistant disables all customizable styling options and does not apply any previous styling selections for the entire report. These settings are not lost. If you decide not to apply a predefined StyleSheet, Report Assistant restores the report's original styling settings and enables you to further customize a report's styling.

Note: When applying a StyleSheet, if you select the Format HTML Output check box, use only the Full or Paged setting. The Fixed setting is not supported with StyleSheets.

Customizing Headings and Footings

Use the Report headings tab to specify the content, style, and placement of headings and footings in your report. Headings and footings can be customized only if a StyleSheet has not been applied. Since Report Assistant applies a StyleSheet by default, you must set the StyleSheet in the Report Options tab to None.

The following image shows the Report headings tab opened with a formatting tool bar, the Page heading and the Page footing text boxes.
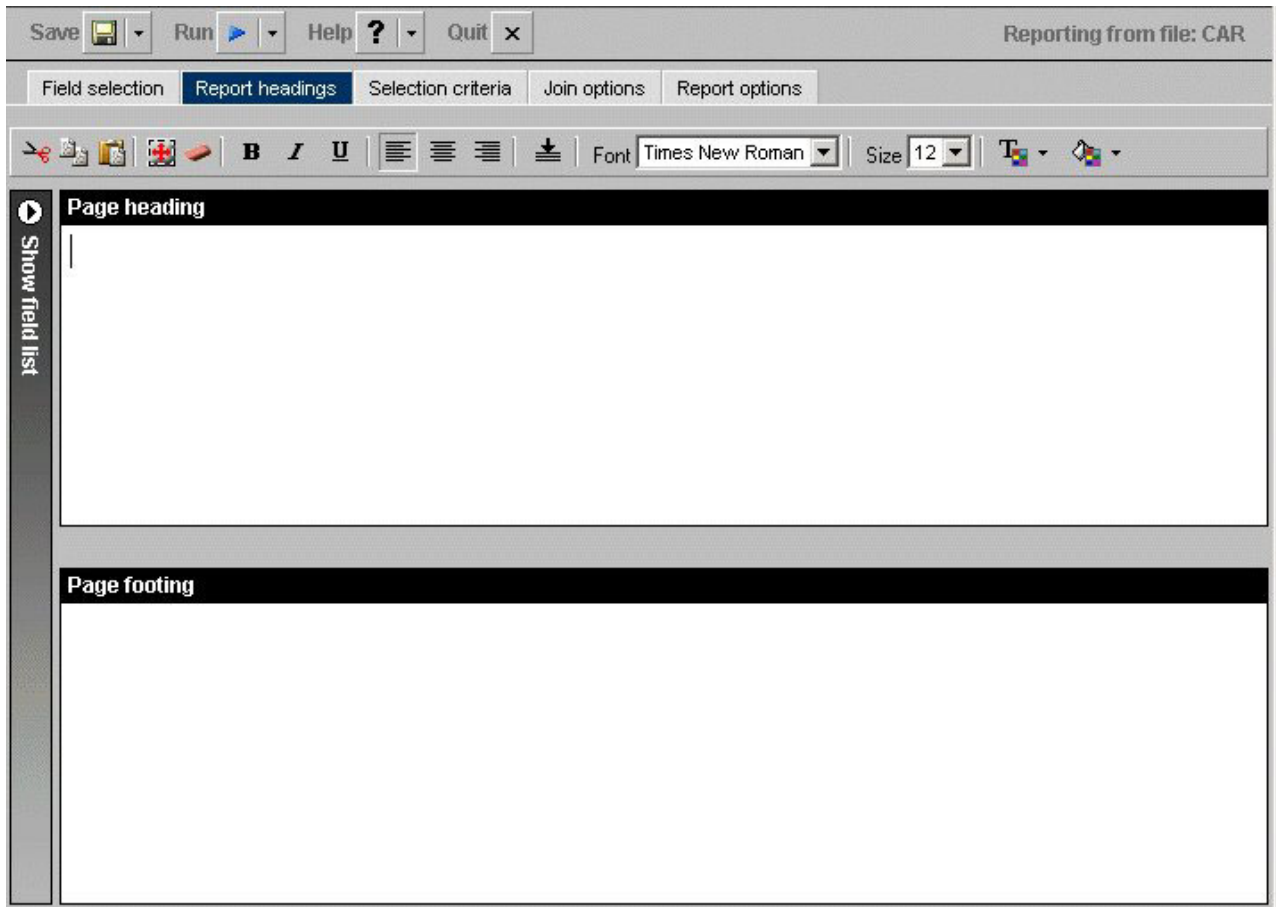
Note: Dragging and dropping items, including images, is not supported in the Report Assistant.

Add and Align a Heading in a Report

1. In the Report headings tab, click in the text box or press **Alt+H** to enter a page heading title.
2. Click the **Center** icon to center the heading in the report. The heading is left-aligned by default.

Add and Align a Footing in a Report

1. In the Report headings tab, click in the text box or press **Alt+F** to enter a page footing title.
2. Click the **Center** icon to center the footing on the report. The footing is left-aligned by default.
3. Click the **Bottom** icon to place the footing at the bottom of the report.

Include a Field in a Heading or Footing

You can include a specified field in a heading or footing. When you run the report, the data value for the field will be included in the heading or footing.

1. Click the **Show fields list** icon to the left of the Report headings box to display the Available fields window.

2. Select a field from the Available fields window and drag the field to the Page heading or Page footing box.

or

Click and hold the left mouse button, and drag the field to the fields box you chose.

or

Use the shortcut keys to quickly go to one of the field boxes.

In the following table, the first column (Field Box) describes where you go when you use the shortcut key combination described in the second column (Shortcut Keys).

Field Box	Shortcut Keys
Page Heading	Alt+H
Page Footing	Alt+F

Press **Alt+M** to go to the Available Fields window, tab to the field and press **Enter** to add the chosen field in the page heading or footing text box.

Note:
- You can add more than one field name to either the Page heading or Page footing box by selecting one field at a time.
- The report's heading or footing will contain the first value retrieved for the field in the heading or footing.

Style a Heading or Footing

To style a heading or footing:
1. Select the field you want to style in the Page heading or the Page footing text box in the Report headings tab.
2. Click the **Cut**, **Copy**, **Paste**, **Select all**, and **Remove formatting** icons to format the text of your choice.
3. Click the appropriate icons to bold, italicize, or underline your text.
4. Click the icons for left, center, or right alignment to specify justification.
5. Click the **Bottom** icon to move your text to the bottom of the report.
6. Click the Font list box or press **Alt+N** to specify your selections.

   Note: When selecting a font, be sure to click in the text box before you select the font name, otherwise the font may return to the original font selection.
7. Click the Font Size list box or press **Alt+S** to specify your selections.
8. Click the **Text Color** icon or press **Alt+C** and **Enter** to open the Color palette window. Choose a color from the color list.
9. Click the **Background Color** icon or press **Alt+B** and **Enter** to open the Color palette window. Choose a color from the color list.
10. Click **Save** or **Save As** to save your settings.

Note: The default styling settings are as follows:
- Font: Times New Roman
- Size: 13 (If you change the font size of your heading, which is a FOCUS title, the code may not properly reflect the font size you specified when you open the procedure in a text editor. This does not affect functionality. You can edit the procedure manually to correct the size.)

- Style: Normal
- Text Color: Black
- Background Color: Transparent
- Justify: Left

Selecting Records for a Report

When generating a report and specifying which fields to display, you may not want to show every instance of a field. By including selection criteria, you can display only those field values that meet your needs. In effect, you can select a subset of the data, which you can easily redefine each time you issue the report request.

When developing a report request, you can define criteria to select records based on:

- The values of an individual field (using a Where statement).
- The aggregate value of a field, for example, the sum or average of a field's values (using a WHERE TOTAL statement).

The Selection criteria tab enables you to create Where or Where Total statements for record selection.

Creating a Where or Where Total Statement

Where and Where Total statements enable you to display only those records that pass your selection criteria. These statements select the data source records to be included in a report. The data is evaluated according to the selection criteria before it is retrieved from the data source. Use the Selection criteria tab to create a new Where or Where Total statement.

Note: In the Report Assistant, when you create a Where statement in the Selection criteria tab for a range of dates using FROM - TO, you will not be alerted if the TO field is chronologically before the FROM field. For example, if you enter something like "from 12/31/05 to 1/1/05", there will be no records in the resulting report.

Select Records Based on Values, Parameters, Fields, or a Constant

1. In the Report Assistant, click the **Selection criteria** tab.
2. Select a field from the Available fields window and drag the field to the **Screening conditions** pane.

   or

   Press **Alt+M** to go to the Available Fields window, tab to the field and press **Enter** to add the chosen field.

   The field name and EQUAL to and Select values appear in the Screening conditions pane.
3. Select **WHERE** or **WHERE TOTAL**.
4. Select an operator from the drop-down list (for example, **EQUAL to**) in the right pane.
5. Click **Select values** to complete the expression.
6. Select one of the following compare type option buttons:
   - **Constant.** Enter a value that will serve as a constant (a value that will not change). For example, enter vehicle.

After you click **OK**, the constant, vehicle, appears in the WHERE statement which reads BODYTYPE EQUAL to vehicle.

- **Parameter.** Enter the name of the variable. For example, moped as shown in the following image with the Parameter option button selected and the variable name entered as moped.

You can set parameter properties, for details see Setting Parameter Properties.

After you click **OK**, the parameter, &moped, appears in the WHERE statement which reads BODYTYPE EQUAL to &moped.

Note: The ampersand is automatically added by DB2 Web Query.

- **Fields.** Select a field from the list.

After you click **OK**, the field BHP appears in the Where statement which reads CAR.COMP.CAR EQUAL to CAR.SPECS.BHP.

- **Values.** Select a value for the field you initially chose (in this case, car).

After you click **OK**, the value, Audi, appears in the Where statement which reads CAR.COMP.CAR EQUAL to AUDI.

Create a Where Statement Using Date and Time Values

In the Report Assistant,

1. Click the **Selection criteria** tab.
2. Select a field from the Available fields window and drag the field to the **Screening conditions** pane.

   or

   Press **Alt+M** to go to the Available Fields window, tab to the field and press **Enter** to add the chosen field.
3. Click **EQUAL to** in the right pane to display a list of operators. Click the operator of your choice.
4. Click **Select values** to complete the expression.
5. Click **Field** to access date and time values for that field. For example, EYEAR.
6. Click **Values** and click the value of your choice. For example, 1989/01/18.

   After you click **OK**, the value, 1989/01/18, appears in the WHERE statement which reads START_DATE EQUAL to 19890118.

Importing Values From External Files for WHERE Statements

When developing a query, you can use a locally saved, external file as selection criteria to limit query results. This enables you to quickly build a query containing a large number of WHERE statement values without having to manually enter repetitive or readily available data.

Create a Value List From an External File

1. Required: Navigate to the **Selection criteria** area.
2. Required: Begin creating a WHERE statement by selecting a field from the list of available fields.
3. Required: Select a data comparison option that can accept multiple values.

   Valid options include EQUAL to, NOT EQUAL to, IN literal list, NOT IN literal list, EXCLUDES literal list, and INCLUDES literal list.
4. Click **Select values**.

   The Values dialog opens.
5. Click the **import** button as shown in the following image.

A dialog box opens as shown in the following image.

6. Select either the Flat file or Excel Spreadsheet (XLS) File Format option.
7. Click the **Browse** button.

   A standard file selection dialog opens.
8. Select an external file to import from your local machine or network.

   Note: The external file must only contain text with new line delimiters.
9. Click **OK**.

   The values contained in the file are displayed in the Multiple values entered list in the right pane of the dialog.
10. Optionally, you can remove specific values from the Multiple values entered list or move values up or down within the list.
11. Click **OK**.

   You can save the query for future use and reopen the query to append new or remove existing values.

Setting Parameter Properties

When you create parameters for Selection Criteria (WHERE statements) you can set the properties for the parameter. For example, you can create a dynamic or static list of values to choose from at run time.

Set Parameter Properties

1. In the Report Assistant, click the **Selection criteria** tab.
2. Create a WHERE or WHERE TOTAL expression.
3. Select an operator. Some operators are not supported with the Variable Editor, see Supported Operators for the Variable Editor for details.
4. Click **Select values**. The Variable Editor dialog box opens
5. Select the **Parameter** option.
6. Click the **Auto Prompt** button.

   The Parameter Properties dialog box opens.
7. Verify the Name and Description fields, which are automatically populated based on the field you selected for the WHERE clause. You can edit the populated values as needed. Note that the Description field cannot contain a period (.) because a period is the delimiter to specify the beginning and end of the description value.
8. Choose the Selection type, **Dynamic** or **Static**.

   The Dynamic option is selected by default.

   Optionally, choose the **Select multiple values at runtime** check box to provide multiple values to the report from the auto prompt page at run time.
9. Set the parameter values.
   - For the Dynamic selection type, the Data Source and Field values are selected by default based on the field you are screening on and can be changed as needed.

     If you want to see the fully qualified Field name, select the **Display fully qualified field name** check box. This setting is for display purposes only and is not preserved.
   - For the Static selection type, select the **Constant** or **Show field values** option. For Constant, enter one or more values. For Show field values, all of the values for the selected field appear in the list box. Move the values you want from the selection list to the Prompt values list.
10. Click **OK** to exit the Parameter Properties dialog box.

Supported Operators for the Variable Editor

Operator	Enabled for the Variable editor?
Equal to	Yes
Not equal to	Yes
Greater than	Yes
Less than	Yes
Greater than or equal to	Yes
Less than or equal to	Yes
In literal list	No
Not in literal list	No
Missing	No

Operator	Enabled for the Variable editor?
Not missing	No
From - to	No
Not-from - to	No
Includes literal list	Yes
Excludes literal list	Yes
Contains characters	Yes
Omits characters	Yes
Like character mask	Yes
Not like character mask	Yes

Combining Expressions

Use the right pane of the Selection criteria tab to use or delete existing Where statements and to combine expressions.

Combine Expressions

1. Create an expression.
2. When you create additional expressions, they are combined with the default operator AND, as shown in the following image with two statements, the first a WHERE and the second an AND statement.
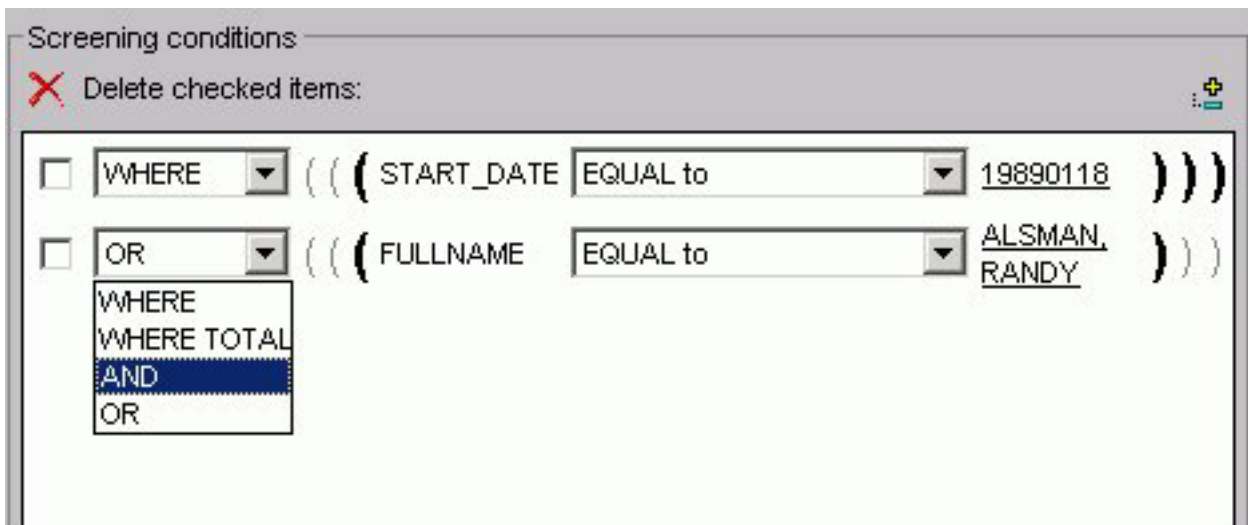
3. Use the drop-down list to select from AND, OR, WHERE, and WHERE TOTAL.

Delete an Active Expression

1. Select the check box next to the expression you want to delete. Click the **Delete** icon above the box.
2. You are prompted to delete the expression. Click **OK** to delete.

Grouping Expressions Together With Parentheses

You can use parentheses to group expressions together to optimize the Where statement.

Group Where Statements Together Using Parentheses

Click the grayed out parentheses to activate the parentheses and group expressions together.

Limiting Data With Filters

Filters enable you to quickly select predefined criteria that limit data included in a report. Filters are selection criteria (Where statements) that an Administrator creates for you to apply as needed, without having to create your own selection criteria.

DB2 Web Query displays filters in filter groups. Each filter group can contain multiple filters. Selecting a single filter within one group creates a report with simple filtering criteria. By selecting multiple filters within a group or combining filters from different groups, you can create complex filtering expressions.

The following image shows a few sample filters as they appear in the Selection criteria tab of the HTML Report Assistant. They are listed by Group and Name.

Simple Filtering Criteria

Simple filtering criteria consists of one or more filters from the same filter group. If you select only one filter, the data must match that filter to be included in the report. If you select multiple filters from the same filter group, the data must match only one filter to be included in the report. This type of criteria is an OR criterion.

Complex Filtering Criteria

Complex filtering criteria consists of one or more filters from multiple filter groups. Data must match one filter from each filter group to be included in the report. This type of criteria is an AND criterion.

Selecting Records With LIKE and NOT LIKE Operators

Alphanumeric fields include the following operators:

- **LIKE the character mask.** This means that the value for the field you selected will be included in the report. All other values for that field will not be included

in the report. For example, if you select the Plant field and specify Boston with LIKE the character mask, only values for Boston will appear in the report. Values for plants in all other locations will not be included.

- **NOT LIKE the character mask.** This means that all values except the one you specify for the field you selected will be included in the report. For example, if you specify Boston with the NOT LIKE character mask, data for Dallas, Los Angeles, New Orleans, Seattle, and St. Louis will display and no data will display for Boston.

These operators generate the appropriate LIKE and NOT LIKE expressions using the character mask available in the expressions list when creating a WHERE statement.

Working With Joins

A join is a temporary connection between two or more data sources that share at least one common field. Once you join two data sources, each time DB2 Web Query retrieves a record from the first data source (host file), it also retrieves the matching records from the second data source (target file).

You use the Join tool to link data sources in both Report Assistant and Graph Assistant. The Join tool provides a graphical method for creating and manipulating joins. You can also create define-based joins and multi-field joins.

When you access the Join tool, the window displays a field list for the host data source (which you specified when you created the report), and a field list for any target data source you add to the window.

Important:

- Only Administrators and Developers have access to the Join options tab.

Create a Join

1. Click the **Join options** tab in Report Assistant or Graph Assistant and click **New**.

   The DB2 Web Query masters list dialog box opens where you select a target data source.

2. Select a data source and click **OK**.

   The following image shows the Join settings window where you select fields from the Host field and the Target field lists to create a join.

3. Select the fields you want to join from the Host field list and the Target field list and add them to their respective Selected Fields boxes.

   or

   Press **Alt+M** to go to the Host field list.

   or

   Press **Alt+T** to go to the Target field list.

   Then, tab to the field and press **Enter** to add the chosen field to the corresponding Selected Fields box.

   Note: Fields must have the same format and be indexed fields in order to use them to create joins.

4. Select your join properties:
   - **Unspecified.** Indicates neither an inner nor left outer join.
   - **Inner Join.** A join that results when a report includes host rows that have corresponding cross-referenced rows.
   - **Left Outer Join.** Extends the results of an Inner Join and retrieves records from both host and cross-referenced tables, including all records from the left table (host) and any records from the right table (cross-referenced) where the condition values match. If there are no matching values in the cross-referenced table, the join still retrieves records from the host table.
   - **Multiple Instance (formerly Non Unique).** A one-to-many join structure that matches one value in the host data source to multiple values in the cross-referenced field. Joining employee ID in a company's employee data source to employee ID in a data source that lists all the training classes offered by that company would result in a listing of all courses taken by each

employee, or a joining of the one instance of each ID in the host file to the multiple instances of that ID in the cross-referenced file. The default join type is inner.

- **Single Instance (formerly Unique).** A one-to-one join structure that matches one value in the host data source to one value in the cross-referenced data source. Joining an employee ID in an employee data source to an employee ID in a salary data source is an example of a unique join.

5. To see the join syntax, click **View**.

6. Click **OK**.

7. In the Description box, give the join a meaningful and unique name.

8. Click **Save & Create** and the new join appears in the Joins list.

Create a Define-based Join

1. Click the **Join options** tab in Report Assistant or Graph Assistant and click **New**.

   The DB2 Web Query masters list dialog box opens where you select a target data source.

2. Select a data source and click **OK**.

3. Click the down arrow of the Define icon and then click **New define field**.

   The following image shows the list of Define field options.

The following image shows the Define field creator dialog box where you create the Define expression.

4. Create the defined field. The new define field appears in the Available fields

   list. The field is prefixed with the define ![Plant1 symbol] symbol.
5. Add the defined field and the target field to their respective Selected Fields boxes.

   Note: Fields must have the same format and be indexed fields in order to use them to create joins.
6. If you want to see the syntax, click **View** to display the define-based join syntax.
7. Give the join a meaningful and unique name in the Description box.
8. Click **Save & Create**. The new join appears in the Joins list.

Create Multi-field Joins

1. Click the **Join options** tab in Report Assistant or Graph Assistant and click **New**.

   The DB2 Web Query masters list dialog box opens where you select a target data source.
2. Select a data source and click **OK**.
3. Enter multiple fields in the Host field list box and one field in the Target field list box.
4. If you want to see the syntax, click **View** to display the multi-field join syntax.
5. In the Description box, give the join a meaningful and unique name.
6. Click **Save & Create**.

   The new join appears in the Joins list.

Delete a Join

1. Select a join from the Select an existing join list box.
2. Click **Delete**.

Edit a Join

1. Select a join from the Select an existing join list box.
2. Make your desired changes.
3. Click **Save & Create**.

Applying Other Report Options

The Report options tab enables you to select from a variety of options for the output of your report.

The features of the Report options tab are shown in the following image.

The following is a list of the Report options tab features:

- Save a report to an output format that supports styling options.
- Save a report to an output format that supports CSS.
- Specify the page-numbering format.
- Set record/read limits.
- Use SQL engine joins for database optimization.
- Enter a report title.
- Enter summary text.
- Style the entire report.
- Style a PDF or PS report.

- Justify an entire report.
- Enable On-demand Paging and grid lines.
- Display pop-up field descriptions for column titles.
- Optimize output with internal Cascading Style Sheets (CSS).
- Run long queries with the Paged option.
- Create a DB2 file as report output.
- Use accessibility features.
- Enable OLAP features.
- Enable the Automatic Drill Down features.
- Direct output to a printer.

Save a Report to an Output Format That Supports Styling Options

1. From the Report options tab, select the **Show styled formats** option from the drop-down list.
2. The **Generate report as** list box offers a choice of file types for the report output. Select one of the following formats from this list:
   - **HTML.** Creates the report in HTML, sending the output to the Web browser, with or without grid lines.
   - **AHTML (Active Report).** Creates an HTML report that is designed for offline analysis. Users can interact with the data, using analysis options similar to those found in an Excel workbook, without any connection to a server. See Creating an HTML Active Report.
   - **PDF.** Captures the entire report and creates a Portable Document Format (.PDF) file, which can be opened in the Adobe® Acrobat Reader (Adobe's PDF viewer).
   - **EXL97.** Generates a report in Excel97 format.
   - **EXL2K.** Generates fully styled reports in the Excel 2000 HTML format. Excel 2000 provides full support for HTML files with embedded XML.
   - **EXL2KFORMULA.** Excel with Formula.
   - **PS.** Creates the report as a postscript file.

Save a Report to an Output Format That Supports CSS

1. From the Report options tab, select the **Show unstyled formats** option from the drop-down list.
2. The Generate report as list box offers a choice of file types for the report output. Select one of the following formats:
   - **EXCEL.** Excel 95.
   - **DOC.** Plain text with page breaks.
   - **WP.** Plain text without page breaks.
   - **WK1.** Import to Lotus® 1-2-3®. See Notes on Using LOTUS Format.
   - **DIF.** Import to Spreadsheet.
   - **LOTUS.** Import to Lotus 1-2-3. See Notes on Using LOTUS Format.
   - **TABT.** Tab delimited.
   - **XML.** eXtensible Markup Language.

Specify Page Numbering Format

The Page Numbering list box offers several options for page numbering. Select one of the following from the list box:

- **Default.** Numbers each page of the report in the format PAGE 1, PAGE 2, and so on. The default value is ON.
- **ON.** Numbers each page of the report in the format PAGE 1, PAGE 2, and so on.
- **OFF.** Suppresses the display of page numbers.
- **NOLEAD.** Suppresses the two leading blank lines included on each page by default.

Set a Record Limit or Read Limit

To limit the number of records retrieved when reporting from a non-FOCUS data source, enter a number greater than zero in the Read Limit box. A read limit specification is ignored when reporting from a FOCUS data source.

To limit the number of records retrieved when reporting from a FOCUS data source, enter a number greater than zero in the Record Limit box.

These options are for development only.

Note:

- A warning is issued to indicate to the user that the report they are viewing is only a subset of the actual data. This occurs when either Record limits or Read limits are checked in the Properties tab, under Warning message displayed in Heading.
- The **This report is only a subset of the data** edit box is checked by default. You can deselect the check box for this message or design a new message. Either message will appear in the heading of the report.

Set Optimization When Running Requests

You can optimize report requests which will reduce the volume of database-to-server communication and improve response time. It also enables the database to exploit its own internal optimization techniques. This function creates SQL statements that take advantage of join, sort, and aggregation capabilities.

To enable this function, select the check box labeled **Use SQL Engine Joins. Multiplicative effect may result. Web Query protection is suppressed.**

Include a Report Title

You can specify a report title in the Report Title field that will appear in the title bar of your report. This option is not available for reports generated in PDF or PS format or for reports that utilize on-demand paging.

Include Summary Text for a Report

You can specify summary text for your report in the Summary Text for Report field. This will place a description of the output into a SUMMARY object inside the

output HTML TABLE. The length limit of this summary is 500 characters. If you try to set the SUMMARY to more than 500 characters, you receive the following message:

```
Limit on length of summary is 500 characters.
```

Note: You can only specify summary text when the Use Accessibility features (Section 508) option is checked.

Style the Entire Report

To style the entire report:

1. Click **Font and Settings** in the Global report styling section on the Report options tab.

   The Styles dialog box opens where you can select report style options.
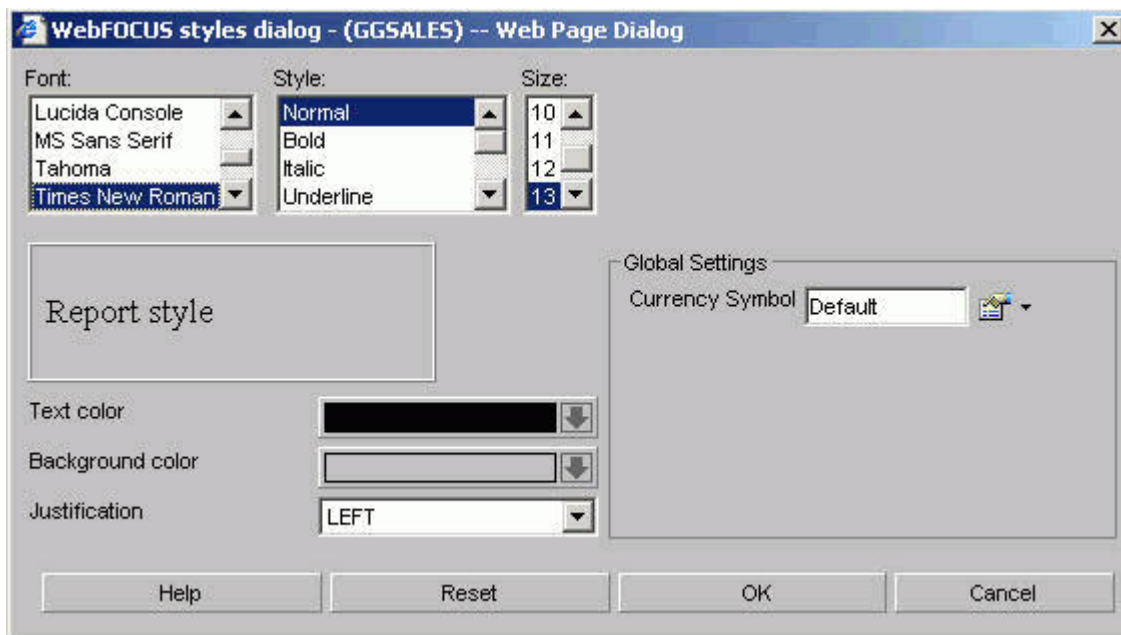
2. Select the font styling options from the Font, Style, and Size boxes.

3. Select a text color from the Text color list box.

4. Select a background color from the Background color list box.

5. Select a justification from the Justification list box.

6. Select a different currency symbol to override the currency symbol set on the reporting server. Choose from Default, USD, GBP, JPY, EUR, $, or Custom in the Currency Symbol menu. When you select Custom, replace "Custom" with the desired one-character custom currency symbol in the Currency Symbol field.

7. Click **OK**.

8. To specify an existing DB2 Web Query Style Sheet or Cascading Style Sheet, make your choice from the relevant list box.

Note: Clicking Reset returns the styling options to the following default settings:

- Font: Times New Roman
- Style: Normal

- Size: 13
- Text Color: Black
- Background Color: Transparent
- Justification: Left

Style a PDF or PostScript Report

When you select PDF or PostScript® (PS) as the display format for a report, DB2 Web Query offers several additional styling features.

1. Select **PDF** or **PS** from the Generate report as list box.
2. Click **Font and Settings**. The Styles dialog box opens.
3. Select a font.
4. Click the Page Orientation list box to select either **Portrait** or **Landscape**.
5. Click the Page size list box to select the page size for the report.
6. Select a different currency symbol to override the currency symbol set on the reporting server. Choose from Default, USD, GBP, JPY, EUR, $, or Custom in the Currency Symbol menu. When you select Custom, replace "Custom" with the desired one-character custom currency symbol in the Currency Symbol field.
7. Click **OK**.

Justify an Entire Report

The following procedure describes how to justify an entire report. This affects headers, footers, fields, and field headings.

1. Click **Font and Settings**. The Styles dialog box opens.
2. In the Justification list box, select **Left**, **Right**, or **Center**.
3. Click **OK**.

Enable On-Demand Paging and Grid Lines

1. Select **HTML - Web browser** or **HTML - Web browser (with grid)** in the Generate report as list box.

   Note: The latter option provides grid lines that separate cells of data in your report.

   You can only use On-demand Paging for reports that use the HTML format.
2. Click the **On-demand Paging** check box.
3. Save and run the report.

   DB2 Web Query displays the first page of the report in the DB2 Web Query Viewer.

Optimize Output With Cascading Style Sheets

When you check Optimize output with Cascading Style Sheets in the Report options tab of Report Assistant, your report will reflect the styling options you select, and generate Internal Cascading Style Sheet code in the HEAD tag of HTML reports. This feature is available only if you select HTML in the Generate report as drop-down list.

Run Long Queries With the Paged Option

To display the first few pages of results prior to the completion of a long-running query, use the Paged option for HTML output. From the Report options tab, select the Format HTML Output check box, then select Paged from the drop-down list.

Note: Output for summary reports will be finalized when the query is complete.

Create a DB2 File as Report Output

To create a DB2 file as report output, complete the following steps.

1. From the Report options tab, select the Show database formats option from the format drop-down list.
2. Accept the default value in the Generate report as list box, which is DB2- DB2 database table.
3. In the Database Output section, supply values for the following options:
   - Destination Library - name of the library destination for the DB2 report file.
   - Name - name of the DB2 file.
4. If you need to overwrite an existing file, select the Overwrite existing file check box.

Note: When specifying to overwrite an existing database file, the file will not be overwritten if the user does not have specific write authority to the file. A message will be displayed which says that the file already exists, even though overwrite was specified.

The following image illustrates the options for the DB2 database format.

Use the Accessibility Feature

If you check this box, the output will be 508 compliant. See Include Summary Text for a Report.

Setting OLAP Reporting Options

OLAP options are available in the Interactive options section of the Report options tab in Report Assistant only when the OLAP license option is activated. The relevant options—**Enable OLAP** and **Automatic Drill Down**—are located here.

Enable OLAP Options

The Enable OLAP options in the Report Assistant control how users can interact with an OLAP report and access OLAP tools.

Administrators and Developers who are creating reports can OLAP-enable them and control the following OLAP interfaces and drill down options.

- **OFF.** OLAP is off when off is selected, OLAP options are not available for the report.
- **Columns only.** Turns off the OLAP Control Panel and the OLAP Selections pane, but enables OLAP functionality from the report itself. You can access options on right-click menus, drag and drop columns within the report, and use up and down arrows to sort columns from high to low or vice versa. This is the default.
- **Columns with panel.** Provides access to the OLAP Selections pane from a square button to the left of the column titles.
- **Show filters on top.** Opens the OLAP Selections pane above the report. The Measures, Graph, and Dimension controls, as well as the band containing the OLAP, Run, and Reset buttons appear above the report output. You can open the Control Panel by clicking the OLAP button on the Selection pane.
- **Show filters on bottom.** Opens the OLAP Selections pane below the report. The Measures, Graph, and Dimension controls, as well as the band containing the OLAP, Run, and Reset buttons appear below the report output. You can open the Control Panel by clicking the OLAP button on the Selection pane.
- **Show Panel in Report.** Opens the OLAP report with the OLAP Selections pane hidden. You can perform a variety of analytic tasks from the report itself. Selection Criteria is shown next to the OLAP button.
- **Show Tabbed.** For OLAP reports that have multiple dimensions, this option groups the dimension elements under a tab labeled with the dimension name.

Automatic Drill Down Options

These options enable you to sort instantly from high to low or low to high for selected report columns:
- **Off.** Disables automatic drill downs.
- **Dimensions only.** Enables automatic drill downs on dimensions in reports and graphs.
- **Measures and Dimensions.** Enables automatic drill downs on dimensions in both reports and graphs and on measures in reports.

Direct Output to a Printer

To create a direct output to a printer, complete the following steps.
1. From the Report Options tab, select the **Send to printer** check box, which activates the Printer Name field.
2. Type the name of the network printer you wish to print to in the Printer Name field, for example, CAN28B1, as shown in the following image.

Displaying Pop-up Field Descriptions for Column Titles

You can have pop-up field descriptions display in an HTML report when the mouse pointer is positioned over column titles. Field description text displays in a pop-up box near the column title using the default font for the report. Pop-up text appears for report column titles including titles created with ACROSS phrases and stacked column titles created with OVER phrases.

The pop-up text displayed for a column title is defined by the Description attribute in the Master File for the corresponding field. If a column title has no Description entry in the Master File, then no pop-up box is generated when your mouse is positioned over the title.

This feature is not supported in the HTML Graph Assistant.

Display Pop-up Field Descriptions in an HTML Report

1. Open the HTML Report Assistant.
2. In the HTML Report Assistant window, select the **Report options** tab.
3. In the Interactive Options area, select the **Show pop-up field description** check box.
4. Click **Save** to save this option in your report.

Creating an HTML Active Report

Note: Active Report functionality is available only when the Active Reports license option is activated.

An HTML Active Report is a report that is designed for offline analysis.

When using an HTML Active report you can:

- Interact with the data, using analysis options similar to those found in an Excel workbook, without any connection to a server.
- Work offline without any additional plug-ins or programs. HTML Active Reports are self-contained reports, meaning they contain all of the data and JavaScript™ within the HTML output file. Packaging the data and the interactive functions in the HTML file also makes the output highly compressible for e-mail and transparent to security systems.
- Save the report on a local machine with the packaged interactivity. Since no connection to a server is required to view the data or use the analysis options, you can save and use the report anywhere.

The following image shows a simple HTML Active Report. The record status and page navigation bar appears at the top of the report.

Create an HTML Active Report With HTML Report Assistant

1. In the HTML Report Assistant, click the **Report options** tab.
2. Ensure that the **Show styled formats** option is selected from the drop-down list.
3. From the Generate report as drop-down list, select **AHTML - Active Report**.
4. Click the **Active Report Styling** button.

   The Active Report Styling dialog box opens.

5. Select styling options for the Active Report.

   For details, see Formatting an HTML Active Report.

Formatting an HTML Active Report

You can apply the following custom formatting to an HTML Active Report:

- **Report view.** You can set the report view as a standard table or as an expandable report. An expandable report is a collapsed report that can be fully expanded for sorting and other purposes.
- **Pagination.** You can set the justification for the information that appears in the record status and page navigation bar and set the number of records that appear per page. Pagination options are not available for expandable report views.
- **Calculations.** You can set certain calculations for fields in the report. You set calculation results to show in a row that appears either above or below the report.
- **Color.** You can set the color for data visualization graphs, the text and background for the record status and page navigation bar, selected rows, the value and background for calculations, and the text, background, and border for the menu.
- **Freeze columns.** Enables you to freeze the report at a particular point so that columns to the left of the freeze point remain in view while the user scrolls through the other report columns.

Active Report Styling Dialog Box
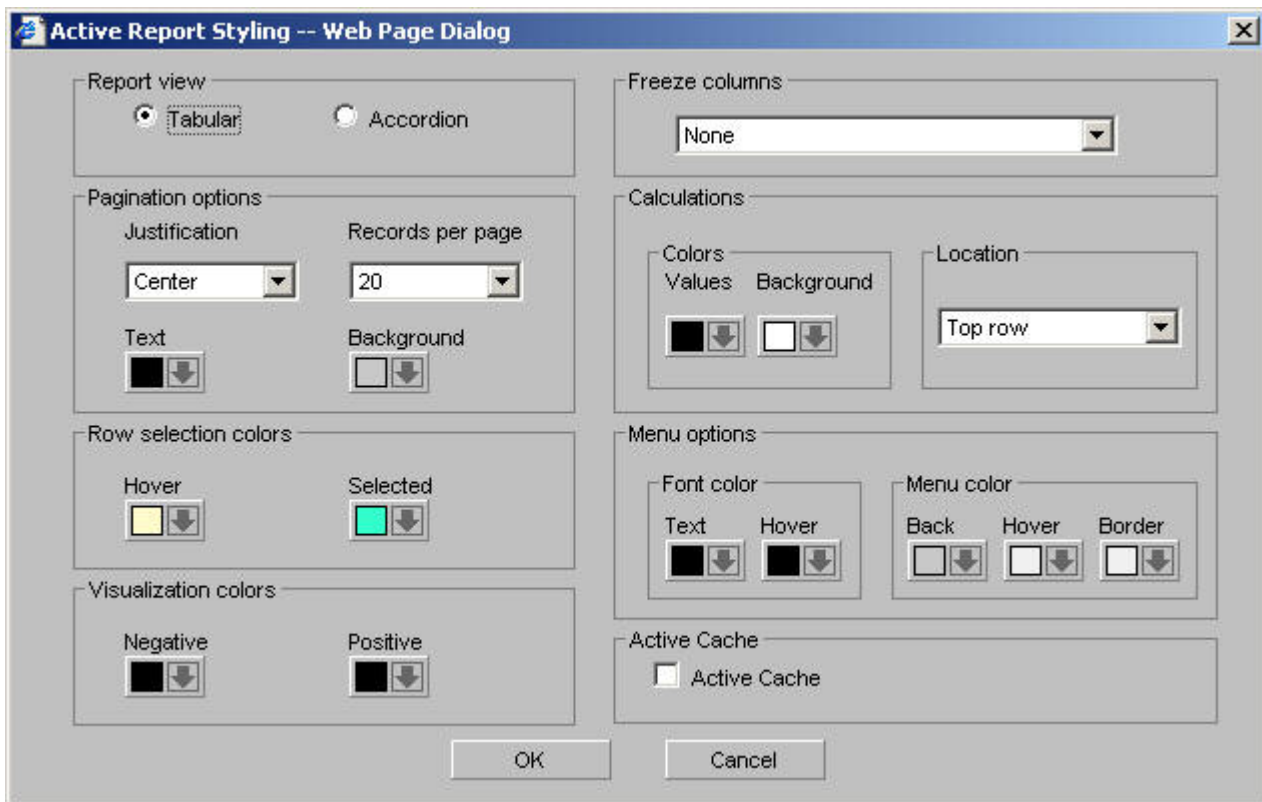
The Active Report styling options are:

Report view

Sets the report view as a standard table (Tabular) or as an Accordion report. Users can fully expand accordion reports for sorting and other purposes.

Pagination options

Following are the pagination options for HTML Active Reports. Pagination options are not available for Accordion report views.

Justification

Justifies the text that appears in the record status and page navigation bar. Options are Left, Center, and Right. Left is the default.

Records per page

Sets the number of records that appear per page. Options are: Default, 10, 20, 30, 40, 50, Show All. The default is 57 lines per page.

Text

Sets the color for the text in the record status and page navigation bar. The default is black.

Background

Sets the background color for the record status and page navigation bar. The default color is silver.

Row selection colors

An HTML Active Report offers visual assistance for viewing data in the report. For example, when a user hovers over a row of data the row is highlighted with a background color and when they click a row of data it is also highlighted with a background color. You can set the colors for these options, Hover and Selected. The default color for Hover is RGB(255 252 204), which is a shade of yellow, and the default color for Selected is RGB(51 255 204), which is a mix of green and blue.

Visualization colors

Users can apply data visualization to numeric fields in an HTML Active Report. Here you can designate the colors of the bars. You can set different colors for positive and negative values. The default color for positive and negative values is black.

Freeze columns

Enables you to freeze the report at a particular point so that columns to the left of the freeze point remain in view while the user scrolls through the other report columns. Options include:

- None turns off the freeze option. This is the default.
- The fields in the request. If the report can be viewed fully viewed in the browser window, then freeze is not applied.

Calculations

Set options for calculations:

Colors

Specifies the colors for the values in a calculation:

Values sets the font color of the calculation results. This color defaults to black.

Background sets the background color of the calculation results. This color defaults white.

Location

Sets the location of the calculations. Options are Top row or Bottom row. Top row is the default.

Menu options

Set options for the menu.

Font color

Enables you to set the colors for the menu text:

Text sets the color of the text

The default is black.

Hover sets the color of the text when the mouse hovers over it

The default color is black (text hover does not show).

Menu color

Set the colors for the menu:

Background sets the background color of the menu. The default color is silver.

Hover sets the background hover color of an individual item on the menu. For example, if the background color of the menu is black, the hover color can be set to white to visually show which option on the menu the mouse is currently hovering over. The default color is white.

Border sets the color of the menu border. The default color is white.

Active Cache

Select to enable the Active Cache option.

Because all post-retrieval processing is performed in the Web browser's memory, an Active Report has a processing limit of approximately 5,000 records or 100 pages of output. To create answer sets in excess of 5,000 records, the Active Cache option enables you to send only the first page of Active Report output to the browser and retrieve subsequent pages from a temporary cache on the WebFOCUS Reporting Server. The server also becomes the resource for performing all calculations, sorting, and filtering when Active Cache is enabled.

Running a Report

You may periodically run a report as you create it as well as when you have finished creating it.

Run a Report

1. Click the **Run** icon in the Report Assistant.

   If you click Run, the report appears in a separate browser session.

2. To return to the Report Assistant, close the browser window displaying the report.

Note: You can continue to make changes to your existing report procedure or create another report by clicking **Save As**.

Saving a Report

When you click the Save or Save As buttons you save your report (procedure). If you make changes in your report without subsequently saving them and then click Quit, you are prompted with the following options:

- **Yes.** Click to save your changes and exit the report.
- **No.** Click to discard your changes and exit the report.
- **Cancel.** Click to return to your report to continue working or use Save or Save As to save your changes and then click Quit to exit your report.

Editing a Report

When you create a report using Report Assistant and you add a new comment as the first line of the report code, the report no longer opens in Report Assistant. Instead, the report opens in the Editor.

When there is an error in a procedure that HTML Report Assistant cannot identify, the code in the procedure appears in a window. At the top of the window, a message appears that describes what line of code the error is occurring on. You must reopen the procedure in the editor and correct the error before the procedure can be reopened in HTML Report Assistant.

HTML Report Assistant Tab References

The topics in this section provide reference information for all of the fields in the various tabs in the HTML Report Assistant.

- Field Selection Tab.
- Report Headings Tab.
- Selection Criteria Tab.
- Join Options Tab.
- Report Options Tab (IBM).

Field Selection Tab

The Field selection tab is used to select fields for a report.

Field Selection Tab: Available Fields Window

The following image shows the Available fields window on the Field selection tab. The fields are sorted by the Name column with corresponding columns containing their Alias, Format, and Segment information.
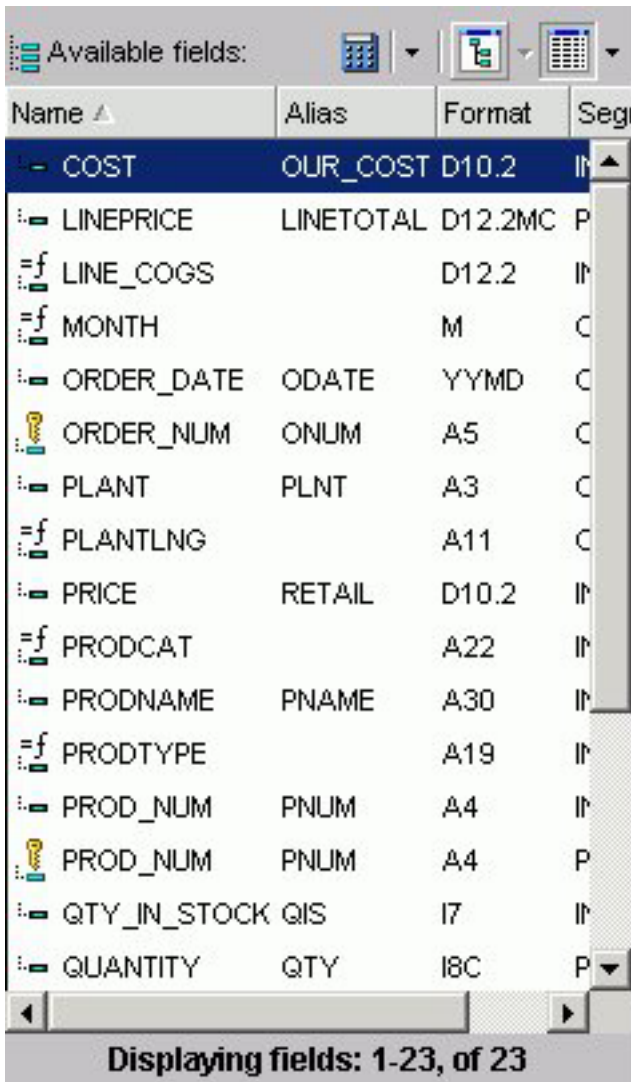
Available Fields Window

Displays a list of field names from the selected data source.

Define icon

Click the Define (calculator) icon to select: New define field, Edit define field, or Delete define field. If you select the New define field or Edit define field, the Define field creator dialog box opens where you can create or edit a defined field.

Note: The New define field is not for use with queries originated in IBM Query/400.

Tree icon

Displays a tree structure of field names by segment with segment, alias, title, description, and format information below the tree. Individual field information appears when you click on the field.

List icon

Displays a list of all field names with information that you select from the adjacent arrow that opens a list box: name, alias, title, format, description, segment, and file name. You can also choose to sort by file order.

Field Selection Tab: Report Fields Window

The following image shows the Report fields section on the Field selection tab. You can view the selected fields with the following options:
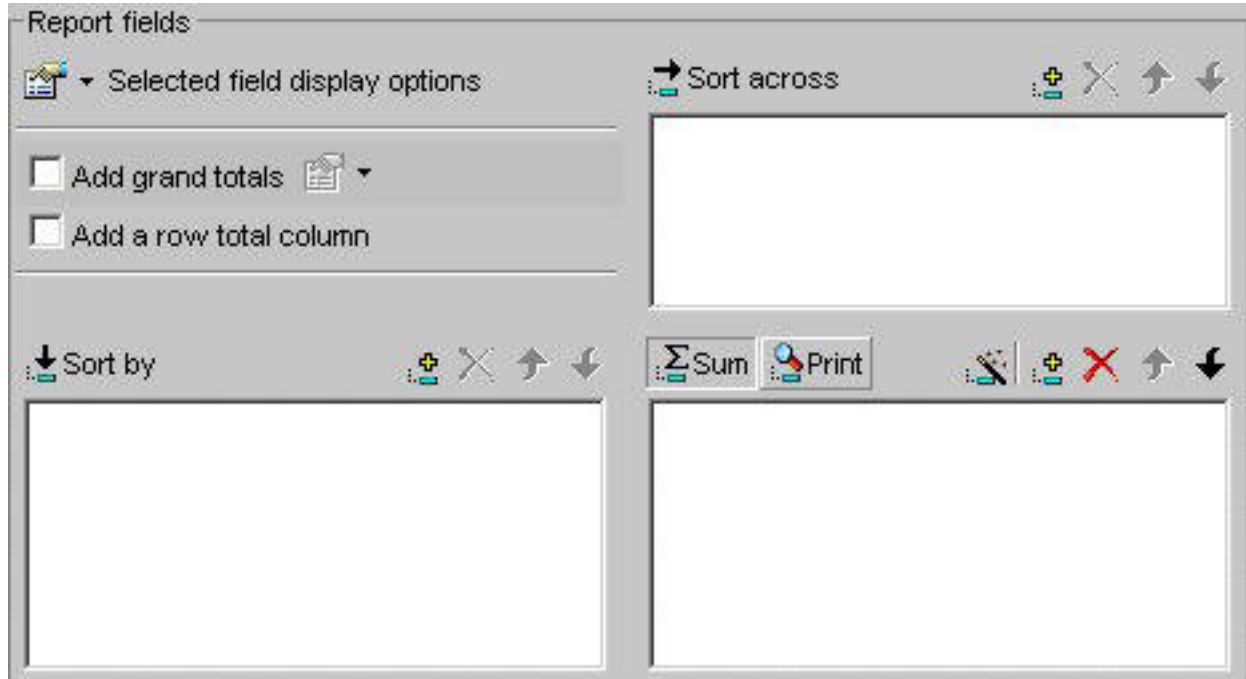
Selected field display options

Lists the following field display options:

Prefix with the Segment Name

Prefixes the field name with the segment name.

Prefix with the Filename

Prefixes the field name with the name of the data source.

Show the field's alias instead of name

Uses the alias defined in the Master File as the field name.

Show field's title

Uses the title in the Master File instead of the field name.

Show field's description

Uses the description in the Master File instead of the field name.

Add grand totals

The Add grand totals drop-down list provides two options. The Totals option displays the totals for each column. The Recompute Totals option recalculates computed fields before displaying the totals for each column.

Add a row total column

Displays the total for each row.

Sort By

Lists the vertical field that you have selected from the Available Fields window.

Add icon

Adds a field to the Sort By window.

Remove icon

Removes a field from the Sort By window.

Move Up or Move Down icons

Moves the position of the field within the report.

Sort Across

Lists the horizontal field you have selected from the Fields window.

Add icon

Adds a field to the Sort Across window.

Remove icon

Removes a field from the Sort Across window.

Move Up or Move Down icons

Moves the position of the field within the report.

Sum

Indicates a report that aggregates data by the horizontal field.

Print

Prints each value on a record-by-record basis.

New Computed Field icon

Opens the Computed Field Creator dialog box where you can define the parameters for a new computed field. For more information on the Field Creator dialog box, see Field Selection Tab: Field Creator Dialog Box.

Add icon

Adds a field to the report fields.

Remove icon

Removes a field from the report fields.

Move Up or Move Down icons

Moves the position of the field within the report.

Field Selection Tab: Field List Searching

The following image shows the Field List Searching options that you can use to search for a field in the Fields List. For more details, see Searching Field Lists.

Pattern Search

Select the Master File attribute for which you want to search. The following table lists the pattern attributes and a description of each.

Attribute	Description
Name	Name of the field.
Alias	Alias for the field.
Title	Display title for the field, as defined in the Master File.
Format	Field format, for example A4.
Description	Field description, as defined in the Master File.
Segment	A segment of fields in the Master File.
Filename	Name of the data source.

Text Box

Type your search string. You can use an asterisk (*) as a wildcard. For example, type c* to find all fields that begin with the letter "c". The character string is not case sensitive.

Field Selection Tab: Show Report Field Options for Sort By

The following image shows the Show report field options where you choose various styles and functions for a selected field.

The Display subtab for a Sort by field has these options:

Make this field invisible

Check to hide the display of a Sort By field.

Font button

Opens the font styling dialog box. See Font Dialog Box.

Conditional Styling button

Opens the conditional styling dialog box where you use to add conditional styling in a report.

Drill Down button

Opens the drill-down dialog box. See Drill Down Dialog Box.

Format in report as

Is grayed out and unavailable.

Calculated as

Is grayed out and unavailable.

Include missing instances

Inserts '.' when no field value exists.

The Title subtab for a Sort by field has this option:

Title

Is the name of the field in the Sort By list box.

The Sorting subtab shown in the following image for a Sort by field has these options:
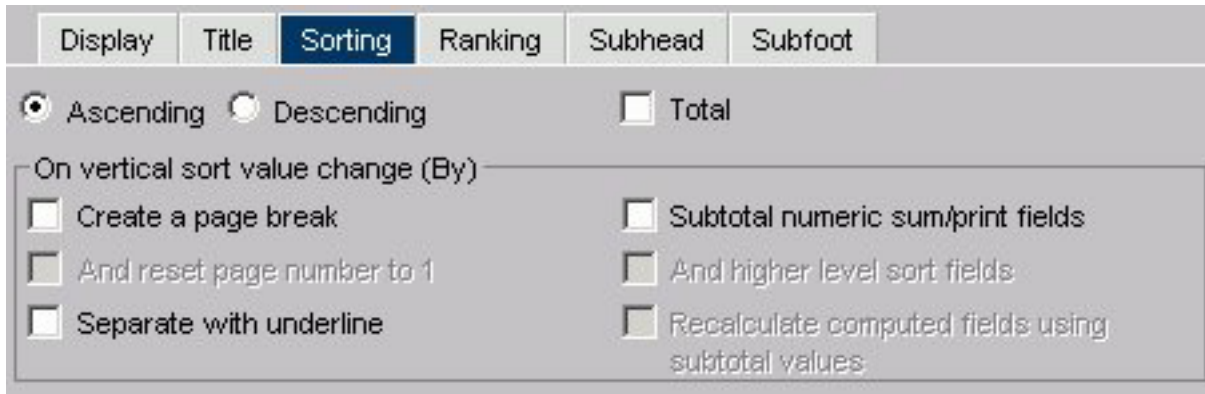
Ascending

>   Click to arrange fields in ascending order.

Descending

>   Click to arrange fields in descending order.

Total

>   Applies aggregation and sorting simultaneously to numeric columns in your report in one pass of the data. See Aggregating and Sorting Fields.

On vertical sort value change (By)

>   Select:
>
>   - Create a page break (on value change).
>   - Separate (from next) with underline.
>   - Subtotal numeric sum/print fields.
>
>   **Note:**
>
>   - When you select the **Create a page break** check box, the **And reset page number to 1** check box is activated.
>   - When you select the **Subtotal numeric sum/print fields** check box, the Subtotal tab appears and the **And higher level sort fields** and the **Recalculate computed fields using subtotal values** check boxes are activated. For more information about subtotals, see Include Subtotals.

The Ranking subtab, shown in the following image for a Sort by field has these options:
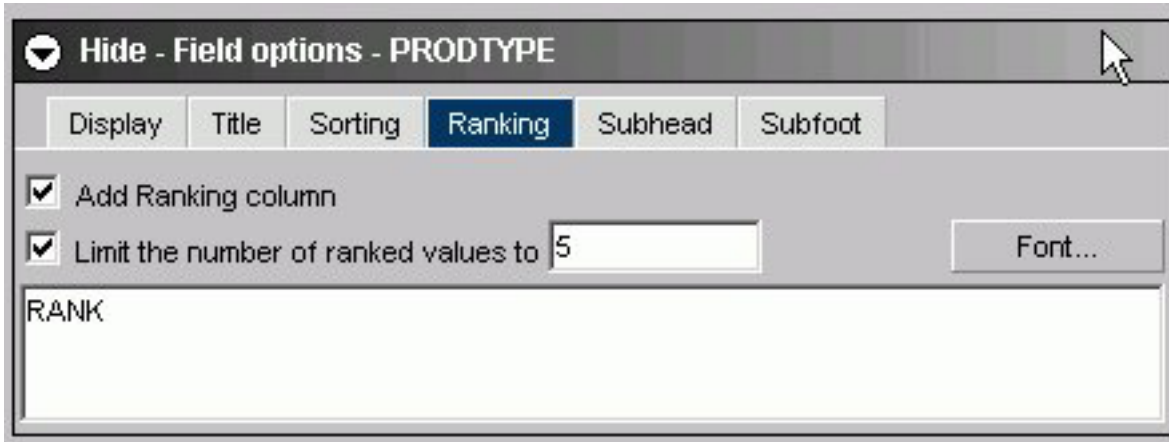
Add Ranking column

This check box is unselected by default and enables a user to add a Ranking column.

Limit the number of ranked values to

This check box is grayed out by default and activated only if the Add Ranking column check box is selected. It enables a user to enter an integer value to select the number of ranked values to display in the report (5 is the default value).

Text Box

The title of the ranking column can be changed from the default of RANK.

Font button

Opens the font styling dialog box. See Font Dialog Box.

The Subhead/Subfoot subtab, shown in the following image for a Sort by field has these options:
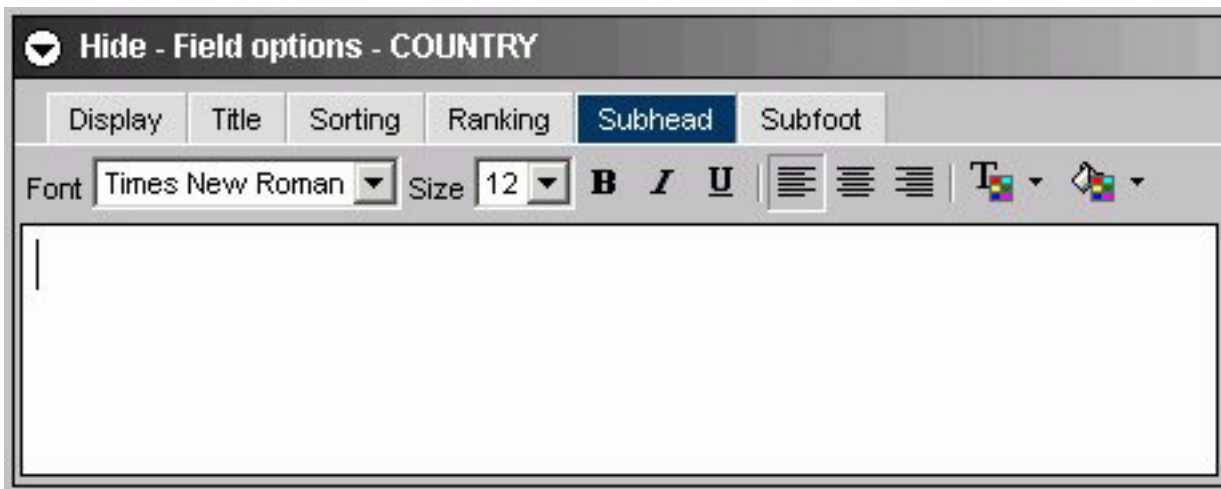
Formatting toolbar

Specify the font, font size, font style, text alignment, text color, and background color of the subhead or subfoot.

Text box

Enter a subhead/subfoot in the text box.

The Subtotal subtab for a Sort by field has:

Text box

Enter text to display for the subtotal.

Note: This sub tab appears when you select the Subtotal numeric sum/print fields option on the Sorting sub tab.

Field Selection Tab: Show Report Field Options for Sort Across

The following image shows the Show report field options where you choose various styles and functions for a selected field.

The Display subtab for a Sort across field has these options:

Make this field invisible

Check to hide the display of a Sort Across field.

Font button

Opens the font styling dialog box. See Font Dialog Box.

Conditional Styling button

Is grayed out and unavailable.

Drill Down button

Opens the drill-down dialog box. See Drill Down Dialog Box.

Format in report as

Is grayed out and unavailable.

Calculated as

Is grayed out and unavailable.

Include missing instances

Is grayed out and unavailable.

The Title subtab for a Sort across field has this option:

Title

Is the name of the field in the Sort Across list box.

The Sorting subtab for a Sort across field has these options:

Ascending

Click to arrange fields in ascending order.

Descending

Click to arrange fields in descending order.

Note: Other options are grayed out and unavailable.

Field Selection Tab: Show Report Field Options for Column

The following image shows the Show report field options where you choose
various styles and functions for a selected field.
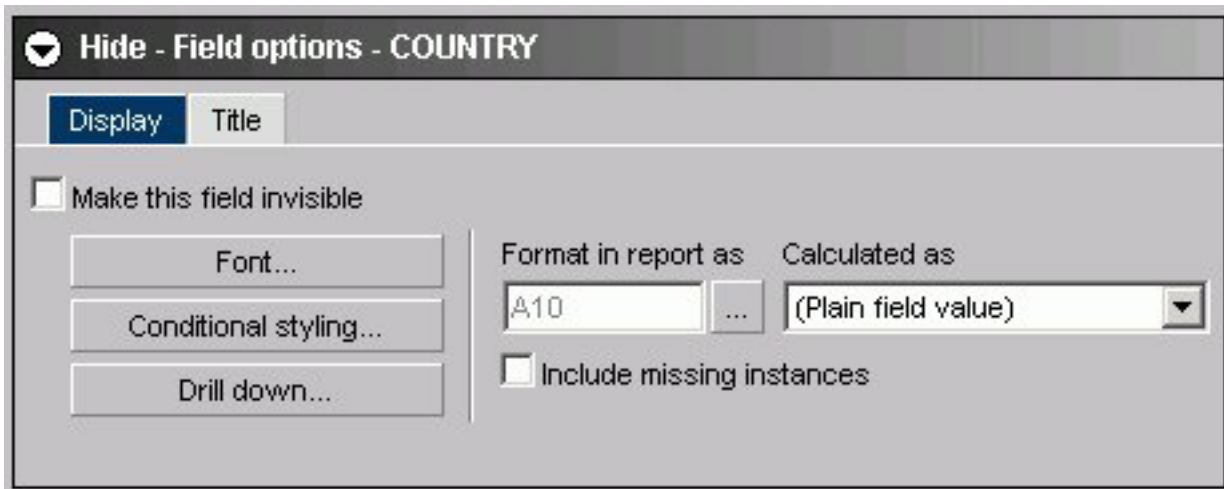
The Display subtab for a Column field has these options:

Make this field invisible

Check to temporarily hide the display of a Column field.

Font button

Opens the font styling dialog box. See Font Dialog Box.

Conditional Styling button

Opens the conditional styling dialog box where you use to add conditional
styling in a report.

Drill Down button

Opens the drill-down dialog box. See Drill Down Dialog Box.

Format in report as

Click ... to open the change format dialog where you can edit the field format. See Change Format Dialog Box.

Calculated as

Make a selection from the drop-down list.

Include missing instances

Inserts '.' when no field value exists.

The Title subtab for a Column field has this option:

Title

Is the name of the field in the Columns list box.

Drill Down Dialog Box

The following image shows the drill down dialog box, where you can add drill down capability to a report.
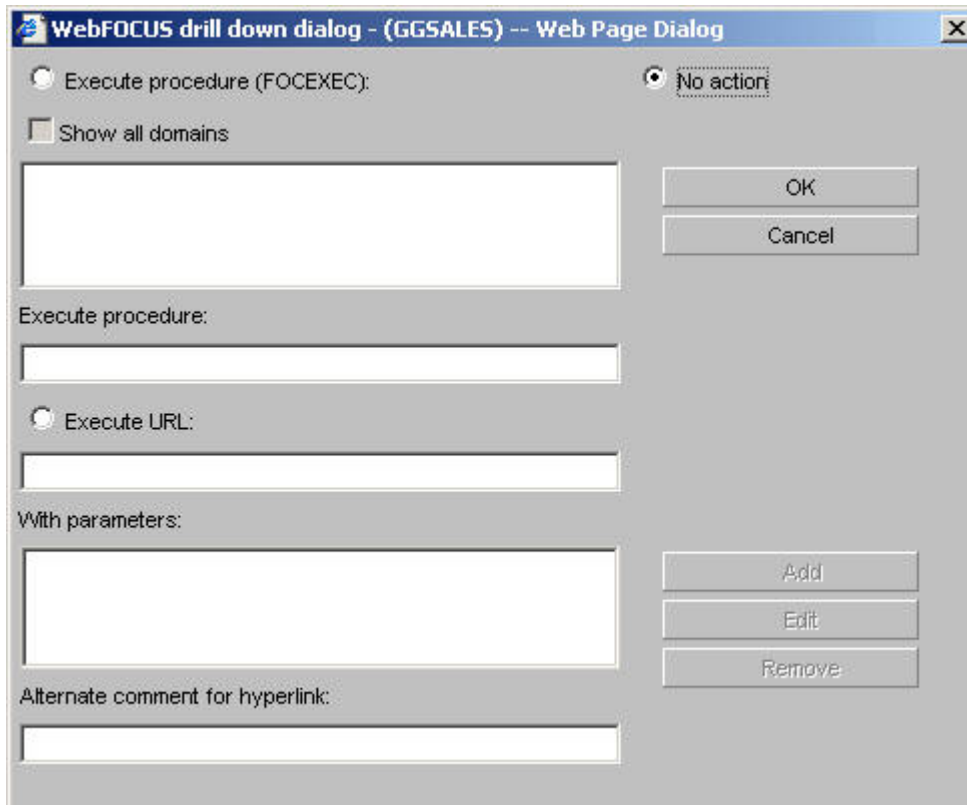
Execute procedure (FOCEXEC)

Lists the domains and procedures that are available as drill-down reports. Click the procedure you want to execute.

Show all domains

When Execute procedure (FOCEXEC) is selected, click this check box to gain access to procedures in the Standard Reports folders of all available domains.

No action

Is the default.

Execute procedure

The procedure you selected appears in this text box.

Execute URL

Enter the URL of a Web page that will open when a user drills down.

With parameters

Once a parameter is created using the Add button, it is automatically added to this area.

Alternate comment for hyperlink

You can place comments here.

Add button

Opens the drill-down parameter dialog shown in the following image.

Parameter name

Enter the parameter name you created in the drill down procedure.

Parameter value: Field

Select the field the user can drill down.

Parameter value: Constant value

The parameter is set to the specified value.

OK button

Returns to the drill down dialog. The parameter is added to the With parameters list box.

Font Dialog Box

The following images shows the font dialog box.
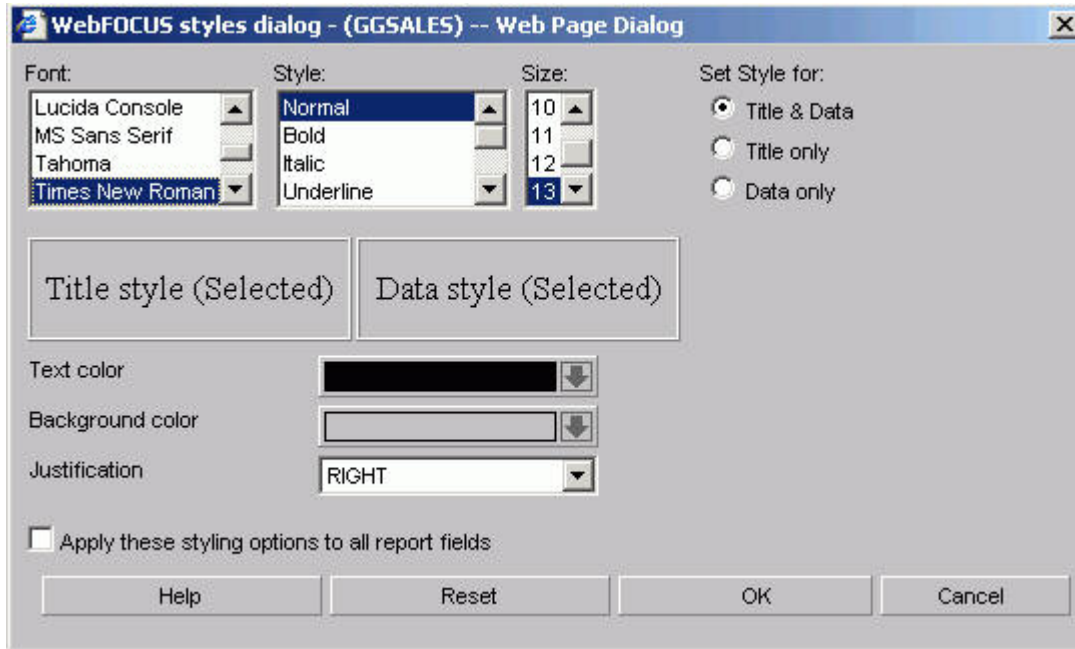
Font

Select from the list box.

Style

Select from normal or bold, italic, underline, and various combinations.

Size

Select a font size from the list box.

Set Style for Title & Data, Title only, or Data only option buttons

Set the style for Title & Data, Title only, or Data only.

Text color

Select a color from the color palette.

Background color

Select a color from the color palette.

Justification

Select left, right, or center from the list box.

Apply these styling conditions to all report fields check box

Apply these styling conditions to all report fields.

Field Selection Tab: Field Creator Dialog Box

Use the Field Creator Dialog Box to create or edit temporary fields, as shown in the following image.
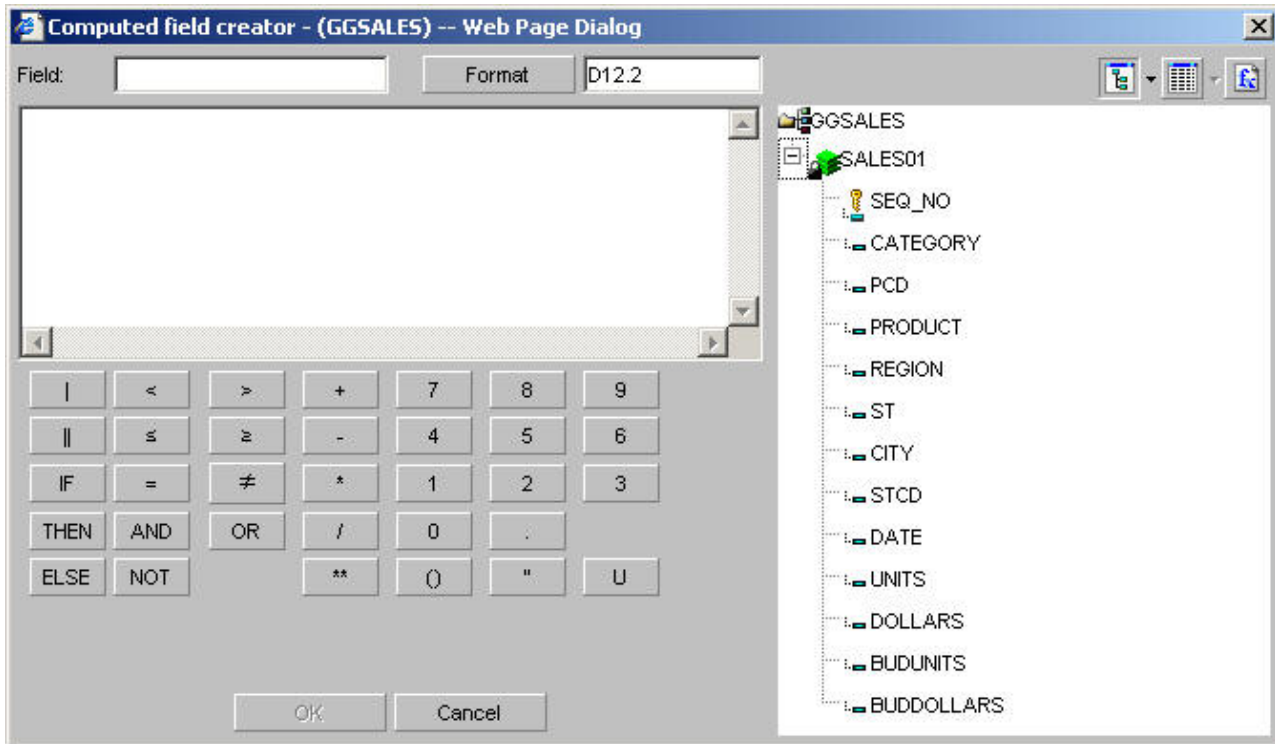
Field

> Enter a name for the temporary field. Note that field names cannot exceed 66 characters.

Format button

> Displays the Change Format dialog box where you can modify the data formats. See Change Format Dialog Box.

Tree or List mode buttons

> Opens the field list in either tree or list mode.

Functions button

> Displays a list of predefined functions.

Fields window

> Double-click a field or function to include it in an expression.

Concatenation operators

> Concatenates two or more alphanumeric constants and/or fields into a single character string. The concatenation operator has two forms: | (weak concatenation, which preserves trailing blanks) and || (strong concatenation, which moves trailing blanks to the end of a concatenated string).

IF... THEN... ELSE... operators

> Creates a conditional expression. A conditional expression assigns a value based on the result of a logical expression. The assigned value can be numeric or alphanumeric. IF, THEN, ELSE logic expressions take this form:
>
> **IF** expression1 **THEN** expression2 **ELSE**expression3

ELSE is an optional operator. All alphanumeric values in conditional expressions must be enclosed in single quotation marks. For example, IF REGION EQ 'West'.

Logical operators

Used to create relational and Boolean expressions. A relational expression returns a value based on a comparison of two individual values (either field values or constants) and a Boolean expression returns a value based on the outcome of two or more relational expressions.

Logical operators are:

*

adds the less than (LT) operator to the expression. This operator returns

a value if the value on the left is less than the value on the right.

*

adds the greater than (GT) operator to the expression. This operator

returns a value if the value on the left is greater than the value on the right.

*

adds the less than or equal to (LE) operator to the expression. This

operator returns a value if the value on the left is less than or equal to the value on the right.

*

adds the greater than or equal to (GE) operator to the expression. This

operator returns a value if the value on the left is greater than or equal to the value on the right.

*

adds the equal (EQ) operator to the expression. This operator returns a

value if the value on the left is equal to the value on the right.

*

add the not equal to (NE) operator to the expression. This operator

returns a value if the value on the left is not equal to the value on the right.
* AND returns a value if both operands are true.
* OR returns a value if either operand is true.
* NOT returns a value if the operand is false.

Arithmetic operators

> Click any of the arithmetic operators to use them in your expression: + (addition), - (subtraction), * (multiplication), / (division), ** (exponentiation).
>
> Two operators cannot appear consecutively. The following expression is invalid:
>
> a* -1
>
> To make it valid, you must add parentheses:
>
> a* (-1)

Parentheses

> Adds parentheses to the expression box. Parentheses affect the order in which the specified operations are performed. For information on when to use parentheses, see Order of Evaluation for Expressions.

Quotation Marks

> Adds quotation marks to the expression box. Use single quotation marks to enclose alphanumeric and date literals.

U option

> Converts entries in the expression box to uppercase. Click the U key in the calculator. Note that field names are case-sensitive.

Order of Evaluation for Expressions

DB2 Web Query performs numeric operations in the following order:

1. Parentheses.
2. Exponentiation.
3. Division and multiplication.
4. Addition and subtraction.

When operators are at the same level, they are evaluated from left to right. Because expressions in parentheses are evaluated before any other expression, you can use parentheses to change this predefined order. For example, the following expressions yield different results because of parentheses:

```
COMPUTE PROFIT/D12.2 = RETAIL_PRICE - UNIT_COST * UNIT_SOLD ;
COMPUTE PROFIT/D12.2 = (RETAIL_PRICE - UNIT_COST) * UNIT_SOLD ;
```

In the first expression, UNIT_SOLD is first multiplied by UNIT_COST, and the result is subtracted from RETAIL_PRICE. In the second expression, UNIT_COST is first subtracted from RETAIL_PRICE, and that result is multiplied by UNIT_SOLD.gives an incorrect result because UNIT_SOLD is first multiplied by UNIT_COST, and then the result is subtracted from RETAIL_PRICE.

Change Format Dialog Box

The following image shows the Change Format dialog box.
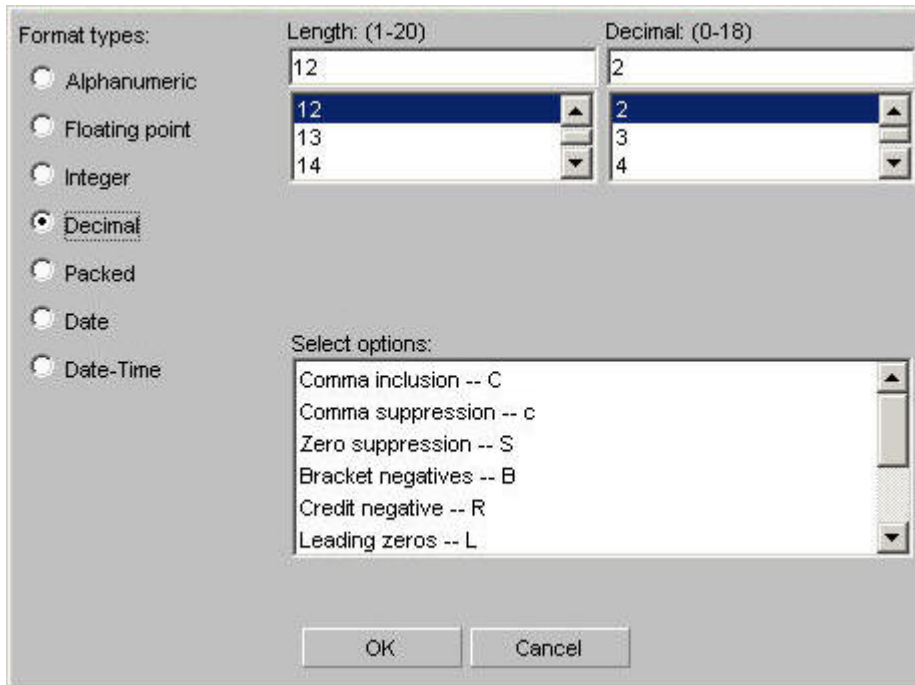
Format types

> Choose:
> - Alphanumeric
> - Floating point
> - Integer
> - Decimal
> - Packed
> - Date
> - Date-Time

Length

> Choose an available number based on the format type you chose.

Select options

> Choose options based on the format type you chose.

Report Headings Tab

The Report headings tab is used to select and style headings and footings for a report.

Report Headings Tab

The following image shows the Report headings tab where you enter and style a page title heading and/or footing. You can also include a field value from the Available fields list in the page heading and/or footing.
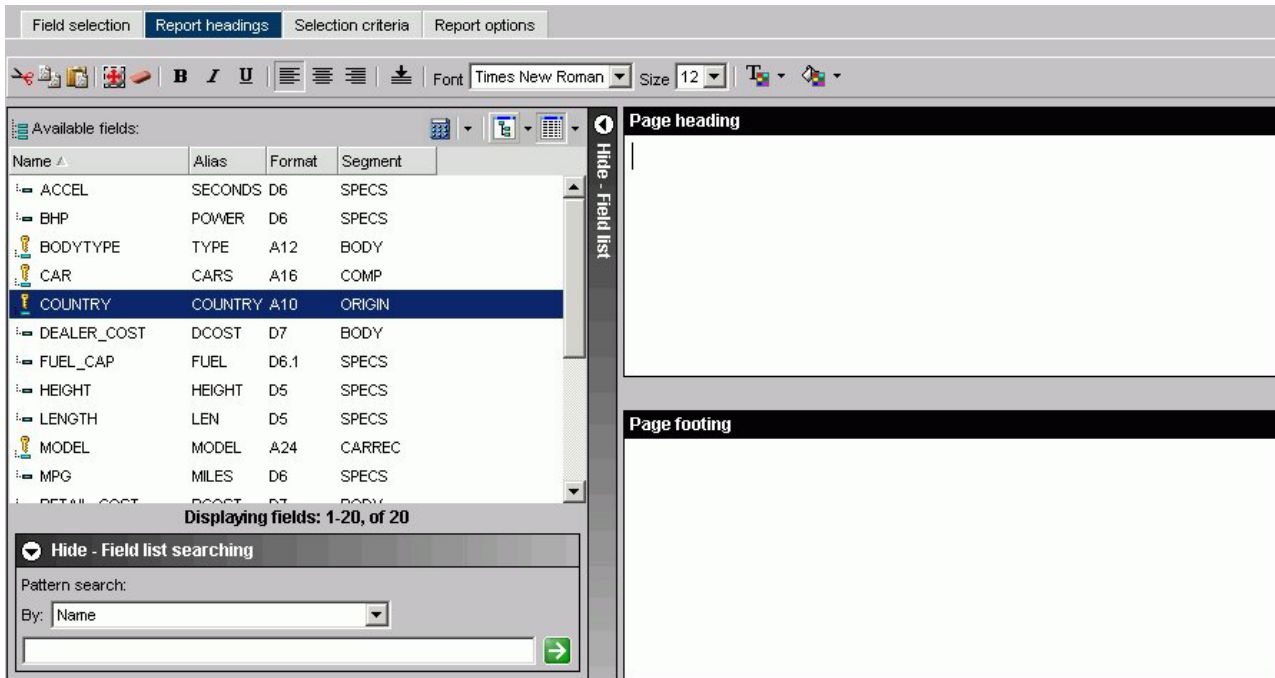
Formatting tool bar
- Cut, copy, paste.
- Select all.
- Remove formatting.

Note: The above icons are not available for Firefox users. You must use the standard key combinations Ctrl+C, Ctrl+V, Ctrl+X, and so on.
- Font styles. Select from bold, italic, and underline.
- Alignment. Select left, center, right, or bottom alignment.
- Font. Select a different font from the list box.
- Font size. Select a font size from the list box.
- Text color. Select the text color from the color menu.
- Background color. Select the background color from the color menu.

Show/Hide Fields list

Show or hide the Available Fields window.

Available fields window

Insert a field in your heading or footing from the fields list.

Define icon

Click the Define (calculator) icon to select: New define field, Edit define field, or Delete define field. If you select the New define field or Edit define field, the Define field creator dialog box opens where you can create or edit a defined field.

Tree icon

Displays a tree structure of field names by segment with segment, alias, title, description, and format information below the tree. Individual field information appears when you click on the field.

List icon

Displays a list of all field names with information that you select from the adjacent arrow that opens a drop-down list: name, alias, title, format, description, segment, and file name. You can also choose to sort by file order.

Page Heading

Enter the text for the heading in the Page Heading box.

Page Footing

Enter the text for the footing in the Page Footing box.

Selection Criteria Tab

The Selection criteria tab is used to create WHERE and WHERE TOTAL statements for a report.

Selection Criteria Tab: Screening Conditions

The following image shows the Screening conditions section on the Selection criteria tab where you create a WHERE statement by dragging and dropping a field from the Available Fields window.
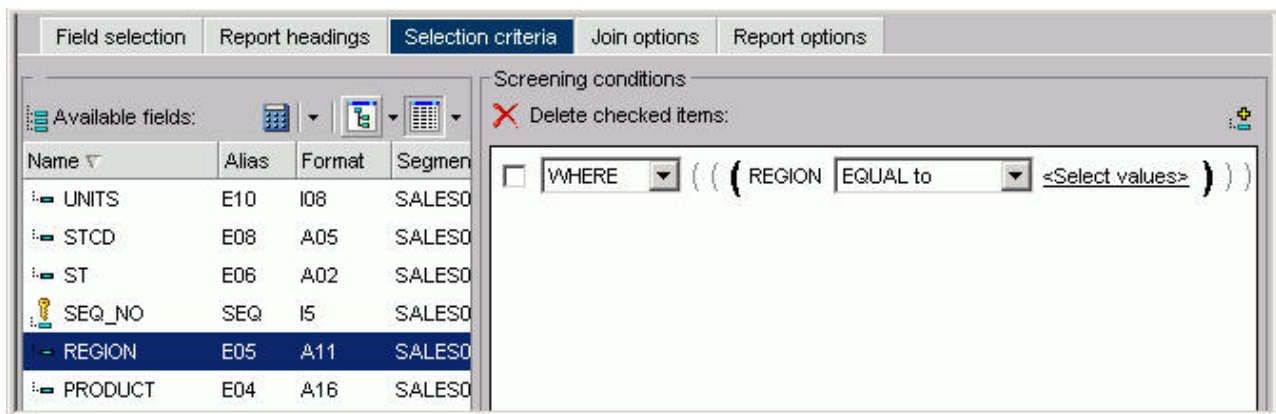
WHERE/WHERE TOTAL

Use WHERE to select records based on the values of an individual field.

Use WHERE TOTAL to select records based on the aggregate value of a field, for example, the sum or average of a field's values.

Fieldname

Is the field you initially chose.

Operator Expression

Select an operator from the drop-down list, for example, EQUAL to.

Select Values

Click to open the Values dialog box, which is shown in the following image.
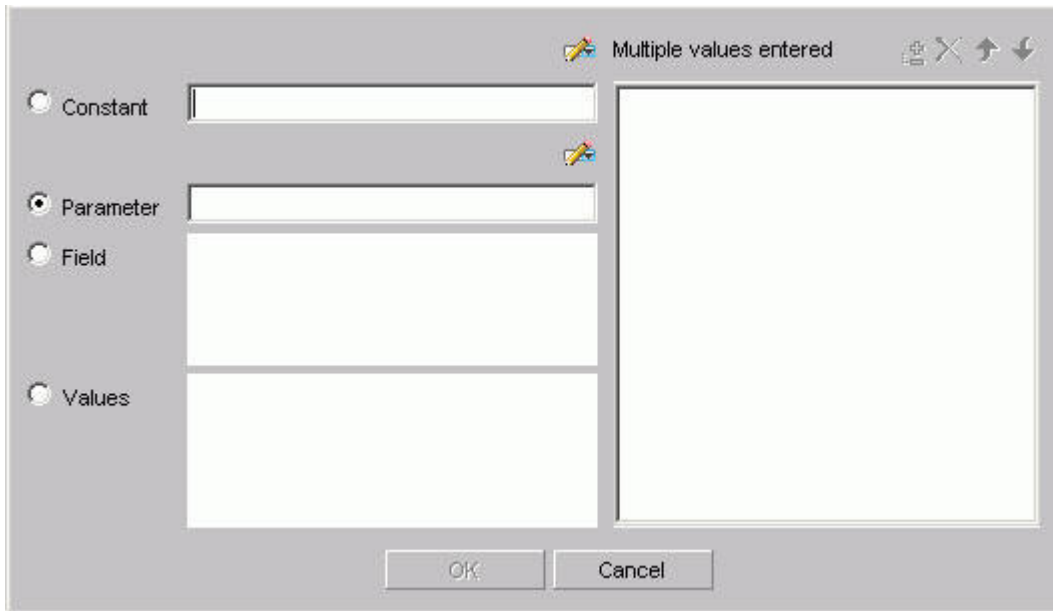
Constant

> Type a constant value. To use multiple constant values, click the plus sign in the Multiple values entered list box after each constant value that you type.

Select from file (Import) button

> Opens the Select file dialog box. For more information, see Importing Values From External Files for WHERE Statements.

Parameter

> Type a parameter name. To use multiple parameter values, click the plus sign in the Multiple values entered list box after each parameter name that you type.

Auto Prompt button

> Opens the Parameter Properties dialog box. See Parameter Properties Dialog Box (Dynamic Parameters) and Parameter Properties Dialog Box (Static Parameters) for details.

Field

> Select a field from the Field list box.

Values

> Select a field value from the Values list box. To use multiple field values, click the plus sign in the Multiple values entered list box after each field value that you select.

Multiple values entered

> Displays the constants, imported values, parameters, or field values you added to the Multiple values list box. Use the available buttons to add, remove, and change the order of the selected values.

Parameter Properties Dialog Box (Dynamic Parameters)

The following image shows the Parameter Properties dialog box with the Dynamic option selected.

Name

> Name is the parameter name. This field is automatically filled in with information from the selected field. You can change the name if desired.

Description

> Text that appears in the Auto prompt page as a prompt for the selection list. This field is automatically filled in with information from the selected field. You can change the description if desired.

Selection

> Select the type of parameter:
> - **Dynamic** retrieves values from the specified data source when the request is run. This is the default selection.
> - **Static** contains a list of values you supply. These values do not change unless you change them.

Select multiple values at runtime

> Selecting this check box enables you to provide more than one value to the report from the auto prompt page. In addition, the user is able to select all values from the values list.

Data Source

> Select a data source that contains the values for the parameter. The data source must be on your APP PATH.

Field

> Select the field from the data source whose values will populate the selection list.

Parameter Properties Dialog Box (Static Parameters)

The following image shows the Parameter Properties dialog box with the Static option selected.

Name

> Is the parameter name. This field is automatically filled in with information from the selected field. You can change the name if desired.

Description

> Text that appears in the Auto prompt page as a prompt for the selection list. This field is automatically filled in with information from the selected field. You can change the description if desired.

Selection

> Select the type of parameter:
> - **Dynamic** retrieves values from the specified data source when the request is run. This is the default selection.
> - **Static** contains a list of values you supply. These values do not change unless you change them.

Select multiple values at runtime

Selecting this check box enables you to provide more than one value to the report from the auto prompt page. In addition, the user is able to select all values from the values list.

Constant

Enter one or more constant values.

Show field values

Shows the values in the selected field. Move the values you want in the selection list to the Prompt values list.

Prompt values

Prompt Values are the values that appear in the selection list. You can use the available buttons to add, remove, and change the order of the Prompt values.

Join Options Tab

The Join options tab is used to create joins for a report.

Join Options Tab: Add or New

The following image shows the Join options tab where you create joins for a report.
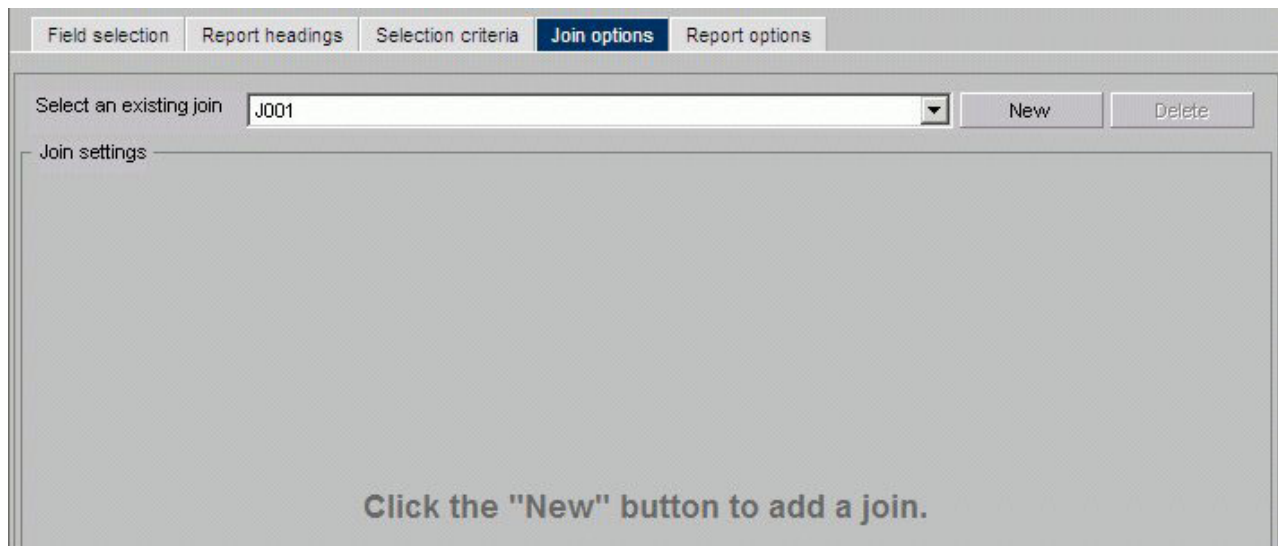
Select an existing join

Use the list box to select the join with which you want to work.

New

Opens the Masters List dialog box where you select the target data source.

Delete

Click to delete the join.

OK button

Click to select a Master File.

Cancel button

Click to return to the Add/New Join window.

When you select a target data source you have multiple join related settings and options, as shown in the following image.

The Join Settings section provides the following fields and buttons for creating joins:

Description

Enter a description for your join.

Internal name

DB2 Web Query assigns an alphanumeric name for the join, for example J001.

Save & Create

Once you have selected your host and target fields, click Save & Create to create the join.

View

Click to view the syntax of your join, which displays the code sent to the DB2 Web Query Reporting Server to validate the join, but may not be the same code required to run the report. This is the case for a DEFINE based

JOIN where only the JOIN statement is displayed for validation, but both the JOIN and DEFINE statements are needed when the report is run.

The Join Type drop-down list provides the following join type options:

Inner Join

A join that results when a report omits host rows that lack corresponding cross-referenced rows.

Unspecified

Indicates neither an inner nor left outer join.

Left Outer Join

Extends the results of an Inner Join and retrieves records from both host and cross-referenced tables, including all records from the left table (host) and any records from the right table (cross-referenced) where the condition values match. If there are no matching values in the cross-referenced table, the join still retrieves records from the host table.

The Instances drop-down list provides the following join instance options:

Multiple Instance (formerly Non Unique)

A one-to-many join structure that matches one value in the host data source to multiple values in the cross-referenced field. Joining employee ID in a company's employee data source to employee ID in a data source that lists all the training classes offered by that company would result in a listing of all courses taken by each employee, or a joining of the one instance of each ID in the host file to the multiple instances of that ID in the cross-referenced file.

Single Instance (formerly Unique)

A one-to-one join structure that matches one value in the host data source to one value in the cross-referenced data source. Joining an employee ID in an employee data source to an employee ID in a salary data source is an example of a unique join.

The Host Field List contains:

Search (binoculars) icon

Opens the Field List Searching dialog box. See Searching Field Lists for details.

Define icon

Click the Define icon to select: New define field, Edit define field, or Delete define field. If you select the New define field or Edit define field, the Define field creator dialog box opens where you can create or edit a defined field.

Tree icon

Displays a tree structure of field names by segment with segment, alias, title, description, and format information below the tree. Individual field information appears when you click the field.

List icon

Displays a list of all field names with information that you select from the adjacent arrow that opens a list box: name, alias, title, format, description, segment, and file name. You can also choose to sort by file order.

Available fields

Lists of fields from the host data source for creating a join.

The Target Field List contains:

Select a target button

Click to select a target Master File.

Tree icon

Displays a tree structure of field names by segment with segment, alias, title, description, and format information below the tree. Individual field information appears when you click the field.

List icon

Displays a list of all field names with information that you select from the adjacent arrow that opens a list box: name, alias, title, format, description, segment, and file name. You can also choose to sort by file order.

Fields

Lists of fields from the target data source for creating a join.

For both the Host and Target field lists, each has its own Selected Fields window that has:

Selected Fields window

Shows the fields selected from the data source for a join.

Add icon

Adds a field.

Remove icon

Removes a field.

Move Up or Move Down icons

Moves the position of the field within the join.

Report Options Tab

The Report options tab is used to select options for a report.

Report Options Dialog Box

The following image shows the Report options tab where you select report output formats.
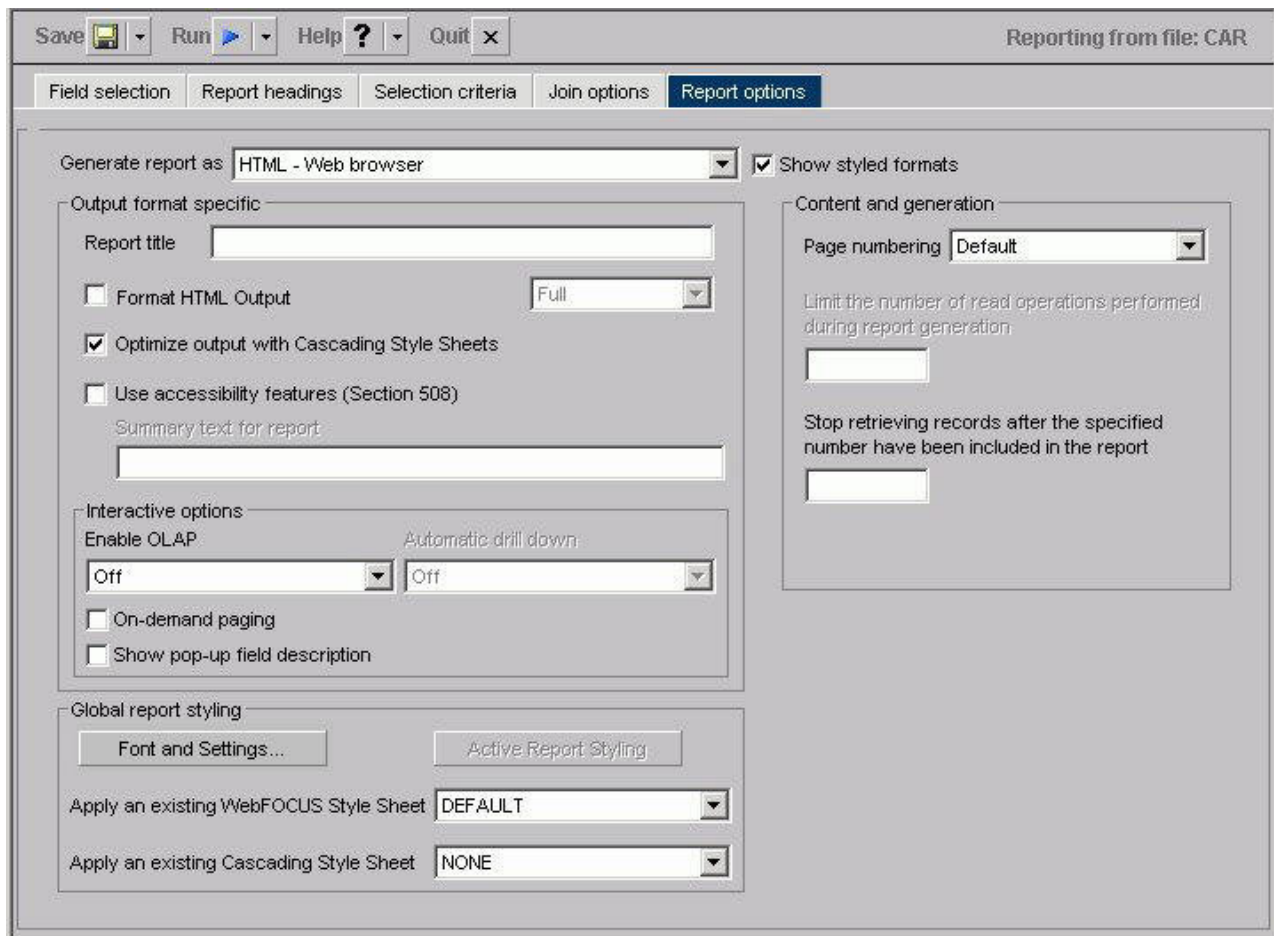
Generate report as list box (when check box is selected)

When the Show styled formats check box is selected, choose from the following formats:

- HTML - Web browser
- HTML - Web browser (with grid)
- AHTML - Active Report
- PDF - Portable Document Format
- EXL97 - Excel 97
- EXL2K - Excel 2000
- EXL2KFORMULA - Excel 2000 with FORMULA
- PS - Postscript

Generate report as list box (when check box is not selected)

When the Show styled formats check box is not selected, choose from the following formats:

- EXCEL 95 - Excel 95
- DOC - Plain text with page breaks
- WP - Plain text without page breaks
- WK1 - Import to Lotus 1-2-3. See Notes on Using LOTUS Format.
- DIF - Import to spreadsheet
- LOTUS - Import to Lotus 1-2-3. See Notes on Using LOTUS Format.

- TABT - Tab-delimited
- XML - eXtensible Markup Language

Show styled formats check box

>Check to show output formats that support styling options.

The Output format specific section has these options:

Report title

>Enter the report title in the text box.

Optimize output with Cascading Style Sheets

>Check to optimize output with Cascading Style Sheets.

Use accessibility features (Section 508)

>Check to use accessibility features (Section 508).

Summary text for report

>Add summary text in the text box. See Include Summary Text for a Report.

The Interactive options subsection has these options:

Enable OLAP

>Choose:
>- OFF
>- Columns only
>- Columns with panel
>- Show filters on top
>- Show filters on bottom
>- Show panel in report
>- Show Tabbed

Automatic drill down

>Choose:
>- Off
>- Dimensions only
>- Dimensions and Measures

On-demand paging

>Check to initiate on-demand paging.

Show pop-up field description

>Check to display pop-up field descriptions for column titles. See Displaying Pop-up Field Descriptions for Column Titles.

The Global report styling section has these options:

Font and Settings button

>Opens the Styles dialog box.

>**Font**

>>Select from the list box.

>**Style**

Select from normal or bold, italic, underline, and various combinations.

Size

Select a font size from the list box.

Text color

Select a color from the color palette.

Background color

Select a color from the color palette.

Justification

Select left, right, or center from the list box.

Currency Symbol

Select a different currency symbol to override the currency symbol set on the reporting server. Choose from Default, USD, GBP, JPY, EUR, $, or Custom in the Currency Symbol menu. When you select Custom, replace "Custom" with the desired one-character custom currency symbol in the Currency Symbol field.

Active Report Styling

Opens the Active Report Styling dialog box. This button is only active when you select the AHTML - Active Report output format. See Creating an HTML Active Report.

Apply an existing WebFOCUS Style Sheet

Select None or a Style Sheet from the list box.

Apply an existing Cascading Style Sheet

Select None or a Style Sheet from the list box.

The Content and generation section has these options:

Page numbering

Select Default, On, Off, or No lead from the list box.

Limit the number of read operations performed during report generation

Enter a number in the text box.

Stop retrieving reports after the specified number have been included in the report

Enter a number in the text box.

Notes on Using Lotus Format

The Display Format selection box on the Report options tab lists Lotus as a display format. This format, however, does not create a valid Lotus™ spreadsheet. To work around this limitation, follow these steps:

1. Select the Excel display format.
2. When the request is run, WebFOCUS prompts you to save or open the file, select **Save**.
3. Save the file with an .xls extension to a valid directory.

   Lotus Release 5.0 or higher can open files with an .xls extension.

Report Options Tab

The Report options tab is used to select options for a report.

Report Options Dialog Box

The following image shows the Report options tab where you select output formats for a report.

Generate report as list box (when Show styled formats is selected)

When the Show styled formats option is selected from the drop-down list, choose from the following formats:

- HTML - Web browser
- HTML - Web browser (with grid)
- AHTML - Active Report
- PDF - Portable Document Format
- EXL97 - Excel 97
- EXL2K - Excel 2000
- EXL2KFORMULA - Excel 2000 with formula
- PS - Postscript

Generate report as list box (when Show unstyled formats is selected)

When the Show unstyled formats option is selected from the drop-down list, choose from the following formats:

- EXCEL 95 - Excel 95
- DOC - Plain text with page breaks
- WP - Plain text without page breaks
- WK1 - Import to Lotus 1-2-3. See Notes on Using LOTUS Format.
- DIF - Import to spreadsheet
- LOTUS - Import to Lotus 1-2-3. See Notes on Using LOTUS Format.
- TABT - Tab-delimited
- XML - eXtensible Markup Language

Generate report as list box (when Show database formats is selected)

When the Show database formats option is selected from the drop-down list, choose from the following formats:

- DB2 - DB2 database table

When the Show database option is selected, the Database Output section becomes active.

Show formats drop-down list

The drop-down list for selecting formats has these options:

- Show styled formats. Displays output formats that support styling options.
- Show unstyled formats. Displays output formats that do not support styling options.
- Show database formats. Displays database formats. When selected, the Database Output section becomes active.

For styled formats, the Output format specific section has these options:

Report title

Enter the report title in the text box.

Format HTML Output

Full - Retains standard HTML formatting. This is the default.

Paged - Specifies that each page of a report is returned to the browser as a separate HTML table.

Fixed - Turns off HTML formatting and allows the report designer to determine where items in the report are placed.

Optimize output with Cascading Style Sheets

Check to optimize output with Cascading Style Sheets.

Use accessibility features (Section 508)

Check to use accessibility features (Section 508).

Summary text for report

Add summary text in the text box. See Include Summary Text for a Report.

In the Interactive options subsection:

Enable OLAP

Choose:
- OFF
- Columns only
- Columns with panel
- Show filters on top
- Show filters on bottom
- Show panel in report
- Show Tabbed

Automatic drill down

Choose:
- Off
- Dimensions only
- Dimensions and Measures

On-demand paging

Check to initiate on-demand paging.

Show pop-up field description

Check to display pop-up field descriptions for column titles. See Displaying Pop-up Field Descriptions for Column Titles.

The Global report styling section has these options:

Font and Settings button

Opens the Styles dialog box.

Font

Select from the list box.

Style

Select from normal or bold, italic, underline, and various combinations.

Size

Select a font size from the list box.

Text color

Select a color from the color palette.

Background color

Select a color from the color palette.

Justification

Select left, right, or center from the list box.

Currency Symbol

Select a different currency symbol to override the currency symbol set on the reporting server. Choose from Default, USD, GBP, JPY, EUR, $, or Custom in the Currency Symbol menu. When you select Custom, replace "Custom" with the desired one-character custom currency symbol in the Currency Symbol field.

Active Report Styling

Opens the Active Report Styling dialog box. This button is only active when you select the AHTML - Active Report output format. See Creating an HTML Active Report.

Apply an existing Style Sheet

Select None or a Style Sheet from the list box.

Apply an existing Cascading Style Sheet

Select None or a Style Sheet from the list box.

The **Send to printer** check box allows you to send your output directly to a designated printer.

The Content and generation section has these options:

Page numbering

Select Default, On, Off, or No lead from the list box.

Limit the number of read operations performed during report generation

Enter a number in the text box.

Stop retrieving reports after the specified number have been included in the report

Enter a number in the text box.

Use SQL Engine Joins. Multiplicative effect may result. Web Query protection is suppressed.

Enables the database to use its internal optimization techniques which reduce the volume of database-to-server communications and improve response time.

The Database Output section has these options:

Destination Library

Enter name of the library destination for the DB2 report file.

Name

Enter the name of the DB2 file.

Overwrite existing file

Select this option if you need to overwrite an existing file. **Note:** When specifying to overwrite an existing database file, the file will not be overwritten if you do not have specific write authority to the file. A message will be displayed which says that the file already exists, even though overwrite was specified.

The Printer section has one option:

Name   Enable when the Send to printer check box is selected. Enter the name of the printer.

Notes on Using Lotus Format

The Display Format selection box on the Report options tab lists Lotus as a display format. This format, however, does not create a valid Lotus™ spreadsheet. To work around this limitation, follow these steps:

1. Select the Excel display format.
2. When the request is run, DB2 Web Query prompts you to save or open the file, select **Save**.
3. Save the file with an .xls extension to a valid directory.

   Lotus Release 5.0 or higher can open files with an .xls extension.

Chapter 5. Analyzing Data in an OLAP Report

Presents the terminology and benefits of using Online Analytical Processing (OLAP). Describes how to customize reports with the OLAP selections panel and the OLAP Control Panel. Describes how to sort and apply various selection criteria (to restrict your data) as well as how to troubleshoot an OLAP⌂enabled report. Explains how the OLAP Control Panel (OCP) provides you with a versatile way to gain more insight from your reports by dynamically manipulating report data. From the Control Panel, you can perform every function available to a DB2 Web Query OLAP user.

If the OLAP license option has been activated, DB2 Web Query Online Analytical Processing (OLAP) enables you to view and quickly analyze data in order to make critical business decisions.

We Do It Every Day: Typical Web Query

Suppose that you own a small business in New York and are exploring a partnership with a company in Oakland, California. You need to get to a Monday morning meeting. How do you go about arranging your flight?

Most likely, you go online.

First, you check available flights on the airline that holds your frequent flyer miles. You discover that your frequent flyer carrier requires a change of planes and you would prefer a direct flight, so you look at routes and fares for other airlines.

In New York, you can get to LaGuardia, JFK, and MacArthur airports on Long Island. In California, you can fly into Oakland or San Francisco.

While you would prefer to fly out on Sunday and return Tuesday morning, you could consider a Saturday flight to California and a return flight on the red-eye Monday night, if fares and schedules are better.

You begin your search by airline and then look at options for each departure point and destination, by day, time and price.

Another approach is to start with an online consolidator, enter the times you can fly, and see what flights and fares are available.

There are a lot of variables to play with, but in ½ hour you have done your research and can make a good decision based on all available factors.

The Web sites you access are designed to facilitate your queries. Various menus and selection boxes make it easy to pursue each line of inquiry. Required and optional information is identified for you. You can move forward down a path of choices, backtrack and start down a different path, or resume the original path with different selections.

You need to keep track of the question you want to answer, but a well-designed site makes your investigation easy. For most of us, this process has become intuitive.

The same process works when analyzing the data in an OLAP-enabled DB2 Web Query report.

Running OLAP Examples

You can run all of the examples in this chapter using several OLAP-enabled Standard Reports. If the reports are not already available in your sample repository, ask your Managed Reporting administrator to provide them for your use. There are 9 reports named olaprep1.fex through olaprep9.fex. and are located in the \ibinccen directory. If you have installed a non-English version of the 'ibincc' directory, you will need to install the English version (ibinccen) in order to access these files.

Each example indicates which Standard Report to run. After the report appears in your browser, you can perform the analytic task shown, or pursue your own line of inquiry.

Suppose that you are an analyst for the fictional Century Corporation, which manufactures electronics equipment. You need to determine which of the stores that sells your products had the highest sales in 2002, and whether there is a pattern in sales periods and/or best selling products that should be considered when planning manufacturing schedules and parts inventories.

You have created a base report that shows sales data only for 2002. You have also OLAP-enabled the report to permit quick analysis of the data.

1. Run the Standard Report **OLAPREP1**.

   Before you begin your analysis, the OLAP report looks like the following image.

Drag Quarter up →	QUARTER	Store Name:	PRODTYPE	Quantity:	Line Cost Of Goods Sold
	Q1	AV VideoTown	Analog	18,449	3,969,296.00
			Digital	22,206	5,109,400.00
		Audio Expert	Analog	78,449	16,467,146.00
			Digital	105,983	25,092,678.00
		City Video	Analog	6,287	1,315,015.00
			Digital	7,196	1,607,513.00
		Consumer Merchandise	Analog	6,980	1,542,036.00
			Digital	14,957	3,251,090.00
		TV City	Analog	19,077	3,772,119.00
			Digital	41,307	10,128,967.00
		Web Sales	Analog	545	124,366.00
			Digital	829	190,201.00
		eMart	Analog	97,128	21,152,262.00
			Digital	108,221	24,990,368.00
	Q2	AV VideoTown	Analog	11,781	2,663,655.00
			Digital	27,377	5,928,507.00
		Audio Expert	Analog	57,944	11,868,758.00
			Digital	111,421	28,064,250.00
		City Video	Analog	1,405	285,323.00

The Quarterly information is spread out over the left-most column. You can try a horizontal display to make comparison easier.

2. Drag and drop **QUARTER** above the report.

   The report changes immediately and appears, as shown in the following image, across the top of the report with the Quantity and Line Cost of Goods Sold columns repeating for each quarter. The store information is more compact, but it is no easier to identify the store with the best sales record, so drag QUARTER back to its original position.

3. Right-click **Line Cost of Goods Sold** and choose **Visualization** as shown in the following image. This applies a data visualization bar graph to each value in the column.

The display changes as shown in the following image. The bar graphs still do not reveal a trend.

| | | | Click to sort → | |
QUARTER	Store Name:	PRODTYPE	Quantity:	Line Cost Of Goods Sold
Q1	AV VideoTown	Analog	18,449	3,969,296.00
		Digital	22,206	5,109,400.00
	Audio Expert	Analog	78,449	16,467,146.00
		Digital	105,983	25,092,678.00
	City Video	Analog	6,287	1,315,015.00
		Digital	7,196	1,607,513.00
	Consumer Merchandise	Analog	6,980	1,542,036.00
		Digital	14,957	3,251,090.00
	TV City	Analog	19,077	3,772,119.00
		Digital	41,307	10,128,967.00
	Web Sales	Analog	545	124,366.00
		Digital	829	190,201.00
	eMart	Analog	97,128	21,152,262.00
		Digital	108,221	24,990,368.00
Q2	AV VideoTown	Analog	11,781	2,663,655.00
		Digital	27,377	5,928,507.00
	Audio Expert	Analog	57,944	11,868,758.00
		Digital	111,421	28,064,250.00

4. Sort the data by highest value by either right-clicking **Line Cost of Goods Sold** and choosing **Sort by Highest**, or clicking the **Up** arrow (the tool tip reads **Sort LINE_COG highest to lowest**).

As shown in the following image, the report shows Audio Expert has the highest sales in the digital product lines in Quarters 1 and 2, with eMart trailing slightly. Each value under the QUARTER, Store Name and PRODTYPE column is hyperlinked for more details.

QUARTER	Store Name:	PRODTYPE	Quantity:	Line Cost Of Goods Sold	
Q2	Audio Expert	Digital	111,421	28,064,250.00	
Q1	Audio Expert	Digital	105,983	25,092,678.00	
Q1	eMart	Digital	108,221	24,990,368.00	
Q2	eMart	Digital	115,102	24,971,512.00	
Q1	eMart	Analog	97,128	21,152,262.00	
Q2	eMart	Analog	74,737	16,789,403.00	
Q1	Audio Expert	Analog	78,449	16,467,146.00	
Q4	eMart	Digital	72,126	14,000,951.00	
Q3	eMart	Digital	66,156	13,867,709.00	
Q2	Audio Expert	Analog	57,944	11,868,758.00	
Q3	Audio Expert	Digital	50,076	11,210,406.00	
Q4	Audio Expert	Digital	53,275	11,190,923.00	
Q1	TV City	Digital	41,307	10,128,967.00	
Q4	eMart	Analog	39,515	9,383,389.00	
Q3	eMart	Analog	36,306	8,308,647.00	
Q2	TV City	Digital	29,627	6,732,303.00	
Q2	AV VideoTown	Digital	27,377	5,928,507.00	
Q4	Audio Expert	Analog	25,897	5,916,936.00	

(Click Q2)

5. Click **Q2** for Audio Expert to check the monthly breakdown.

    In the monthly report, both stores recorded their highest sales in June (06). Filter out the other stores and focus on Audio Expert in June.

6. Click **Audio Expert** as shown in the following image.

MONTH	Store Name:	PRODTYPE	Quantity:	Line Cost Of Goods Sold	
06	Audio Expert	Digital	42,473	11,119,112.00	
06	eMart	Digital	45,895	10,518,058.00	
05	Audio Expert	Digital	36,505	8,930,974.00	
04	Audio Expert	Digital	32,443	8,014,164.00	
04	eMart	Digital	35,336	7,797,162.00	
05	eMart	Digital	33,871	6,656,292.00	
04	eMart	Analog	27,584	6,061,370.00	
05	eMart	Analog	24,253	5,728,792.00	
04	Audio Expert	Analog	24,312	5,087,601.00	
06	eMart	Analog	22,900	4,999,241.00	
05	Audio Expert	Analog	17,073	3,606,229.00	
06	Audio Expert	Analog	16,559	3,174,928.00	
05	TV City	Digital	10,296	2,745,088.00	
06	AV VideoTown	Digital	11,466	2,507,452.00	
06	TV City	Digital	9,848	1,997,234.00	
04	TV City	Digital	9,483	1,989,981.00	
05	AV VideoTown	Digital	7,336	1,728,971.00	
04	AV VideoTown	Digital	8,575	1,692,084.00	

(Click Audio Expert)

You now see information for digital and analog sales at Audio Expert. Since the significant sales for Audio Expert are in the digital area, let's see which digital products contributed to the June figures.

7. Click **Digital** as shown in the following image.

The breakdown shows clearly that PDAs drove Audio Expert digital sales.

8. Click **PDA Devices** to see the details as shown in the following image.

As shown in the following image, **ZT Digital PDA - Commercial** was by far the top seller.

Let's now see what drove digital sales at eMart, the second highest producer.

9. Click the browser's **Back** arrow until you return to the following window. This time, click **eMart**, as shown in the following image.

MONTH	Store Name:		PRODTYPE	Quantity:	Line Cost Of Goods Sold	
06	Audio Expert		Digital	42,473	11,119,112.00	
06	eMart	←	Digital	45,895	10,518,058.00	
05	Audio Expert	Click eMart	Digital	36,505	8,930,974.00	
04	Audio Expert		Digital	32,443	8,014,164.00	
04	eMart		Digital	35,336	7,797,162.00	
05	eMart		Digital	33,871	6,656,292.00	
04	eMart		Analog	27,584	6,061,370.00	
05	eMart		Analog	24,253	5,728,792.00	
04	Audio Expert		Analog	24,312	5,087,601.00	
06	eMart		Analog	22,900	4,999,241.00	
05	Audio Expert		Analog	17,073	3,606,229.00	
06	Audio Expert		Analog	16,559	3,174,928.00	

Once again, the digital category leads sales.

10. Click **Digital** as shown in the following image.

PRODTYPE		Quantity:	Line Cost Of Goods Sold	
Digital	← Click Digital	45,895	10,518,058.00	
Analog		22,900	4,999,241.00	

PDA is the strong seller here too.

11. Click **PDA Devices** as shown in the following image to examine the models that comprise these sales.

PRODCAT		Quantity:	Line Cost Of Goods Sold	
PDA Devices	← Click PDA Devices	16,020	5,348,080.00	
Camcorders		3,443	2,456,730.00	
DVD		5,518	1,029,502.00	
Digital Tape Recorders		13,378	923,082.00	
CD Players		7,390	731,610.00	
Cameras		146	29,054.00	

The report shows sales figures for the two digital models as shown in the following image.

Product Name:	Quantity:	Line Cost Of Goods Sold	
ZT Digital PDA - Commercial	13,591	4,743,259.00	
ZC Digital PDA - Standard	2,429	604,821.00	

ZT Digital PDA - Commercial far outsells **ZC Digital PDA -Standard**.

This information from the two top selling stores suggests that Century Corporation should evaluate and adjust available parts inventories for each model and consider shifting production schedules of plants to produce more Commercial units.

You have done all of your data manipulation from the report. But, because of the options you selected when OLAP-enabling this report, it is easy to expose the OLAP Selections pane where you can review the selections that are currently in effect, and make additional selections if you like. For details on OLAP set-up options, see OLAP-Enabling a Report.

12. To expose the OLAP Selections pane, right-click **Product Name** and select **Show Panel** from the menu as shown in the following image.

The selection panel appears above the report as shown in the following image.
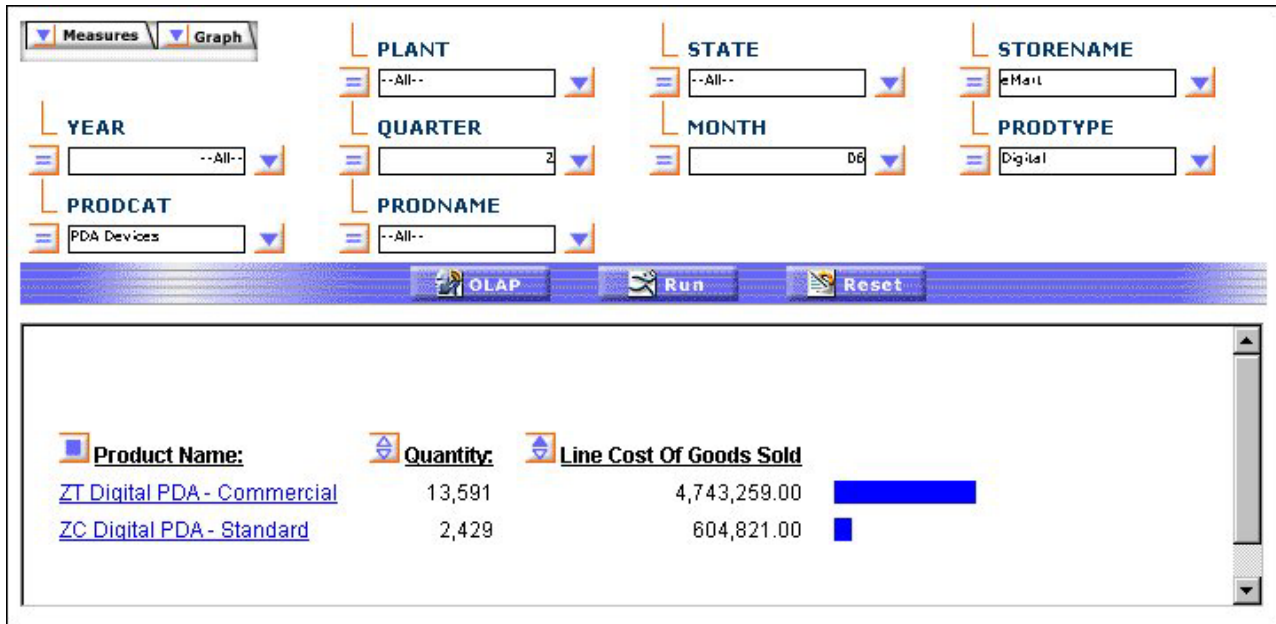
Notice that STORENAME is eMart, PRODTYPE is Digital, and PRODCAT is PDA Devices.

OLAP Reporting Requirements

OLAP reporting requires some preparation both of the data to be reported against and of the report itself. In many instances, this preparation is entirely transparent, having been done before a user encounters an OLAP report. However, for developers who are charged with OLAP-enabling data and reports and for users who wish, and are authorized, to OLAP-enable their personal reports, the following summary will be useful.

OLAP-Enabling Data

Behind the scenes of any DB2 Web Query OLAP report is a hierarchical data structure. For example, a typical hierarchy of sales regions might contain a GEOGRAPHY category including the fields (in descending order) Region, State, and City. Region, the highest level in this hierarchy, would contain a list of all available regions within GEOGRAPHY. State, the second highest level in the hierarchy, would contain a list of all available states within those regions, and so on.

In DB2 Web Query, the hierarchical structure is generally built into the Master File for a data source, where it becomes active for any report that uses that data source. Developers or administrators who are responsible for describing data in a Master File can use the underlying language. The keyword WITHIN defines the elements in each dimension in the hierarchy.

In addition, those working in Developer's Workbench have access to a variety of graphical tools that make it easy to drag and drop fields into position to form a hierarchy. The hierarchy may be global to all procedures or local to one procedure. To define a:

- Global hierarchy in a Master File for use with multiple procedures, use the Dimension Builder.

- Local hierarchy as a component of a particular procedure, use the Dimension tool. The hierarchy you define with this tool does not affect the source Master File.

OLAP-Enabling a Report

In addition to using OLAP-enabled data, a report must be enabled to support OLAP analysis. OLAP-enabling a report consists of specifying how a user will interact with and drill down on OLAP data.

The primary interactions occur in the report itself. In addition, you can choose to expose two supplementary tools, the OLAP Selections pane and the Control Panel.

Setting OLAP Reporting Options

OLAP Interface Options

Users who are creating their own reports can OLAP-enable them and control the OLAP interfaces and following drill-down options.

- **OFF.** Provides an OLAP button to open the OLAP Control Panel.
- **Columns only.** Turns off the OLAP Control Panel and the OLAP Selections pane, but allows OLAP functionality from the report itself. You can access options on right-click menus, drag and drop columns within the report, and use up and down arrows to sort columns from high to low or vice versa. This is the default.
- **Columns with panel.** Provides access to the OLAP Selections pane from a square button to the left of the column titles.
- **Show filters on top.** Opens the OLAP Selections pane above the report. The Measures, Graph, and Dimension controls, as well as the band containing the OLAP, Run, and Reset buttons appear above the report output. You can open the Control Panel by clicking the OLAP button on the Selection pane.
- **Show filters on bottom.** Opens the OLAP Selections pane below the report. The Measures, Graph, and Dimension controls, as well as the band containing the OLAP, Run, and Reset buttons appear below the report output. You can open the Control Panel by clicking the OLAP button on the Selection pane.
- **Show Panel in Report.** Opens the OLAP report with the OLAP Selections pane hidden. You can perform a variety of analytic tasks from the report itself. Selection Criteria is shown next to the OLAP button.
- **Show Tabbed.** For OLAP reports that have multiple dimensions, this option groups the dimension elements under a tab labeled with the dimension name.

Drill Down options

These options enable you to sort instantly from high to low or low to high for selected report columns:

- **None**

  Disables automatic drill downs.
- **Dimensions**

  Enables automatic drill downs on dimensions in both reports and graphs.
- **Dimensions and Measures**

  Enables automatic drill downs on dimensions in both reports and graphs and, also, on measures in reports.

Note: Explicit drill downs in a StyleSheet (if they exist) take precedence over OLAP-enabled hyperlinks. If you click a hyperlink associated with an explicit drill-down, the behavior will be defined by the StyleSheet rather than by the AutoDrill On or All settings.

OLAP Terminology

The following table describes OLAP terms that may be useful as you work in the DB2 Web Query OLAP tools. Some of these terms are directly reflected in the interfaces of the OLAP Selections pane and the OLAP Control Panel. Others provide useful background information.

The first column of the following table provides the term and the second column provides the definition.

Term	Definition
Dimension	Group or list of related elements, usually structured in a hierarchy. For example, a Location dimension could include the elements Country, Region, State, and City arranged in a hierarchy where Country is the top level and City is the base level. Dimensional data usually describes the measured item.
Hierarchy	Logical parent-child structure of elements within a dimension.
Measure	Type of item that specifies the **quantity** of another element with which it is associated. A measure typically defines how much or how many. For example, Units, Revenue, and Gross Margin are measures in the Account dimension and specify how many units were sold, how much revenue was generated, and at what profit margin, respectively.
Pivot	Manipulating (or rotating) the view of a report by moving a field (or a group of fields) from a column to a row, or vice versa.

Characteristics of an OLAP Report

An OLAP-enabled report has a number of features that distinguish it from other DB2 Web Query reports.

A basic OLAP report is shown in the following image.

■ QUARTER	■ Store Name:	■ PRODTYPE	⬍ Quantity:	⬍ Line Cost Of Goods Sold
Q1	AV VideoTown	Analog	18,449	3,969,296.00
		Digital	22,206	5,109,400.00
	Audio Expert	Analog	78,449	16,467,146.00
		Digital	105,983	25,092,678.00
	City Video	Analog	6,287	1,315,015.00
		Digital	7,196	1,607,513.00
	Consumer Merchandise	Analog	6,980	1,542,036.00
		Digital	14,957	3,251,090.00
	TV City	Analog	19,077	3,772,119.00
		Digital	41,307	10,128,967.00
	Web Sales	Analog	545	124,366.00
		Digital	829	190,201.00
	eMart	Analog	97,128	21,152,262.00
		Digital	108,221	24,990,368.00
Q2	AV VideoTown	Analog	11,781	2,663,655.00
		Digital	27,377	5,928,507.00
	Audio Expert	Analog	57,944	11,868,758.00

Every OLAP user can take advantage of the analytic features that are built into the OLAP report:

- **Hyperlinks.** The values in an OLAP report are usually hyperlinks from which you can drill down to related information.

  Depending on your OLAP settings, the hyperlinks may be active for both the dimension fields (by which the report is sorted) and the measures fields (which display quantitative data), or only for the dimension fields. For related information, see OLAP-Enabling a Report.

- **Context menus.** You can right-click any column title to access a menu of options that facilitate analysis. The options vary slightly to suit the tasks associated with dimensions and measures.

- **Sorting diamonds** ⬍ . The measures (fields that make up the body of the report) have blue diamonds adjacent to them. You can click either the top or bottom of the diamond to instantly sort data from high to low or low to high.

- **Drag and drop capabilities for dimensions and measures.**
  - You can drag and drop sort fields to shift sorting from vertical (By) to horizontal (Across) or vice versa.
  - You can change the order in which sorting occurs by dragging sort fields from inner to outer positions (and vice versa).
  - You can drag measures from one position to another to affect the order in which data appears.

Beyond the features in the report itself, your OLAP options depend on the interface and drill down settings that are in effect for a particular report. Those choices determine whether you have access to the following tools:

- **Selections Pane.** When this tool is available, a pane may appear above, as shown in the following image, or below your report. For details, see Selections Pane.

- **OLAP Control Panel.** When this tool is available, the blue squares adjacent to the sort fields (By or Across) in the report become active. You can click a square or the **OLAP** button to open the Control Panel, as shown in the following image. For details, see OLAP Control Panel.
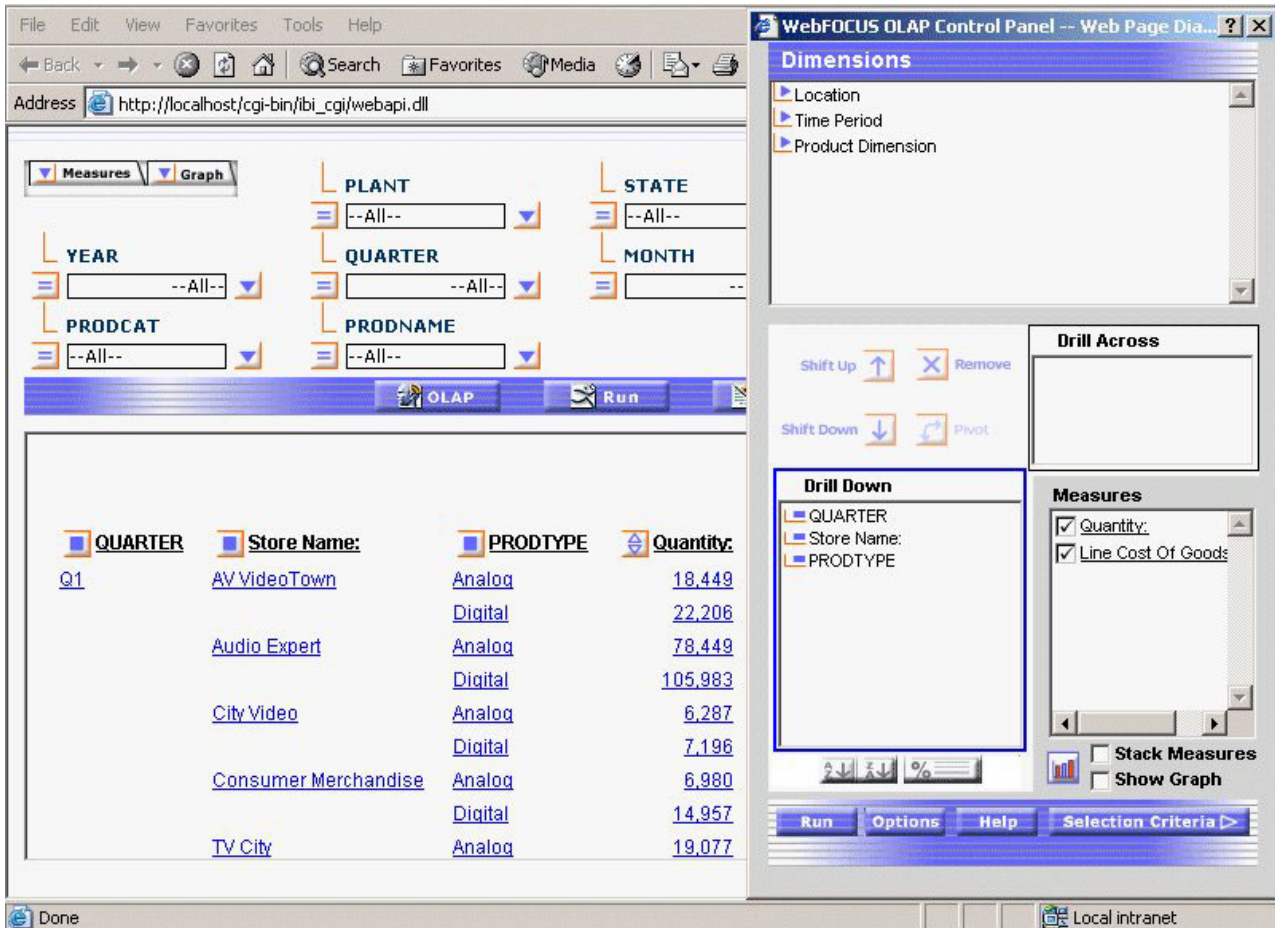
Three Ways of Working With OLAP Data

There are three ways to work with OLAP data: from the report itself, from the Selections pane, and from the Control Panel. This documentation is organized to help you understand what you can do from each location and which method is most suitable and efficient for your particular OLAP settings.

The Report

You can perform a wide range of basic analytic functions from the report itself. Changes you make in the report are implemented instantly. Every OLAP user can perform these tasks:

- Sort the data in measures in either ascending (lowest value to highest) or descending order (highest value to lowest).
- Drill down on measures, dimensions, or both (depending on the settings described in Setting OLAP Reporting Options).
- Hide fields in the current report.
- View hidden fields in the dimensions hierarchy and add them to the report.
- Change a vertical (By) sort field to a horizontal (Across) sort field and vice versa.
- Delete sort fields.

- Add a column of small bar graphs that help you visualize trends in numeric data (measures).
- Display a graphical representation of your data in a frame above the tabular report.

For an illustration of report-powered OLAP analysis, see We Do It Every Day: Typical Web Query.

Selections Pane

When the OLAP Selections pane is turned on, you can quickly limit the data in the report by selecting specific values for the dimensions in the hierarchy. A drop-down list is available for each dimension. You can multi-select values from one or more dimension lists to refine your report output.

If you wish to add a dimension element to the report you can drag it from the Selections pane into the report frame. (The cursor changes to a + sign to indicate an acceptable location.)

Each dimension has a relational operator button located to its left. This button toggles through a selection of basic numeric operators that enable you to quickly define your selection criteria. The operators are: Equal to, Not equal to, Less than or equal to, Less than but not equal to, Greater than or equal to, Greater than but not equal to. For details, see Selection Criteria Relational Operators.

The following image shows the Equal to operator as the selection for each dimension in the Selection Pane.

The name of the dimension field appears as defined in the Master File, even if an alternate column title has been specified.

In addition, you can customize the display of the measures in your report from the Selections pane. You can click either the Measures or the Graphs arrow in the upper left corner of the pane to list the measures.

- From the Measures arrow, you can display or hide the selected measure(s) or request a column of simple bar graphs to reveal trends.
- From the Graphs arrow, you can choose the measure(s) you wish to graph and specify one of seven basic graph types: vertical and horizontal bar, line, area charts and pie charts.

Note that the Selections pane is resizable; the controls for dimensions, measures, and graphs float as you resize the report window so that they continue to be visible in the frame.

Three buttons appear below the Selections pane: OLAP, Run, and Reset.
- **OLAP** opens the OLAP Control Panel (OCP).
- **Run** executes the report with the current set of selections.
- **Reset** resets all the controls in the report to their previous state (that is, before the current set of selections was made and after the last execution of the report).

OLAP Control Panel

From the Control Panel you can perform every analytic function available to a DB2 Web Query OLAP user as shown in the following image.
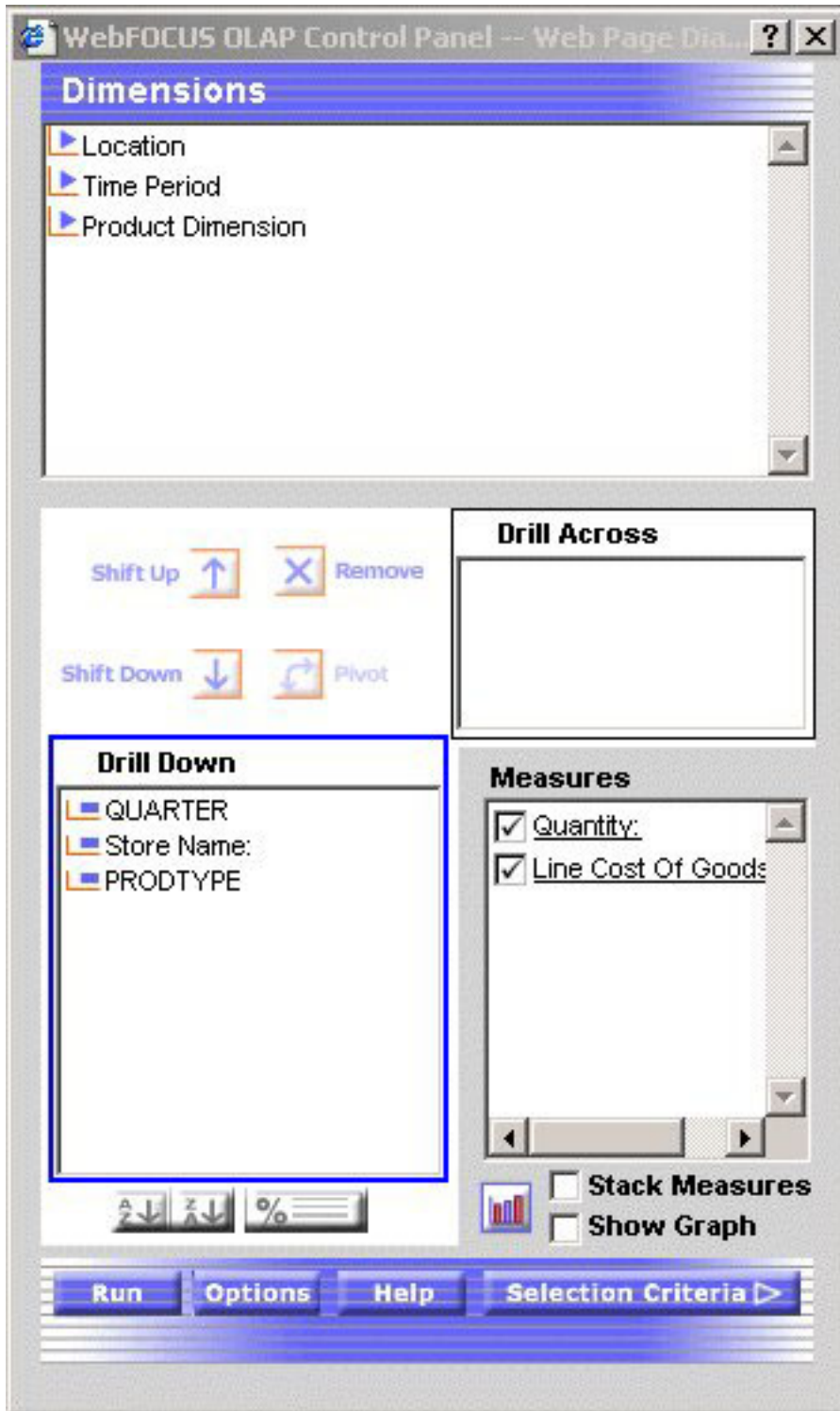
The main window of the Control Panel contains the following components:

- **Dimensions box** reflects the hierarchical structure of the data source being used by the current report (for example, the Location dimension contains the Region, State, and City fields; the Region is made up of several States, and each State contains several Cities). You click the arrow to the left of a dimension name to view the elements that comprise it. (The fields shown here are also listed in the Selections pane.)

- **Drill Down and Drill Across boxes** list the fields being used to sort the report. You can 'pivot' a Drill Down field to a Drill Across field and vice versa, and shift their positions in the report. (You can also accomplish these tasks by dragging fields within the report.)
- **Measures Properties box** contains the body of your report (usually numeric fields). You can change the display mode of a measure by clicking the check box next to the measure; the options are display, hide, and show a column of associated bar graphs. (This is equivalent to the options available from the Measures control in the Selections pane.)

Although the most frequently used functions are available directly from an OLAP report and/or from the Selections pane, several can only be performed from the Control Panel.

Unique Control Panel operations include:
- Sorting options for dimensions: from highest to lowest and vice versa (A>Z or Z>A); restricting sort field values to a specified number of either highest or lowest values; assigning a rank number to each row in the report. For details, see Sorting Data.
- Options for grouping numeric data by tile (for example, percentile, decile, quartile, etc.). For details, see Grouping Numeric Data Into Tiles.
- Defining selection criteria based on omitted or existing characters, dates, and range specifications. For details, see Limiting Data.
- Saving OLAP output in PDF and Excel formats. In Managed Reporting, users can also save OLAP output in the My Reports folder. For details, see Saving and Displaying OLAP Reports and Graphs in Other Formats.
- Stacking multiple measures to limit the width of the report. For details, see Stacking Measures.

Drilling Down On Dimensions and Measures

You can drill down on dimensions in OLAP reports and graphs and, also, on measures in reports. The settings activate the required hyperlinks:
- **Dimensions** enables automatic drill downs on dimensions in reports and graphs.
- **Dimensions and Measures** enables automatic drill downs on dimensions in both reports and graphs and on measures in reports.
- **None** disables automatic drill downs. This is the default.

In Developer's Workbench, you can set drill down options from the Report Options Format tab. For details about this setting, see Setting OLAP Reporting Options.

Drilling Down on Dimensions in a Report

What result can you expect when you drill down on a dimension?

It is easiest to discern the pattern by looking at a concrete example.

This report you are about to run uses data from a hierarchy that contains three dimensions, each of which has three elements. The report is sorted by the specified field from each dimension. The following table outlines three dimensions, Time Period, Location and Product Dimension to which each contains three elements.

Time Period	Location	Product Dimension
Year	Manufacturing Plant	**PRODTYPE**
Quarter	State	PRODCAT
Month	**Store Name**	PRODNAME

The report will show data at different levels in each dimension: Quarter is down one level in its dimension; Store Name is at the lowest level in its dimension; PRODTYPE is the top level in its dimension. This determines how much farther you can drill down within each dimension. If you drill down on a value of Quarter, the report shows information broken down by Month within that Quarter. The Quarter column itself will no longer appear.

1. Run the Standard Report **OLAPREP2**.

   In this quarterly report, drill down hyperlinks are active for both dimensions and measures.

2. Click **Q1** in the quarterly report to see a monthly report as shown in the following image.

QUARTER	Store Name:	PRODTYPE	Quantity:	Line Cost Of Goods Sold
Q1	AV VideoTown	Analog	18,449	3,969,296.00
		Digital	22,206	5,109,400.00
	Audio Expert	Analog	78,449	16,467,146.00
		Digital	105,983	25,092,678.00
	City Video	Analog	6,287	1,315,015.00
		Digital	7,196	1,607,513.00
	Consumer Merchandise	Analog	6,980	1,542,036.00
		Digital	14,957	3,251,090.00
	TV City	Analog	19,077	3,772,119.00
		Digital	41,307	10,128,967.00
	Web Sales	Analog	545	124,366.00
		Digital	829	190,201.00
	eMart	Analog	97,128	21,152,262.00
		Digital	108,221	24,990,368.00
Q2	AV VideoTown	Analog	11,781	2,663,655.00
		Digital	27,377	5,928,507.00
	Audio Expert	Analog	57,944	11,868,758.00
		Digital	111,421	28,064,250.00
	City Video	Analog	1,405	285,323.00

The monthly report looks like this. Since Month is the bottom level in its dimension, if you drill down on a month value, you will no longer see the month column; however, you will see the data that relates to the selected month in subsequent columns.

3. Click the **01** in the monthly report to see details for January as shown in the following image.

MONTH	Store Name:	PRODTYPE	Quantity:	Line Cost Of Goods Sold
01	AV VideoTown	Analog	147	35,280.00
		Digital	1,426	299,504.00
	Audio Expert	Analog	11,061	2,281,228.00
		Digital	14,062	3,432,741.00
	City Video	Analog	1,097	199,968.00
		Digital	1,382	339,594.00
	Consumer Merchandise	Analog	1,801	369,868.00
		Digital	3,580	800,913.00
	TV City	Analog	3,257	683,014.00
		Digital	6,281	1,470,194.00
	Web Sales	Analog	86	18,889.00
		Digital	136	33,877.00
	eMart	Analog	6,407	1,282,935.00
		Digital	10,737	2,632,355.00
02	AV VideoTown	Analog	9,124	1,982,103.00
		Digital	10,823	2,331,172.00
	Audio Expert	Analog	34,413	7,254,037.00

As shown in the following image, the January report shows product type, quantity, and sales for each store.

Store Name:	PRODTYPE	Quantity:	Line Cost Of Goods Sold
AV VideoTown	Analog	147	35,280.00
	Digital	1,426	299,504.00
Audio Expert	Analog	11,061	2,281,228.00
	Digital	14,062	3,432,741.00
City Video	Analog	1,097	199,968.00
	Digital	1,382	339,594.00
Consumer Merchandise	Analog	1,801	369,868.00
	Digital	3,580	800,913.00
TV City	Analog	3,257	683,014.00
	Digital	6,281	1,470,194.00
Web Sales	Analog	86	18,889.00
	Digital	136	33,877.00
eMart	Analog	6,407	1,282,935.00
	Digital	10,737	2,632,355.00

Next, see what happens when you drill down in the Location dimension—in this case, on a value of Store Name in the **second** column of the report. When you drill down on a dimension column other than the first, output is affected to the right and left of that column.

4. Click the browser's **Back** button to return to the monthly report, then click **AV VideoTown** in the second column.

Since Store Name is the lowest level in its dimension, the Store Name column no longer appears, nor does the Time Period column to its left. Nevertheless, both the store name (AV VideoTown) and the current time period (January) set the context for the information you see, which now consists of types of products sold, quantity sold, and line cost of sold goods for AV VideoTown in January as shown in the following image.

PRODTYPE	Quantity:	Line Cost Of Goods Sold
Analog	147	35,280.00
Digital	1,426	299,504.00

Drill Down on Measures in Reports

By drilling down on a measure, you expose the next level of detailed information associated with that measure for each displayed dimension in the hierarchy. In other words, when you drill down on a measure, the current dimension is used as a limiting criterion; the rest of the hierarchy is then expanded based on that limitation.

Remember that a measure contains quantitative information about fields in each dimension.

In this example, Quantity and Line of Sold Goods provide data about products at particular stores during particular time periods.

1. Run the Standard Report **OLAPREP2**.

   Notice that quantity of sales for all digital products at AV VideoTown in the first quarter of the year is 22,206. You want to find out how much each digital product contributed to the total quantity.

2. Click **22,206** under Quantity as shown in the following image.

QUARTER	Store Name:	PRODTYPE	Quantity:	Line Cost Of Goods Sold
Q1	AV VideoTown	Analog	18,449	3,969,296.00
		Digital	22,206	5,109,400.00
	Audio Expert	Analog	78,449	16,467,146.00
		Digital	105,983	25,092,678.00
	City Video	Analog	6,287	1,315,015.00
		Digital	7,196	1,607,513.00
	Consumer Merchandise	Analog	6,980	1,542,036.00
		Digital	14,957	3,251,090.00
	TV City	Analog	19,077	3,772,119.00
		Digital	41,307	10,128,967.00
	Web Sales	Analog	545	124,366.00
		Digital	829	190,201.00
	eMart	Analog	97,128	21,152,262.00
		Digital	108,221	24,990,368.00
Q2	AV VideoTown	Analog	11,781	2,663,655.00

As shown in the following image, the report now shows total quantity for digital products sold at AV VideoTown broken out by month, product category, and product name. Notice that Store Name no longer appears; since it is the lowest level of the Location dimension, there is no lower level of detail.

MONTH	PRODCAT	Product Name:	Quantity:	Line Cost Of Goods Sold
01	DVD	Combo Player - 4 Hd VCR + DVD	147	42,483.00
		DVD Upgrade Unit for Cent. VCR	147	20,433.00
	Digital Tape Recorders	R5 Micro Digital Tape Recorder	566	39,054.00
	PDA Devices	ZT Digital PDA - Commercial	566	197,534.00
02	CD Players	QX Portable CD Player	2,161	213,939.00
	Camcorders	650DL Digital Camcorder 150 X	545	386,950.00
	DVD	Combo Player - 4 Hd VCR + DVD	624	180,336.00
		DVD Upgrade Unit for Cent. VCR	621	86,319.00
	Digital Tape Recorders	R5 Micro Digital Tape Recorder	2,815	194,235.00
	PDA Devices	ZC Digital PDA - Standard	1,465	364,785.00
		ZT Digital PDA - Commercial	2,592	904,608.00
03	CD Players	QX Portable CD Player	538	53,262.00
	Camcorders	650DL Digital Camcorder 150 X	1,101	781,710.00
	DVD	Combo Player - 4 Hd VCR + DVD	1,063	307,207.00
		DVD Upgrade Unit for Cent. VCR	589	81,871.00

Since all relevant information is now visible, no further drill downs are possible and the measure is no longer represented as a hyperlink.

Next, verify this behavior at another level in the hierarchy.

3. Click the **Back** button in your browser to return to the original report.

4. Click **Q1** to see the monthly breakdown for that quarter.

5. Click **AV Video Town**. You are now looking at types of products sold, quantity sold, and line cost of goods sold at AV Video Town.

6. Drill down on **1, 426** under Quantity as shown in the following image.

PRODTYPE	Quantity:	Line Cost Of Goods Sold
Analog	147	35,280.00
Digital	1,426	299,504.00

PRODTYPE **Digital** serves as the limiting criterion. Therefore, the expanded hierarchy shows the next level of detail for each digital product. This level is comprised of digital product categories (PRODCAT) and the names of the products in each category (Product Names).

The report displays, as shown in the following image, the detailed data for each element in the PRODTYPE dimension—in this case, the product categories and product names that comprise the quantity figure of 1,426. The total quantity and, correspondingly, the line cost of goods data, is now broken down by product.

PRODCAT	Product Name:	Quantity:	Line Cost Of Goods Sold
DVD	Combo Player - 4 Hd VCR + DVD	147	42,483.00
	DVD Upgrade Unit for Cent. VCR	147	20,433.00
Digital Tape Recorders	R5 Micro Digital Tape Recorder	566	39,054.00
PDA Devices	ZT Digital PDA - Commercial	566	197,534.00

Note that when you drill down on a measure value, results may differ depending on the combination of sort fields in the report. The examples that follow show several variations.

Drilling Down on a Measure in a Report With ACROSS Fields

When you drill down on a measure in a report with at least one dimension Across field and no By fields, all Across fields are removed from the report and all of the dimension elements under the removed Across fields become By fields from left to right in the resulting report. (This convention ensures that the maximum number of Across values supported by DB2 Web Query is not exceeded.)

The values that appear for the new By fields are controlled by internally generated selection criteria. The measure values in the resulting report depend on the values of the new By fields.

1. Run the Standard Report **OLAPREP3**.

   In the report, RISK_CLASS and Continent are dimension Across fields on which you can drill down.

2. Click the Balance value **671,290** under RISK_CLASS Low and Continent AMERICAS as shown in the following image.

The report now looks like the following image.

In the new report, the RISK_CLASS and Continent fields are removed based on
two internally generated criteria: IF RISK_CLASS EQ 'Low' and IF Continent
EQ 'AMERICAS.'

The only dimension element under RISK_CLASS is Risk_Factor; the dimension elements under Continent are Region and Country. These become By fields in the new report, from left to right. The data displayed for the measures in the resulting report are those that satisfy the values in the current By fields.

Drill Down on a Measure When BY/ACROSS Fields Are Under the Same Dimension

When you drill down on a measure in a report with at least one By and one Across dimension field under the same root dimension, both the By and Across fields are hidden and the subordinate element(s) in the same dimension becomes By fields in the new report. In effect, the report is filtered based on the values of the dimensions. As a result, the sorting controlled by both hidden and visible dimensions remains in effect.

1. Run the Standard Report **OLAPREP4**.

   In the report, Continent is a By field and Region is an Across field; both are in the Geographic Area dimension.

2. Click the CANADA_DOLLAR value of **56,280,934** in the Continent row for AMERCAS under the Region CENTRAL AMERICA as shown in the following image.

The report now looks like the following image.

The new report, data is filtered based on the internally generated criteria: IF Continent EQ 'AMERICAS' and IF REGION EQ 'CENTRAL AMERICA.' (Continent and Region are no longer visible.)

REGION is replaced by the last element in the Geographic Area dimension, Country, which becomes the controlling By field in the report. The data displayed for the measures are those that satisfy the values in the current By field.

Drill Down on a Measure When BY/ACROSS Fields Are Under Different Root Dimensions

When you drill down on a measure in a report with at least one By and one Across dimension field from different root dimensions, the By fields are broken down to their last dimension level, then the Across fields are broken down.

The original By and Across fields are removed. The dimension elements under the removed By fields become the first set of By fields from left to right. The dimension elements under the removed Across fields follow the first set of By fields from left to right.

1. Run the Standard Report **OLAPREP5**.

   In the report, Continent is a By field from the Geographic Area dimension and Risk Class is an Across field from the Risk dimension.

2. Click the CANADA_DOLLAR value of **67,021,020** in the Continent row for EUROPE under the RISK_CLASS High as shown in the following image.

The report looks like the following image.

In the new report, the Continent and RISK_CLASS fields are removed based on the internally generated criteria: IF CONTINENT EQ 'EUROPE' and IF RISK_CLASS EQ 'High.'

The By field (Continent) is broken down to its last dimension element. Then, the Across field (RISK_CLASS) is broken down to its last dimension level. The

resulting By fields in the report, from left to right, are Region, Country, and Risk Factor. The data displayed for the measures satisfy the values in the current By fields.

Sorting Data

You can sort the data in an OLAP report based on the values of dimensions in the hierarchy and/or the values of the quantitative measures that constitute the body of the report. Sorting options vary depending on the nature of the data being sorted. For details, see Sorting Measures and Sorting Dimensions.

You can also group numeric data into any number of tiles (percentiles, quartiles, deciles, etc.). See Grouping Numeric Data Into Tiles.

Sorting Measures

You can apply aggregation and sorting simultaneously to a numeric measure in an OLAP report, and sort the data from high to low (descending order) or from low to high (ascending order). All other columns are sorted correspondingly.

For the measure being sorted, you can restrict the report to a specified number of highest values (when sorting high to low) or lowest values (when sorting from low to high).

When you sort a measure, any subtotals, subheadings, or subfootings in the report are automatically suppressed since these elements relate to a specific sort field and are not meaningful when the report is resorted by the values in a measure column. For an illustration, see "Applying a Percent Calculation to a Measure" on page 221.

Note: Sorting by measures is not available in a report in which measures have been stacked. See Hiding and Displaying Measures.

Sort Measures High to Low/Low to High in an OLAP Report

To sort the values of a measure from high to low:

- Click the top half of the diamond button.

  or
- Right-click the measure and select **Sort By Highest** from the menu.

The report runs automatically. The highest value is now first in the column. The top of the diamond button becomes solid blue to indicate the current sort direction.

To sort the values of a measure from low to high:

- Click the bottom half of the diamond button.

  or
- Right-click the measure and select **Sort By Lowest** from the menu.

The lowest value is first in the column. The bottom of the diamond button becomes solid blue.

Tip: After a measure has been sorted, clicking the upper or lower half of the diamond button inverts the sort order of that measure. Place your mouse over

either half of the diamond to see a message that indicates the next sort order that will occur if you click that half of the diamond.

Sorting a Measure From High-to-Low in the Report

The following is an example of sorting a measure from high-to-low in the report.

1. Run the Standard Report **OLAPREP2**.

   The OLAP report shows sales information sorted by quarter, store, and product type.

   You are interested in seeing where the greatest quantity of goods have been sold.

2. Click the top half of the diamond button next to the Quantity measure, as shown in the following image, to sort the values from high to low.

QUARTER	Store Name:	PRODTYPE	Quantity:	Line Cost Of Goods Sold
Q1	AV VideoTown	Analog	18,449	3,969,296.00
		Digital	22,206	5,109,400.00
	Audio Expert	Analog	78,449	16,467,146.00
		Digital	105,983	25,092,678.00
	City Video	Analog	6,287	1,315,015.00
		Digital	7,196	1,607,513.00
	Consumer Merchandise	Analog	6,980	1,542,036.00
		Digital	14,957	3,251,090.00
	TV City	Analog	19,077	3,772,119.00
		Digital	41,307	10,128,967.00
	Web Sales	Analog	545	124,366.00
		Digital	829	190,201.00
	eMart	Analog	97,128	21,152,262.00
		Digital	108,221	24,990,368.00
Q2	AV VideoTown	Analog	11,781	2,663,655.00
		Digital	27,377	5,928,507.00
	Audio Expert	Analog	57,944	11,868,758.00
		Digital	111,421	28,064,250.00
	City Video	Analog	1,405	285,323.00
		Digital	8,835	2,025,508.00

As shown in the following image, the report now shows data values for the Quantity measure in descending order. The top half of the diamond next to Quantity is blue and solid to indicate the current sort order of the measure. This is now the controlling sort in the report; all other values are reordered correspondingly.

QUARTER	Store Name:	PRODTYPE	Quantity:	Line Cost Of Goods Sold
Q2	eMart	Digital	115,102	24,971,512.00
Q2	Audio Expert	Digital	111,421	28,064,250.00
Q1	eMart	Digital	108,221	24,990,368.00
Q1	Audio Expert	Digital	105,983	25,092,678.00
Q1	eMart	Analog	97,128	21,152,262.00
Q1	Audio Expert	Analog	78,449	16,467,146.00
Q2	eMart	Analog	74,737	16,789,403.00
Q4	eMart	Digital	72,126	14,000,951.00
Q3	eMart	Digital	66,156	13,867,709.00
Q2	Audio Expert	Analog	57,944	11,868,758.00
Q4	Audio Expert	Digital	53,275	11,190,923.00
Q3	Audio Expert	Digital	50,076	11,210,406.00
Q1	TV City	Digital	41,307	10,128,967.00
Q4	eMart	Analog	39,515	9,383,389.00
Q3	eMart	Analog	36,306	8,308,647.00
Q2	TV City	Digital	29,627	6,732,303.00
Q2	AV VideoTown	Digital	27,377	5,928,507.00
Q4	Audio Expert	Analog	25,897	5,916,936.00

Tip: To invert the sort order, click either the solid or hollow part of the diamond button.

Sort Measures High to Low/Low to High From the Control Panel

1. Open the OLAP Control Panel.
2. Click a measure in the Measures box to open the sort options pane (do not click the Measures check box which controls the display of a measure, not its sorting).

   Verify that the Sort box is checked (this setting is required to apply sorting specifications to the selected measure).
3. Select the **High-to-Low** or **Low-to-High** options button to specify the sort order you wish to apply. The default sort order is high to low.
4. Click **OK**.

   The sort pane is replaced by the Measures box, where the measure becomes blue to indicate that sorting specifications have been defined.
5. Click **Run** to display the report with sorting applied to the selected measure.

   The diamond button next to the sorted measure changes to reflect the sort order: if High to Low, the top half of the diamond is solid blue; if Low to High, the bottom half is solid blue.

Note:
- Report execution is automatic when you sort a measure in an OLAP report. However, if the Control Panel is open, all current changes in the Control Panel are applied.
- If an OLAP request contains a horizontal (Across) sort field, the measures appear several times in the report, once for each Across value. If you apply

sorting to a measure, the sort is performed on the first column occurrence of the measure, and reflected in all subsequent instances. The appropriate half of the diamond button becomes solid only for the first instance. Any additional sorting you wish to perform must be done from the first occurrence of the measure.

View a Subset of Data for Sorted Measures

You can select to view only a subset of the total number of records in your report.

1. Open the OLAP Control Panel.
2. Click a measure name in the Measures box to open the sort options pane (do not click the Measures check box which controls the display of a measure, not its sorting).

   Verify that the **Sort** check box is selected (this setting is required to apply sorting specifications to a measure).
3. Select the **Rank** check box, then specify the number of sort field values to be included in the report. (Notice that **Highest** or **Lowest** appears to the left of the input box to reflect the current sort order.)

   - Use the spin buttons located to the right of the word Highest or Lowest to increase or decrease the number of sort fields.

     or

   - Position the cursor in the input box and type a number.

   The default number of sort fields values is 5.
4. Click **OK**.

   The sort pane is replaced by the Measures box, where the measure becomes blue to indicate that sorting specifications have been defined.
5. Click **Run** to display the report with the designated number of sorted values.

Displaying a Subset of Sorted Data for a Measure

The following is an example of displaying a subset of sorted data for a measure.

1. Run the Standard Report **OLAPREP2**.

   The report shows sales information sorted by quarter, store, and product type.
2. Click the blue square next to QUARTER to open the Control Panel (notice that the original report is open on the left).
3. Click **Quantity** in the Measures box.

   The sort pane opens as shown in the following image on top of the report.

4. If not already selected, click the **Sort** check box.

   High to Low sorting is selected by default.

5. Click the **Rank** check box.

   Because the report is being sorted from high to low, you can indicate the number of values you wish to see, beginning with the highest.

6. Specify Highest **4**.

7. Click **OK**.

   The main Control Panel window appears. In the Measures box the Quantity measure is blue to indicate that sorting specifications have been defined.

8. Click the **Run** button at the bottom of the Control Panel.

   As shown in the following image, the report now shows Quantity sorted from high to low with the highest four values appearing.

QUARTER	Store Name:	PRODTYPE	Quantity:	Line Cost Of Goods Sold
Q2	eMart	Digital	115,102	24,971,512.00
Q2	Audio Expert	Digital	111,421	28,064,250.00
Q1	eMart	Digital	108,221	24,990,368.00
Q1	Audio Expert	Digital	105,983	25,092,678.00

Remove Sorting Criteria for a Measure

You can remove sorting specifications for a measure whether the measure is appears or hidden.

1. Open the OLAP Control Panel.
2. In the Measures box, click the measure for which you want to remove sorting specifications.
3. Clear the **Sort** check box.
4. Click **OK**.

Sorting Dimensions

There are several ways in which you can sort dimensions in an OLAP hierarchy. You can:

- Control the order in which data is sorted: ascending or descending.
- Restrict sort field values to a specified number of either highest or lowest values.
- Assign a rank number to each row in a vertically sorted report.
- Shift the positions of sort fields in the report. For example, you can change from sorting by State and then by Product to sorting by Product and then by State.
- Pivot a vertical (By) sort field to make it a horizontal (Across) sort field and vice versa.
- Hide a sort field in the report while retaining the sorting associated with it. For example, you can sort data by quarters without showing the Quarter column.
- Group numeric data in tiles (for example, percentile, decile, etc.).

Change Sort Order for a Dimension

1. Open the Control Panel.
2. Select a field from the Drill Down or Drill Across box.
3. Click the **Sort** button.

   The sort pane opens.
4. Under Sort Order, choose the **Low to High** or **High to Low** options button (Low to High is the default for a dimension).
5. Click **OK**.

   The main Control Panel window reopens.
6. Click **Run** to execute the report.

Inverting the Sort Order of a Dimension

The following is an example of inverting the sort order of a dimension.

1. Run the Standard Report **OLAPREP4**.

In the report, the values of both sort fields—Continent and Region—are sorted from low to high (A to Z) as shown in the following image.

2. To sort the report in reverse alphabetical order, click the **OLAP** button on the band below the Selections pane to open the Control Panel.

3. Select **Continent** in the Drill Down box and click the **Sort** button. The sort pane opens.

4. Under Sort Order, choose the **High to Low** options button as shown in the following image on the OLAP Control Panel.
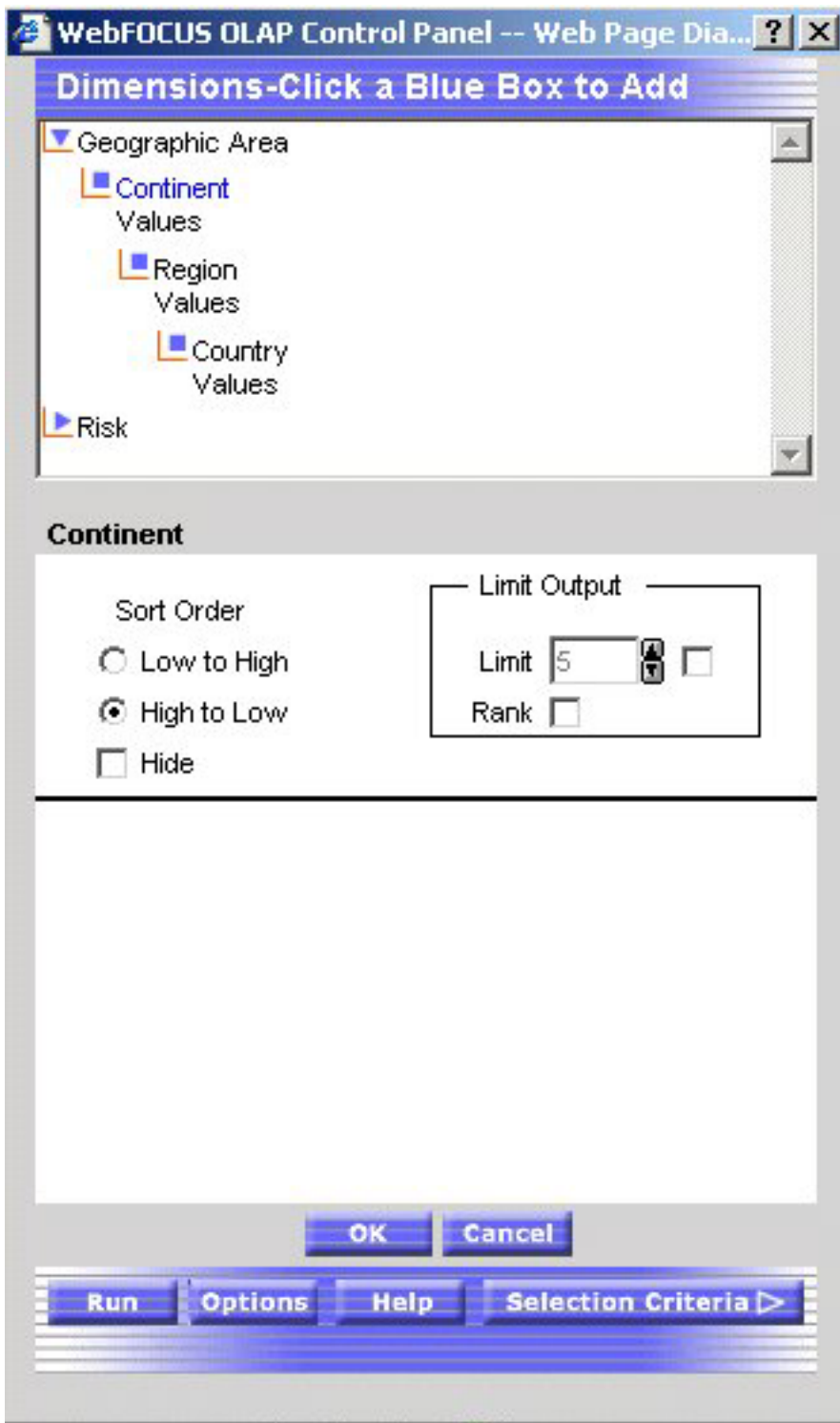
5. Click **OK**.

   The main Control Panel window reopens.

6. Repeat the process for Region: select **Region** in the Drill Across box and click the **Sort** button. When the sort pane opens, select the **High to Low** options button and click **OK**.

   The main Control Panel window opens.

7. Click the **Run** button.

Both dimensions are now sorted in inverse alphabetical order (Z to A) as shown in the following image.

	Region					
	WESTERN EUROPE		SOUTH AMERICA		NORTH AMERICA	
Continent	Balance	CANADA_DOLLAR	Balance	CANADA_DOLLAR	Balance	CANADA
EUROPE	210,550,374	294,770,524	.	.	.	
ASIA	.	.	.	.	.	
AMERICAS	.	.	87,661,381	122,725,933	367,610,683	51

Restrict the Display of Sort Values

To restrict the display of sort field values to a certain number of highest or lowest values:

1. Open the OLAP Control Panel.
2. Select a field from the Drill Down box.
3. Click the **Sort** button.

   The sorting pane opens.
4. Under Sort Order, choose the **Low to High** or **High to Low** options button as shown in the following image on the OLAP Control Panel.
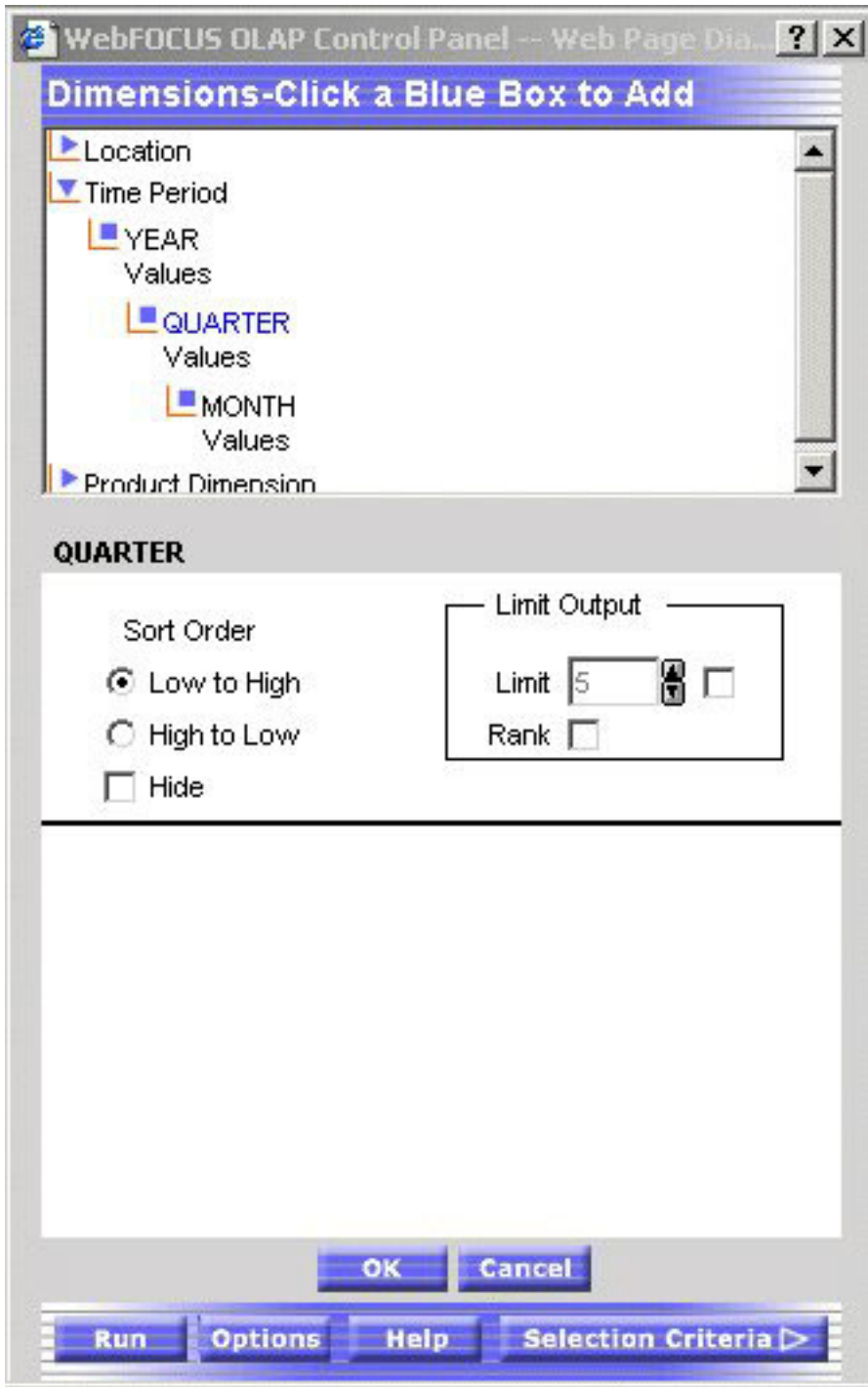
5. Under Limit Output, click the **Limit** check box and choose or type a value in the input area.

6. Click **OK**.

   The main Control Panel window reopens.

7. Click **Run** to execute your report.

Rank Rows in a Vertically Sorted Report

1. Open the OLAP Control Panel.

2. Select a field from the Drill Down box.

3. Click the **Sort** button.

   The sort pane opens.

4. Under Sort Order, choose the **Low to High** or **High to Low** options button.

5. Click the **Rank** check box.

6. If you wish to place a restriction on the number of sort field values to rank, click the Limit check box, choose or type a value in the input area.

   • If the **High to Low** option button is selected, you can rank a specified number of Highest values.

   • If the **Low to High** option button is selected, you can rank a specified number of Lowest values.

7. Click **OK**.

   The main Control Panel window reopens.

8. Click **Run** to execute your report.

Ranking and Restricting the Number of Sort Values

The following is an example of ranking and restricting the number of sort values.

1. Run the Standard Report **OLAPREP2**.

   Information for all stores is shown for each quarter. You want to see quarterly information for only the first two stores in alphabetical order (low to high).

2. Click the blue square next to QUARTER to open the Control Panel (notice that the original report is open at the left).

3. Choose **Store Name** in the Drill Down box and click the **Sort** button.

   The sort pane opens.

   The following image shows these three selections on the OLAP Control Panel.

a. Accept the default sort order: **Low to High**.

b. Click the **Limit** check box and choose **2** from the input area.

c. Click the **Rank** check box.

4. Click **OK** to return to the main Control Panel window.

5. Click the **Run** button at the bottom of the Control Panel.

   Notice that only two values now appear for each Quarter and ranked low to high within each group as shown in the following image.

QUARTER	RANK	Store Name:	PRODTYPE	Quantity:	Line Cost Of Goods Sold
Q1	1	AV VideoTown	Analog	18,449	3,969,296.00
			Digital	22,206	5,109,400.00
	2	Audio Expert	Analog	78,449	16,467,146.00
			Digital	105,983	25,092,678.00
Q2	1	AV VideoTown	Analog	11,781	2,663,655.00
			Digital	27,377	5,928,507.00
	2	Audio Expert	Analog	57,944	11,868,758.00
			Digital	111,421	28,064,250.00
Q3	1	AV VideoTown	Analog	7,700	1,792,498.00
			Digital	17,379	3,625,972.00
	2	Audio Expert	Analog	19,508	4,216,289.00
			Digital	50,076	11,210,406.00
Q4	1	AV VideoTown	Analog	7,761	1,844,696.00
			Digital	21,915	4,553,762.00
	2	Audio Expert	Analog	25,897	5,916,936.00
			Digital	53,275	11,190,923.00

Reposition Sort Fields in an OLAP Report

You can change the order in which data is sorted and presented in the report. For example, you can change from sorting by State and then by Product to sorting by Product and then by State.) If you want to reposition:

- Vertical (By) sort fields, drag and drop a field into a new column position.
- Horizontal (Across) sort fields, drag and drop the lower field above the higher one or vice versa.

In each case, the cursor changes to a plus (+) sign to indicate acceptable places into which you can drop the field. Unacceptable positions are shown by a circle with a slash across the center.

Repositioning Sort Fields in an OLAP Report

The following is an example of repositioning sort fields in an OLAP report.

1. Run the Standard Report **OLAPREP2**.
2. Click the top half of the diamond button next to **Quantity** to sort values from high to low.

   The dimension values adjust accordingly. The report now shows the Quantity values from high to low but according to the QUARTER sort order as shown in the following image.

QUARTER	Store Name:	PRODTYPE	Quantity:	Line Cost Of Goods Sold
Q2	eMart	Digital	115,102	24,971,512.00
Q2	Audio Expert	Digital	111,421	28,064,250.00
Q1	eMart	Digital	108,221	24,990,368.00
Q1	Audio Expert	Digital	105,983	25,092,678.00

You would like to change the sort order in the report, making Store Name the first sort field, followed by PRODTYPE and QUARTER.

3. Drag **QUARTER** after PRODTYPE.

The cursor changes to a plus (+) sign to indicate acceptable places into which you can drop the field.

The report changes immediately as shown in the following image with the Store Name being the first sort order.

Store Name:	PRODTYPE	QUARTER	Quantity:	Line Cost Of Goods Sold
eMart	Digital	Q2	115,102	24,971,512.00
Audio Expert	Digital	Q2	111,421	28,064,250.00
eMart	Digital	Q1	108,221	24,990,368.00
Audio Expert	Digital	Q1	105,983	25,092,678.00

Reposition Sort Fields From the Control Panel

1. Open the OLAP Control Panel.
2. Select a field in the Drill Down or Drill Across box.
3. Click the **Shift Up** or **Shift Down** arrow until the field is in the desired position.

   Repeat for other fields as needed.
4. Click **Run** to execute your report.

Repositioning Sort Fields From the Control Panel

The following is an example of repositioning sort fields from the Control Panel.

1. Run the Standard Report **OLAPREP2**.
2. Click the top half of the diamond button next to **Quantity** to sort values from high to low.

   The dimension values adjust accordingly. The report now shows the Quantity values from high to low but according to the QUARTER sort order as shown in the following image.

	QUARTER		Store Name:		PRODTYPE		Quantity:		Line Cost Of Goods Sold
	Q2		eMart		Digital		115,102		24,971,512.00
	Q2		Audio Expert		Digital		111,421		28,064,250.00
	Q1		eMart		Digital		108,221		24,990,368.00
	Q1		Audio Expert		Digital		105,983		25,092,678.00

You would like to change the sort order in the report, making Store Name the first sort field, followed by PRODTYPE and QUARTER.

3. Click the blue square button next to QUARTER to open the Control Panel.

4. Select **Quarter** from the Drill Down box.

5. Click the **Shift Down** arrow twice.

   QUARTER is now the third item in the Drill Down list as shown in the following image.

6. Click the **Run** button at the bottom of the Control Panel.

   QUARTER appears in the third column of the report as shown in the following image.

Store Name:	PRODTYPE	QUARTER	Quantity:	Line Cost Of Goods Sold
eMart	Digital	Q2	115,102	24,971,512.00
Audio Expert	Digital	Q2	111,421	28,064,250.00
eMart	Digital	Q1	108,221	24,990,368.00
Audio Expert	Digital	Q1	105,983	25,092,678.00

Pivot Rows and Columns In an OLAP Report

You can quickly change a field from one that sorts data vertically, creating rows, to one that sorts data horizontally, creating columns, or vice versa. To change a:

- Vertical (By) sort field to a horizontal (Across) sort field, drag and drop a field above the row of column titles.
- Horizontal (Across) sort field to a vertical (By) sort field, drag and drop the field into the desired location in the row of column titles.

In each case, the cursor changes to a plus (+) sign to indicate acceptable places into which you can drop the field. (Unacceptable positions are shown by a circle with a slash across the center.)

Pivoting Rows and Columns in a Report

The following is an example of pivoting rows and columns in a report.

1. Run the Standard Report **OLAPREP2**.
2. Click **Q1**.

   The report is now sorted vertically, by month, store, and product type as shown in the following image.

MONTH	Store Name:	PRODTYPE	Quantity:	Line Cost Of Goods Sold
01	AV VideoTown	Analog	147	35,280.00
		Digital	1,426	299,504.00
	Audio Expert	Analog	11,061	2,281,228.00
		Digital	14,062	3,432,741.00
	City Video	Analog	1,097	199,968.00
		Digital	1,382	339,594.00
	Consumer Merchandise	Analog	1,801	369,868.00
		Digital	3,580	800,913.00
	TV City	Analog	3,257	683,014.00
		Digital	6,281	1,470,194.00
	Web Sales	Analog	86	18,889.00
		Digital	136	33,877.00
	eMart	Analog	6,407	1,282,935.00
		Digital	10,737	2,632,355.00
02	AV VideoTown	Analog	9,124	1,982,103.00
		Digital	10,823	2,331,172.00
	Audio Expert	Analog	34,413	7,254,037.00
		Digital	39,905	9,674,622.00
	City Video	Analog	576	145,184.00
		Digital	805	136,405.00

You want to create a matrix in which data is sorted horizontally by month, and vertically by store and product type.

3. Drag **Month** above the report to sort data horizontally (Across).

The cursor changes to a plus (+) sign to indicate acceptable places into which you can drop the field.

In the new report, Quantity and Line Cost of Goods Sold are repeated horizontally for each month as shown in the following image.

Store Name:	PRODTYPE	MONTH				
		01		02		03
		Quantity:	Line Cost Of Goods Sold	Quantity:	Line Cost Of Goods Sold	
AV VideoTown	Analog	147	35,280.00	9,124	1,982,103.00	
	Digital	1,426	299,504.00	10,823	2,331,172.00	
Audio Expert	Analog	11,061	2,281,228.00	34,413	7,254,037.00	
	Digital	14,062	3,432,741.00	39,905	9,674,622.00	
City Video	Analog	1,097	199,968.00	576	145,184.00	
	Digital	1,382	339,594.00	805	136,405.00	
Consumer Merchandise	Analog	1,801	369,868.00	2,835	639,082.00	
	Digital	3,580	800,913.00	3,232	766,052.00	
TV City	Analog	3,257	683,014.00	7,959	1,466,033.00	
	Digital	6,281	1,470,194.00	14,269	3,761,723.00	
Web Sales	Analog	86	18,889.00	165	37,831.00	
	Digital	136	33,877.00	204	47,392.00	
eMart	Analog	6,407	1,282,935.00	44,944	9,869,545.00	
	Digital	10,737	2,632,355.00	42,786	9,826,681.00	

Pivot Rows and Columns From the Control Panel

You can change a field from one that sorts data vertically, creating rows, to one that sorts data horizontally, creating columns, or vice versa.

1. Open the OLAP Control Panel.
2. Select the title of the row or column you want to pivot in the Drill Down or Drill Across box.
3. Click the **Pivot** button. The title appears in the new location.
4. Click **Run** to execute your report.

Pivoting Rows Into Columns From the Control Panel

The following is an example of pivoting rows into columns from the Control Panel.

1. Run the Standard Report **OLAPREP2**.
2. Click **Q1**.

   The report is now sorted vertically, by month, store, and product type as shown in the following image.

MONTH	Store Name:	PRODTYPE	Quantity:	Line Cost Of Goods Sold
01	AV VideoTown	Analog	147	35,280.00
		Digital	1,426	299,504.00
	Audio Expert	Analog	11,061	2,281,228.00
		Digital	14,062	3,432,741.00
	City Video	Analog	1,097	199,968.00
		Digital	1,382	339,594.00
	Consumer Merchandise	Analog	1,801	369,868.00
		Digital	3,580	800,913.00
	TV City	Analog	3,257	683,014.00
		Digital	6,281	1,470,194.00
	Web Sales	Analog	86	18,889.00
		Digital	136	33,877.00
	eMart	Analog	6,407	1,282,935.00
		Digital	10,737	2,632,355.00
02	AV VideoTown	Analog	9,124	1,982,103.00
		Digital	10,823	2,331,172.00
	Audio Expert	Analog	34,413	7,254,037.00
		Digital	39,905	9,674,622.00
	City Video	Analog	576	145,184.00
		Digital	805	136,405.00

You want to create a matrix in which data is sorted horizontally by month, and vertically by store and product type.

3. Click the blue square next to MONTH to open the Control Panel.

4. Select **Month** in the Drill Down box and click the **Pivot** button.

Month moves into the Drill Across box as shown in the following image.

MONTH	Store Name:	PRODTYPE	Quantity:
01	AV VideoTown	Analog	147
		Digital	1,426
	Audio Expert	Analog	11,061
		Digital	14,062
	City Video	Analog	1,097
		Digital	1,382
	Consumer Merchandise	Analog	1,801
		Digital	3,580
	TV City	Analog	3,257
		Digital	6,281
	Web Sales	Analog	86
		Digital	136
	eMart	Analog	6,407
		Digital	10,737
02	AV VideoTown	Analog	9,124
		Digital	10,823
	Audio Expert	Analog	34,413

WebFOCUS OLAP Control Panel -- Web Page Dia...

Dimensions-Click a Blue Box to Add

- ▶ Location
- ▼ Time Period
  - ■ YEAR
    - Values
      - ■ QUARTER
        - Values
          - ■ MONTH
            - Values
- ▶ Product Dimension

Shift Up ↑   ✕ Remove
Shift Down ↓   ↻ Pivot

Drill Across
■ MONTH

Drill Down
- ■ Store Name:
- ■ PRODTYPE

Measures
- ☑ Quantity:
- ☑ Line Cost Of Goods

☐ Stack Measures
☐ Show Graph

Run   Options   Help   Selection Criteria ▷

5. Click the **Run** button on the Control Panel.

In the new report, Quantity and Line Cost of Goods Sold are repeated horizontally for each month as shown in the following image.

		MONTH				
		01		02		03
Store Name:	PRODTYPE	Quantity:	Line Cost Of Goods Sold	Quantity:	Line Cost Of Goods Sold	
AV VideoTown	Analog	147	35,280.00	9,124	1,982,103.00	
	Digital	1,426	299,504.00	10,823	2,331,172.00	
Audio Expert	Analog	11,061	2,281,228.00	34,413	7,254,037.00	
	Digital	14,062	3,432,741.00	39,905	9,674,622.00	
City Video	Analog	1,097	199,968.00	576	145,184.00	
	Digital	1,382	339,594.00	805	136,405.00	
Consumer Merchandise	Analog	1,801	369,868.00	2,835	639,082.00	
	Digital	3,580	800,913.00	3,232	766,052.00	
TV City	Analog	3,257	683,014.00	7,959	1,466,033.00	
	Digital	6,281	1,470,194.00	14,269	3,761,723.00	
Web Sales	Analog	86	18,889.00	165	37,831.00	
	Digital	136	33,877.00	204	47,392.00	
eMart	Analog	6,407	1,282,935.00	44,944	9,869,545.00	
	Digital	10,737	2,632,355.00	42,786	9,826,681.00	

Sort by a Field Without Displaying the Sort Column

To use a field to sort your data, but not show the sort field as a column in the report:

1. Open the OLAP Control Panel.
2. Select a field in the Drill Down or Drill Across box.
3. Click the **Sort** button.

   The sort pane opens.
4. Under Sort Order, click the **Hide** check box.
5. Click **OK**.

   The main Control Panel window reopens.
6. Click **Run** to execute the report.

Tip: To expose the hidden sort field, repeat the process and deselect the **Hide** check box.

Sorting by a Hidden Field

The following is an example of sorting by a hidden field.

1. Run the Standard Report **OLAPREP2**.

   The first sort field in the report is QUARTER. You want to retain the sorting but not display this field.
2. Click the blue square next to QUARTER to open the Control Panel.
3. Select **QUARTER** in the Drill Down box, then click the **Sort** button.

   The sort pane opens.
4. Select the **Hide** check box as shown in the following image.
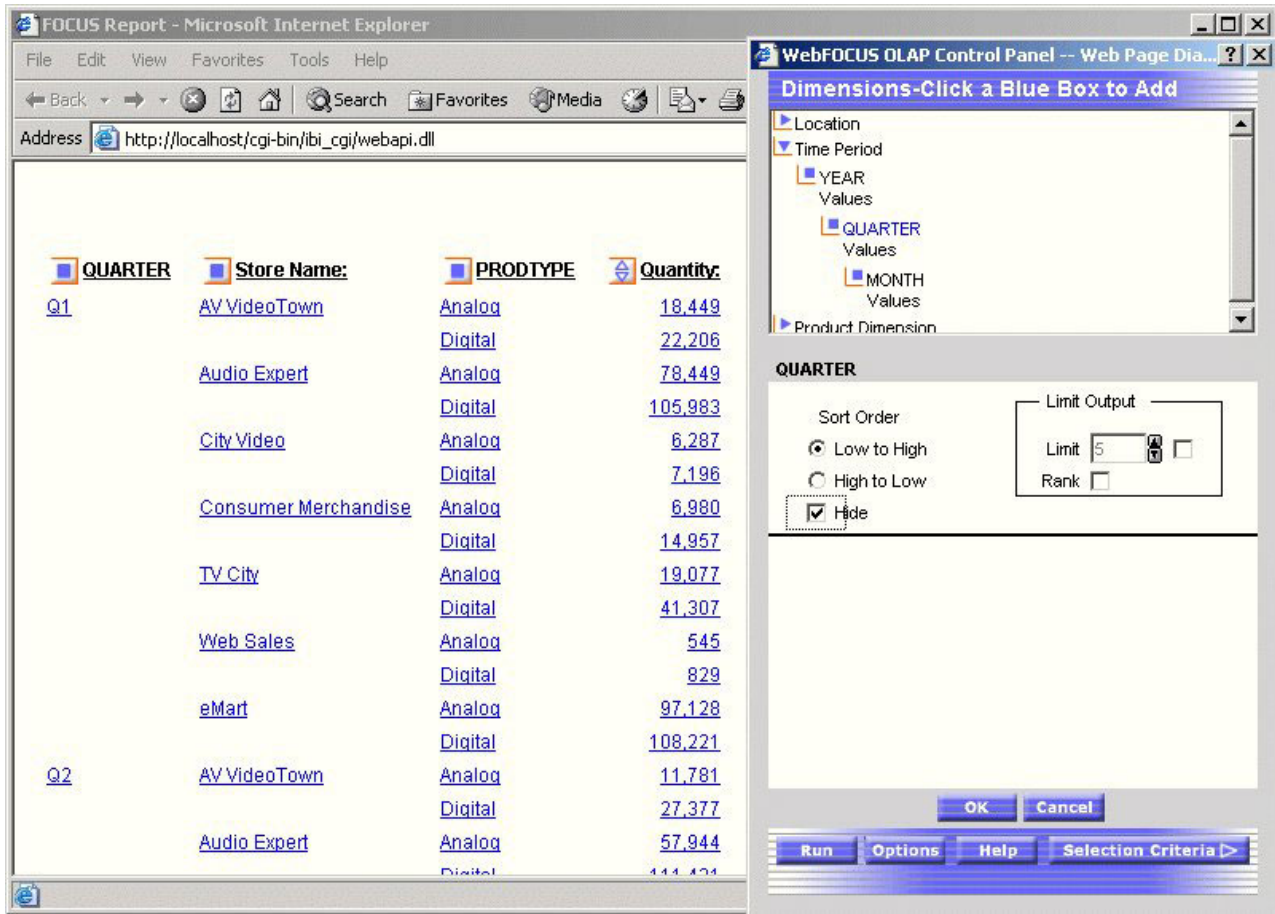
5. Click **OK**.

   The main Control Panel window reopens.

6. Click the **Run** button in the Control Panel.

   Report sorting is unchanged, but the QUARTER column no longer appears as shown in the following image.

Store Name:	PRODTYPE	Quantity:	Line Cost Of Goods Sold
AV VideoTown	Analog	18,449	3,969,296.00
	Digital	22,206	5,109,400.00
Audio Expert	Analog	78,449	16,467,146.00
	Digital	105,983	25,092,678.00
City Video	Analog	6,287	1,315,015.00
	Digital	7,196	1,607,513.00
Consumer Merchandise	Analog	6,980	1,542,036.00
	Digital	14,957	3,251,090.00
TV City	Analog	19,077	3,772,119.00
	Digital	41,307	10,128,967.00
Web Sales	Analog	545	124,366.00
	Digital	829	190,201.00
eMart	Analog	97,128	21,152,262.00
	Digital	108,221	24,990,368.00
AV VideoTown	Analog	11,781	2,663,655.00
	Digital	27,377	5,928,507.00

Grouping Numeric Data Into Tiles

You can group numeric data into any number of tiles (percentiles, deciles, quartiles, etc.) in tabular reports. For example, you can group student's test scores into deciles to determine which students are in the top ten percent of the class or determine which salesmen are in the top half of all salesmen based on total sales.

Grouping is based on the values in the selected vertical (BY) field and data is apportioned as equally as possible into the number of tile groups you specify.

The following occurs when you group data into tiles:

- A new column (labeled TILE by default) is added to the report output and displays the tile number assigned to each instance of the tile field. You can change the column title in the Tiles section of the OLAP Control Panel.
- Tiling is calculated within all of the higher-level sort fields in the request and restarts whenever a sort field at a higher level than the tile field's value changes.
- Instances are counted using the tile field. If the request displays fields from lower level segments, there may be multiple report lines that correspond to one instance of the tile field.
- Instances with the same tile field value are placed in the same tile. For example, consider the following data, which is to be apportioned into three tiles:

  1

  5

  5

  5

8

9

In this case, dividing the instances into groups containing an equal number of records produces the following table:

Group	Data Values
1	1,5
2	5,5
3	8,9

However, because all of the same data values must be in the same tile, the fives (5) that are in group 2 are moved to group 1. Group 2 remains empty. The final tiles look like the following table:

Tile Number	Data Values
1	1,5,5,5
2	
3	8,9

Group Data Into Tiles in an OLAP Report

1. Open the OLAP Control Panel.
2. Select a numeric or date field from the Drill Down box.
3. Click the **Tiles** button.

   The Tiles pane opens at the bottom of the OLAP Control Panel as shown in the following image.
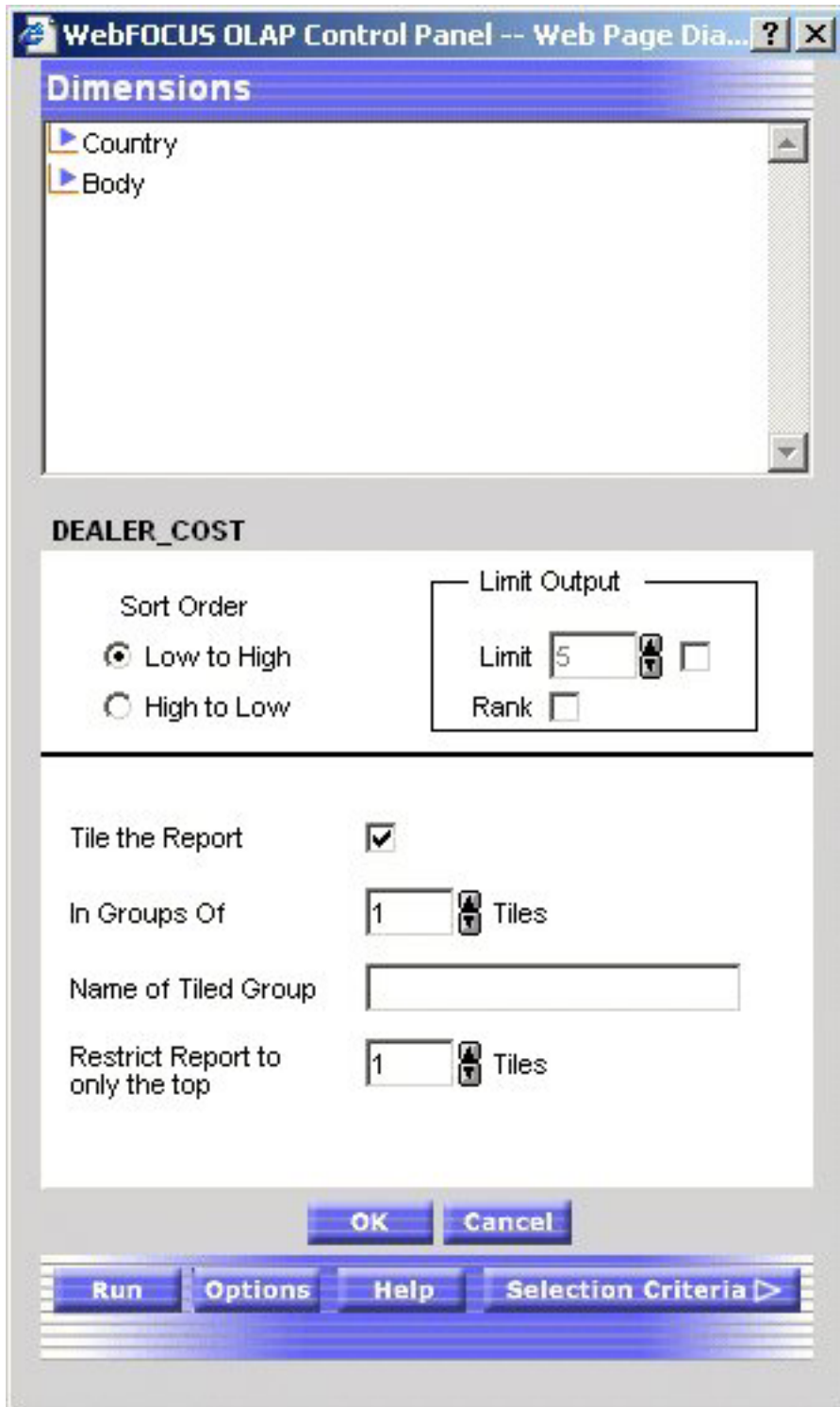
4. Click the **Tile the Report** check box.

5. In the **In Groups Of** input area, select the number of tiles to be used in grouping the data. For example, 100 tiles produces percentiles, 10 tiles produces deciles, etc.

6. In the **Name of Tile Group** input box, type a name for the Tile column.

7. In the **Restrict Report to only the Top** input area, select the number of tile groups to display in the report.

8. Optionally, select a Sort Order option button:
   - Choose **High to Low** to sort data in descending order so that the highest data values are placed in tile 1.
   - Choose **Low to High** to sort data in ascending order so that the lowest data values are placed in tile 1. This is the default.
9. If you wish to specify the highest tile value to appear in the report, select a value from the Limit input area. For example, if you enter a Limit of 3, the report will not display any data row that is assigned a tile number greater than 3.
10. Click **OK** to accept the selections and return to the main Control Panel window.
11. Click **Run** to re-execute and view the report.

Performing a Calculation on a Measure

You can perform standard calculations, such as average, percent, and summarize, on the numeric data in measures on an OLAP report.

Apply a Calculation to a Measure

1. Run the Standard Report.
2. Open the OLAP Control Panel.
3. Click a measure in the Measures box.

   The sort options pane opens. Do not click the Measures check box, which controls the display of a measure, not its sorting.
4. Required: Click the arrow under Measure Calculations and select a calculation from the drop-down list.

   None is the default value. For details, see Calculations You Can Perform on a Measure.
5. Click **OK**.

   The sort pane is replaced by the Measures box, where the selected calculation appears as a prefix to the measure.
6. Click **Run**.

   The applied calculation is added to the column title.

Calculations You Can Perform on a Measure

The following table lists the types of calculations in the first column and describes their functions in the second column.

Calculation	Function
Average Sum of Squares	Computes the average sum of squares for standard deviation in statistical analysis.
Average	Computes the average value of the field.
Count	Counts the number of occurrences of the field.
Count Distinct	Counts the number of distinct values within a field when using -REMOTE. For other modes of operation, this behaves like Count.
Maximum	Generates the maximum value of the field.
Minimum	Generates the minimum value of the field.

Calculation	Function
Percent	Computes a field's percentage based on the total values for the field. The Percent can be used with detail as well as summary fields.
Percent Count	Computes a field's percentage based on the number of instances found.
Row	Computes a field's percentage based on the total values for the field across a row.
Summarize	Sums the number of occurrences of the field.
Total	Counts the occurrences of the field for use in a heading (includes footings, subheads, and subfoots).

Applying a Percent Calculation to a Measure

The following is an example of applying a percent calculation to a measure.

1. Run the Standard Report **OLAPREP6**.

   The report shows Quantity and Line Cost of Goods sold sorted by plant and product category, with a subtotal at each sort break.

   You want to create a report column that shows the percent of total sales for each plant.

2. Click the blue square next to PLANT to open the Control Panel as shown in the following report image.

PLANT	PRODCAT	⇕ Quantity:	⇕ Line Cost Of Goods Sold
Boston	CD Players	56,723	5,615,577.00
	Camcorders	172,592	55,400,795.00
	Cameras	21,554	1,985,822.00
	DVD	38,986	7,586,704.00
	Digital Tape Recorders	98,312	6,783,528.00
	PDA Devices	118,550	39,002,750.00
	VCRs	32,119	4,143,351.00
*TOTAL Boston		538,836	120,518,527.00
Dallas	CD Players	18,377	1,819,323.00
	Camcorders	64,712	19,836,296.00
	Cameras	6,496	686,960.00
	DVD	21,724	4,271,836.00
	Digital Tape Recorders	38,251	2,639,319.00
	PDA Devices	43,492	14,490,008.00
	VCRs	10,885	1,404,165.00
*TOTAL Dallas		203,937	45,147,907.00

3. Click **Line Cost of Goods Sold** in the Control Panel's Measures box.

   The sort pane opens.

4. Under Measures Calculations, choose **Percent** from the drop-down list as shown in the following image, then click **OK** to see the calculation as a prefix for the measure in the Measures box.
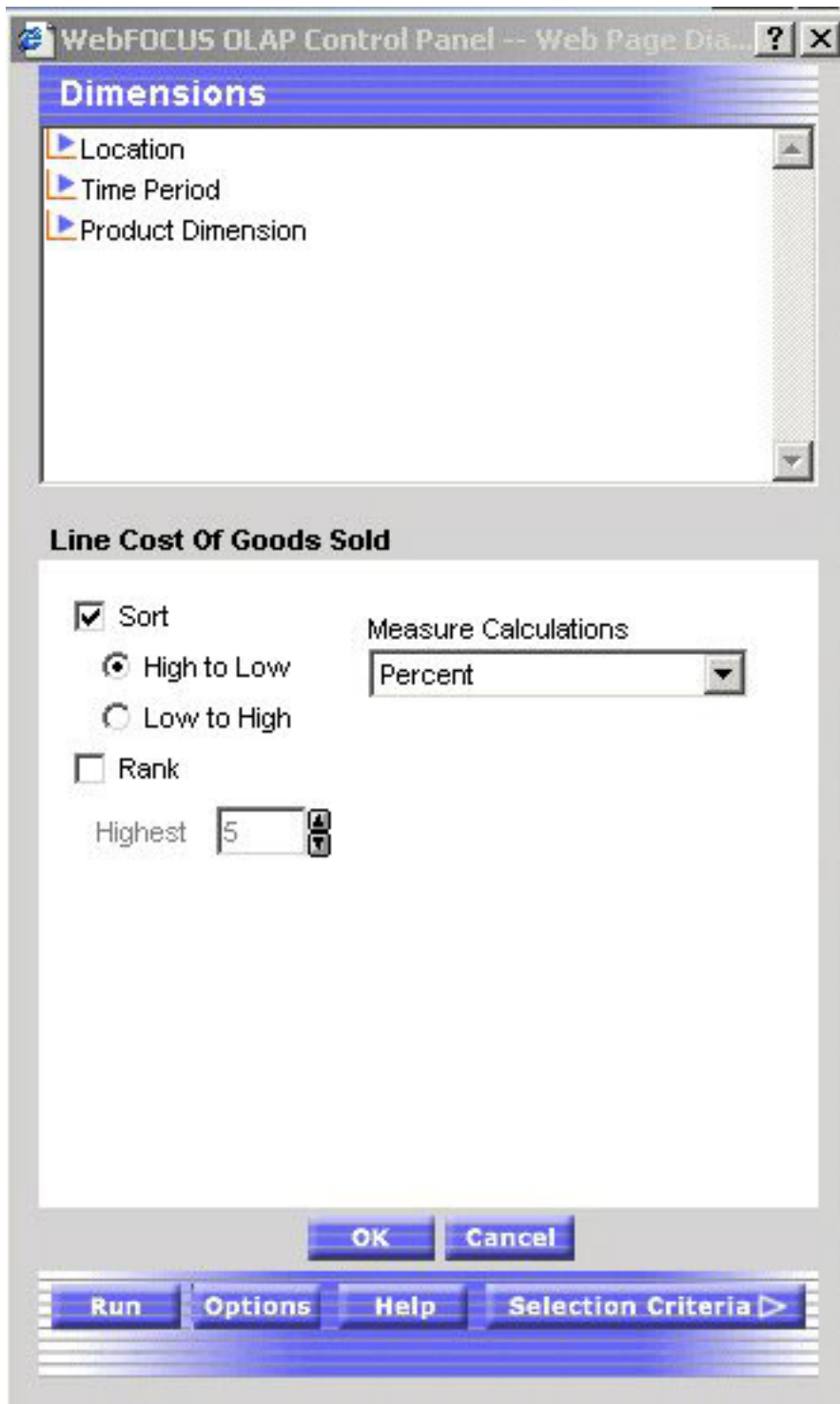
5. Click the **Run** button at the bottom of the Control Panel.

   The report now breaks down sales for each product at each plant by as a percentage of total sales as shown in the following image.

PLANT	PRODCAT	Quantity:	PCT LINE COGS
Boston	Camcorders	172,592	16.42
St Louis	Camcorders	135,629	12.55
Boston	PDA Devices	118,550	11.56
St Louis	PDA Devices	84,158	8.24
Orlando	Camcorders	69,611	6.49
Dallas	Camcorders	64,712	5.88
Dallas	PDA Devices	43,492	4.29
Orlando	PDA Devices	37,421	3.71
Los Angeles	Camcorders	39,595	3.52
Los Angeles	PDA Devices	23,980	2.41
Boston	DVD	38,986	2.25
Boston	Digital Tape Recorders	98,312	2.01
St Louis	DVD	28,667	1.69
Boston	CD Players	56,723	1.66
St Louis	Digital Tape Recorders	74,607	1.53
Seattle	Camcorders	13,968	1.44
St Louis	CD Players	43,580	1.28
Dallas	DVD	21,724	1.27
Boston	VCRs	32,119	1.23

Notice that the subtotals have been removed from the report because the breakdown by plant is no longer suitable for the data.

Limiting Data

An OLAP report is limited to values belonging to the parent categories in the dimensions hierarchy. There are several ways to further limit the data that appears in the report.

From the Selections pane or the Control Panel, you can explicitly limit the data in an OLAP report by selecting dimension values and relational operators (such =, >, <). For a list of the relational operators, see Selection Criteria Relational Operators.

- The Selections pane provides the easiest approach since you can choose both dimension values and relational operators with a few mouse clicks, while the report is fully exposed to view.

- The Control Panel offers several options that are not available from the Selections pane, including record selection based on a date or date range and on the existence or absence of specific characters.

Note that changes you make in the Selections Pane are immediately implemented in the Control Panel (even if the Control Panel is closed). However, the inverse is not true; changes you make in the Control Panel are not reflected in the Selections pane until the report is run.

From the report, you can limit data indirectly by drilling down on measures and dimensions to hone in on subset of information. For details, see Drilling Down On Dimensions and Measures.

Selection Criteria Relational Operators

You can define selection criteria using several relational operators, most of which are supported in both the Selections pane and the Control Panel. Those supported only in the Control Panel are noted in the following table which describes the operators.

The first column shows the operator and the second column describes the operator.

Equal to	Are equal to the criteria you specified. This is the default operator.
- Not Equal to	Are not equal to the criteria you specified.
- Less than or equal to	Are less than or equal to the criteria you specified.
- Less than	Are less than, but not equal to, the criteria you specified.
- Greater than or equal to	Are greater than or equal to the criteria you specified.
- Greater than	Are greater than, but not equal to, the criteria you specified.
- Contains	Contain the criteria you specified. **Note:** This operator is available only for alphanumeric fields and is only supported in the Control Panel.
- Not contain	Do not contain the criteria you specified. **Note:** This operator is available only for alphanumeric fields and is only supported in the Control Panel.

Note: You can select more than one value using the same relational operator.

The following relational operators are available for selecting a range of dates. They are only available from the Control Panel.

The following table shows the relational operators in the first column and describes them in the second column.

Within range	The value in the indicated date field falls within the specified range. Note: To use this relational operator, you must select the Range check box in the Date Selection panel.
 - Not within range	The value in the indicated date field does **not** fall within the specified range. Note: To use this relational operator, you must select the Range check box in the Date Selection panel.

Apply Selection Criteria From the Selections Pane

When the Selections pane is turned on, there is one control (drop-down list) for every dimension in the OLAP hierarchy. (Note that the name of the dimension field appears as defined in the Master File, even if an alternate column title has been specified.)

To limit data for the dimensions that are included in the report:

1. Click the arrow to the right of the dimension to open the list of values.
2. Select one or more values from the list (**All** is the default value).

   To select multiple values, click the desired values while holding the **Ctrl** key on the keyboard.
3. Select a relational operator from the button to the left of the dimension to indicate the basis for selection (equals (=) is the default).

   You can toggle through a list of operators. See Selection Criteria Relational Operators.
4. Repeat steps 1-3 for each dimension whose values you wish to limit.
5. Click the **Run** button on the band below the Selections pane.

Tip: To change or eliminate selection criteria, reopen the values list and choose another value or choose **All**.

Limiting Continents and Regions From the Selections Pane

The following is an example of limiting continents and regions from the selections pane.

1. Run the Standard Report **OLAPREP7**.

   As shown in the following image, the Selections pane above the report, the controls for Continent and Region are set to **All** to show all values of each dimension. You wish to focus on the data for one continent and one region.

2. In the Selections pane, click the arrow to the right of CONTINENT and select **AMERICAS** from the list of values. Use the default operator = to limit the data.

3. Next, click the arrow to the right of REGION and select **NORTH AMERICA**. Once again, accept the default operator =.

4. Click the **Run** button on the band below the Selections pane.

   The output is now limited to data for the selected continent and region as shown in the following image

Apply Selection Criteria From the Control Panel

1. Open the Control Panel.
2. Click the **Selection Criteria** button at the bottom right of the window.

   The Selection Criteria pane opens.
3. In the Dimensions box above the Selection Criteria pane, expand a dimension and click **Values**.

   A secondary window opens; select one or more values (press the **Ctrl** key to multi-select).
4. Click **OK** to return to the Selection Criteria pane, where the selected values appear in the drop-down lists.
   - If a Developer has applied selection criteria to the Reporting Object from which you create an OLAP report, you only see the selected acceptable values of the field.
   - If no selection criteria have been applied, you see all the values of the field in the drop-down lists.
5. In the **Selection Criteria** pane, click a relational operator next to the dimension to specify the relationship that you want to base selection on—for example, =, >, or <. For a complete list, see Selection Criteria Relational Operators.
6. Repeat the process for other dimensions whose values you wish to limit.
7. Click **Run** to execute your report.

Limiting Continents and Countries From the Control Panel

The following is an example of limiting continents and countries from the Control Panel.

Tip: If you have access to the Selections pane, it provides the quickest way to limit data. For an illustration, see "Limiting Continents and Regions From the Selections Pane" on page 226.

1. Run the Standard Report **OLAPREP8**.

   The report shows data for continents and countries. You want to restrict the information to the Countries ARGENTINA and BRAZIL in the Continent AMERICAS.
2. Click the blue square next to **Continent** to open the Control Panel as shown in the following image.

■ Continent	■ Country	■ RISK_CLASS	⬍ Balance	⬍ CANADA_DOLLAR
AMERICAS	ARGENTINA	Medium	36,921,658	51,690,321
		Unknown	5,246,222	7,344,711
	BRAZIL	Medium	29,600,684	41,440,958
		Unknown	15,892,817	22,249,944
	CANADA	Low	26,263,063	36,768,288
		Medium	29,949,571	41,929,399
	GUATEMALA	Medium	727,810	1,018,934
	HONDURAS	Medium	39,472,857	55,262,000
	MEXICO	Medium	74,781,444	104,694,022
		Unknown	8,280,274	11,592,384
	UNITED STATES	Low	159,861,394	223,805,952
		Medium	68,474,937	95,864,912
ASIA	HONG KONG	Medium	88,068,897	123,296,456
	ISRAEL	Medium	25,368,704	35,516,186
	JAPAN	Medium	114,591,945	160,428,723
	SAUDI ARABIA	Medium	19,291,067	27,007,494
EUROPE	CZECH. REP	High	42,854,041	59,995,657
		Medium	8,267,160	11,574,024
	ENGLAND	Low	10,214,318	14,300,045
		Medium	115,542,880	161,760,032

3. Click the **Selection Criteria** button at the bottom right to open the Selection Criteria pane.

4. In the Dimensions box above the Selection Criteria pane, expand the **Geographic Area** dimension and click **Values** under Country.

   A secondary window lists the acceptable values.

5. In this window, choose **ARGENTINA** and **BRAZIL** as shown in the following image (hold down the **Ctrl** key to multi-select values).
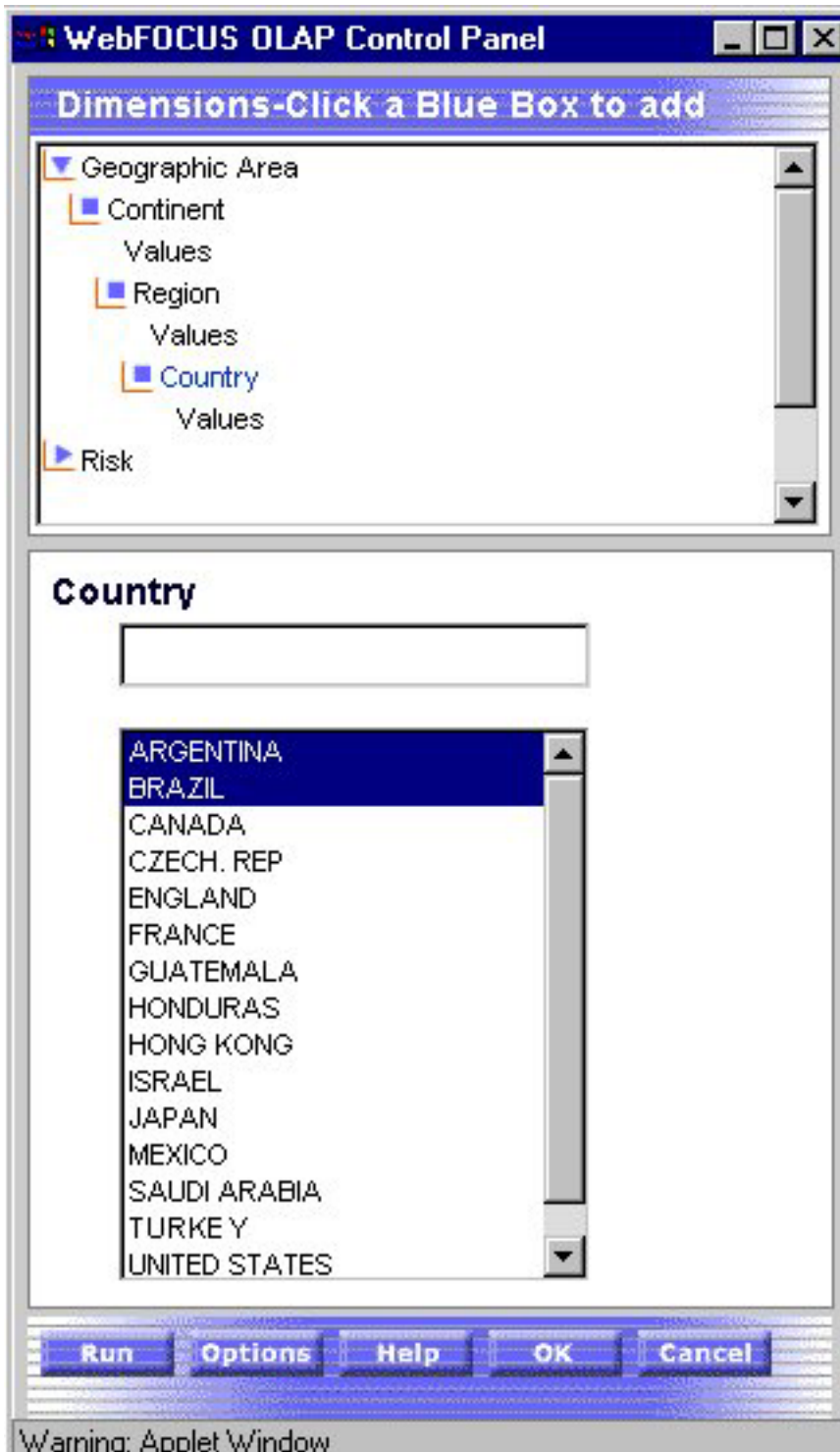
6. Click **OK** to return to the Selection Criteria pane.
7. Repeat step 3, but click **Values** under Continent and choose **AMERICAS**, then click **OK**.

   The selected values now appear in the drop-down lists in the Selection Criteria pane as shown in the following image.
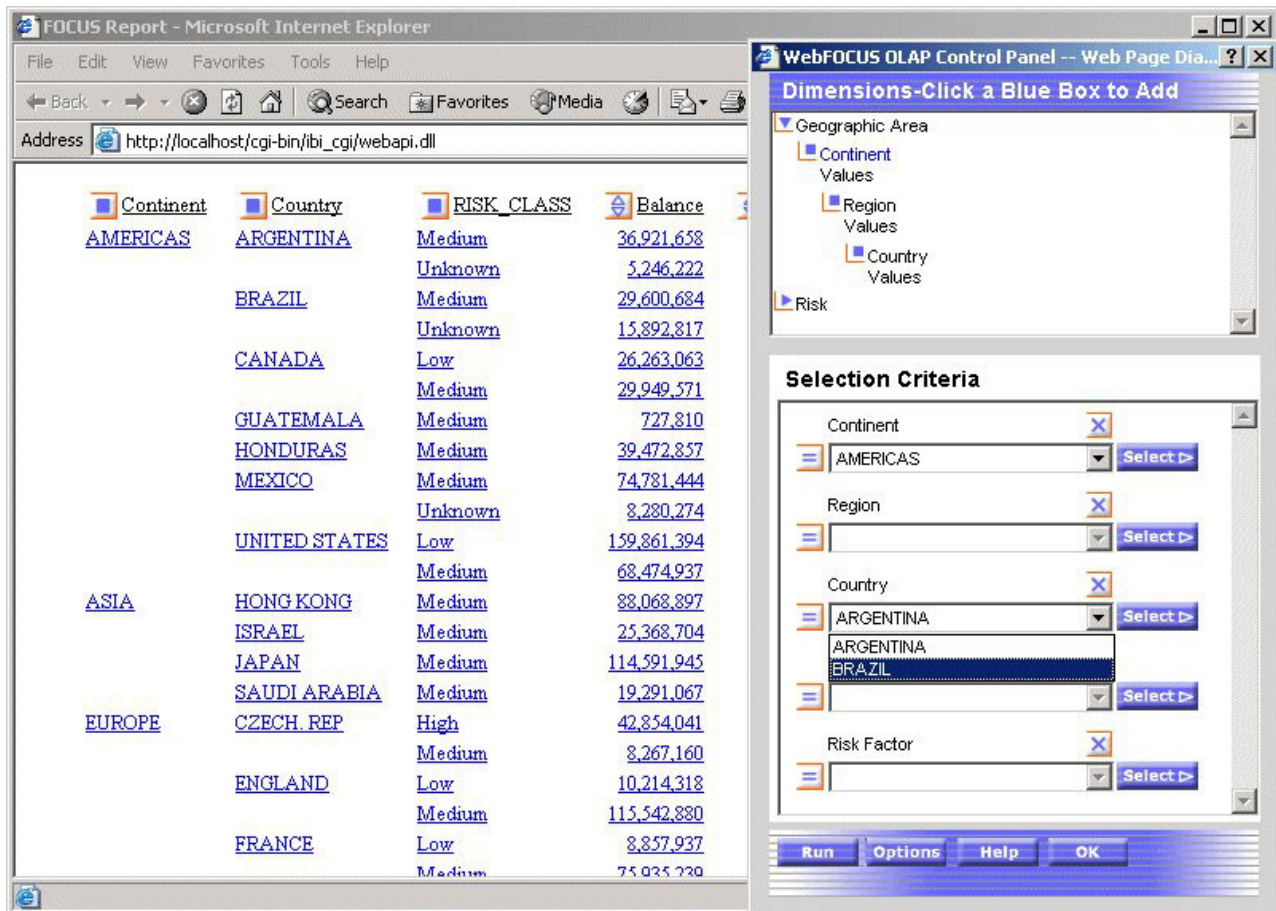
8. Verify that you want to use the default operator =, then click the **Run** button at the bottom of the Control Panel.

   The new report displays the data by Continent, Americas followed by Country as shown in the following image.

Change Selection Criteria From the Control Panel

Tip: If you have access to the Selections pane, it provides the easiest way to adjust or remove selection criteria. See Apply Selection Criteria From the Selections Pane.

From the Selections Criteria pane in the Control Panel:

1. Click the **Select** button next to the dimension value you wish to modify.

   A secondary window opens.

To change a value, type the new value in the text box or select one or more values from the list. (The value you type must be in the same case as the value in the data source.)

You can input only one value in the text box. If you select more than one value from the list, only the first value appears. However, all values appear in your report.

To deselect a value, hold down the **Ctrl** key while clicking the value.

2. Click **OK** to return to the Selection Criteria pane where you can verify the revised value and/or change the relational operator if required.

3. Click **OK** again to confirm your choice and return to the main Control Panel window.

4. Click **Run** to execute your report.

Remove Selection Criteria From the Control Panel

Tip: If you have access to the Selections pane, it provides the easiest way to adjust or remove selection criteria. See Apply Selection Criteria From the Selections Pane.

From the Selections Criteria pane in the Control Panel:

1. Select the criterion you want to remove.

2. Click the **Delete** button.

   The selection category is removed from the list.

3. Click **Run** to execute your report with all values.

Applying Selection Criteria to Date Elements

You can apply selection criteria to date elements just as you apply them to other types of elements. The results are limited by the date(s) you select. For example, you can select to view data associated with a particular date or to exclude data from the specified date.

Note: Like other dimension elements, date fields must have been defined in the Master File by a Managed Reporting developer. The Master File specifies the date formats available for selection criteria.

This feature is only supported from the Control Panel, where you choose the selection criteria from a Date selection pane that contains the appropriate controls for the date format.

You can also select a range of dates in a designated year by specifying a **From** and **To** date. Two relational operators are available for selecting a range of dates:

- The **Within range** operator displays records when the value in the indicated date field falls within the specified range.

- The **Not within range** operator displays records when the value in the indicated date field does **not** fall within the specified range.

For more information on supported date formats, see Date Format Limitations.

Apply Selection Criteria to a Date Field

1. Required: Open the Control Panel.
2. Required: Click the **Selection Criteria** button.

   The Selection Criteria pane opens.
3. In the Dimensions box above the Selection Criteria pane, expand the dimension that includes the date field, and click the **Values** button.

   A secondary window displays controls for the dimension's date format. For example, if the date format is YYM, only the year and month controls appear. If the format is YYMD, year, month, and day controls appear.

   Note: The date selection pane appears only when a supported date format is provided. See Date Format Limitations.
4. Specify a date using the spin controls, drop-down lists, or by typing the value.

   If your date format includes edit masking such as Y.M.D, the date appears with forward slashes in the Date selection list box, the Selection Criteria pane, and the drop-down list at the bottom of the report. However, the date edit mask appears as specified within the body of the report.
5. Click **Add** to display the date in the Selections list box.
6. Click **OK** to return to the Selection Criteria pane and verify the selected date.
7. In the Selection Criteria pane, click a relations button to the left of the date field (for example, =, >, or <) to indicate a basis for record selection.
8. Optionally, define additional date selection criteria by repeating steps 2-7.
9. Click **Run** to execute your report.

Applying Selection Criteria to a Date Field

The following is an example of applying selection criteria to a date file.

1. Run the Standard Report **OLAPREP9**.

   As shown in the following images, the multi-page OLAP report shows several years of data about reported problems falling into five categories: incorrect labeling, missing components, physical damage, power failure, remote failure.

	Date		
Problem ■ Category	Problem ■ Reported	Problem ⬍ Number	Problem ⬍ Occurrence
Incorrect Labeling	1998/04/05	693	1
		694	1
	1998/04/19	695	1
	1998/05/03	701	1
	1998/05/31	879	1
	1998/06/07	856	1
	1998/06/21	880	1
		614	1
	1998/07/05	859	1
	1998/07/26	824	1
		825	1
		996	1
	1998/08/02	826	1

OLAP

You want to investigate problems reported on June 6, 2001. You can limit data based on a single date from the Control Panel.

Problem Category	Date Problem Reported	Problem Number	Problem Occurrence
Mechanical Failure	2000/09/15	2885	1
		3909	1
		2742	1
		3766	1
	2000/09/25	3304	1
	2000/09/29	2616	1
		3640	1
		2747	1
		2889	1
		3771	1
		3913	1
		2615	1

OLAP

Note: To show the selection of a particular date, a dimension component has been added to the procedure. This dimension places **Date Prob. Reported** in the Time Period dimension hierarchy directly below the root.

2. Click the **OLAP** button below the report to open the Control Panel.

   The OLAP button appears at the bottom of this report because the OLAP CONTROL setting was selected. For details, see Setting OLAP Reporting Options.

3. Click the **Selection Criteria** button at the bottom of the Control Panel.

   The Selection Criteria pane opens.

4. In the Dimensions box above the Selection Criteria pane, expand the **TIMEPERIOD** hierarchy.

5. Click **Values** under Date Problem was Reported.

   The PROBLEM_DATE pane replaces the Selections Criteria pane, with a drop-down list for each selectable value (Year, Month, and Date) based on the date format of the selected field as shown in the following image.
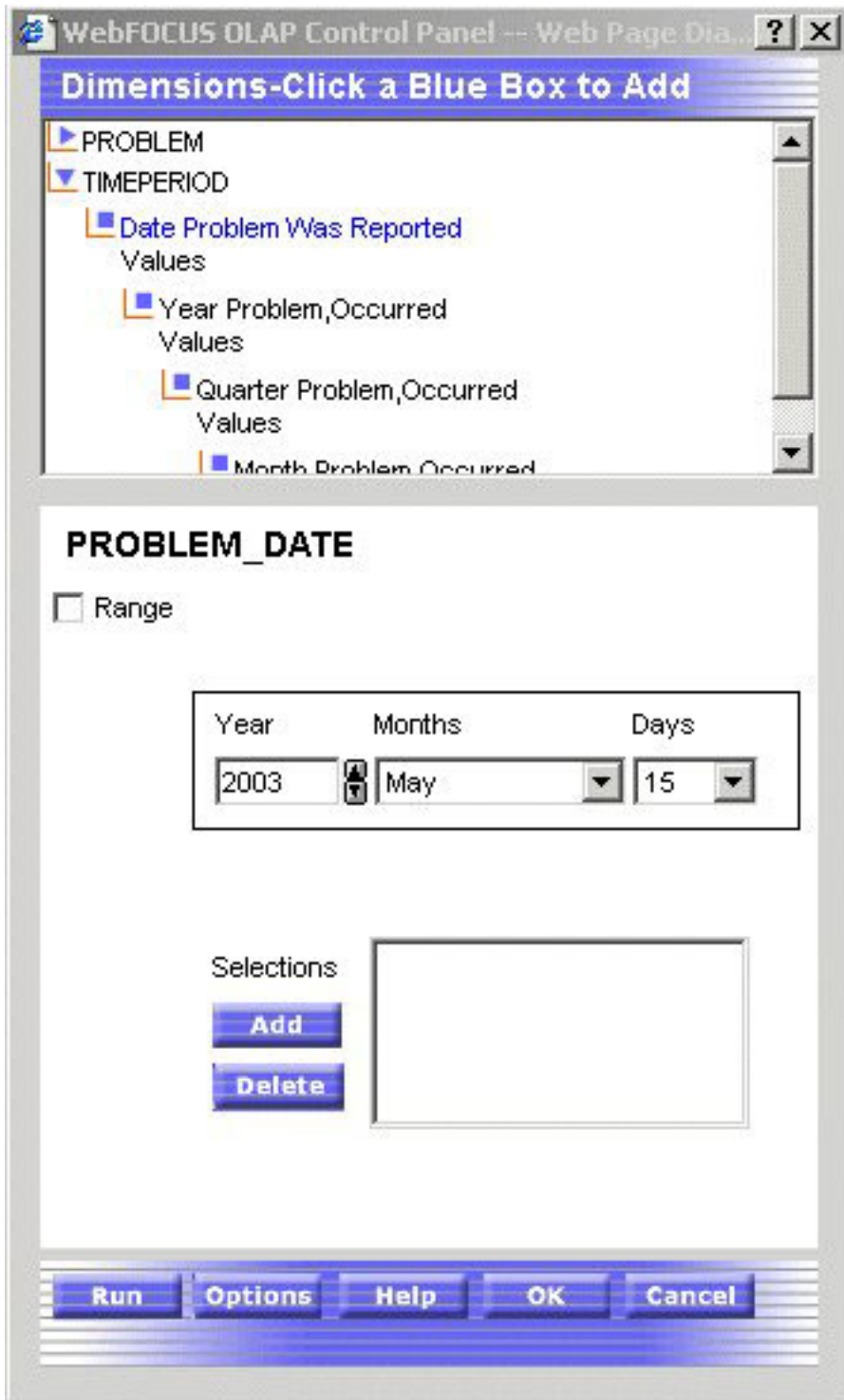
6. Select values. For example:

   a. Change the year to 2003 in the Year box by using the spin buttons or typing the value.

   b. Select May from the Months drop-down list.

   c. Select 15 from the Days drop-down list.

   d. Click **Add** to enter these criteria in the input box.

7. Click **OK** to return to the Selection Criteria pane, which now reflects your entries as shown in the following image.
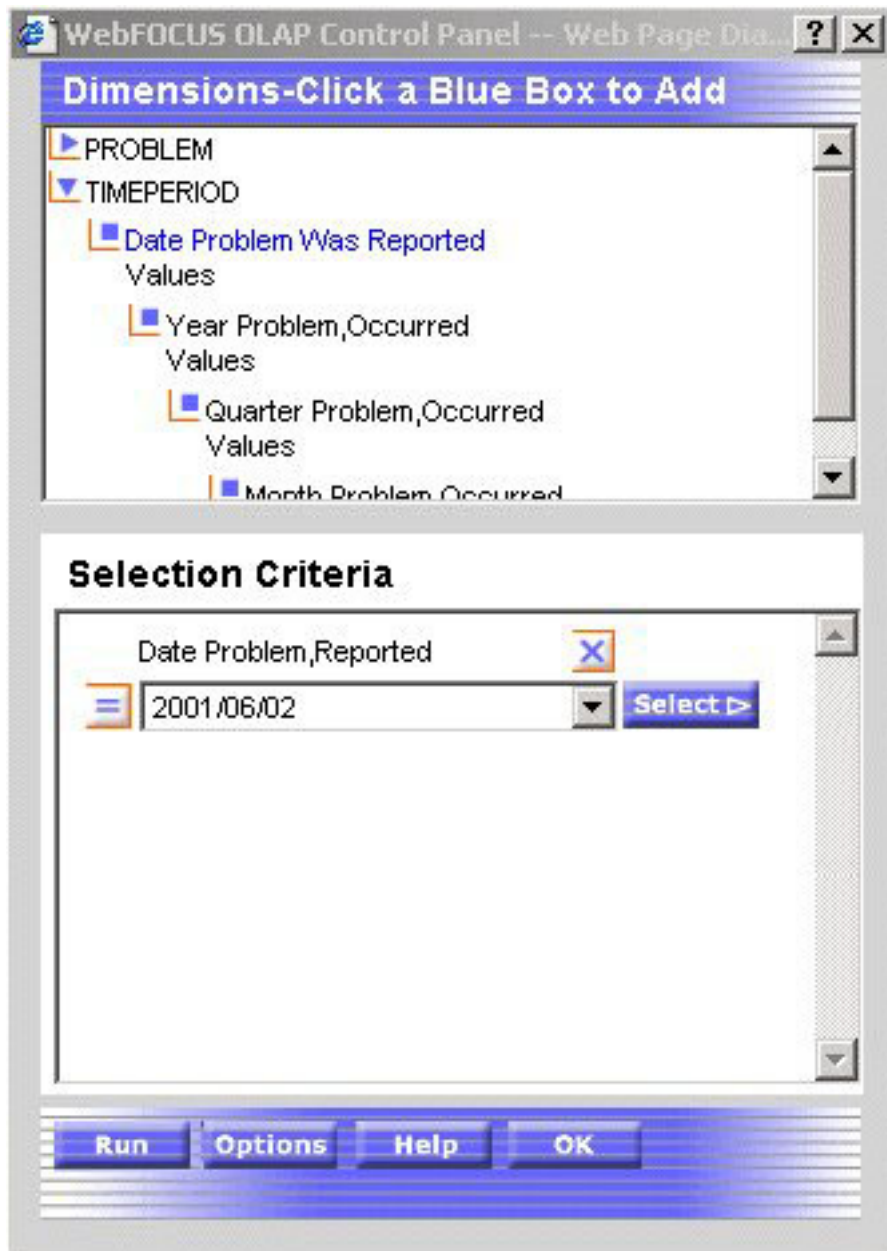
The relational operator to the left of the Date box indicates that your report will contain data only for those rows where date is equal to (=) the values you entered. This default operator is correct for this example.

8. Click **Run** to see the problem report for the specified date.

Your selection criteria are listed beside the **OLAP** button at the bottom of the report as shown in the following image.

Apply Selection Criteria to a Date Range

1. Open the OLAP Control Panel.

2. Click the **Selection Criteria** button.

   The Selection Criteria pane opens.

3. In the Dimensions box above the Selection Criteria pane, expand the dimension that includes the date field, and click the **Values** button.

   A secondary window displays controls for the dimension's date format. For example, if the date format is YYM, only the year and month controls appear. If the format is YYMD, year, month, and day controls appear.

   Note: The Date selection pane appears only when a supported date format is provided. See Date Format Limitations.

4. Required: Click the **Range** check box.

   Inclusive and Exclusive options buttons appear:

   - Choose **Inclusive** to show the range including the dates specified.
   - Choose **Exclusive** to show the range excluding the dates specified.

   Note:

   - You can select only one range of dates at a time.
   - You can apply selection criteria to a range of dates only if the date format contains a year. See Date Format Limitations.

   From and To drop-down lists open for all selectable options. By default, the current date appears.

5. Specify a **From** date and a **To** date by using the spin controls and drop-down lists.

6. Click **OK** to return to the Selection Criteria pane.

7. To view both the From and To dates of the range selected, click the down arrow on the drop-down list.

8. Click a relational operator to the left of the date element in the Selection Criteria pane:

   - Choose **Within range** operator to display records when the value falls within the specified range.

- Choose **Not within range** operator to display records when the value does *not* fall within the specified range.

9. Click **Run** to execute your report.

Applying Selection Criteria to a Range of Date Fields

The following is an example of applying selection criteria to a range of date fields.

1. Run the Standard Report **OLAPREP9**.

   As shown in the following images, your report shows problem information reported over the course of several years.

Problem Category	Date Problem Reported	Problem Number	Problem Occurrence
Incorrect Labeling	1998/04/05	693	1
		694	1
	1998/04/19	695	1
	1998/05/03	701	1
	1998/05/31	879	1
	1998/06/07	856	1
	1998/06/21	880	1
		614	1
	1998/07/05	859	1
	1998/07/26	824	1
		825	1
		996	1
	1998/08/02	826	1

OLAP

The information falls into the following categories: incorrect labeling, missing components, physical damage, power failure, remote failure.

Problem Category	Date Problem Reported	Problem Number	Problem Occurrence
Mechanical Failure	2000/09/15	2885	1
		3909	1
		2742	1
		3766	1
	2000/09/25	3304	1
	2000/09/29	2616	1
		3640	1
		2747	1
		2889	1
		3771	1
		3913	1
		2615	1

OLAP

You want to restrict the information to problems reported between June 6, 2001 and July 6, 2001. From the Control Panel, you can limit data based on a range of dates.

2. Click the **OLAP** button below the report to open the Control Panel.

3. Click the **Selection Criteria** button at the bottom right of the Control Panel.

   The Selection Criteria pane opens.

4. In the Dimensions box above the Selection Criteria pane, expand the **TIMEPERIOD** hierarchy.

5. Click **Values** under Date Problem was Reported.

   The PROBLEM_DATE pane opens over the Selections Criteria pane, with a drop-down list for each selectable value (Year, Month, and Date) based on the date format of the selected field.

6. Select the **Range** check box:
   - Inclusive and Exclusive options buttons appear. To show the range including the dates specified, choose Inclusive (the default).
   - From and To drop-down lists open for all selectable options. By default, the current date appears.

7. Specify values for the From date. For example:
   a. Change the current year to 2001 by using the spin buttons.
   b. Select June from the Months drop-down list to change the current calendar month.

    c. Select 6 from the Days drop-down list to change the current calendar day.

8. Specify values for the To date. For example:

    a. Change the current year to 2001 by using the spin buttons.

    b. Select July from the Months drop-down list to change the current calendar month.

    c. Select 6 from the Days drop-down list to change the current calendar day.

      The following image shows the selections.
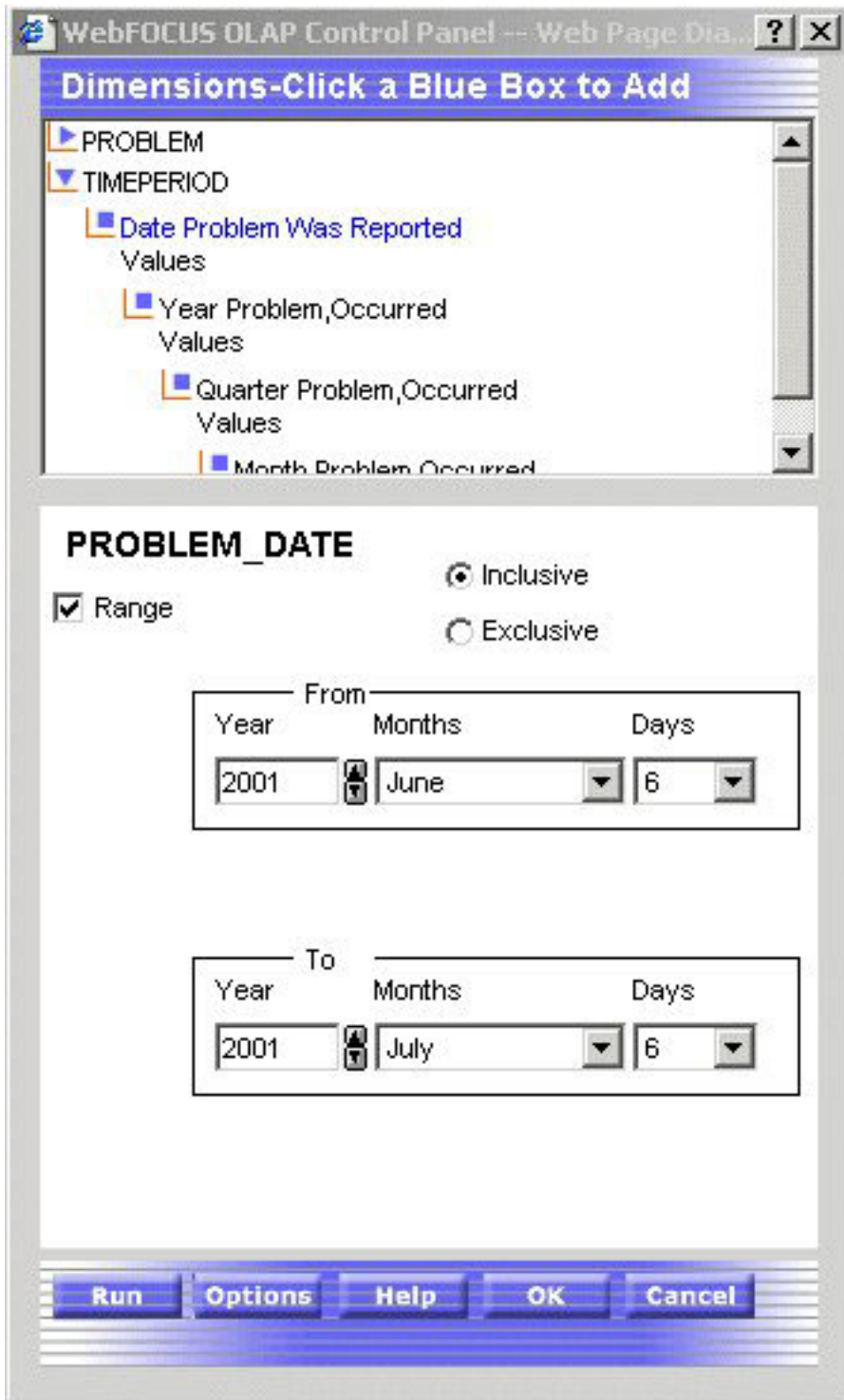
9. Click **OK** to return to the Selection Criteria pane.

    a. To view the range of dates, click the down arrow in the drop-down list, then click **OK** again.

    b. To report on information within the specified range of dates, accept the

        default, **Within range** operator.

10. Click **Run** to execute the report, which now only displays problem information from June 6, 2001 to July 6, 2001 as shown in the following image.

Problem Category	Date Problem Reported	Problem Number	Problem Occurrence
Incorrect Labeling	2001/06/07	6636	1
		7660	1
	2001/06/10	7101	1
	2001/06/14	6639	1
		7663	1
		6638	1
		7662	1
	2001/06/17	7103	1
		7102	1
	2001/06/24	7108	1
	2001/06/28	6645	1
		7669	1
	2001/07/01	7112	1
		7239	1
Mechanical Failure	2001/06/09	6800	1

Date Problem,Reported

OLAP   R   2001/06/06   ▼
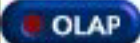
The date element appears at the bottom of the window.

11. To view the range of dates, click the arrow in the drop-down list.

Remote Failure	2001/06/08	6637	1
		7661	1
		6798	1
		7822	1
	2001/06/15	6748	1
		7772	1
	2001/06/29	6805	1
		7829	1
		6749	1
		7773	1
	2001/07/02	7010	1
		7113	1
	2001/07/06	6649	1
		7673	1
		6808	1
		7832	1

Date Problem,Reported

OLAP R 2001/06/06 ▼

Add Dates to the Selections List Box

1. Open the Control Panel.
2. Click **Selection Criteria** to open the Selection Criteria pane.
3. Click the **Select** button to open the Date selection pane.
4. Specify the date you want to add by using the spin buttons, drop-down lists, or by typing the value.
5. Click **Add**.

   The date appears inside the Selections list box.
6. Click **OK** to return to the Selection Criteria pane.

Delete Dates From the Selections List Box

1. Open the Control Panel.
2. Click **Selection Criteria** to open the Selection Criteria pane.
3. Click **Select** to open the Date selection pane.
4. Select one or more dates that you want to remove from the Selections list box.
5. Click **Delete**.

   The date is removed.
6. Click **OK** to return to the Selection Criteria pane.

Date Format Limitations

Note the following limitations when applying selection criteria to date elements:

- The Date selection pane does not support Julian dates. However, if you are using Julian dates, the Date controls still open.
- Dates containing only a day format (D, I2D, A2D) are not supported from the Date selection pane. Instead, the data source provides a list of values.
- The Range check box is enabled on the Date selection pane when the date format contains one of the following formats:
  - Any smart date format—for example, YMD, MDY, YYMD, MDYY, Q, M.
  - A4YY
  - I4YY
  - I8YYMD
  - A8YYMD
  - I6YYM
  - A6YYM

Visualizing Trends

To make your reports more powerful, you can insert visual representations of selected data directly into the report output. These visual representations, which appear as a column of vertical or horizontal bar graphs adjacent to the numeric data, make relationships and trends among data more obvious.

You can apply data visualization graphs to selected measures from:

- Context menus in the report itself.

  This is the quickest way to apply data visualization bar graphs to numeric measures.
- The Measures control in the Selections pane.
- Check boxes in the Measures box on the Control Panel.

For details about data visualization graphs, see Visualizing Trends in Reports.

Add a Column of Bar Graphs for a Numeric Measure

The quickest way to apply data visualization graphics is from the report itself:

1. Right-click the title of a measure column.
2. Choose **Visualization** from the menu.

The report runs automatically, displaying a column of bar graphs following the selected measures column.

Tip: To remove the bar graphs, right-click the measure column title and choose **Remove Visualization** from the menu.

For other methods of applying bar graphs to columns, see Visualizing Trends in Reports.

Displaying Graphs and Reports

When you graph a measure in an OLAP report, you select the specific data elements to include and view the tabular report and a graphical representation of the identical information simultaneously in a split window. The graph appears in a frame in the top half of the window to facilitate comparison.

To be graphable, the data in the report must include at least one numeric measure and one sort field (By or Across). The Graph control is activated in the Selections pane or the Control Panel when these basic requirements are met.

As shown in the following image, it includes three sort fields (PRODCAT, Store Name, and Manufacturing Plant) and three numeric measures (Quantity, Our Cost, and Price), displayed as horizontal bar charts for quick comparison.
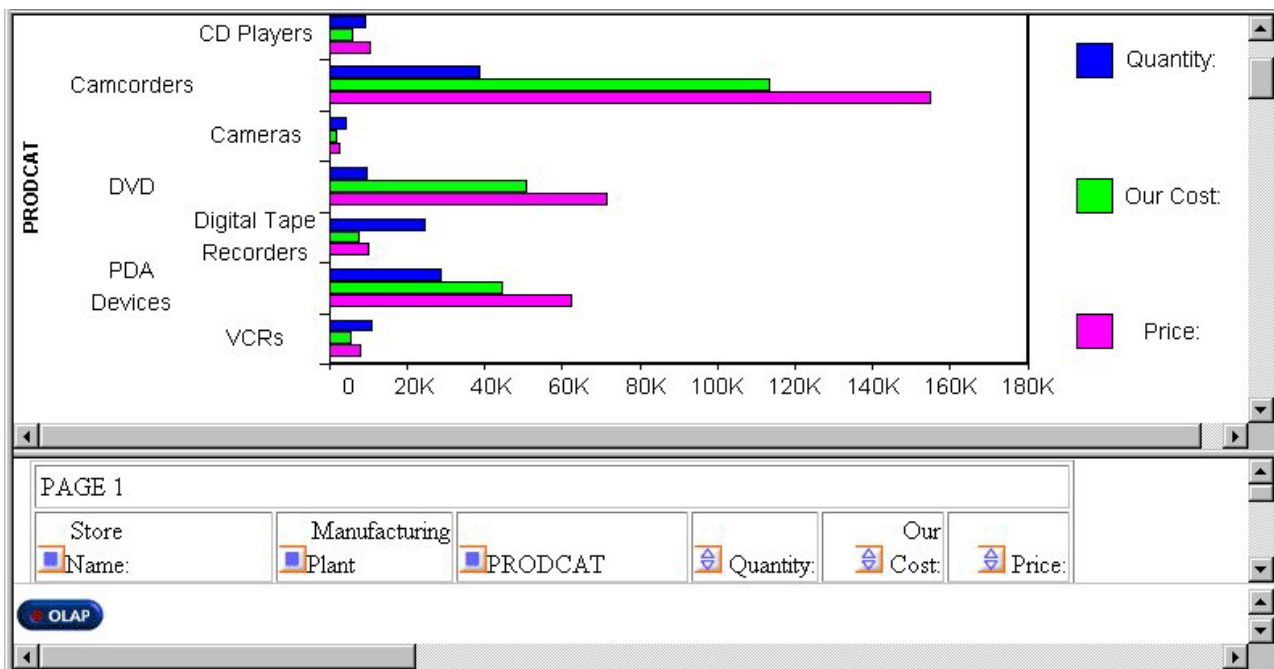
You can request a graph from an OLAP report, from the Selections pane, or from the Control Panel:

- **From an OLAP report,** you can create a vertical bar chart to represent the data in a selected measure.
- **From the Selections pane or the Control Panel,** you can create seven different types of graphs and apply them to one or more measures:
  - Vertical Bar (This is the default graph type.)
  - Vertical Line
  - Vertical Area
  - Horizontal Bar
  - Horizontal Line
  - Horizontal Area
  - Pie

If you choose to graph more than one measure, you can employ different graph types to suit the data in each column, with the following restrictions:

- When you select Vertical or Horizontal Bar, Line, or Area as the controlling graph style for a measure, you can apply any combination of these styles to other measures. For example, the first measure can appear as bars, the second measure as lines, and the third measure as areas. All measures must have the same orientation (vertical or horizontal).

- When you choose Pie as the controlling graph style, you can use only pie charts for other measures.

For details about supported combinations, see Combining Graph Styles and Measure Styles in OLAP Graphs.

Note: If drill down capability has been enabled for the dimensions in a report, the same functionality is automatically enabled for graphs. You can, therefore, drill down from one graphical representation of your data to another.

Combining Graph Styles and Measure Styles in OLAP Graphs

The following table lists the available style combinations in the second column for each graph style in the first column.

Controlling Graph Style	Potential Measure Styles
Vertical Bar (default)	Vertical Bar (default)
	Vertical Line
	Vertical Area
Vertical Line	Vertical Line (default)
	Vertical Bar
	Vertical Area
Vertical Area	Vertical Area (default)
	Vertical Bar
	Vertical Line
Horizontal Bar	Horizontal Bar (default)
	Horizontal Line
	Horizontal Area
Horizontal Line	Horizontal Line (default)
	Horizontal Bar
	Horizontal Area
Horizontal Area	Horizontal Area (default)
	Horizontal Line
	Horizontal Area
Pie	Pie

Graph a Measure From the Selections Pane

1. Click the down arrow to the left of the Graph control to open a pane that contains all the numeric measures in the current report.

There is a check box to the left of each measure and a graph button to the right of each measure. All check boxes are unchecked by default and all graph buttons are grayed (inactive) by default.

2. Select a check box associated with a measure.

   The graph button to the right of the measure becomes active. The default graph style is Vertical bar.

3. Toggle through the seven graph style icons until you reach the one you want to apply to the selected measure.

4. Repeat steps 2 and 3 for any other measures you want to graph.

   For a list of graph types that can be defined, see Combining Graph Styles and Measure Styles in OLAP Graphs.

5. Click the **Run** button on the band below the Selections pane.

   The graph opens in a separate frame above the report and Selections pane.

Graphing Multiple Measures From the Selections Pane

This example contains two measures, BALANCE and CANADA_DOLLARS, sorted by Continent. You would like to see graphical representations of both measures. To contrast the graphical information, you use a different graph type for each one.

1. Run the Standard Report **OLAPREP4**.

2. Right-click the **Region** field and select **Delete** from the menu to limit the report to the fields you want to graph—one dimension, Continent, and two measures, BALANCE and CANADA DOLLARS.

3. In the Selections pane above the report, click the arrow to the left of the Graph control to list the measures.

   • Click the **BALANCE** measure check box, then choose the **vertical bar** icon to the right of the measure. (This is the default graph type.)

   • Click the **CANADA_DOLLAR** measure check box, then toggle through the graph icons until you see the **vertical area** graph.

   As shown in the following image, the Selections pane has the Graph control listing BALANCE represented as a vertical bar and CANADA_DOLLAR represented as a vertical area.

4. Click the **Run** button on the band below the Selection pane to generate the graphs.

The following image shows the results of the graph selections.
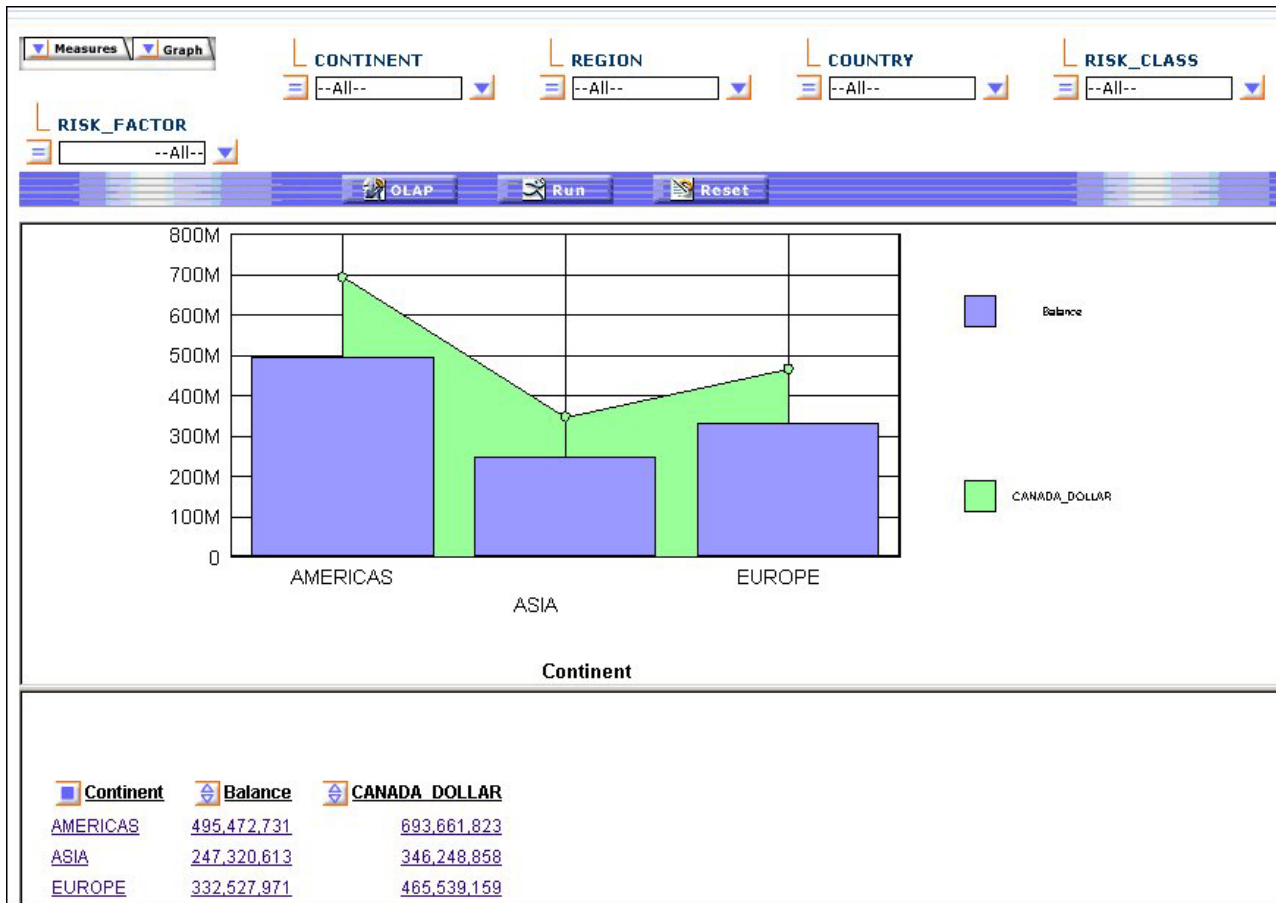
Create a Pie Chart From the Selections Pane

The following procedure is an example of creating a pie chart from the Selections pane.

1. Run the Standard Report **OLAPREP2**.

   The report shows order information for stores that sell Century Corporation's electronic products. Audio Expert shows the highest numbers, with orders of digital products significantly exceeding analog.

   You want a clearer picture of how the digital orders breakdown by product so you decide to create a pie chart.

2. Click **Digital** for Audio Expert in Q2 to hone in on the data you want to graph as shown in the following image.

QUARTER	Store Name:	PRODTYPE	Quantity:	Line Cost Of Goods Sold
Q1	AV VideoTown	Analog	18,449	3,969,296.00
		Digital	22,206	5,109,400.00
	Audio Expert	Analog	78,449	16,467,146.00
		Digital	105,983	25,092,678.00
	City Video	Analog	6,287	1,315,015.00
		Digital	7,196	1,607,513.00
	Consumer Merchandise	Analog	6,980	1,542,036.00
		Digital	14,957	3,251,090.00
	TV City	Analog	19,077	3,772,119.00
		Digital	41,307	10,128,967.00
	Web Sales	Analog	545	124,366.00
		Digital	829	190,201.00
	eMart	Analog	97,128	21,152,262.00
		Digital	108,221	24,990,368.00
Q2	AV VideoTown	Analog	11,781	2,663,655.00
		Digital	27,377	5,928,507.00
	Audio Expert	Analog	57,944	11,868,758.00
		Digital	111,421	28,064,250.00

The report now shows the Quantity and Line Cost of Goods sold for several digital products sold at Audio Expert in Q2 as shown in the following image.

PRODCAT	Quantity:	Line Cost Of Goods Sold
CD Players	16,328	1,616,472.00
Camcorders	12,841	9,159,550.00
Cameras	305	60,695.00
DVD	13,959	2,669,901.00
Digital Tape Recorders	29,396	2,028,324.00
PDA Devices	38,592	12,529,308.00

3. Right-click **Quantity** and choose **Show Panel** to open the Selection pane.
4. In the Selections pane, click the arrow to the left of the Graph control, then click the check box for **Quantity** and toggle through the graph options until you reach the **pie** icon as shown in the following image.

PRODCAT	Quantity:	Line Cost Of Goods Sold
CD Players	16,328	1,616,472.00
Camcorders	12,841	9,159,550.00
Cameras	305	60,695.00
DVD	13,959	2,669,901.00
Digital Tape Recorders	29,396	2,028,324.00
PDA Devices	38,592	12,529,308.00

5. Click the **Run** button on the band below the Selections pane.

As shown in the following image, the graph appears in a pane above the report. You can see at a glance that PDA Devices constituted about 1/3 of digital sales at the Audio Expert store in Q2.
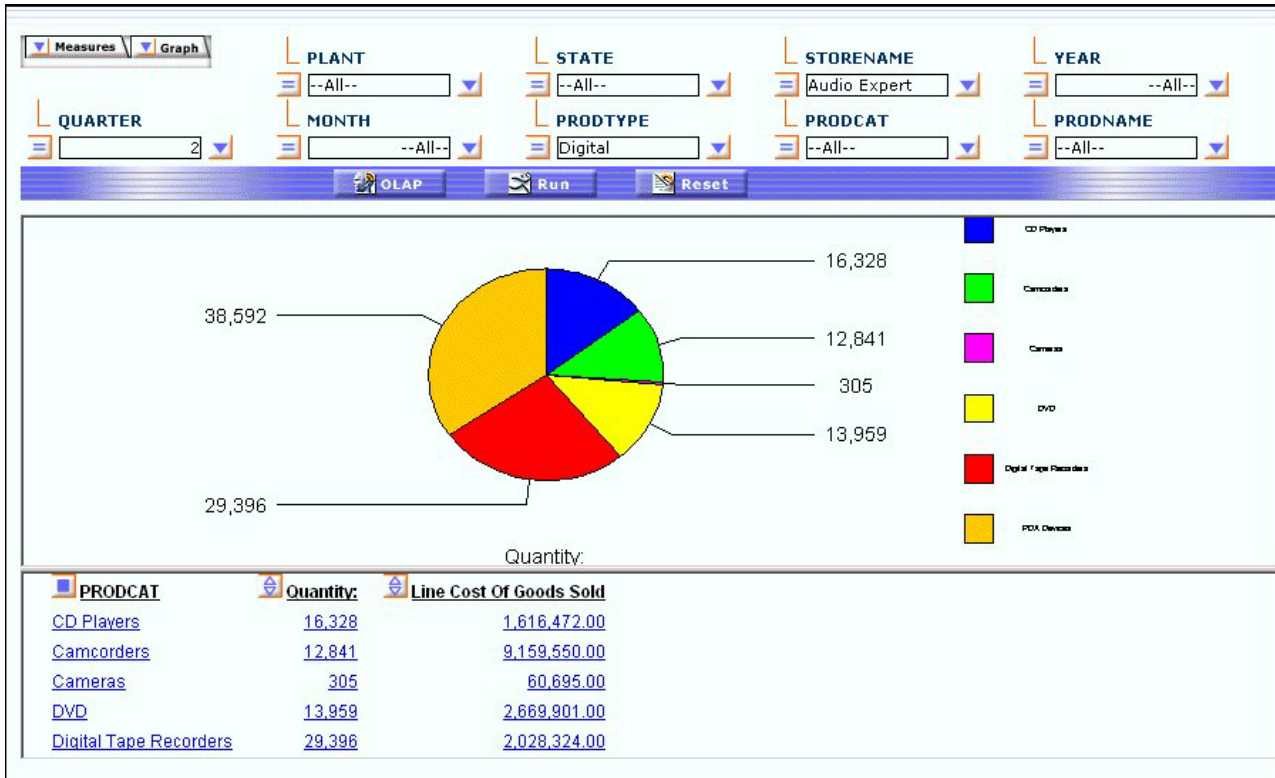
Graph a Measure From the Control Panel

1. Open the Control Panel.

2. Select the **Show Graph** check box located below the Measures Properties box.

   Note that the contents of the Drill Down and Drill Across boxes determine the X-axis fields. When there are multiple drill (X-axis) fields, multiple graphs appear vertically stacked in the same frame. The measures appear as Y-axis fields on the graphs you display.

3. Click the **Graph** icon adjacent to the Show Graph check box.

   The Measures and GraphStyle pane opens.

   Check boxes associated with the available measures are unchecked by default.

4. Click one of the seven icons at the bottom of the window to set a controlling graph style.

5. Select the check box(es) for the measure(s) you wish to graph.

   The graph icon corresponding to the controlling graph style appears next to each selected measure.

6. Click the icon next to a measure to choose a different graph style from the supported combinations as shown in the following image.
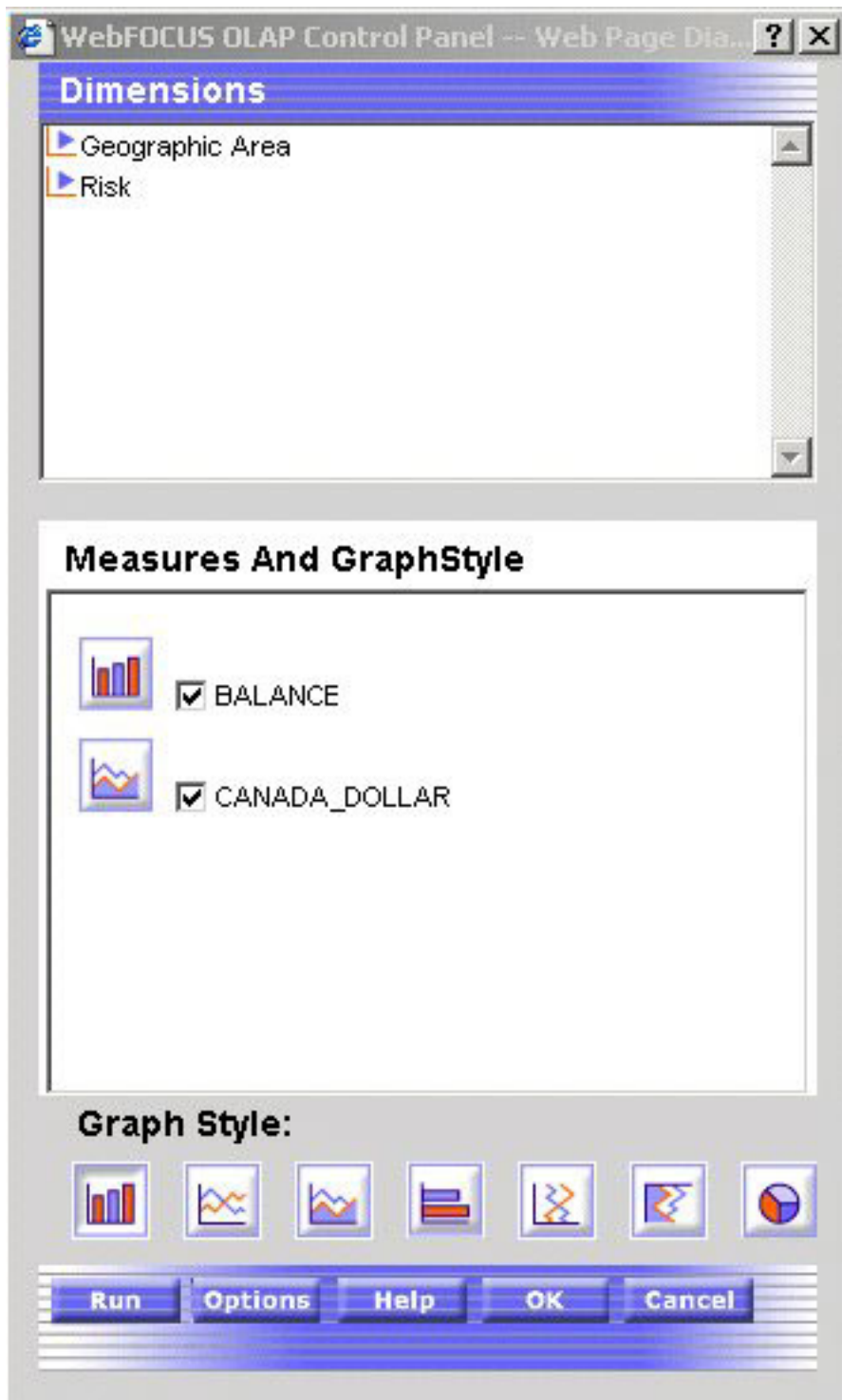
7. Click **OK** to return to the main Control Panel window with all the graph settings retained.

8. Click **Run** to display the graph(s) and the tabular report in a split window.

Note:

- If you select the Show Graph check box and click Run without selecting a controlling graph style, the default style (Vertical Bar) is applied.

- If you click Run without selecting the Show Graph check box, a tabular report appears without a graph.
- If you select at least one measure in the Measures and GraphStyle pane without selecting the Show Graph check box, when you click OK the system automatically selects the Show Graph check box. The tabular report appears with a graph.
- You cannot choose to graph alphanumeric or date fields. If there are no numeric measures, the Show Graph check box and the Graph button are disabled (grayed out).

Controlling the Display of Measures in a Report

While you cannot add new measures to an OLAP report without returning to the original report request, you can adjust the display of the measures in the report in several ways. You can:

- Stack measures in rows.
- Change the order of measure columns.
- Hide and expose measures.
- Add a column of data visualization bar graphs following any numeric measure.

Stacking Measures

When you have more than one measure in an OLAP report, you can stack the measures in separate rows within the same column to reduce the width of the report.

You cannot apply data visualization bar graphs to stacked measures.

Display Stacked Measures

1. Open the Control Panel.
2. Select the **Stack Measures** check box to display measures in separate rows under one column.
3. Click **Run** to execute your report.

Tip: To restore the standard display, deselect the **Stack Measures** check box and rerun the report.

Displaying Stacked Measures

The following is an example of displaying stacked measures.

1. Run the Standard Report **OLAPREP4**.

   Initially, this report is sorted vertically by Continent and Risk_Class and horizontally by Region, and the measures—Balance and CANADA_DOLLARS—appear as separate columns.
2. For this example, you will not need the Region dimension, but you will need the Country dimension. You can quickly make these changes to the report:
   a. Right-click **Region** and select **Delete** from the menu.
   b. Right-click **Continent** and select **Unhide** from the menu, then select **Country** from the secondary menu.

      The report now displays data by Continent followed by Country as shown in the following image.

You wish to show the measure titles and data values in rows.

3. Click the **OLAP** button on the band below the Selections pane to open the Control Panel.

4. Click the **Stack Measures** check box below the Measures box as shown in the following image.
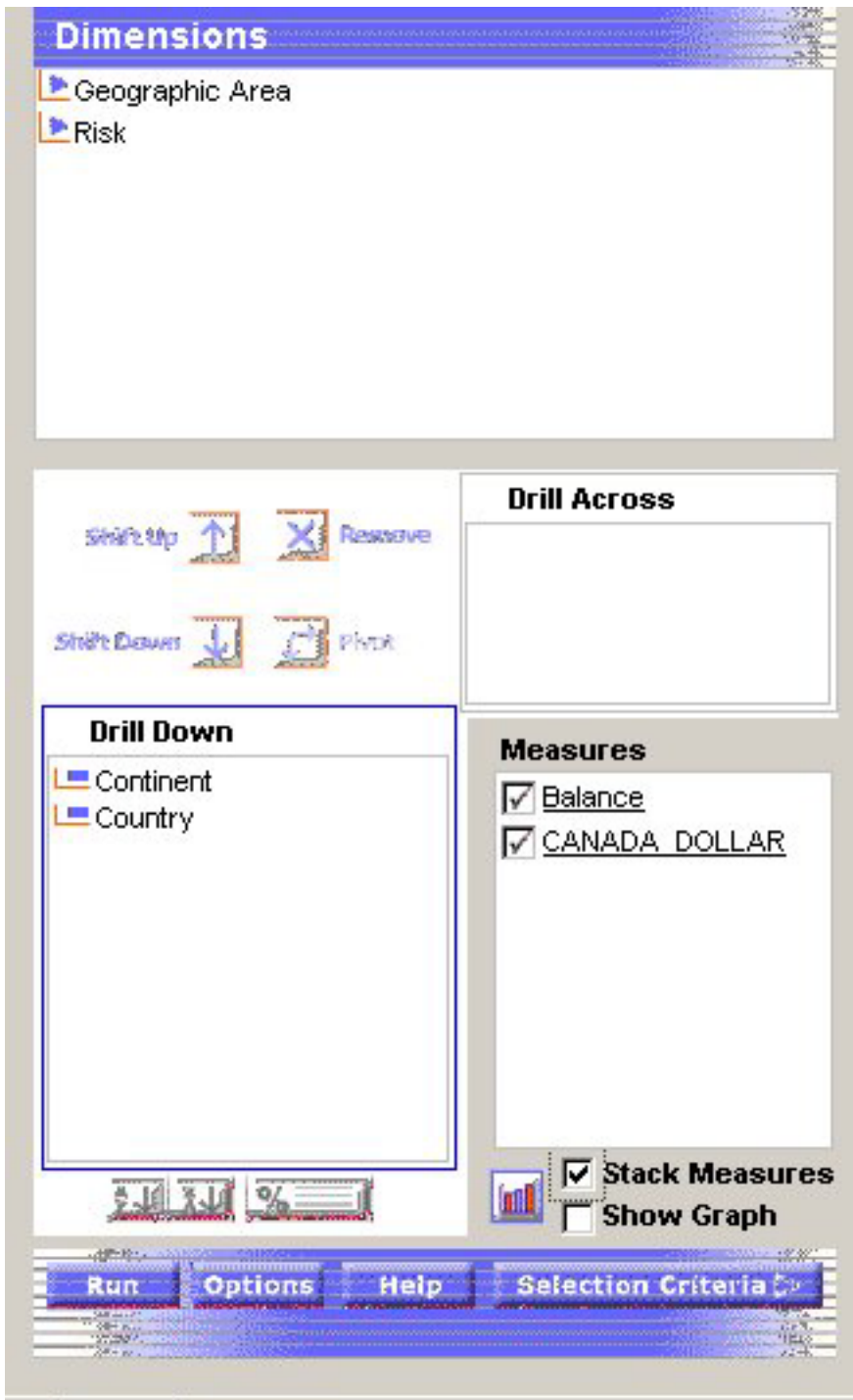
5. Click **Run** to execute the report and display the titles and values of the measures stacked over each other in separate rows as shown in the following image.
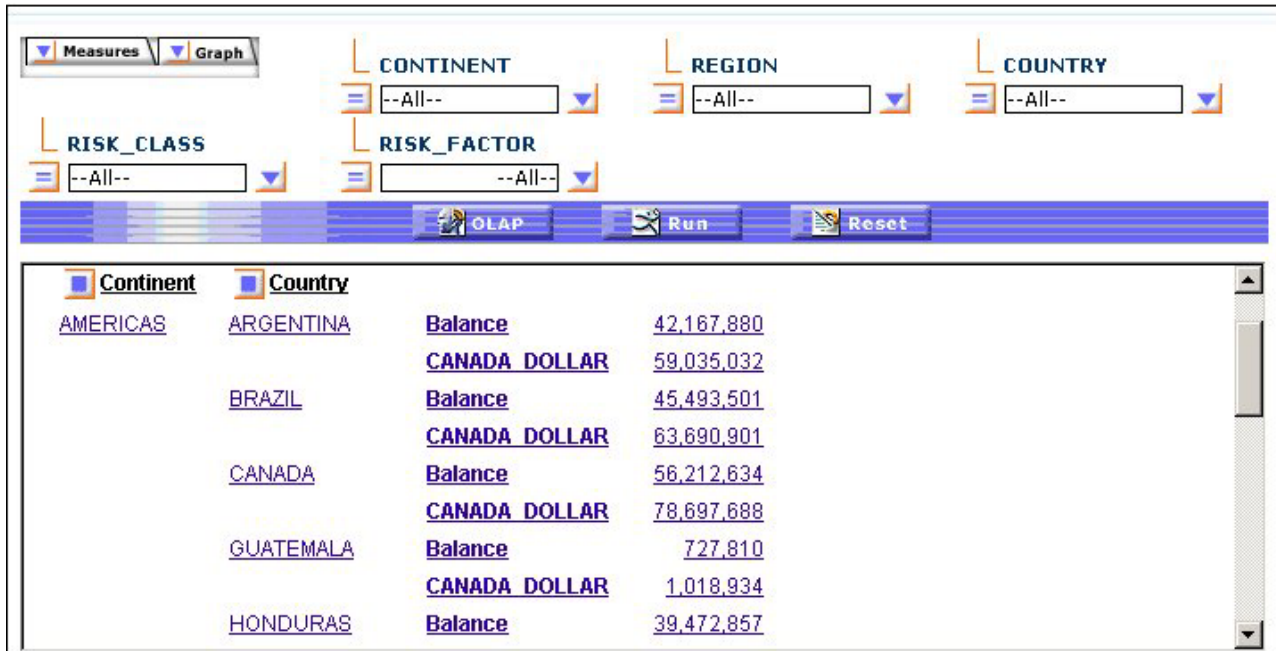
Changing the Order of Measure Columns

You can change the order in which measure columns are presented in the report.

Reposition Measure Columns in an OLAP Report

To reposition a numeric column, drag and drop the field into a new column position.

The cursor changes to a plus (+) sign to indicate acceptable places into which you can drop the field. (Unacceptable positions are indicated by a circle with a slash cross the center.)

Repositioning Measure Columns

The following is an example of repositioning measure columns.

1. Run the Standard Report **OLAPREP2**.

   As shown in this image, the column for the Quantity measure precedes the column for the Line Cost of Goods Sold measure.

QUARTER	Store Name:	PRODTYPE	Quantity:	Line Cost Of Goods Sold
Q1	AV VideoTown	Analog	18,449	3,969,296.00
		Digital	22,206	5,109,400.00
	Audio Expert	Analog	78,449	16,467,146.00
		Digital	105,983	25,092,678.00
	City Video	Analog	6,287	1,315,015.00
		Digital	7,196	1,607,513.00
	Consumer Merchandise	Analog	6,980	1,542,036.00
		Digital	14,957	3,251,090.00
	TV City	Analog	19,077	3,772,119.00
		Digital	41,307	10,128,967.00
	Web Sales	Analog	545	124,366.00
		Digital	829	190,201.00
	eMart	Analog	97,128	21,152,262.00
		Digital	108,221	24,990,368.00

2. To change the order of columns, drag and drop Line Costs of Goods Sold before Quantity.

The cursor changes to a + sign to designate where you can drop the field. The report, as shown in the following image, now displays the Quantity column as its last column.

QUARTER	Store Name:	PRODTYPE	Line Cost Of Goods Sold	Quantity:
Q1	AV VideoTown	Analog	3,969,296.00	18,449
		Digital	5,109,400.00	22,206
	Audio Expert	Analog	16,467,146.00	78,449
		Digital	25,092,678.00	105,983
	City Video	Analog	1,315,015.00	6,287
		Digital	1,607,513.00	7,196
	Consumer Merchandise	Analog	1,542,036.00	6,980
		Digital	3,251,090.00	14,957
	TV City	Analog	3,772,119.00	19,077
		Digital	10,128,967.00	41,307
	Web Sales	Analog	124,366.00	545
		Digital	190,201.00	829
	eMart	Analog	21,152,262.00	97,128
		Digital	24,990,368.00	108,221

Hiding and Displaying Measures

You can hide and expose measures from an OLAP report, the Selections pane, or the Control Panel.

Hide or Expose a Measure From the Report

To hide a measure column, right-click the column title and choose **Hide** from the menu. The column is automatically removed from the display.

To expose a hidden measure column, right-click a displayed measure and choose **Unhide** from the menu. A secondary menu lists any hidden measures.

Choose the one you want to re-expose in the report.

Tip: If you want to add a new measure to the report, you must return to the original request and add the field there.

Hiding and Exposing a Measure From the Report

The following is an example of hiding and exposing a measure from the report.

1. Run the Standard Report **OLAPREP2**.

   The report includes two measures: Quantity and Line Cost of Goods Sold.

2. Right-click the **Line Cost of Goods Sold** and choose **Hide** from the menu as shown in the following image.

The report runs and displays only the Quantity measure.

3. Right-click **Quantity** and select **Unhide**.

   A secondary menu displays the hidden measure.

4. Select **LINE_COGS** to redisplay Line Cost of Goods Sold as shown in the following image.

QUARTER	Store Name:	PRODTYPE	Quantity:
Q1	AV VideoTown	Analog	
		Digital	
	Audio Expert	Analog	
		Digital	
	City Video	Analog	
		Digital	
	Consumer Merchandise	Analog	
		Digital	14,957
	TV City	Analog	19,077
		Digital	41,307

Menu overlay: Sort by Highest, Sort by Lowest, Graph, Hide, Unhide, Visualize, Full Screen, Help

Button: LINE_COGS

The report now displays the Line Cost of Goods Sold column as shown in the following image.

QUARTER	Store Name:	PRODTYPE	Quantity:	Line Cost Of Goods Sold
Q1	AV VideoTown	Analog	18,449	3,969,296.00
		Digital	22,206	5,109,400.00
	Audio Expert	Analog	78,449	16,467,146.00
		Digital	105,983	25,092,678.00
	City Video	Analog	6,287	1,315,015.00
		Digital	7,196	1,607,513.00
	Consumer Merchandise	Analog	6,980	1,542,036.00
		Digital	14,957	3,251,090.00
	TV City	Analog	19,077	3,772,119.00
		Digital	41,307	10,128,967.00
	Web Sales	Analog	545	124,366.00
		Digital	829	190,201.00

Hide or Display a Measure From the Selections Pane

1. Click the down arrow to the left of the Measures control to display a list of the measures in the report.
2. Click the check box next to a measure to display or hide it. The check box toggles through three positions.
   - **To hide the measure,** click the check box until it is blank.
   - **To expose a hidden measure,** click the check box until you see a check mark.

Tip: You can use the same check box to display a column of data visualization bar graphs for numeric measures. This setting is represented as a graph in the check box. For details, see Visualizing Trends.

Hiding and Exposing a Measure Column From the Selections Pane

The following is an example of hiding and exposing a measure column from the Selections pane.

1. Run the Standard Report **OLAPREP2**.

   Because of the OLAP settings selected for this report, the Selections pane is hidden. For this example, you will need to expose it.

2. Right-click **QUARTER** and select **Show Panel** from the menu.

   As shown in the following image, the report now looks like this: two measures—Quantity and Line Cost of Goods Sold—appear.

3. In the Selection pane, click the arrow to the left of the Measures control to list the measures in the report. Notice that both measures are checked.

4. To hide **Line Cost of Goods Sold**, click the check box until it is blank as shown in the following image.

5. Click the **Run** button on the band below the Selections pane.

   Only **Quantity** now appears as shown in the following image.

6. Open the Measures control again and recheck **Line Cost of Goods Sold** as shown in the following image.
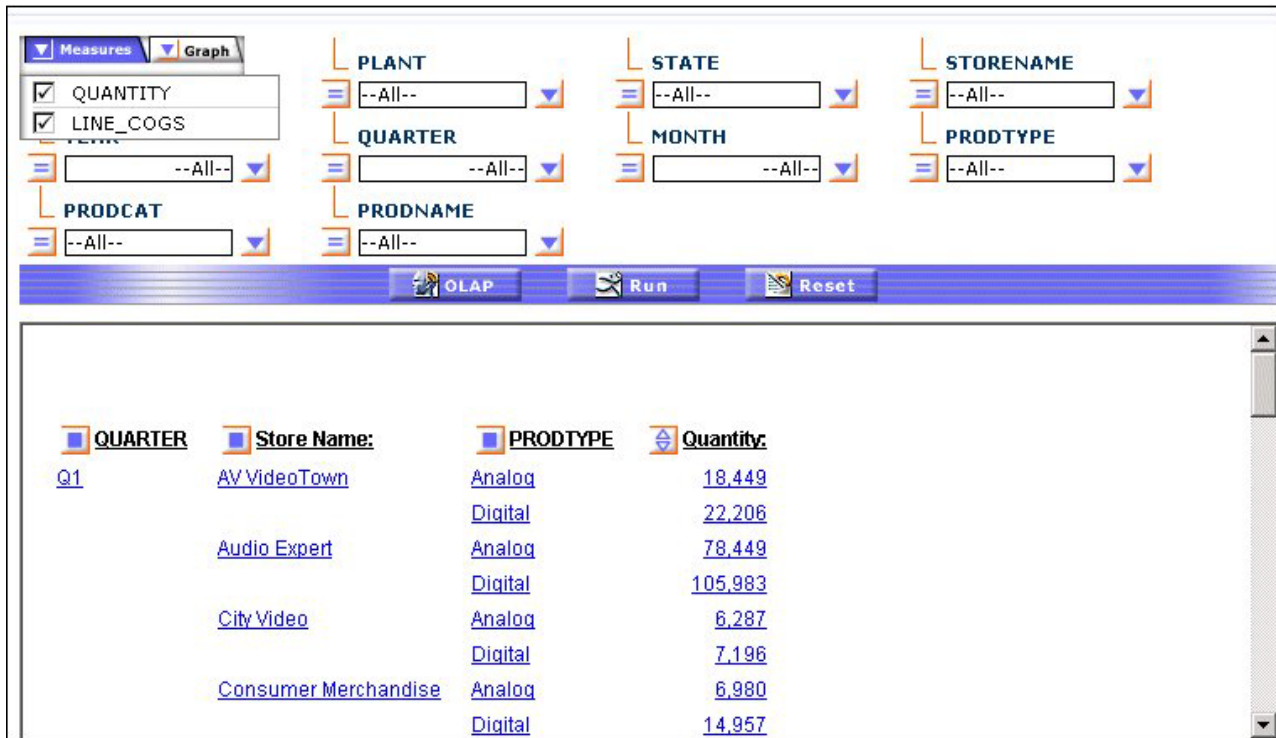
7. Run the report again.

   The output now looks as it originally did.

Display or Hide a Measure From the Control Panel

1. Open the Control Panel.
2. In the Measures box, click the check box next to a measure to display or hide it. The check box toggles through three positions.
   - **To hide the measure,** click the check box until it is blank.
   - **To expose a hidden measure,** click the check box until you see a check mark.

   Tip: You can use the same check box to display a column of data visualization bar graphs for numeric measures. This setting is represented as a graph in the check box. For details, see Visualizing Trends.

3. Click **Run** to execute your report.

Adding and Removing Dimensions

Since all of the values in a dimensions hierarchy are available in an OLAP report, you can add dimensions to the OLAP report at any time, without returning to the original report request. You can add dimensions from:
- An OLAP report.
- The Control Panel.

Add a Dimension Element From the Control Panel

1. Open the Control Panel.
2. Select a report layout box (Drill Down or Drill Across) to indicate how you want the new sort dimension to be used in the report.

3. Expand a dimension in the Dimensions box at the top of the window, then click the dimension element you want to add to the designated layout box. The new dimension is added to the bottom of the list.

4. If you wish to change the position of the new sort field, click the up arrow to reposition it.

5. Click **Run** to execute your report with the new settings.

Adding a Dimension Element From the Control Panel

The following is an example of adding a dimension element from the Control Panel.

1. Run the Standard Report **OLAPREP2**.

   Initially the report is sorted by quarter, store, and product type as shown in the following image.

QUARTER	Store Name:	PRODTYPE	Quantity:	Line Cost Of Goods Sold
Q1	AV VideoTown	Analog	18,449	3,969,296.00
		Digital	22,206	5,109,400.00
	Audio Expert	Analog	78,449	16,467,146.00
		Digital	105,983	25,092,678.00
	City Video	Analog	6,287	1,315,015.00
		Digital	7,196	1,607,513.00
	Consumer Merchandise	Analog	6,980	1,542,036.00
		Digital	14,957	3,251,090.00
	TV City	Analog	19,077	3,772,119.00
		Digital	41,307	10,128,967.00
	Web Sales	Analog	545	124,366.00
		Digital	829	190,201.00
	eMart	Analog	97,128	21,152,262.00
		Digital	108,221	24,990,368.00
Q2	AV VideoTown	Analog	11,781	2,663,655.00
		Digital	27,377	5,928,507.00
	Audio Expert	Analog	57,944	11,868,758.00
		Digital	111,421	28,064,250.00

You want to sort by month within each quarter.

2. Click the square button next to QUARTER to open the Control Panel.

3. In the Control Panel:

   a. Click in the Drill Down box to activate the buttons immediately above the box.

   b. Expand the **Time Period** dimension and click **MONTH**. It is added to the bottom of the Drill Down list.

   c. Click the **Shift Up** arrow twice to move MONTH below QUARTER as shown in the following image.

4. Click the **Run** button at the bottom of the Control Panel.

   The report is now sorted by quarter, month, store, and product type as shown in the following image.

Delete a Dimension Element From the Report

Right-click the dimension column you wish to remove and choose **Delete** from the menu.

The report runs automatically.

Deleting a Dimension Element From the Report

The following is an example of deleting a dimension element from the report.

1. Run the Standard Report **OLAPREP2**.

   Initially the report is sorted by quarter, store, and product type. You wish to remove PRODTYPE as a sort category.

2. Right-click the **PRODTYPE** column and choose **Delete** from the menu as shown in the following image.

QUARTER	Store Name:	PRODTYPE	Quantity:	Line Cost Of Goods Sold
Q1	AV VideoTown		18,449	3,969,296.00
			22,206	5,109,400.00
	Audio Expert		78,449	16,467,146.00
			105,983	25,092,678.00
	City Video		6,287	1,315,015.00
		Digital	7,196	1,607,513.00
	Consumer Merchandise	Analog	6,980	1,542,036.00
		Digital	14,957	3,251,090.00
	TV City	Analog	19,077	3,772,119.00
		Digital	41,307	10,128,967.00
	Web Sales	Analog	545	124,366.00
		Digital	829	190,201.00
	eMart	Analog	97,128	21,152,262.00
		Digital	108,221	24,990,368.00
Q2	AV VideoTown	Analog	11,781	2,663,655.00
		Digital	27,377	5,928,507.00

(Context menu overlay: Delete, Unhide, Move To Across, Full Screen, Help)

The report runs automatically. The new report is sorted by quarter and store as shown in the following image.

QUARTER	Store Name:	Quantity:	Line Cost Of Goods Sold
Q1	AV VideoTown	40,655	9,078,696.00
	Audio Expert	184,432	41,559,824.00
	City Video	13,483	2,922,528.00
	Consumer Merchandise	21,937	4,793,126.00
	TV City	60,384	13,901,086.00
	Web Sales	1,374	314,567.00
	eMart	205,349	46,142,630.00
Q2	AV VideoTown	39,158	8,592,162.00
	Audio Expert	169,365	39,933,008.00
	City Video	10,240	2,310,831.00
	Consumer Merchandise	23,795	5,515,318.00
	TV City	45,344	10,236,165.00
	Web Sales	2,507	562,332.00
	eMart	189,839	41,760,915.00
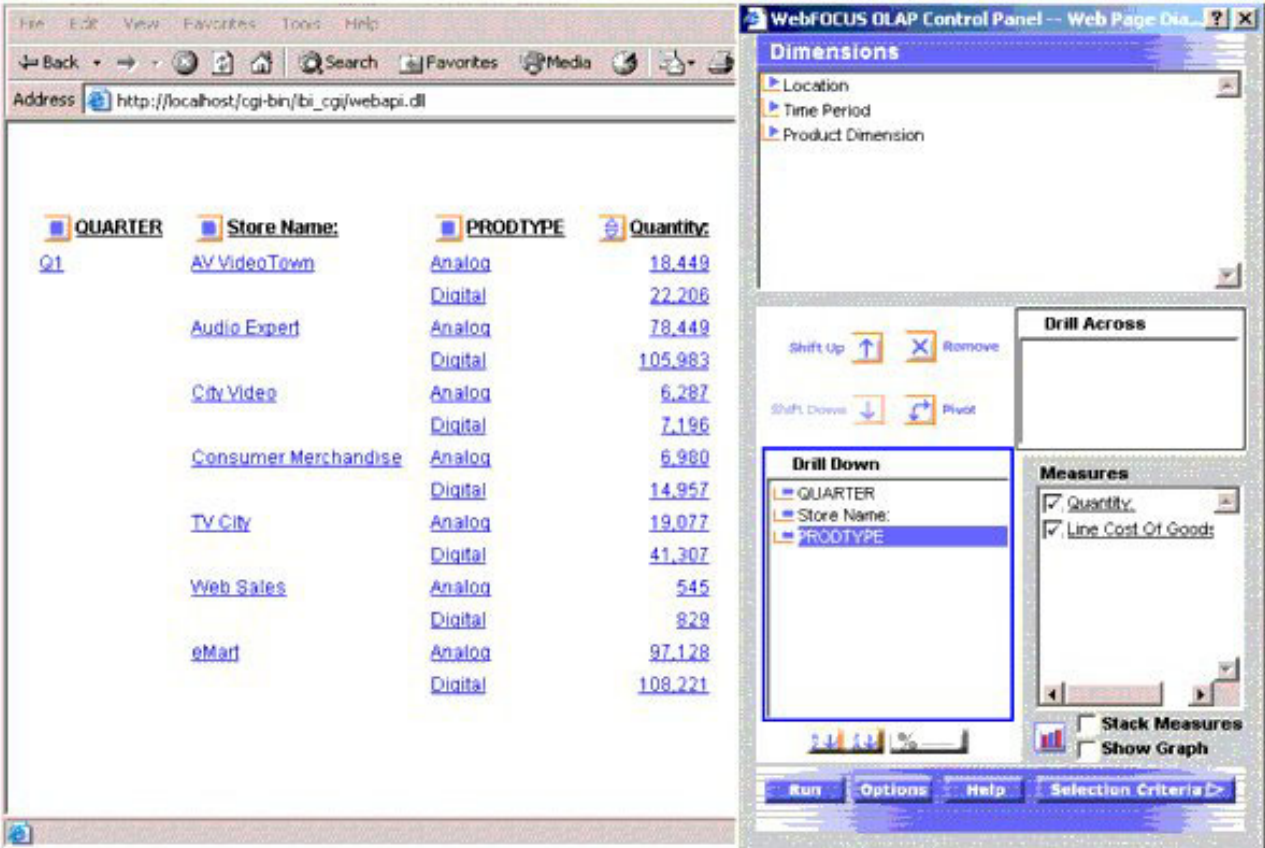
Delete a Dimension Element From the Control Panel

1. Select the element in the Drill Down or Drill Across box. The buttons above the box become active.

2. Click **Remove** . The element is deleted from the Drill Down or Drill Across box.

3. Click **Run** to see the new report.

Deleting a Dimension Element From the Control Panel

The following is an example of deleting a dimension element from the Control Panel.

1. Run the Standard Report **OLAPREP2**.

   Initially the report is sorted by quarter, store, and product type. You wish to remove PRODTYPE as a sort category.

2. Click the blue square button next to QUARTER to open the Control Panel.

3. Select **PRODTYPE** in the Drill Down box as shown in the following image.

4. Click the **Remove** button.

5. Click the **Run** button at the bottom of the Control Panel.

   The new report is sorted by quarter and store as shown in the following image.

QUARTER	Store Name:	Quantity:	Line Cost Of Goods Sold
Q1	AV VideoTown	40,655	9,078,696.00
	Audio Expert	184,432	41,559,824.00
	City Video	13,483	2,922,528.00
	Consumer Merchandise	21,937	4,793,126.00
	TV City	60,384	13,901,086.00
	Web Sales	1,374	314,567.00
	eMart	205,349	46,142,630.00
Q2	AV VideoTown	39,158	8,592,162.00
	Audio Expert	169,365	39,933,008.00
	City Video	10,240	2,310,831.00
	Consumer Merchandise	23,795	5,515,318.00
	TV City	45,344	10,236,165.00
	Web Sales	2,507	562,332.00
	eMart	189,839	41,760,915.00

Saving OLAP Reports

DB2 Web Query includes several saving features for OLAP reports:

- The dialog for saving reports allows them to be placed in subfolders of the user's choice under Reports; subfolders can also be created from the OLAP tool.
- Users can save their reports with the OLAP functionality removed, so the OLAP tool can serve as a simple report writer tool.
- Field name referencing is uniform throughout the OLAP product. For example, the AS or TITLE phrases will appear in reports generated using the OLAP Selections pane or the OLAP Control Panel (OCP).

Note: When saving OLAP reports to Managed Reporting, you must refresh the Domain to see the newly saved reports. Otherwise, the new reports will not be listed in the Managed Reporting tree.

Saving Options

If you are working in the OLAP Selections pane, you can save your report by using the Save button on the toolbar. You can also access other saving options in the OCP by selecting the Options button.

Save a Report in OLAP

1. Click the **Save** button in the OLAP Selections Pane.

   The following image shows the OLAP Selections Pane containing five buttons, OLAP, Run, Reset, Save, and Help.

or

Click the **Options** button in the OLAP Control Panel (OCP).

The following image shows the OLAP Control Panel where the Options button appears at the bottom of the panel.
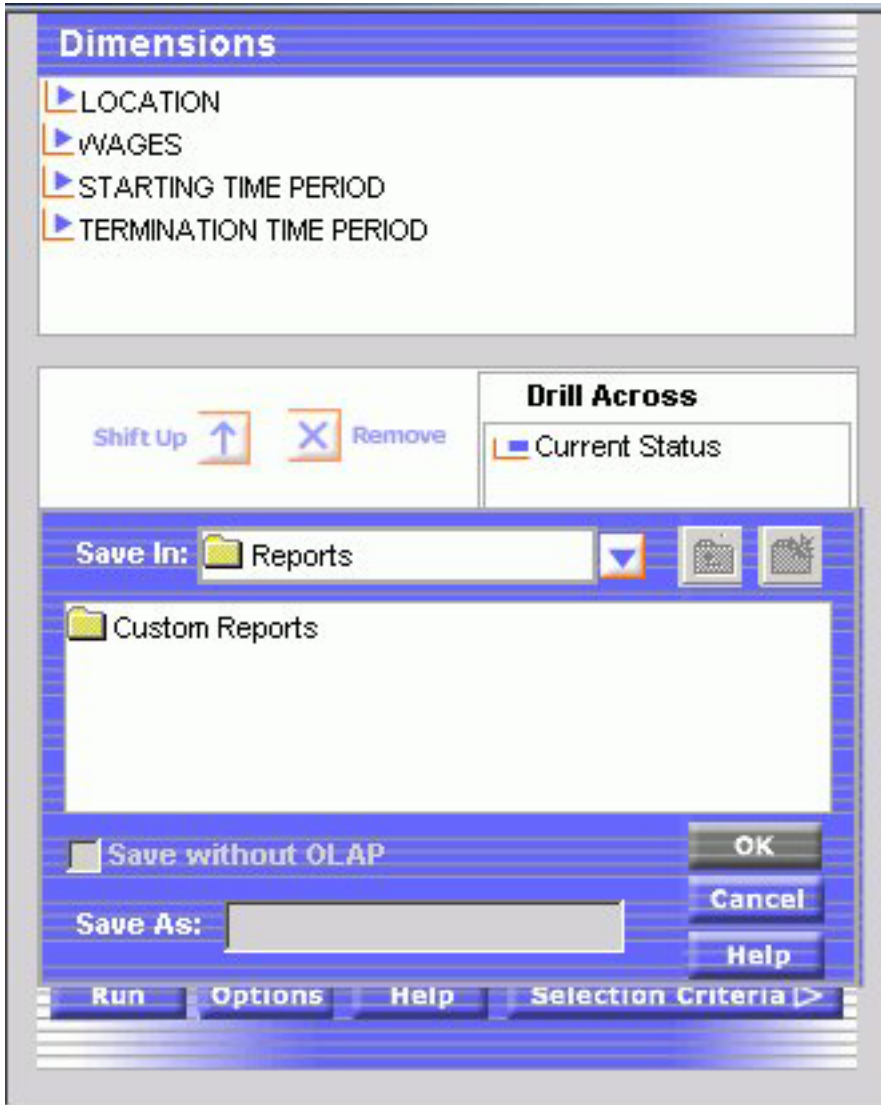
You are now presented with several saving options.

The following image shows the Options menu containing five options, Save the data in an Excel file, Save the data in an Excel 2000 file, Save the data in an Excel 2000 file with formulas, Display as a PDF Report, and Save Report.

Save the data in an Excel file
Save the data in an Excel 2000 file
Save the data in an Excel 2000 file with formulas
Display as a PDF Report
Save Report

2. Click **Save Report** to open a new dialog box as shown in the following image.

Save In: Reports

Custor | Reports
        | Other Files

Save without OLAP

OK

Cancel

Save As:

Help

3. Click **Reports** to enable the New Folder icon in the upper right corner as shown in the following image.
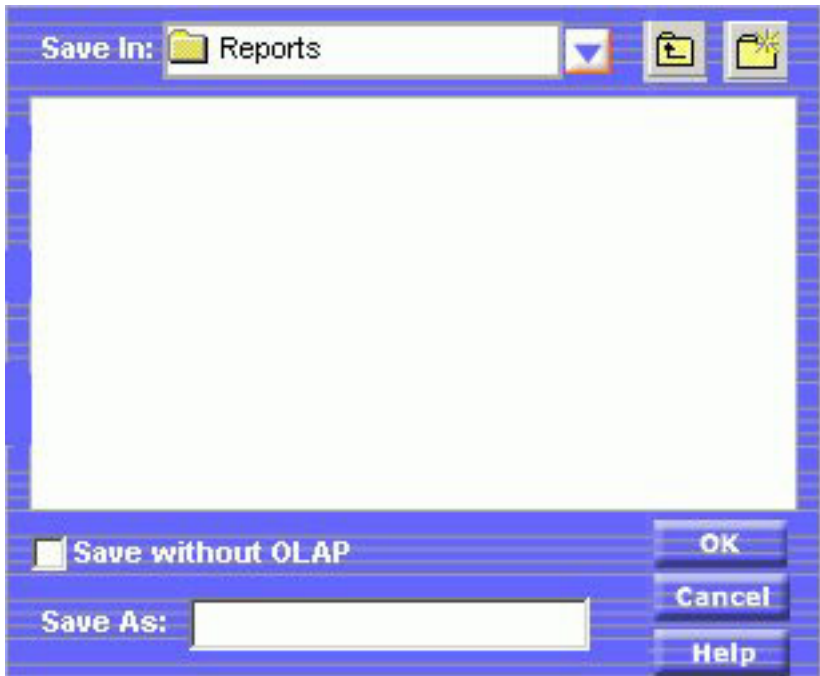
4. A pop-up prompts you to create a new folder, for example, a subfolder called OLAP Reports as shown in the following image.

5. Name your report in the Save As box and click **OK** to return to your report.

Save an OLAP-enabled Report as a Plain Report Without OLAP Functionality

Users can save their reports with the OLAP functionality removed, so the OLAP tool can be used as a simple report writer tool.

1. Create a subfolder by following these procedures: Save a Report in OLAP.

2. After a subfolder is selected, check the **Save without OLAP** box.
3. Enter a new name for your report in the **Save As** box and click **OK**.

DB2 Web Query generates a new report without OLAP functionality. The following image shows a normal report of employees' pay levels which is sorted by employee's last name.

Uniform Field Name Referencing in OLAP

However the developer designs the report with regard to field referencing will carry through to both the OCP and the OLAP Selections pane. Field referencing does not differ between the report and the OCP and OLAP Selections pane. Field references by AS, TITLE, or Field Name, are uniform in the report output and OLAP controls.

Saving and Displaying OLAP Reports and Graphs in Other Formats

OLAP reports and graphs appear in your browser in HTML format. You can display the report and corresponding graph in PDF and Excel formats, and in folders within Managed Reporting.

The following image shows a pop up menu listing the available formats to save and/or display.
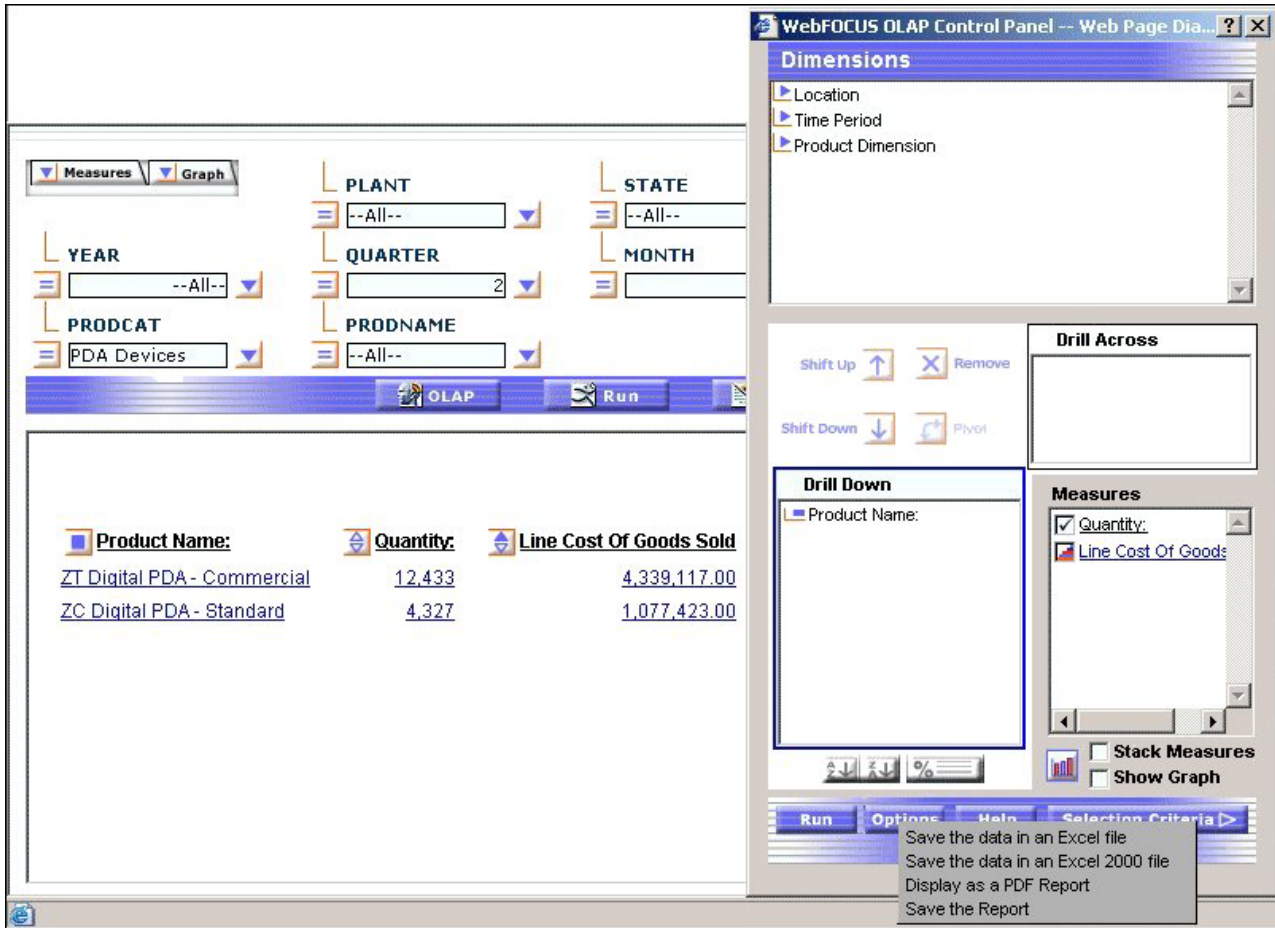
- PDF (Adobe Acrobat's Portable Document Format) is useful when you want a report or graph to maintain its presentation and layout regardless of a user's browser or printer type.

  When you choose PDF format, the report appears in Adobe Acrobat Reader; the graph continues to appear above it in a browser window. If you print from Acrobat, only the report will be printed.

- Excel is useful when you want to convert a large database to a spreadsheet or save a report and graph in a commonly used Office tool. Two Excel formats are available:

  – Excel 2000 supports most StyleSheet attributes, allowing for full report formatting. The computer on which the report is being displayed must have Microsoft® Excel 2000 or higher installed.

    When you choose Excel 2000, the report and graph are displayed in the same tool where you can manipulate the data using Excel options. From Excel you can print both the report and the graph.

    When you save in Excel 2000 format, only explicit drill-downs (based on parameters passed from the base report to the drill-down report) continue to work. Automatic drill downs on Dimensions and Measures are not supported in Excel.

  – Excel is a binary display format with limited formatting support. The computer on which the report is being displayed must have Microsoft Excel installed.

    Drill-downs of any kind are not supported.

In Managed Reporting, a user can save the HTML output in the Reports folder.

Display an OLAP Report and Graph in PDF Format

1. Open the Control Panel.
2. Click the **Options** button at the bottom of the window.
3. Select **Display as a PDF Report**.

   The graph appears in the browser above the report, while a second browser opens and launches the report output in Adobe Acrobat as shown in the following image.
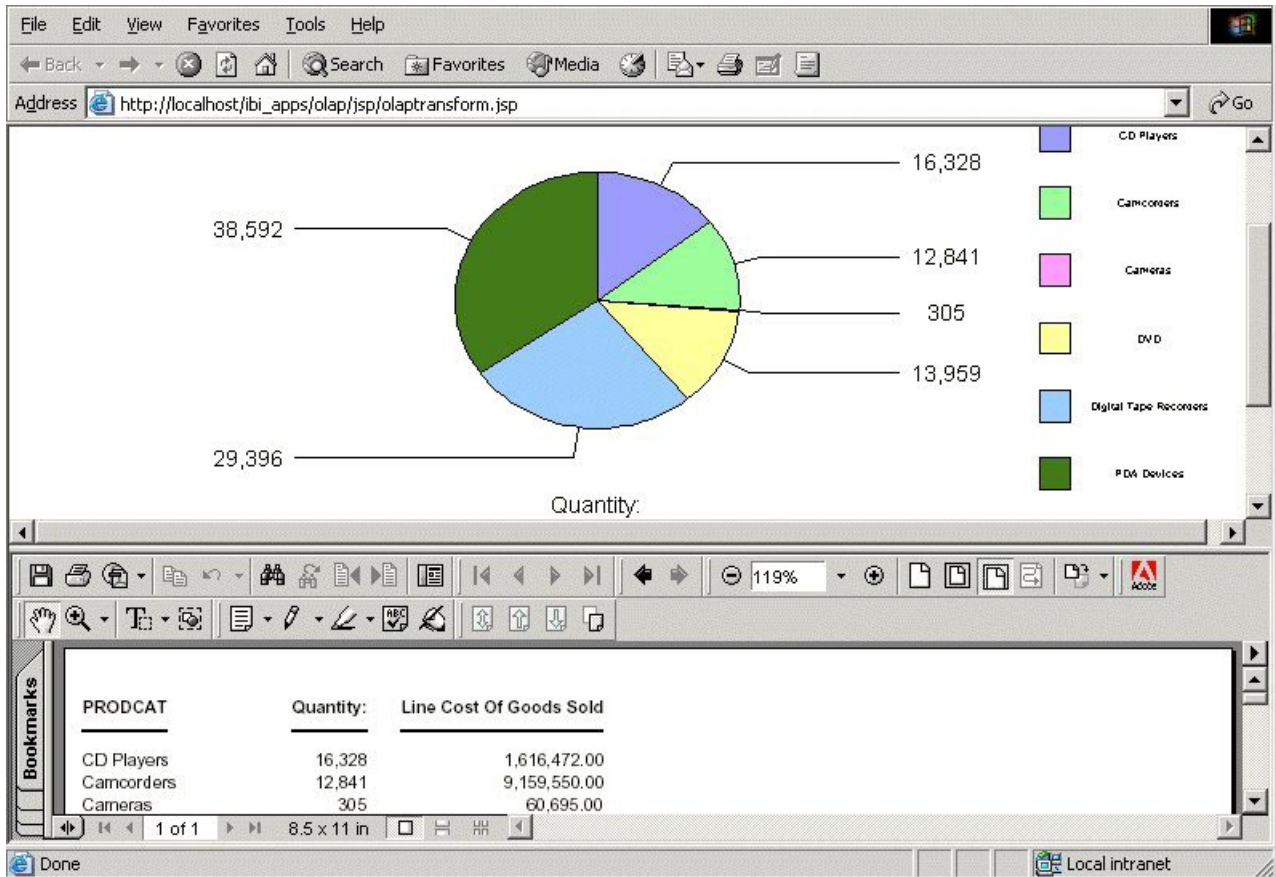
Tip: If you wish, you can save and print the PDF report from Adobe Acrobat.

Save an OLAP Report and Graph as an Excel File

1. Open the Control Panel.
2. Click the **Options** button at the bottom of the window.
3. Select **Save the data in an Excel file** or **Save the data in an Excel 2000 file**.
4. Follow the instructions to export the data.

Saving OLAP Reports and Graphs in the Reports Folder

You can save an OLAP report and graph in your Reports folder.

1. Open the Control Panel.
2. Click the **Options** button at the bottom of the window.
3. Select **Save as My Reports**.

A secondary window opens.

4. Enter a descriptive name and click **OK** to save the graph(s) and the tabular report.

Note: There is no limit to the number of characters in a graph legend's label, but long labels may appear truncated.

Chapter 6. Creating a Graph With Graph Assistant

Describes how to use the HTML Graph Assistant to create graphical reports.

This chapter provides information on graphs and how to use the Graph Assistant.

Graph Overview

Graphs often convey meaning more clearly than data listed in tabular format. Using the Graph Assistant, you can easily transform almost any type of data into an effective graph that you can customize to suit your needs.

You can link your graph to other resources. You may also select from a multitude of graph styles, which include the standard graph formats bar, line, pie, and scatter as well as many variations on these types.

You can also represent data graphically using data visualization. For details, see Visualizing Trends in Reports.

Content Analysis: Determining Graphing Objectives

DB2 Web Query offers a range of reporting tools that allow you to create reports that deliver critical information to your users. By selecting a tool that is well suited to your particular needs, you can design the information you deliver to users. One effective option with almost any type of data is a graphical presentation.

Graphs enable you to display multivariate or complex data efficiently, precisely, and in a way that a viewer can intuitively grasp. A graph is an effective presentation tool because it presents a visual idea, communicating meaningful changes in data to a user in a memorable way. By viewing your graph, a user can identify and track a change that you want them to notice.

Creating a meaningful graph is not simply a matter of applying aesthetics to your data. Instead, graphs allow you to design your presentation to capture the essential information in your data.

The first step in creating excellent graphs is determining your graphing objectives. You can break this process into several stages.

1. Assess your data:

   Look for meaningful patterns or changes in the data. Does your data change most dramatically over time or in relationship to some other value? Are there two sets of data that you would like to compare to each other?

   Determine what movement or changes you would like to highlight. Which of the patterns in the data would you most want the viewer to picture?

2. Select the graph type and style that best suits your argument and the overall shape of your data. Determine what will lead viewers to the cognitive task/connection that you want them to make.

3. Begin developing your graph.

4. Refine your graph:

   Are the labels meaningful/useful?

How can the data be organized in a meaningful way? Consider customizing the scales you use with your graph.

Accessing the Graph Assistant

HTML Graph Assistant is accessible:
- During the creation of a new Report.
- When editing a Report within Domains.

The HTML Graph Assistant is best viewed using a resolution of 1024 x 768.

You may be prompted for a DB2 Web Query Reporting Server ID.

Note: If you are using the HTML Graph Assistant with a Firefox browser, the display may be slightly different than Internet Explorer. These differences do not effect functionality.

Options Available on All Tabs

The following options are available from any tab in the Graph Assistant:

Save/Save As

Click Save to save your graph. Click Save As to open the Save New Report dialog box.

Run

Click run to run your graph.

Help

Click the icon to access help about the area of the tab in which you are working. Click About to open a window describing the version of Graph Assistant in which you are working.

Quit

Click to quit the application.

Selecting a Graph Type and Style

When creating a graph, it is important to select the appropriate graph type with which to display your data. You may select from a number of basic graph types, as well as refinements on these types, known as graph styles. Basic graph types include line graphs (connected point plots), bar graphs, pie graphs, and scatter graphs. Use the brief descriptions (see Graph Types) to select a graph type that suits the data set you are displaying and the change you want to highlight. Keep in mind that the data are the sets of numbers that you are displaying, and the scales are the numbers or variable measures that appear along the axes of the graph.

The Graph types tab of the Graph Assistant provides a list and brief descriptions of the many graph types and graph styles available in DB2 Web Query.

Note: When using a stacked chart of any type at least 2 series are required. Negative values are not supported in stacked charts.

The following image shows the Graph types tab with a sample cluster bar chart.

Save 🖫 ▾   Run ▶ ▾   Help ❓ ▾   Quit ✖   Graphing from file: CAR

| Graph types | Field selection | Headings | Selection criteria | Join options | Properties |

Options

Select from the following graph types

BAR ▾

Select from the following Graph Styles:

Bi-Polar Clustered
Bi-Polar Stacked
Clustered
Dual-Axis Clustered
Dual-Axis Stacked
Percent
Stacked

General options

◉ Vertical   ☐ Use 3D effect

○ Horizontal

Apply template

None ▾

Reset all styling information
back to default values   Reset

☑ Automatically arrange the chart elements
so that they make the best use of
available space

Graph type description

A Clustered Bar chart consists of side-by-side groups of bars. This is the standard type of two-dimensional bar chart.

LOOKGRAPH type VBAR

Show graph preview

Note:
This image is just an example of the selected graph type. To see an actual preview press the "Show graph preview" button.

Graph Types

Following are descriptions of the types of graphs you can create:

- **Line graphs.** Line graphs are useful for emphasizing the movement or trend of numerical data over time, since they allow a viewer to trace the evolution of a particular point by working backwards or interpolating. Highs and lows, rapid or slow movement, or a tendency towards stability are all types of trends that are well suited to a line graph.

  Line graphs can also be plotted with two or more scales to suggest a comparison of the same value, or set of values, in different time periods. The number of scales your graph has depends on the type of graph you select. There is a description of each available graph type on the Graph types tab of the Graph Assistant.

- **Bar graphs.** Bar graphs plot numerical data by displaying rectangular blocks against a scale. The length of a bar corresponds to a value or amount. Viewers can develop a clear mental image of comparisons among data series by distinguishing the relative heights of the bars. Use a bar graph to display numerical data when you want to present distributions of data. You can create horizontal as well as vertical bar graphs.

- **Pie graphs.** Pie graphs emphasize where your data fits in relation to a larger whole. Keep in mind that pie graphs work best when your data consists of several large sets. Too many variables divide the pie into small segments that are difficult to see. Use color or texture on individual segments to create visual contrast.

- **Scatter graphs.** Scatter graphs share many of the characteristics of basic line graphs. Data can be plotted using variable scales on both axes. When you use a scatter graph, your data is plotted using a basic line pattern. Use a scatter graph to visualize the density of individual data values around particular points or to demonstrate patterns in your data. A numeric X-axis, or sort field, will always yield a scatter graph by default.

  It is important to note that scatter graphs and line graphs are distinguishable from one another only by virtue of their X-axis format. Line graphs can appear without connecting lines (making them look like scatter graphs) and scatter graphs can appear with connecting lines (making them look like line graphs).

- **Area graphs.** Area graphs are similar to line graphs except that the area between the data line and the zero line (or axis) is usually colored or textured. Area graphs allow you to stack data on top of each other. Stacking enables you to highlight the relationship between data series, showing how some data series approach or shadow a second series.

- **3D graphs.** 3D graphs add dimension to your graphing presentation. Dimensionality enables your viewers to recognize trends based on two or more data sets easily. 3D graphs also add impact to your presentation.

- **Bubble charts.** Bubble charts are used to display three dimensions, requiring only three COLUMN fields representing X, Y, and Z data values in that order. The data points can be either opaque or transparent bubbles. If you want to visualize the precise data point, you can add a dot in the center of the bubble. The size of the bubbles are proportional to the values they represent.

- **Polar graphs.** The Graph Assistant supports two styles of Polar graphs: an XY Polar Chart or a polar coordinate scatter chart and an XY Polar Dual-Axis Chart or a Dual-Y polar coordinate chart. In both styles, only one Column field is allowed in the following order: X (degree) for the Column field and Y (distance from the center) for the Across/By field.

- **Radar graphs.** Radar graphs are used to compare two or more data sets. You can use axes or polygons to represent values in a star or spider configuration. They are essentially analogous to a line chart, except that the scale wraps around. Radar graphs work well with any data that are cyclical, such as the months of a year.

- **Stock Chart.** Stock charts are used to track the trend of a particular stock. Stock charts show the stock's volume, opening and closing values as well as the stock's high and low values over a specific time period. The data is represented by sets of bars and/or lines.

- **Histogram.** Histograms depict the distribution of a data set by grouping all of the data together and assigning it to buckets based on their value. You only measure one variable in histograms. After the data is sorted by this variable, you count the number of instances of this data. This determines the height of each bar. For example, if a company wanted to measure how many franchises there are in each class of annual revenue, a histogram would show the number of franchises that have 1,000,000 to 2,000,000; 2,000,000 to 5,000,000; 5,000,000 to 10,000,000; and 10,000,000 to 15,000,000.

- **Multi-y.** A Multi-y is a vertical, triple/quadruple/quintuple-Y-axis clustered bar chart. Any series can be assigned to any of the triple/quadruple/quintuple axes.

- **Waterfall.** A Waterfall chart is a cumulative stacked chart. The waterfall chart will automatically perform the cumulative sum when using Subtotal or Total. Waterfall charts essentially require one data value for each series or group marker to be drawn in a chart.

- **Miscellaneous.** The Miscellaneous Graph Type includes a variety of Graph Styles:

- **Default (Circular Gauge).** A Gauge Chart usually indicates the current position of a value within a given spectrum.
- **Funnel.** A Funnel Chart is basically a pie chart displaying only one group of data at a time from the first series to the last series at the bottom of the funnel.
- **Pareto.** The X-axis scale is for the group members. The Y-axis scale is the percent of the total accumulation for each selected series.
- **Product Position.** A Product Position chart provides a visual representation of market share and growth versus revenue and measurement (past, present, and future). Product Position charts require a set of three display fields.
- **Resource Return.** Resource charts plot X- and Y-axis data cumulatively. They are useful for plotting two independent variables against each other, particularly in cases where one or both of the axes specify percentages and the user wants to examine issues such as capacity utilization.
- **Spectral Map.** This is a Spectral Map Chart. This is a chart with row or column matrix of markers that are colored according to data values.

Graph Styles

You can choose among several graph styles for each graph type. First you select the graph type with which you want to work in the Graph Type drop-down list box, and then you select the graph style from the Graph Styles drop-down list box. A thumbnail graph with a graph type and style definition appears to the right on the Graph types tab.

Previewing Styling Options for Your Graph

The graph in the Graph Preview window displays the graph type and style you chose as well as any styling information you specified from the template drop-down list or in the Properties tab. The Preview does not open by default for other resolutions. In this case, you have to click the Show Preview button.

The following image is of a bar graph that uses the 3DBAR graph style. This generic chart is a two-dimensional chart with three-dimensional type bars.
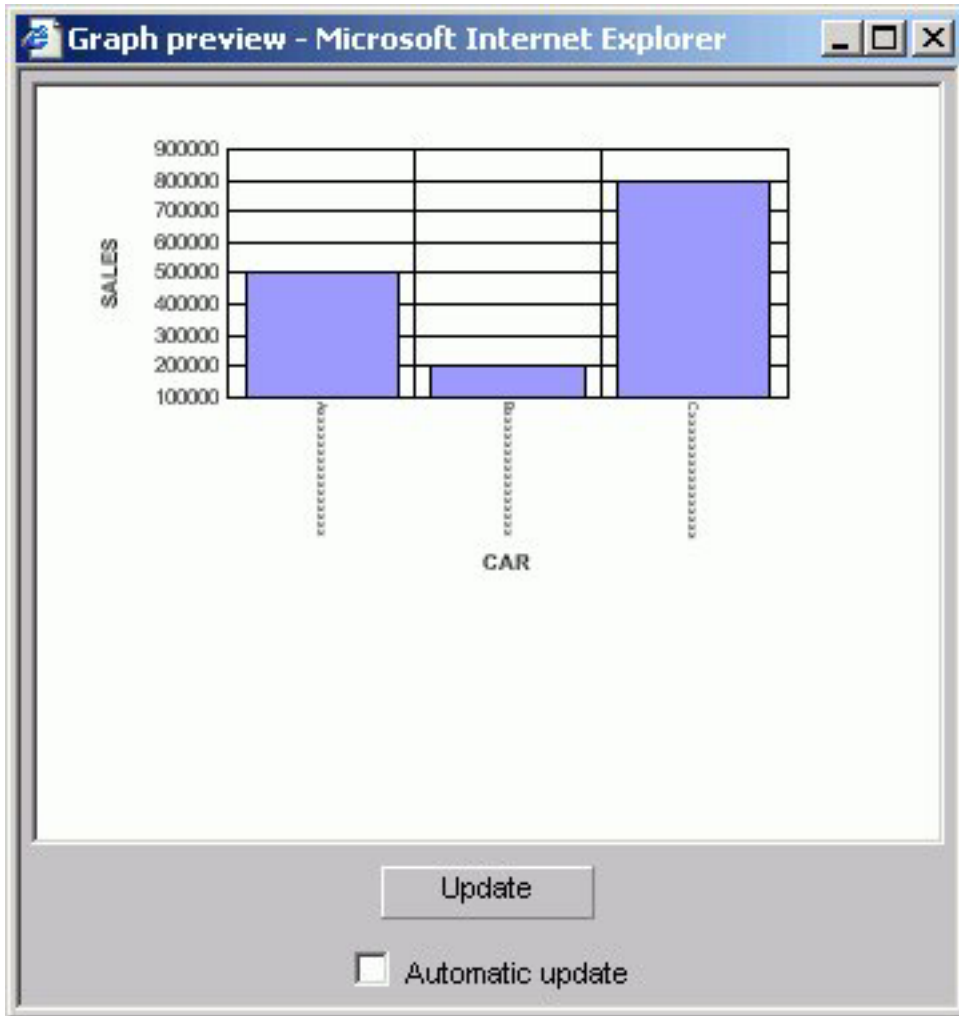
The graphical representation in the Graph Preview is based on an internal algorithm to help show the type and style of the graph. It is not a true representation of your particular data. However, legends and axis labels will reflect selected fields, formats, and positioning. The data used for this Graph Preview is built with the selected field format types and lengths of the data. In this case, the X-axis field (car) is alphanumeric (A16) and will be displayed as the series label

where:

A, B, C

> Represents the data type alphanumeric.

xxxxxxxxxxxxxxxx

> Indicates the number of characters or digits allowed.

The Y-axis (dealer cost and retail cost) scale is based on a data range of selected Y-axis fields.

You can manually update the Graph Preview by clicking the Update button to see how your changes will look in a graph. You can choose to have the Graph Preview updated automatically any time you make changes by selecting the Automatic update check box.

Important: Merged graphs are not supported in the Preview panel.

Auto Arranging Graph Elements

The auto arrange feature enables the Graph Preview to arrange all the elements in a graph report efficiently. The auto arrange feature is checked by default.

Arrange the Elements of a Graph Efficiently

Check the **Automatically arrange the chart elements so that they make the best use of the available space** option.

Note: This option is checked by default. If you uncheck this feature, DB2 Web Query will arrange the elements.

Comparing the Arrangement of the Elements of a Graph

In the Graph types tab:
1. Uncheck the **Automatically arrange the chart elements so that they make the best use of the available space** option.
2. Click **Show Graph Preview**.

   The Graph preview window opens. The following image shows the Graph types tab with a sample bi-polar stacked bar chart of a sales report with sales and mpg dimensions.

Note how the MPG symbol partially blocks the data along the right Y-axis.

3. Now check the **Automatically arrange the chart elements so that they make the best use of the available space** option.

A new Graph preview window opens. The following image shows the same bi-polar stacked bar chart with the elements arranged automatically.

Note how the Y-axis and X-axis labels fit within the size of the report.

Select a Graph Type and Style

1. Highlight a graph type from the Graph Type list on the Graph types tab of the Graph Assistant. Next, highlight a graph style from the Graph Styles drop-down list.

   A thumbnail image and a brief description of the graph are provided when you highlight a graph type and style. Be sure to read the description of the graph before selecting a graph type and style. Certain graph types require a particular number and/or type of data values; therefore, if your data does not satisfy the requirements, your graph will not accurately represent your data.

2. After you have found the graph type and style you desire, click the next tab to continue creating your graph.

Selecting Scales

After you have chosen a graph type, you should select an appropriate scale. A scale is a classification scheme or series of measures that you select for application to the axes of your graph. The scale provides the framework against which your data are plotted. When you choose an appropriate scale for your data, meaningful patterns can emerge, and when you modify a scale, the overall shape of your graph changes.

Steps or measures in the scale are represented along the axes of your graph by marks. The type of scale you choose determines the number of divisions along the scale. There are two general types of scales you can apply to the y-axis of your graph:

- Linear scales
- Logarithmic scales

A linear scale is a scale in which the values increase arithmetically. Each measure along the scale is one unit higher than the one that precedes it. Linear scales are useful when the data you are plotting are relatively small in range.

A logarithmic scale is a scale in which the values increase logarithmically. Each measure along the scale represents an exponential increase in the data value. Logarithmic scales are useful when you need to accommodate a large range of numbers.

Select Scales

1. Click the **Properties** tab of the Graph Assistant.
2. Click the **Y axis** subtab.
3. Find the **Log Scale** check box. When this option is checked, the Y-axis scale uses logarithmic scaling. When unchecked, the Y-axis scale uses linear scaling.

Selecting Values for the X and Y Axes

The values you select for the X- and Y-axes determine what data is included in the graph you are creating, and how it appears.

You can also:

- Select a second horizontal category (X-axis), which will produce multiple graphs. For details, see Creating Multiple Graphs.
- Temporarily hide the display of a Y-axis field. For details, see Hiding the Display of a Y-Axis Field.
- Interpolate X and Y-axis values using linear regression.

You can search fields lists. For details, see Searching Field Lists.

Note: Field names that contain a hyphen "-" cannot be used in the HTML Graph Assistant.

Notes on the Fields List

- Key fields are represented with a key icon in all field lists (tree and list displays) in the HTML Report and Graph Assistant.
- The number of fields that are currently displayed shows at the bottom of the fields list. If a field list contains more than 200 fields, the first 200 fields display initially. Additional fields are retrieved and displayed when you select the next set of fields.

  To select the next set of fields, scroll to the bottom of the field list and select the Next option that appears. The Next option describes which fields will be returned, for example in the following image it shows that the next 201-400 fields of 417 will be retrieved:

  To go back, scroll to the top of the field list and select the previous set of fields. For example, the following image shows the Previous option to select the previous 1-200 of 417 fields:

Name △	Alias
Previous (1-200) of 417	▲

- If you are viewing a Fields List in the Dimension Tree mode and you create a recursive join (joining a file to itself), you will not be able to view the Dimensions in the Fields List after the join is created. This is due to duplication of field names in a recursive join. If you delete the join, you will be able to view dimensions again.

Select Values for the Y-axis

1. Click the **Field selection** tab from the Graph Assistant.
2. If available, click the **List** or **Tree** icon.
3. Select the field from the Available fields window and click the **Add** icon to the Field value(s) plotted on Y-axis fields box.

   or

   Click and hold the left mouse button, and drag the field to the Field value(s) plotted on Y-axis fields box.

   or

   Press **Alt+C** to go the Field value(s) plotted on Y-axis fields box. Then press **Alt+M** to go to the Available Fields window, tab to the field and press **Enter** to add the chosen field.

To add more than one field name at a time:

1. Ensure the **List** icon is selected.
2. Click the field names you wish to include. The fields will highlight as you click on them.
3. Drag and drop or use the **Add** icon to add all the selected fields to the Field value(s) plotted on Y-axis fields box.

Note: The display field(s) determines the vertical Y-axis of the graph. When the number of Y-axis labels is greater than one, the labels do not appear along the Y-axis. Instead, they appear in a legend that provides the names of the fields being graphed.

Select Values for the X-axis

1. Click the **Field selection** tab from the Graph Assistant.
2. Select the field from the Available fields window and click the **Add** icon to the Field value(s) plotted on X-axis fields box.

   or

   Click and hold the left mouse button, and drag the field to the Field value(s) plotted on X-axis fields box.

   or

   Press **Alt+S** to go the Field value(s) plotted on X-axis fields box. Then press **Alt+M** to go to the Available Fields window, tab to the field and press **Enter** to add the chosen field.

Hiding the Display of a Y-axis Field

You can hide the display of a Y-axis field in a graph. This option is useful when you want to temporarily remove a particular Y-axis field while retaining all of the original graph properties.

To hide a Y-axis field temporarily, use the **Make this field invisible** check box located on the subtab of the Show graph field options of the Field Selection tab. By default, the check box is not selected for each Y-axis field. If there are no Y-axis fields, the Show graph field options subtab is disabled.

Hide the Display of a Y-axis Field

On the Field selection tab:
1. From the Field value(s) plotted on Y-axis fields box, highlight the Y-axis field you want to hide.
2. If hidden, click **Show graph field options**.
3. Check the **Make this field invisible** check box.

Assigning Characteristics to Y-axis Values

You can assign a characteristic for each Y-axis field in your graph. In the following table, the first column lists the prefix operators or characteristics, and the second column lists their descriptions.

Summary Type	Description
Field_Name	(Plain field value)
ASQ	Average Square
AVE	Average
CNT	Count
FST	Show first in group
LST	Show last in group
MAX	Maximum
MIN	Minimum
PCT	Percentage
PCT.CNT	Count Percentage
RPCT	Row Percentage
TOT	Total

Assign Characteristics to Y-axis Values
1. From the Field selection tab, select **Summary Type** under Y-axis values.
2. From the Field value(s) plotted on Y-axis fields box, select the field you would like to assign a characteristic (or prefix operator) to.
3. If hidden, click **Show graph field options**.
4. Select the characteristic from the Summary Type list box.

The field name in the Field value(s) plotted on Y-axis fields box is updated with the appropriate prefix. For example, if you have a field named SEQ_NO, and you select the Average Square summary type, the field name changes to ASQ.SEQ_NO.

Selecting Records for a Graph

When generating a report and specifying which fields to display, you may not want to show every instance of a field. By including selection criteria, you can display only those field values that meet your needs. In effect, you can select a subset of the data, which you can easily redefine each time you issue the report request.

When developing a report request, you can define criteria to select records based on:

- The values of an individual field (using a Where statement).
- The aggregate value of a field, for example, the sum or average of a field's values (using a WHERE TOTAL statement).

The Selection criteria tab enables you to create Where or Where Total statements for record selection.

Creating a Where or Where Total Statement

Where and Where Total statements enable you to display only those records that pass your selection criteria. These statements select the data source records to be included in a report. The data is evaluated according to the selection criteria before it is retrieved from the data source. Use the Selection criteria tab to create a new Where or Where Total statement.

Note: In the Graph Assistant, when you create a Where statement in the Selection Criteria tab for a range of dates using FROM - TO, you will not be alerted if the TO field is chronologically before the FROM field. For example, if you enter something like "from 12/31/05 to 1/1/05". There will be no records in the resulting report.

Select Records Based on Values, Parameters, Fields, or a Constant

1. In the Graph Assistant, click the **Selection criteria** tab.
2. Select a field from the Available fields window and drag the field to the **Screening conditions** pane.

    or

    Press **Alt+M** to go to the Available Fields window, tab to the field and press **Enter** to add the chosen field.

    The field name and Equal to and Select values appear in the Screening conditions pane.
3. Select **WHERE** or **WHERE TOTAL**.
4. Select an operator from the drop-down list (for example, **Equal to**) in the right pane.
5. Click **Select values** to complete the expression.
6. Required: Select one of the following compare type option buttons:
    - **Constant.** Enter a value that will serve as a constant (a value that will not change). For example, enter vehicle.

        After you click **OK**, the constant, vehicle, appears in the WHERE statement which reads BODYTYPE Equal to vehicle.
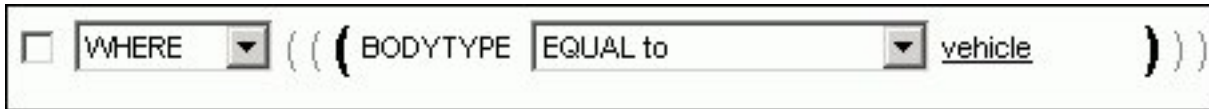
- **Parameters.** Enter the name of the variable. For example, moped as shown in the following image with the Parameter option button selected and the variable name entered as moped.

You can set parameter properties, for details see Setting Parameter Properties.

After you click **OK**, the parameter, &moped, appears in the WHERE statement which reads BODYTYPE Equal to &moped.

| ☐ | WHERE ▾ | (((| BODYTYPE | EQUAL to ▾ | &moped |))) |

Note: The ampersand is automatically added by DB2 Web Query.

- **Fields.** Select a field from the list.

  After you click **OK**, the field BHP appears in the Where statement which reads CAR.COMP.CAR Equal to CAR.SPECS.BHP.

| ☐ | WHERE ▾ | (((| CAR.COMP.CAR | EQUAL to ▾ | CAR.SPECS.BHP |

- **Values.** Select a value for the field you initially chose (in this case, car).

  After you click **OK**, the value, Audi, appears in the Where statement which reads CAR.COMP.CAR Equal to AUDI.

| ☐ | WHERE ▾ | (((| CAR.COMP.CAR | EQUAL to ▾ | AUDI |))) |

Create a Where Statement Using Date and Time Values

In the Graph Assistant,

1. Click the **Selection criteria** tab.
2. Select a field from the Available fields window and drag the field to the **Screening conditions** pane.

   or

   Press **Alt+M** to go to the Available Fields window, tab to the field and press **Enter** to add the chosen field.
3. Click **Equal to** in the right pane to display a list of operators. Click the operator of your choice.
4. Click **Select values** to complete the expression.
5. Click **Field** to access date and time values for that field. For example, EYEAR.
6. Required: Click **Values** and click the value of your choice. For example, 1989/01/18.

   After you click **OK**, the value, 1989/01/18, appears in the WHERE statement which reads START_DATE Equal to 19890118.

| ☐ | WHERE ▾ | (((| START_DATE | EQUAL to ▾ | 19890118 |))) |

Importing Values From External Files for WHERE Statements

When developing a query, you can use a locally saved, external file as selection criteria to limit query results. This enables you to quickly build a query containing a large number of WHERE statement values without having to manually enter repetitive or readily available data.

Create a Value List From an External File

1. Required: Navigate to the **Selection criteria** area.
2. Required: Begin creating a WHERE statement by selecting a field from the list of available fields.
3. Required: Select a data comparison option that can accept multiple values.

   Valid options include EQUAL to, NOT EQUAL to, IN literal list, NOT IN literal list, EXCLUDES literal list, and INCLUDES literal list.
4. Click **Select values**.

   The Values dialog opens.
5. Click the **import** button as shown in the following image.

A dialog box opens as shown in the following image.

6. Select either the Flat file or Excel Spreadsheet (XLS) File Format option.
7. Click the **Browse** button.

A standard file selection dialog opens.

8. Select an external file to import from your local machine or network.

   Note: The external file must only contain text with new line delimiters.

9. Click **OK**.

   The values contained in the file are displayed in the Multiple values entered list in the right pane of the dialog.

10. Optionally, you can remove specific values from the Multiple values entered list or move values up or down within the list.

11. Click **OK**.

    You can save the query for future use and reopen the query to append new or remove existing values.

Setting Parameter Properties

When you create parameters for Selection Criteria (WHERE statements) you can set the properties for the parameter. For example, you can create a dynamic or static list of values to choose from at run time.

Set Parameter Properties

1. In the Graph Assistant, click the **Selection criteria** tab.
2. Create a WHERE or WHERE TOTAL expression.
3. Select an operator. Some operators are not supported with the Variable Editor, see Supported Operators for the Variable Editor for details.
4. Click **Select values**.
5. Select the **Parameter** option button.
6. Click the **Auto Prompt** button.

   The Parameter Properties dialog box opens.

7. Required: Choose the Selection type, **Dynamic** or **Static**.

   The Dynamic option is selected by default.

   The Name and Description fields are automatically populated based on the field you are screening on, but you can edit the populated values.

   Note: The Description field cannot contain a period (.) because a period is the delimiter to specify the beginning and end of the description value.

8. Required: Set the parameter values. For the:
   - Dynamic selection type, the Data Source and Field are selected by default based on the field you are screening on. You may change these as necessary.
   - Static selection type, select the **Constant** or **Show field values** option button. For constant, enter a value or values. For Show field values, all of the values for the selected field appear in the list box. Move the values you want in the selection list to the Prompt values list.

9. Click **OK** to exit the Parameter Properties dialog box.

Supported Operators for the Variable Editor

Operator	Enabled for the Variable editor?
Equal to	Yes
Not equal to	Yes
Equal to multiple	No
Not equal to multiple	No
Greater than	Yes

Operator	Enabled for the Variable editor?
Less than	Yes
Greater than or equal to	Yes
Less than or equal to	Yes
In literal list	No
Not in literal list	No
Missing	No
Not missing	No
From - to	No
Not-from - to	No
Includes literal list	No
Excludes literal list	No
Contains characters	Yes
Omits characters	Yes
Like character mask	Yes
Not like character mask	Yes

Combining Expressions

Use the right pane of the Selection criteria tab to use or delete existing Where statements and to combine expressions.

Combine Expressions

1. Create an expression.
2. When you create additional expressions, they are combined with the default operator AND as shown in the following image with two statements, the first a WHERE and the second an AND statement.
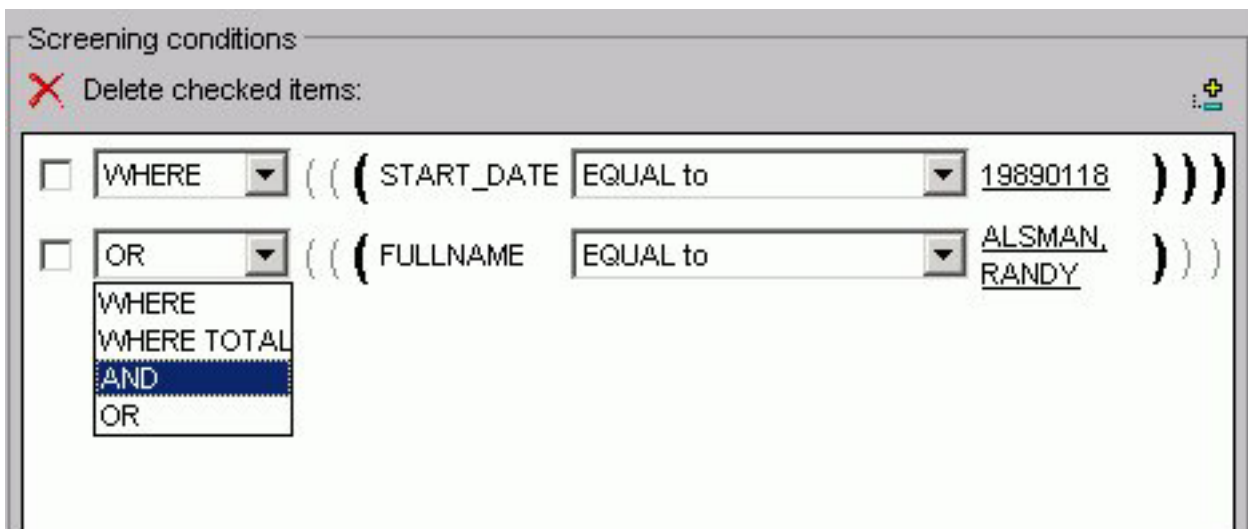
3. Toggle between AND, OR, WHERE, and WHERE TOTAL by clicking the AND to make your choice.

Delete an Active Expression

1. Select the check box next to the expression you want to delete. Click the **Delete** icon above the box.

2. You are prompted to delete the expression. Click **OK** to delete.

Grouping Expressions Together With Parentheses

You can use parentheses to group expressions together to optimize the Where statement.

Group Where Statements Together Using Parentheses

Click the grayed out parentheses to activate the parentheses and group expressions together.

Limiting Data With Filters

Filters enable you to quickly select predefined criteria that limit data included in a graph. Filters are selection criteria (WHERE statements) that an Administrator creates for you to apply as needed, without having to create your own selection criteria.

DB2 Web Query displays filters in filter groups. Each filter group can contain multiple filters. Selecting a single filter within one group creates a report with simple filtering criteria. By selecting multiple filters within a group or combining filters from different groups, you can create complex filtering expressions.

The following image shows a few sample filters as they appear in the Selection Criteria tab of the HTML Graph Assistant. They are listed by Group and Name.

Creating Temporary Fields

You can create temporary fields from the HTML Graph Assistant. Temporary fields are created the same way in the HTML Report and Graph Assistant. For complete details, see Creating Temporary Fields.

Creating Multiple Graphs

You can create multiple graphs by selecting a second horizontal category (X-axis).

The number of graphs created depends on the number of values in the field you select. For example, if you select a field with two values, two graphs are generated. If you select a field with ten values, ten graphs are generated.

You can select the second horizontal category from the Field selection tab in the Graph Assistant.

Multiple graphs can appear in either merged format or in columns. For details, see Merging Multiple Graphs and Displaying Multiple Graphs in Columns.

Create Multiple Graphs

1. Click the **Field selection** tab of the Graph Assistant.
2. Select the field from the Available fields window and click the **Add** icon to the Create a separate graph for each value of this field box.

   or

   Click and hold the left mouse button, and drag the field to the **Create a separate graph for each value of this field** box.

   or

   Press **Alt+A** to go the Create a separate graph for each value of this field box. Then press **Alt+M** to go to the Available Fields window, tab to the field and press **Enter** to add the chosen field.
3. If you want to change the title of the field name, click **Show graph field option** if hidden.
4. In the Axis label text box located under the General subtab, type the new field title name.

Merging Multiple Graphs

When you create a graph with a second horizontal category, multiple graphs are generated. You can merge these graphs into a single graph.

You can do this from the Field selection tab in the Graph Assistant.

Merge Multiple Graphs

1. Click the **Field selection** tab of the Graph Assistant.
2. Add a field to the **Create a separate graph for each value of this field** box.
3. If hidden, click **Show graph field options**.
4. Click the **Merge graphs into a single graph** option button. When you run your graph, the display shows an X, Y, and Z axis to represent the dual X-axis and the Y-axis.

Merging Multiple Graphs

The following image illustrates a graph with two categories (Manufacturing Plant and Product Name) that have been merged.

BOS

1600		
1400		
1200		
1000		
800		
600		
400		
200		
0		

CNT PRODCAT: BOS
CNT PRODCAT: DAL
CNT PRODCAT: LA
CNT PRODCAT: ORL
CNT PRODCAT: SEA
CNT PRODCAT: STL

ZC Digital PDA - Standard
QX Portable CD Player
Combo Player - 4 Hd VCR + DVD
AR2 35MM Camera 8 X
650DL Digital Camcorder 150 X
250 8MM Camcorder 40 X
150 8MM Camcorder 20 X
110 VHS-C Camcorder 20 X
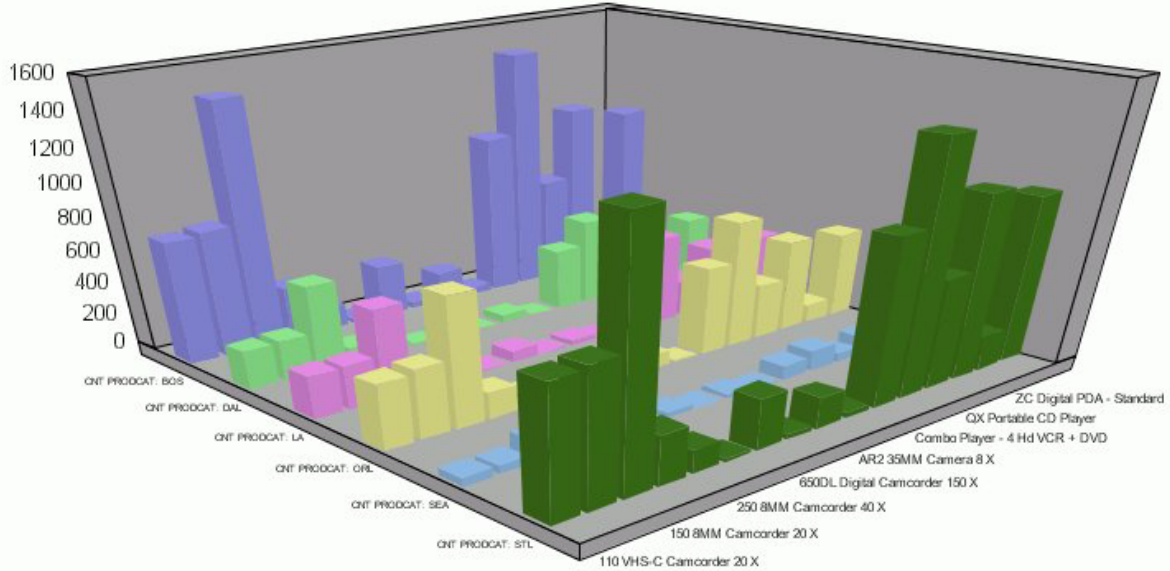
**Manufacturing
Plant**

**Product
Name:**

Displaying Multiple Graphs in Columns

When you create a graph with a second horizontal category, multiple graphs are
generated. You can display these graphs in columns.

Display Multiple Graphs in Columns

1. Click the **Field selection** tab of the Graph Assistant.
2. Add a field to the **Create a separate graph for each value of this field** box.
3. If hidden, click **Show graph field options**.
4. Select the **Display graphs in number of columns** option button. This is the
   default selection.
5. Select the number of columns to display the graphs. The default value of 1
   displays all of the graphs in one single column, one underneath the other.
   Selecting a number greater than 1 places the graphs in an HTML table, with the
   number of cells across the table corresponding to the number of columns
   specified.

Displaying Multiple Graphs in Columns

The following image shows a graph that has a second horizontal category. The
multiple graphs that are generated appear in 2 columns since the Display graphs
in columns option button was selected when creating the graph.

Working With Joins

A join is a temporary connection between two or more data sources that share at least one common field. Once you join two data sources, each time DB2 Web Query retrieves a record from the first data source (host file), it also retrieves the matching records from the second data source (target file).

You use the Join tool to link data sources in both Report Assistant and Graph Assistant. The Join tool provides a graphical method for creating and manipulating joins. You can also create define-based joins and multi-field joins.

When you access the Join tool, the window displays a field list for the host data source (which you specified when you created the report), and a field list for any target data source you add to the window.

Create a Join

1. Click the **Join options** tab in Report Assistant or Graph Assistant and click **New**.

   The DB2 Web Query masters list dialog box opens where you select a target data source.

2. Select a data source and click **OK**.

   The following image shows the Join settings window where you select fields from the Host field and the Target field lists to create a join.

3. Select the fields you want to join from the Host field list and the Target field list and add them to their respective Selected Fields boxes.

   or

   Press **Alt+M** to go to the Host field list.

   or

   Press **Alt+T** to go to the Target field list.

   Then, tab to the field and press **Enter** to add the chosen field to the corresponding Selected Fields box.

   Note: Fields must have the same format and be indexed fields in order to use them to create joins.

4. Select your join properties:

   • **Unspecified.** Indicates neither an inner nor left outer join.

   • **Inner Join.** A join that results when a report includes host rows that have corresponding cross-referenced rows.

   • **Left Outer Join.** Extends the results of an Inner Join and retrieves records from both host and cross-referenced tables, including all records from the left table (host) and any records from the right table (cross-referenced) where the condition values match. If there are no matching values in the cross-referenced table, the join still retrieves records from the host table.

   • **Multiple Instance (formerly Non Unique).** A one-to-many join structure that matches one value in the host data source to multiple values in the cross-referenced field. Joining employee ID in a company's employee data source to employee ID in a data source that lists all the training classes offered by that company would result in a listing of all courses taken by each

employee, or a joining of the one instance of each ID in the host file to the multiple instances of that ID in the cross-referenced file. The default join type is inner.

- **Single Instance (formerly Unique).** A one-to-one join structure that matches one value in the host data source to one value in the cross-referenced data source. Joining an employee ID in an employee data source to an employee ID in a salary data source is an example of a unique join.

5. To see the join syntax, click **View**.

6. Click **OK**.

7. In the Description box, give the join a meaningful and unique name.

8. Click **Save & Create**.

   The new join appears in the Joins list.

Create a Define-based Join

1. Click the **Join options** tab in Report Assistant or Graph Assistant and click **New**.

   The DB2 Web Query masters list dialog box opens where you select a target data source.

2. Select a data source and click **OK**.

3. Click the down arrow of the Define icon and then click **New define field**.

   The following image shows the list of Define field options.

The following image shows the Define field creator dialog box where you create the Define expression.

4. Create the defined field. The new define field appears in the Available fields list. The field is prefixed with the define symbol as shown in the following image.

5. Add the defined field and the target field to their respective Selected Fields boxes.

   Note: Fields must have the same format and be indexed fields in order to use them to create joins.

6. If you want to see the syntax, click **View** to display the define-based join syntax.

7. Give the join a meaningful and unique name in the Description box.

8. Click **Save & Create**. The new join appears in the Joins list.

Create Multi-field Joins

1. Click the **Join options** tab in Report Assistant or Graph Assistant and click **New**.

   The DB2 Web Query masters list dialog box opens where you select a target data source.

2. Select a data source and click **OK**.

3. Enter multiple fields in the Host field list box and one field in the Target field list box.

4. If you want to see the syntax, click **View** to display the multi-field join syntax.

5. In the Description box, give the join a meaningful and unique name.

6. Click **Save & Create**.

   The new join appears in the Joins list.

Delete a Join

1. Select a join from the Select an existing join list box.
2. Click **Delete**.

Edit a Join

1. Select a join from the Select an existing join list box.
2. Make your desired changes.
3. Click **Save & Create**.

Displaying Missing Data Values in a Graph

You can display missing data values (in a bar graph, line graph, area graph, or any variation of these graph types) in one of the following formats:

- **Graph as zero.** In bar graphs, a bar appears on the zero line. In line graphs, a solid line connects the missing value with the succeeding value. In area graphs, the area appears on the zero line.
- **Graph as gap.** In all graph types (bar, line, or area), missing values appear as a gap in the graph.
- **Dotted line to zero.** In line graphs, a dotted line connects the missing value with the succeeding value. In 3D bar graphs, solid lines outline the flat bar corresponding to the missing value. In 2D bar graphs, a gap appears in the graph. In area graphs, a transparent area extends down to the zero line and then up to the succeeding value.
- **Interpolated dotted line.** In a line graph, missing values appear as an interpolated dotted line that connects the plot points immediately preceding and succeeding the missing value. In bar and area graphs, missing values display as an interpolated (transparent) bar or area.

Display Missing Values in a Graph

1. In the Graph Assistant, click the **Properties** tab.
2. Click the **Y-axis** subtab.
3. Required: From the Missing values display list box, select one of the following values:
   - Graph as zero.
   - Graph as gap.
   - Dotted line to zero.
   - Interpolated dotted line.

Displaying Missing Values as Zero in a Graph

The following image illustrates how missing values are represented in a bar graph when designated to appear as zero. The CURR_SAL values for McCoy and Stevens are missing.
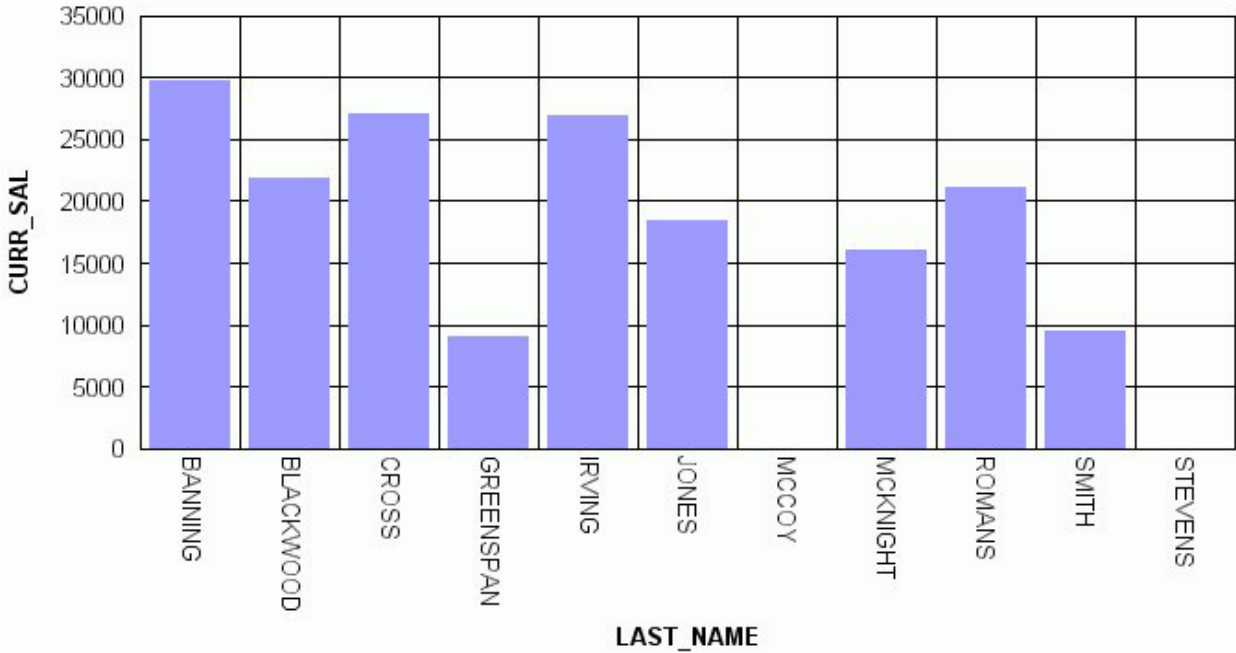
Displaying Missing Values as a Gap

The following image illustrates how missing values are represented in a bar graph when designated to appear as a gap. The CURR_SAL value for McCoy is missing.
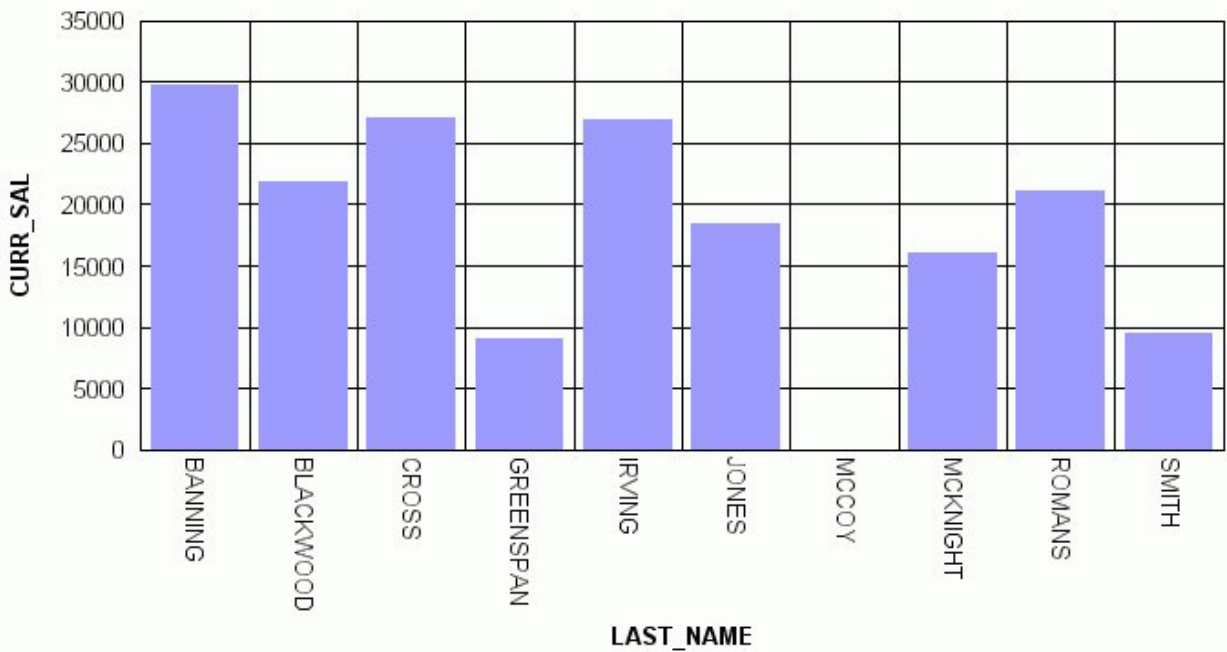
Displaying Missing Values as a Dotted Line to Zero

The following image illustrates how missing values are represented in a line graph when designated to appear as a dotted line to zero. The CURR_SAL value for McCoy is missing.
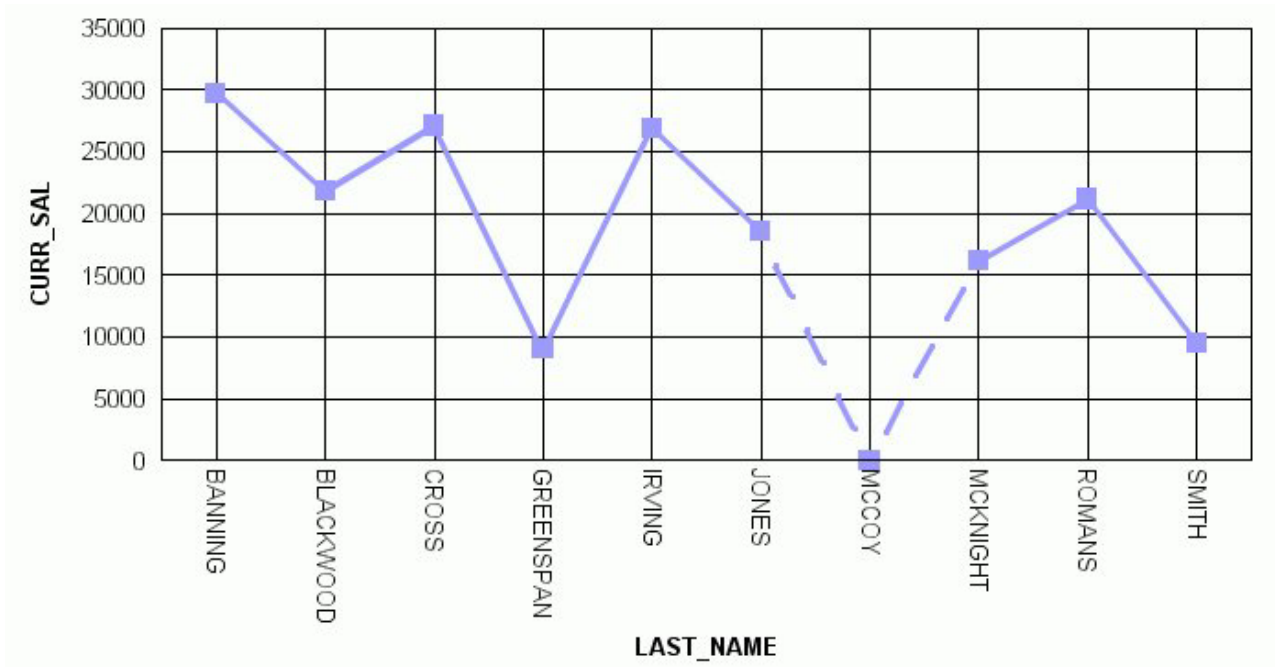
Displaying Missing Values as an Interpolated Dotted Line

The following image illustrates how missing values are represented in a line graph when designated to appear as an interpolated dotted line. The CURR_SAL value for McCoy is missing.
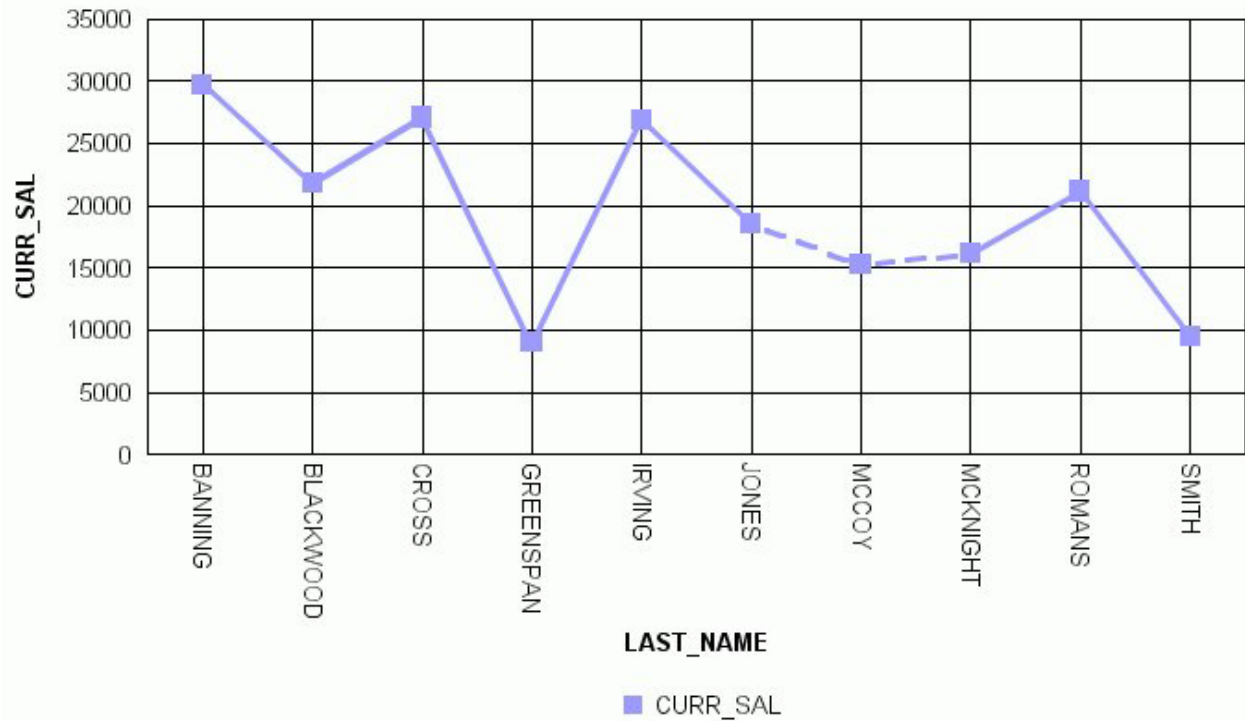
Applying Conditional Styling to a Graph

You can add further value to your graph by using conditional styling to highlight certain defined data with specific styles and colors. Conditional styling, also referred to as stoplighting, enables you to define conditions that determine when to apply particular fonts, point size, text style, foreground and background color, and drill down procedures to your report's data when the report is run.

For example, you can apply the color red to all departments that did not reach their sales quotas and apply the color black to all departments that did reach their sales quotas. In this example, the user can view quickly which departments did or did not reach their quotas. To examine how the results of one department may impact the results of a second department, you may want to provide a drill-down to a report that examines this possibility.

You can apply color to the following graph types:
- Bar graphs.
- Three-dimensional graphs with noncontinuous plot points.
- Pie graphs.
- Stack charts.

Conditional styling is only supported for Y-axis values.

Applying Conditional Styling to a Graph

The following illustrates how you can apply conditional styling to a graph. In this example:
- Blue is specified as the default color for all data in the graph.

- Firebrick is specified as the color for data in the DOLLARS column.
- Silver is specified as the color for data in the DOLLARS column when the value is greater than five million.
- Lime is specified as the color for data in the UNITS column when the value is less than two hundred thousand.

The output is shown in the following image.
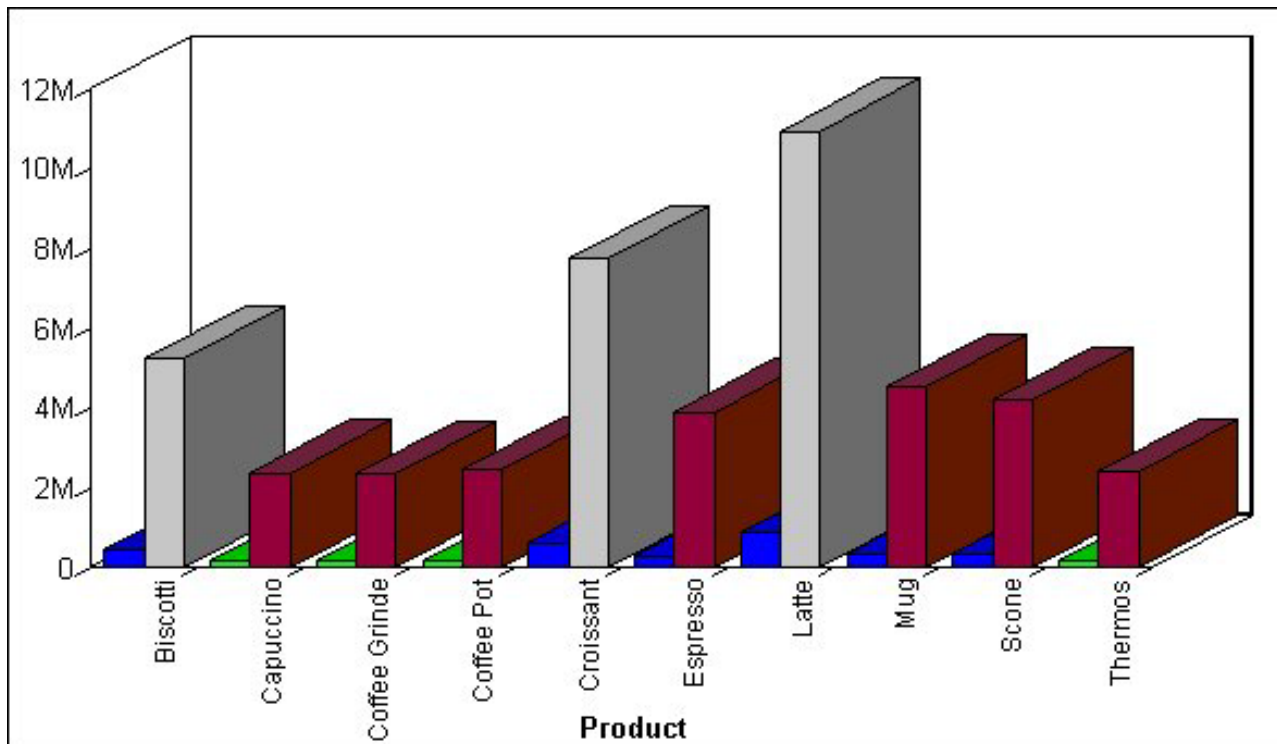
Linking Graphs to Other Resources

To drill down to a more detailed level of information in a graph, you can link a procedure (FOCEXEC) or a URL to one or more values in your graph. When you run your graph, the selected values become "hot spots" that invoke the underlying procedure or URL.

Link a Graph to Other Resources

1. Click the **Field selection** tab in the Graph Assistant.
2. Select a field from the Field value(s) plotted on Y-axis list.
3. If hidden, click **Show graph field options**.
4. Click the **Drill down** button. The Drill Down dialog box opens.
5. Select either **Execute Procedure (FOCEXEC)** or **Execute URL** option button.

   You can select the **No action** option button to disable a drill-down link.
6. If you are linking your graph to a:
   - URL, type the URL in the URL text box. If you don't want the URL to appear when a user moves the mouse over a link in the graph, you can enter different text in the Alternate Comment for Hyperlink text box.
   - Procedure, select the procedure from the hierarchy.

7. Use the **Add** button to add any parameters. For details, see Creating Parameters.

8. Click **OK**.

Note: You can disable drill-down by following steps 1-3 and then selecting the **No action** option button from the Drill Down dialog box.

Creating Parameters

Parameters enable you to specify criteria and conditions for the associated (drill-down) report. By defining parameters, you can control the amount and type of information to retrieve when you click on a hot spot.

Note: To avoid conflicts, do not name variables beginning with Date, IBI, or WF, as variables beginning with these values are reserved for IBM use.

Create a Parameter in the Main and Drill Down Procedures

1. Click the **Selection criteria** tab.

2. Create an expression (WHERE statement) that defines a parameter. For details on creating WHERE statements, see Selection Criteria Tab in HTML Graph Assistant.

3. Select the **Field selections** tab.

4. Select the Sort by field that you want to drill down on, for example, Product.

5. Click **Drill down**. The DB2 Web Query Drill Down dialog box opens.

6. Click **Execute Procedure (FOCEXEC)** or **Execute URL**.

   The following image shows a sample drill down dialog box with Execute Procedure (FOCEXEC) option selected.
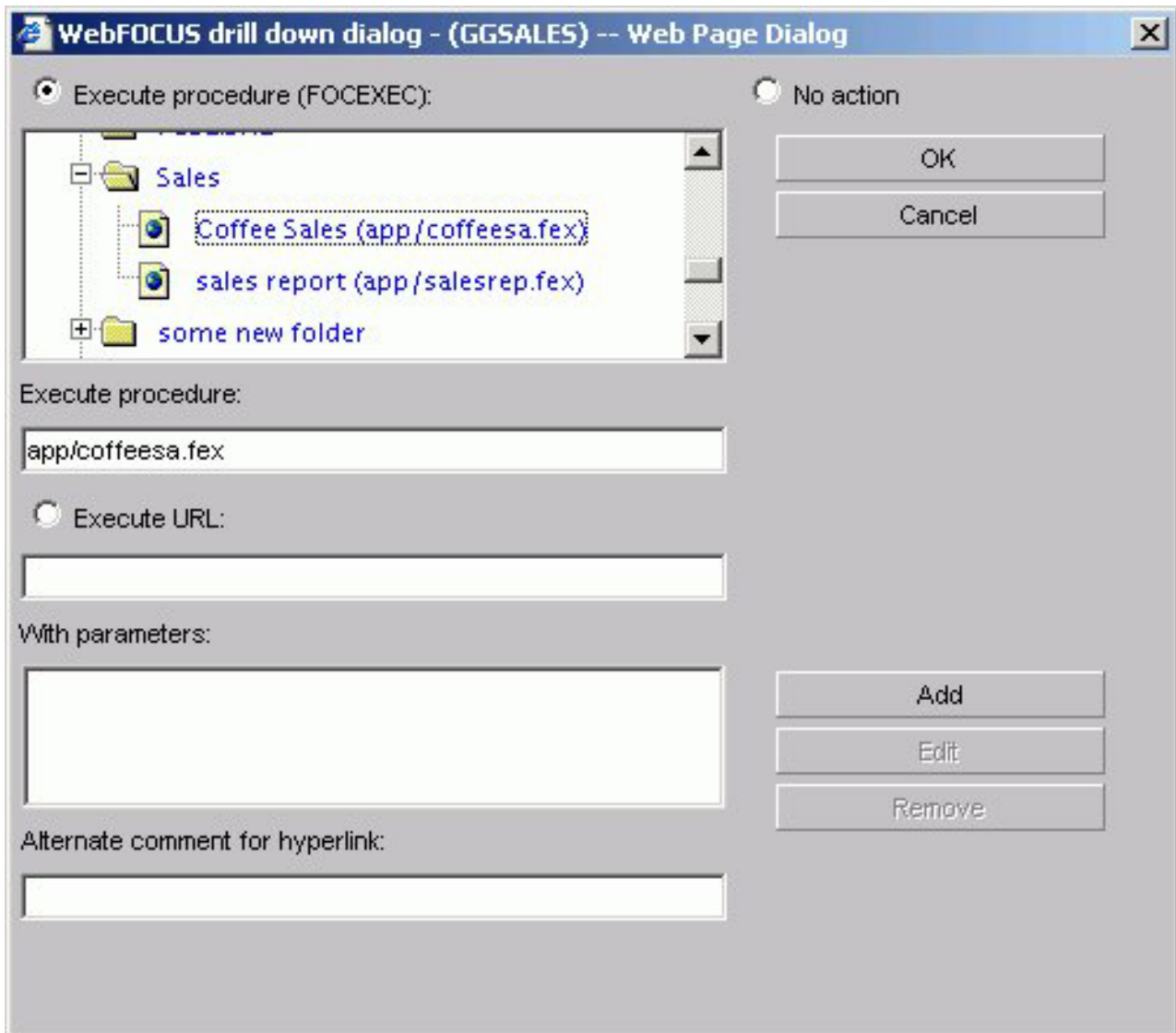
7. Click **Add**. The Drill Down Parameter dialog box opens as shown in the
following image.
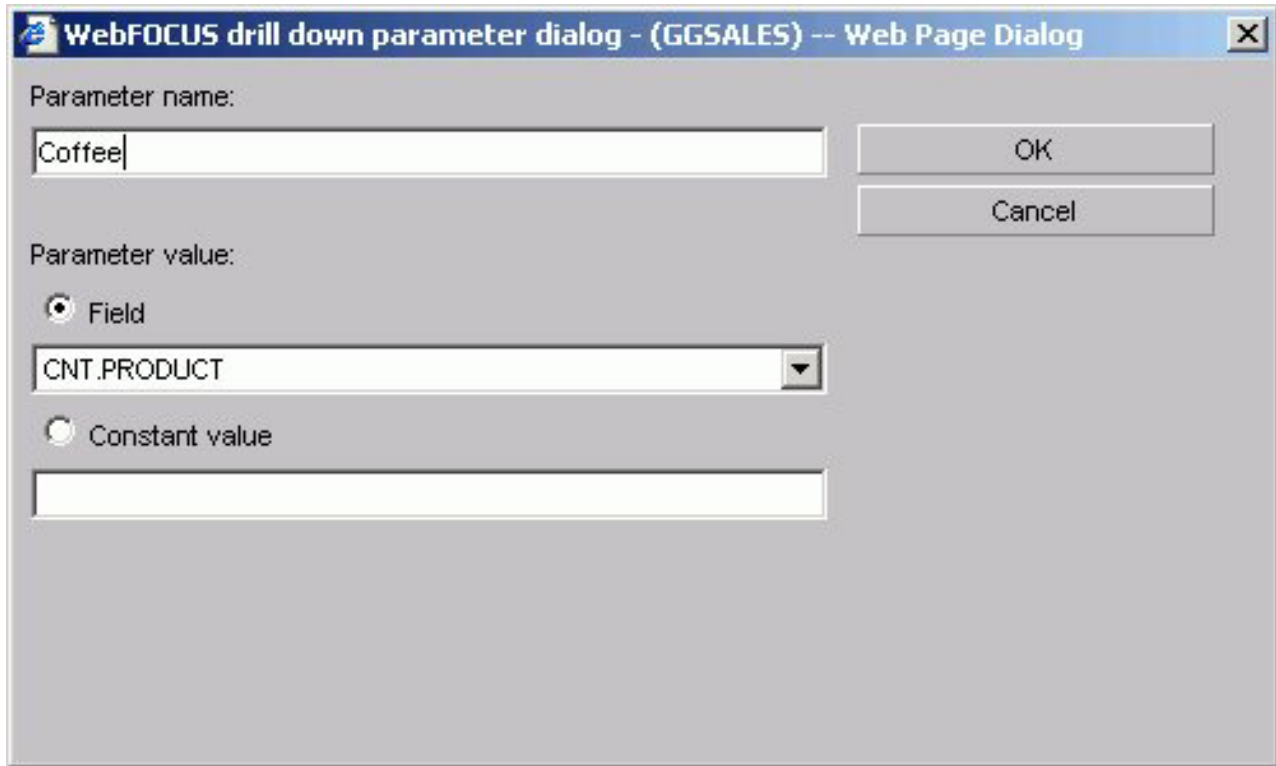
8. Enter the name of the parameter you created in the drill down procedure in the Parameter Name text box, for example, Coffee.

   Note: When passing parameters to a drill-down procedure, you must use local variables (&variables). Global variables (&&variables) cannot be used as drill-down parameters. Also, it is not necessary to type an ampersand (&) before the parameter name. This will create a global variable that cannot be used as a drill-down parameter.

9. When you pass the parameter to the drill down procedure, you must set a value for it in the Drill Down Parameter Dialog box. If you select:

   • **Field,** the parameter will be set to the corresponding value of the object the user drills down on in the specified field.

   • **Constant value,** the parameter is set to the specified value.

   Note: If the drill-down report contains a -DEFAULTS statement that sets a default value to the same variable passed from the main report, the variable value passed down overwrites the -DEFAULTS statement in the target procedure.

10. Once a value has been supplied, click **OK** to return to the Drill Down Dialog box. The parameter is added to the With Parameters list box as shown in the following image.
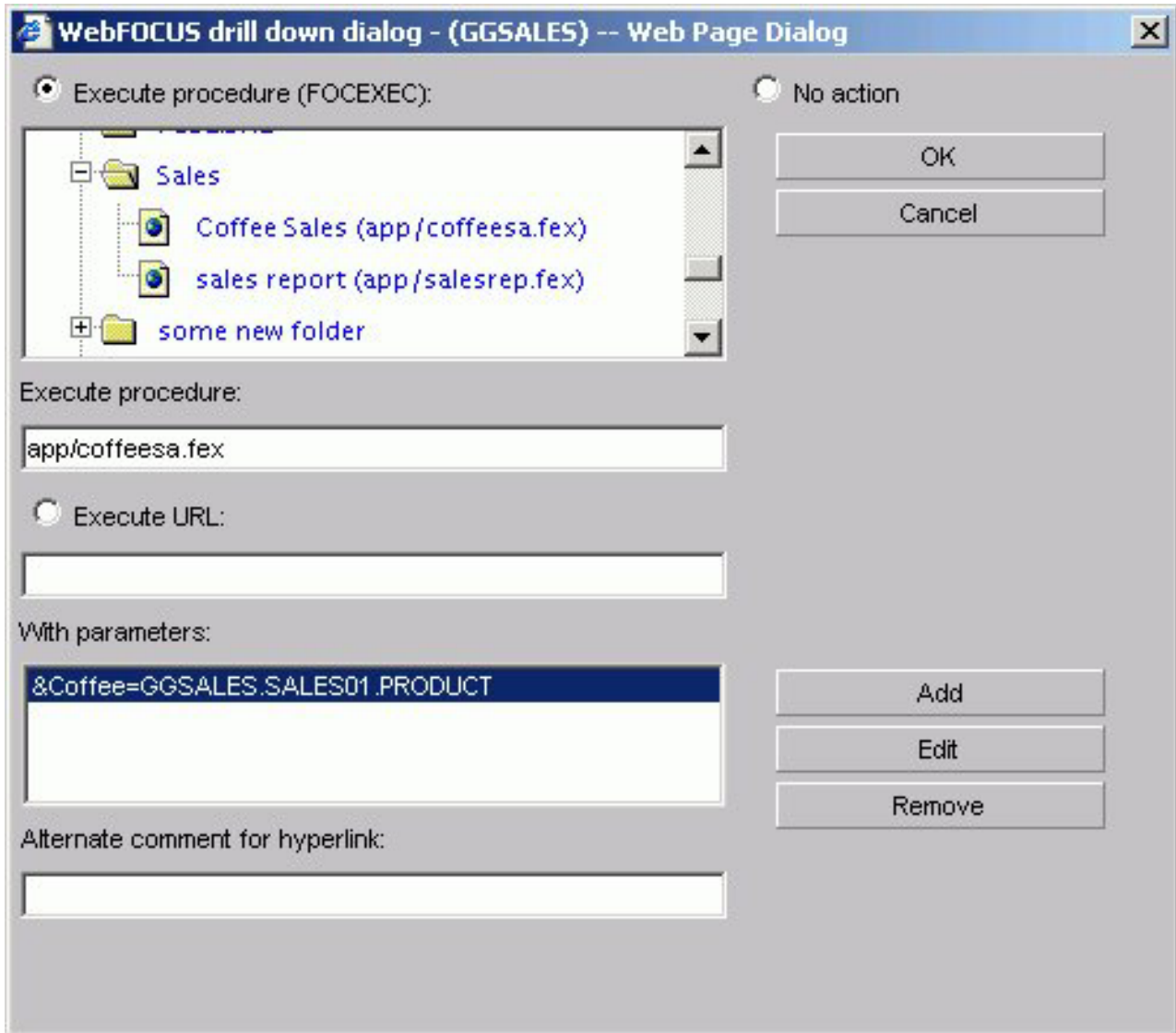
Adding Labels to a Graph

Adding labels to your graph helps provide important information about what the data in your graph represents. You may choose to add headings and/or footings to your graph, as well as style legend text and horizontal (X) and vertical (Y) axis labels.

From the Headings tab of the Graph Assistant, you can add titles, headings, and footings and customize them. For details on customizing, see Customizing Fonts in a Graph.

Note:

- There is a 95-character limit for Graph Titles.
- Dragging and dropping items, including images, is not supported in the HTML Graph Assistant.

The following image shows the Headings tab with a list of available fields on the left and the Page Heading and Page Footing fields on the right.
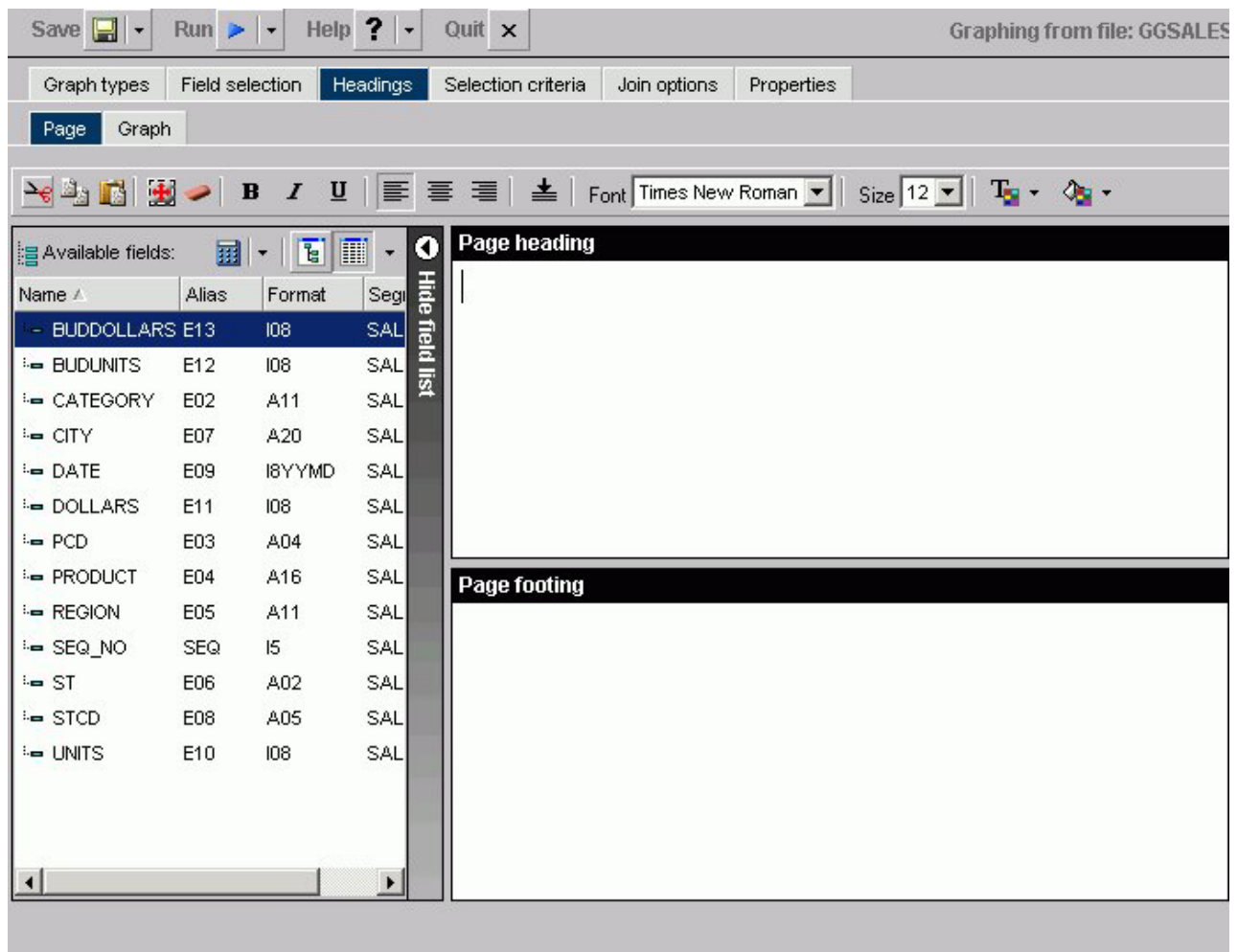
Note: If your graph labels or legends do not appear correctly when you run your graph, see Change Color Settings for details on correcting this.

Location of Headings and Footings in Graph Output

The following image illustrates where FOCUS Titles and Graph Titles appear in graph output. Note that none of the headings in this sample have been formatted and appear in their default format. Also note that the Graph Titles chart title will appear in the graph output as well as in your browser's title bar.

This is a FOCUS Titles page heading.

This is a Graph Titles chart title.
This is a Graph Titles chart subtitle.
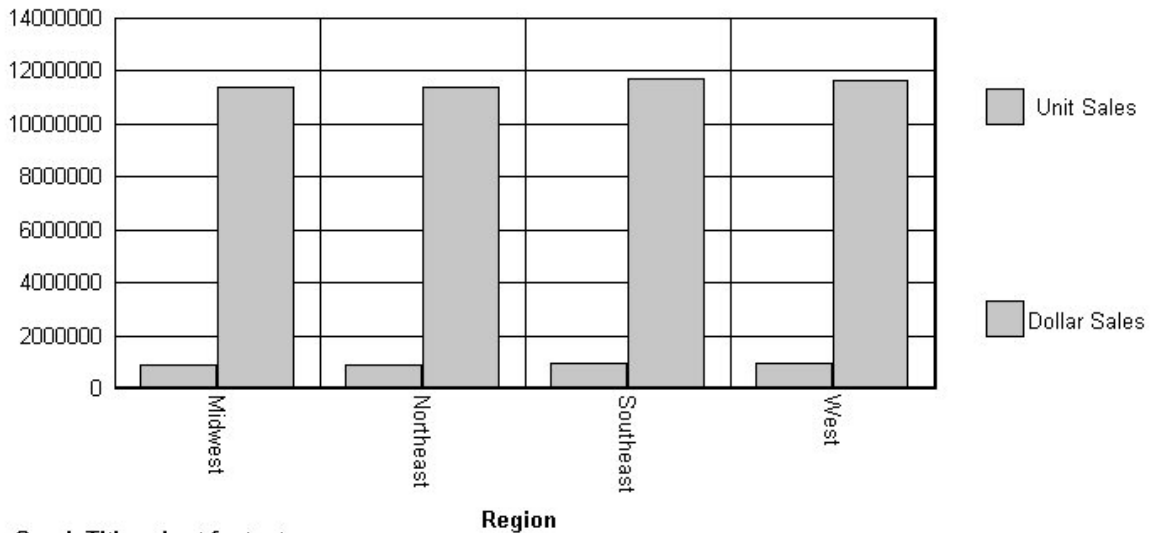
This is a Graph Titles chart footnote.

This is a FOCUS Titles page footing.

Note: The graph preview reflects all display changes to the graph. FOCUS Titles are not shown in the graph preview.

When you use the Graph Assistant to create a graph and add it to a report in the PDF Layout Painter, if the graph contains FOCUS Titles (Page Heading, Page Footing), they are automatically removed from the graph. To utilize headings or footings in a graph you intend to add to the PDF Layout Painter, use the Graph Assistant to create Graph Titles (Chart Title, Chart Subtitle, Chart Footnote). You can also add Text objects to a graph in the PDF Layout Painter and position them to function as a heading or footing.

Add a Heading or Footing to Your Graph

1. Click the **Headings** tab of the Graph Assistant.
2. Click the **Page** subtab.
3. Required: Click in the text box or press **Alt+H** to enter a page heading title.

   or

   Click in the text box or press **Alt+F** to enter a page footing title.

Note:
- If you change the font size of your heading (FOCUS titles), the code may not properly reflect the font size you specified when you open the procedure in a text editor. This does not affect functionality. You can edit the procedure manually to correct the size.
- When selecting a font for the page heading or footing, be sure to click in the text box before you select the font name, otherwise the font may return to the original font selection.

Add a Field to a Graph Heading or Footing

1. Click the **Headings** tab of the Graph Assistant.
2. Click the **Page** subtab.
3. Select a field from the Available fields window and drag the field to the Page heading or Page footing box.

   or

   Click and hold the left mouse button, and drag the field to the text box you chose.

   The following table lists the shortcut keys you can use to quickly go to one of the text boxes. The first column specifies the text box you go to when using the shortcut keys and the second column shows the shortcut key combination.

Text Box	Shortcut Keys
Page Heading	Alt+H
Page Footing	Alt+F

   Press **Alt+M** to go to the Available Fields window, tab to the field and press **Enter** to add the chosen field.

Add Titles, Subtitles, and Footnotes to a Graph

1. Click the **Headings** tab of the Graph Assistant.
2. Click the **Page** subtab.
3. Enter the text for the Chart Title, Chart Subtitle, and Chart Footnote in the respective boxes.

Note: There is a 95-character limit for Graph Titles.

Adding Vertical (Y-axis) and Horizontal (X-axis) Labels to a Graph

Vertical (Y-axis) and horizontal (X-axis) graph labels are placed on the graph according to the display fields and sort fields specified in the request. The titles that appear on the graph are the titles that appear in the Master File for that particular field.

The vertical (Y-axis) title of the graph is determined by the display field. Note that when the number of Y-axis labels is greater than one, the labels do not appear along the Y-axis. Instead, the labels appear in a legend that provides the names of the fields being graphed.

The horizontal (X-axis) title of the graph is determined by the sort field.

You can change the titles for the X and Y-axis from the Graph tab in the Headings tab in the Graph Assistant.

Applying Custom Styling to a Graph

Using the Graph Assistant, you may customize your graph using the subtabs (Options, Settings, X-axis, Y-axis, Pie) on the Properties tab. The customizations you can apply to your graph include customizing fonts, output destination, graph on server, label locations for all graph types, legend location, and much more.

You can also select the orientation of the X- and Y- axes and 3D effects as well as choose a predefined template using the Graph types tab.

Selecting Graph Orientation and 3D Effects

You can select either vertical or horizontal orientation for your graph on the Graph types tab. The default selection is vertical. If horizontal is selected, the X-axis appears along the side of the graph while the Y-axis appears along the bottom. You can choose vertical or horizontal orientation for Bar, Line, and Area graphs.

You can apply 3D effects to your graph, also on the Graph types tab. This option is not supported for histograms, spectral, stock, or radar graphs. To display bubble help when using 3D Effects, markers must be checked under Settings in the Properties tab.

Some graph types and graph styles require vertical and horizontal orientations, with and without 3D effects. In such cases, the relevant option buttons are selected and grayed out. You will only be able to make changes to orientation and 3D effects by changing the Graph Type and Graph Style. For example, if you select the 3D Graph Type and Bar Graph Style, the vertical option button is selected and grayed out.

Using Predefined Templates

When you check the Use a predefined template check box on the Graph types tab, you can choose one of the color and texture templates from the drop-down list: Pastels, Beveled on Tan, Color to White, Cylinder on Grey, Fade to Grey, Bumps, Greys, Plum On White, Red Accent, Speckled, and True colors.

Customizing Fonts in a Graph

You can change the font for the X and Y-axis data and labels, legend text, headings, footings, subheadings, and footnotes. From the text properties dialog box you can customize the:

- **Font.** Select the font type. Note that depending on what fonts you have installed on your machine, some fonts in the text properties dialog box may not be distinguishable from other fonts. In particular, Monospaced and Dialog Input may look the same as well as Dialog and San Serif.
- **Style.** Select the text style from normal, italic, bold, underline, or a combination of these.
- **Size.** Select the font size.
- **Set Style for.** Select either Title or Data to apply style changes.
- **Text Color.** Select the font color.
- **Text Justification.** Select left, center, or right justification.
- **Autofit.** Automatically adjusts the font size to fit the graph and disables the Size drop-down list. To select a different font size, uncheck this box and select the font size in the Size drop-down list.
- **Wrap.** Check this box to wrap the text.
- **Text Rotation.** Select the data label to display horizontally, vertically (from top to bottom or from bottom to top), and at a 45-degree angle.

Arrange X-axis Property Data Styles

1. Click the **Properties** tab and then the **X axis** subtab.
2. Click the **Style Labels** button to open the X-axis properties dialog box.

3. To enable the text rotation features, select the check box at the top of the X-axis properties dialog box as shown in the following image.
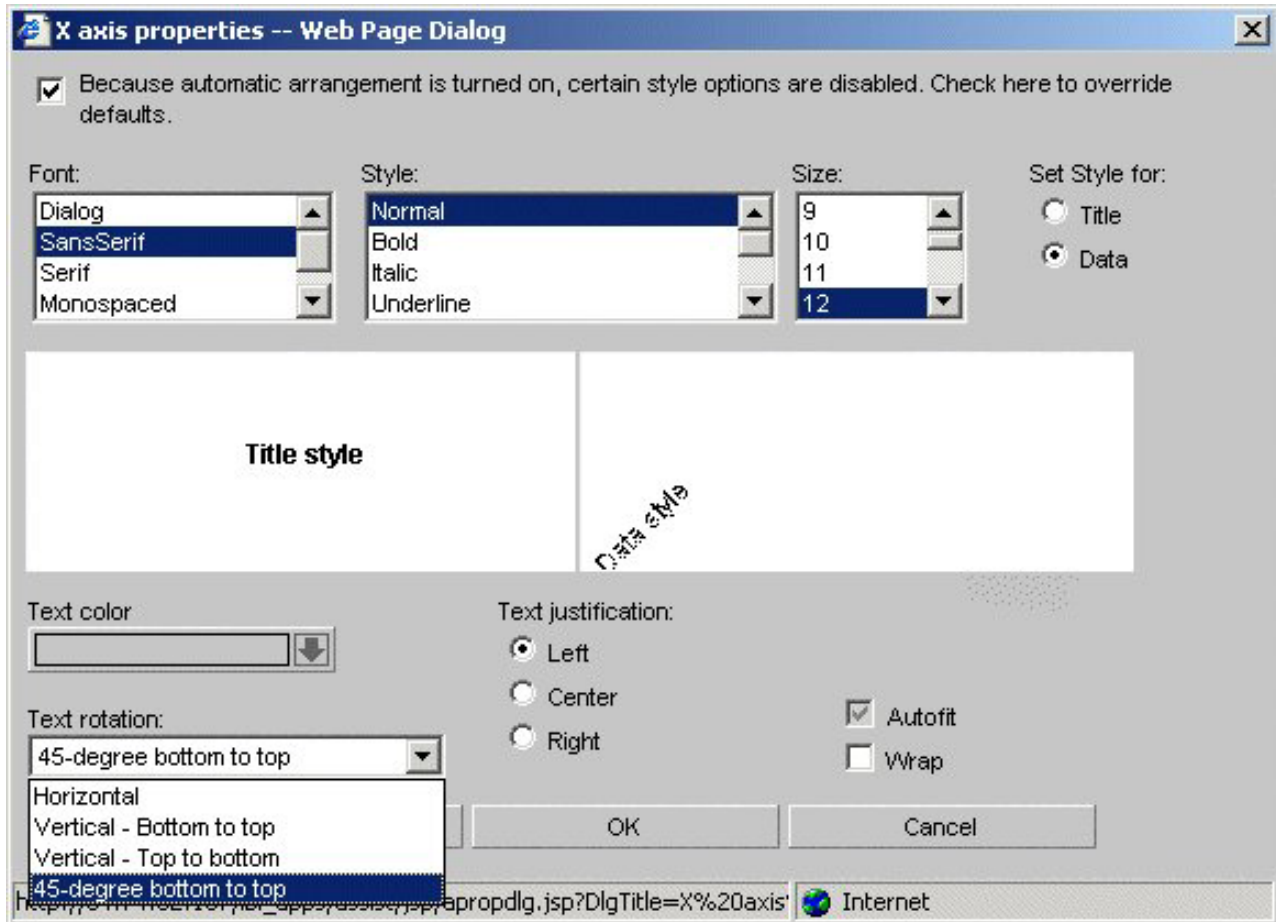
Note: This option is grayed out if the Automatic Arrangement option is de-selected on the Graph Types tab.

4. Required: Under Text rotation, choose one of the following:
   - Horizontal
   - Vertical - Bottom to top
   - Vertical - Top to bottom
   - 45-degree bottom to top

   In this case, 45-degree bottom to top is highlighted. Accordingly, the Data Style label is positioned at a 45-degree angle.

As you make changes to the text properties, sample text appears to the right and reflects all selections. If you leave the Autofit check box checked for a particular object, the font size will be automatically adjusted to fit your graph. You can also wrap the text by checking the **Wrap** check box. Note that when you select the Wrap option, text only wraps to the next line if there is not enough room to display the entire text in the chart.

The text properties dialog box can be invoked from many places in the Graph Assistant. Depending on where it is invoked from, different options are available.

Note: You can choose Background and Frame colors in the Options tab in the Properties tab.

Change X-axis Label Fonts

1. Click the **Properties** tab of the Graph Assistant.
2. Click the **X axis** subtab.
3. Click the **Style Labels** button. The X-Axis Properties dialog box opens.
4. Select the font properties. You can change the fonts for the data text and for the x-axis title text by selecting either the **Data** or **Title** option button.
5. Click **OK**.

Change Y-axis Label Fonts

1. Click the **Properties** tab of the Graph Assistant.
2. Click the **Y axis** subtab.
3. Click the **Style Labels** button. The Y1 Axis Label Properties dialog box opens. If you are selecting fonts for the Y2 axis, the Y2 Axis Label Properties dialog box opens; it contains the same style options.
4. Select the font properties. You can change the fonts for the data text and for the y-axis title text by selecting either the **Data** or **Title** option button.
5. Click **OK**.

Change Graph Legend Fonts

1. Click the **Properties** tab of the Graph Assistant.
2. Click the **Settings** subtab.
3. Click the **Style legend text** button.

    The Legend Properties dialog box opens.
4. Select the font properties.
5. Click **OK**.

Change Graph Title Fonts

1. Click the **Headings** tab of the Graph Assistant.
2. Click the **Graph** subtab.
3. Click **Style Title**, **Style Subtitle**, or **Style Footnote** for Chart title, Chart subtitle, or Chart footnote.

    Depending on the option you select, the Heading Properties, Subheading Properties, or Footnote Properties dialog box opens.
4. Select the font properties.
5. Click **OK**.

Setting the Graph Height and Width

The width (or horizontal axis) of each graph, which includes any surrounding text, is automatically set to 770 pixels. When setting the graph width, you should allow for the inclusion of any text required for the vertical axis and its labels along the left margin.

The height (or vertical axis) of your graph is automatically set to 405 pixels.

The vertical axis is automatically set to cover the total range of plotted values. The height of the axis is set as high as possible (taking into consideration the presence of any headings or footings and the need to provide suitably rounded vertical class markers). The range is divided into intervals called "classes." The scale is

normalized to provide class values rounded to the appropriate multiples and powers of 10 for the intervals plotted on the axis.

Set the Graph Height and Width

1. Click the **Properties** tab of the Graph Assistant.
2. Click the **Options** subtab.
3. In the Graph Size field, select the **Custom size** option button.
4. Enter the horizontal (width) value and the vertical (height) value, both in pixels.

   The horizontal value can be any number between 20 and 985.

   The vertical value can be any number between 20 and 518.

Running, Saving, and Printing Your Graph

When you run your graph, you may print the output directly from the browser.

Note: If your graph labels or legends are not appearing correctly when you run your graph, see Change Color Settings for details on correcting this.

If you log off DB2 Web Query without first closing the HTML Graph Assistant, you must manually close the tool.

Save Your Graph From the Graph Assistant

1. Click **Save** or **Save As** from the Graph Assistant.
2. From the Save New Report dialog box, enter a name for the graph.

Note: If you are editing an existing graph, clicking Save will automatically save the edits to the current graph file. You will not be prompted to name the file again.

Saving Options

When you click the Save or Save As buttons you save your report (procedure). If you make changes in your report without subsequently saving them and then click Quit, you are prompted with the following options:

- **Yes.** Click to save your changes and exit the report.
- **No.** Click to discard your changes and exit the report.
- **Cancel.** Click to return to your report to continue working or use Save or Save As to save your changes and then click Quit to exit your report.

Note: In the HTML Graph Assistant, if you close the window by clicking the X, your changes will not be saved.

Run Your Graph

After creating your graph in the Graph Assistant, click **Run** from any tab in the Graph Assistant to view your graph output.

Print Your Graph

1. Run your graph.
2. From the browser, select **Print** from the File menu.

Note: Some older printers, such as the IBM model 6262, do no support graphical output.

Send Graph Output Directly to a Printer

1. From the Properties tab, click the **Options** subtab.
2. Select **Printer** from the Output list box.

   You do not need to provide a name or select a format.
3. Click **Run**.

   The graph prints to your default printer setting.

Change Color Settings

1. From the Windows® Control Panel, select **Display**.
2. Click the **Settings** tab.
3. In the Color palette box, click the drop-down arrow and select True Color (32 bit) or a higher color count.
4. Click **OK**.

If you use different color settings from this recommended value, your graphs may appear in grayscale format.

Graph Display Formats

Following are descriptions of the graph display formats:

- PNG (Portable Network Graphics). PNG graphs are bitmapped and can support 16 million colors. In addition, PNG graphs have lossless compression (data is decompressed 100% back to the original). Therefore, saving, altering, and resaving a PNG does not degrade its overall quality. The PNG graph appears in the browser.
- GIF (Graphics Interchange Format). GIF compresses well but only supports 256 colors. The GIF graph appears in the browser.
- JPEG (Joint Photographic Experts Group). A compressed file format.
- SVG (Scalable Vector Graphics). SVG is a text-based graph file format that enables Web application developers to design powerful, interactive graphs. SVG is compact and generates high quality images on any media regardless of resolution. The SVG is viewable with an Adobe plug-in.

   Note: To include a graph in the PDF Layout Painter, the graph must be saved in .SVG or .jpg format.
- PDF/SVG (Portable Document Format/Scalable Vector Graphics). The SVG graph is automatically embedded into a PDF and shown only on the screen. Drill down is not available with this format.
- PDF/GIF (Portable Document Format/Graphics Interchange Format). The GIF graph is automatically embedded into a PDF and shown only on the screen. Drill down is not available with this format.

Choosing Formats for Output to the Screen or Printer

When you create or edit a graph in Graph Assistant, you can choose among these formats in the Output section in the Options tab. If you are designating Output To the Screen, all options listed in Graph Display Formats are available, as shown in the Format drop-down list.

The following image shows the Options subtab.
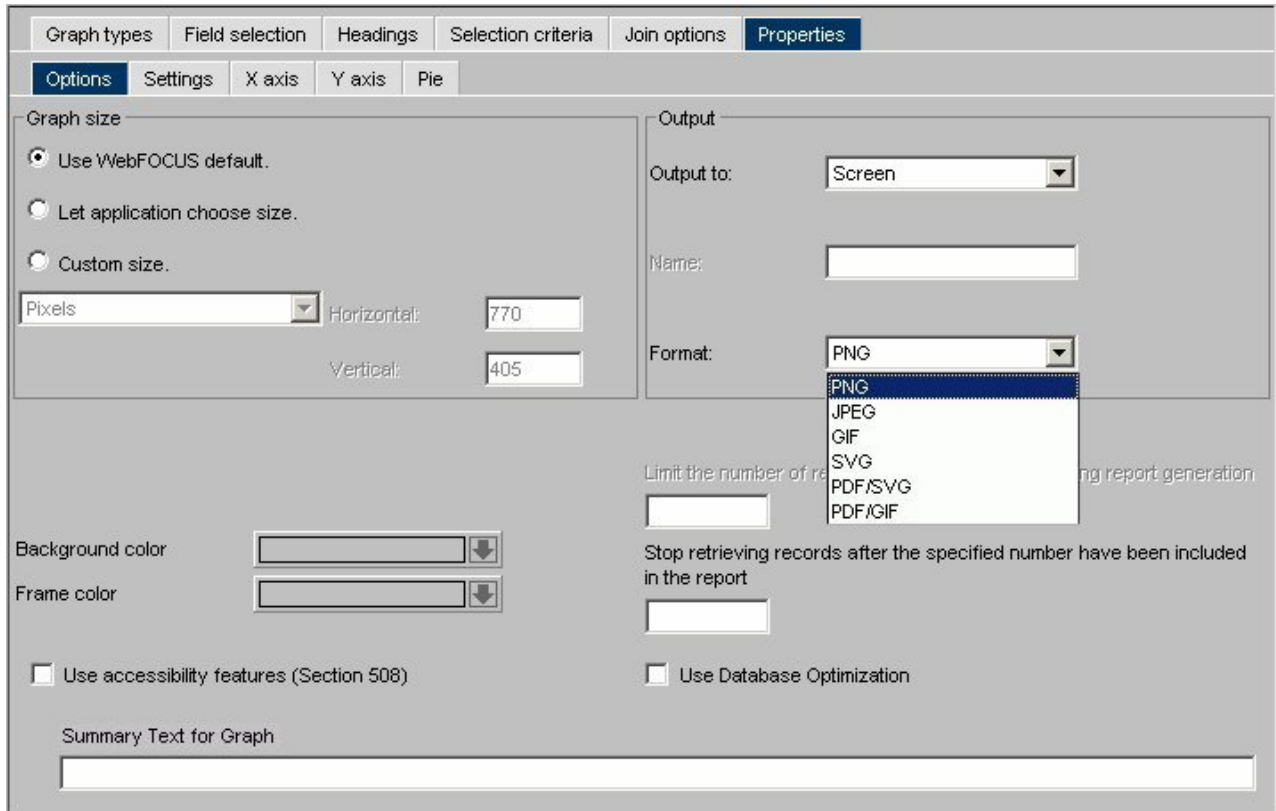
If you are designating Output to a Printer, the PNG, GIF, and SVG options also are available. Note that the printer must be able to print images.

The following image shows the Output section of the Options tab with the Output to Printer option selected.
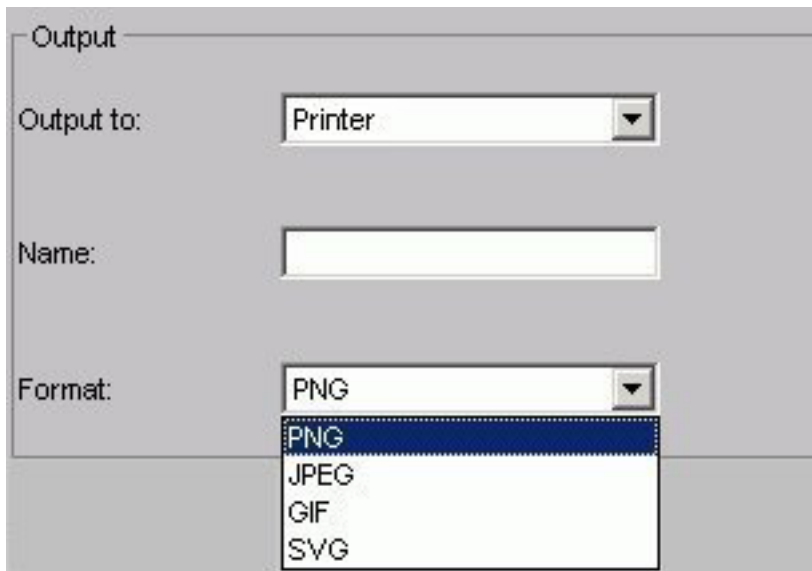
Graph Types Tab

The Graph Types tab is used to create and view a graph type and style.

Graph Types Tab: Graph Types Window

The following image shows the Graph types tab with a sample cluster bar chart.
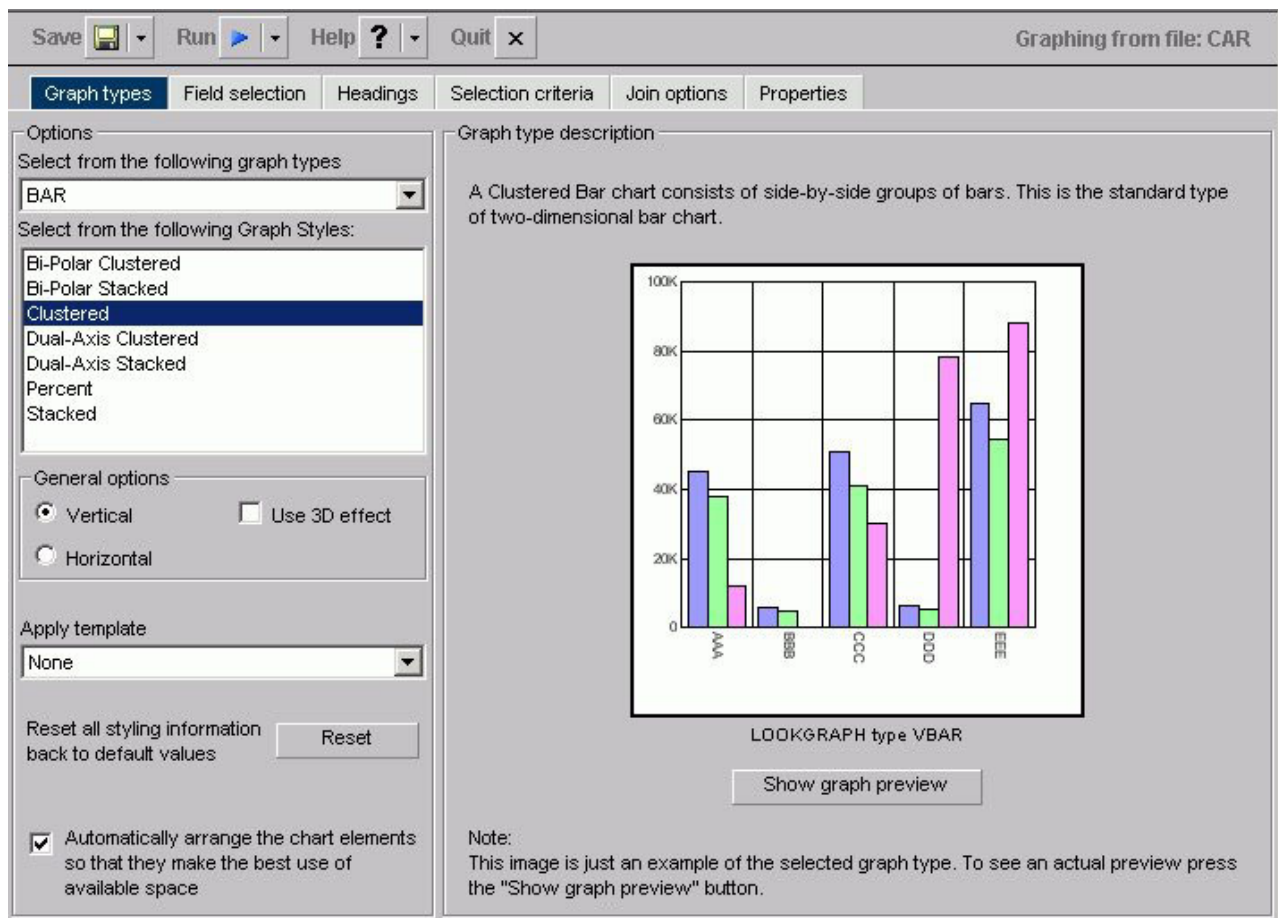
In the Graph types component:

Select from the following graph types

> Displays Line, Bar, Pie, Scatter, Area, 3D, Bubble, Polar, Radar, Spectral Map, Stock, Histogram, Multi-Y, Waterfall, and Miscellaneous graph types.

Select from the following Graph Styles

> Displays several graph styles customized for each graph type.

General options:

Vertical

> Displays the date vertically, along the Y-axis.

Horizontal

> Displays the data horizontally, along the X-axis.

Use 3D effect

Applies a three-dimensional look to either the vertical or horizontal data.

Apply template

Displays the Pastels, Beveled on Tan, Color to White, Cylinder on Grey, Fade to Grey, Bumps, Greys, Plum on White, Red Accent, Speckled templates, and True colors.

Reset button

Resets all styling information back to the default values.

Automatically arrange the chart elements so that they make the best use of available space

Specify this option with a check in the check box.

The Graph type description component of the Graph types subtab includes a description of the Graph type and style you have chosen, along with a sample graph. In addition, you have the following options:

Show Graph Preview button

Displays the actual graph in a separate window with the options you specified.

Update button

Updates a graph after you make changes.

Automatic Update check box

Check to have your graph automatically updated each time a change is made.

Field Selection Tab in HTML Graph Assistant

The Field selection tab, shown in the following image, is used to choose the fields you want to use in a graph.
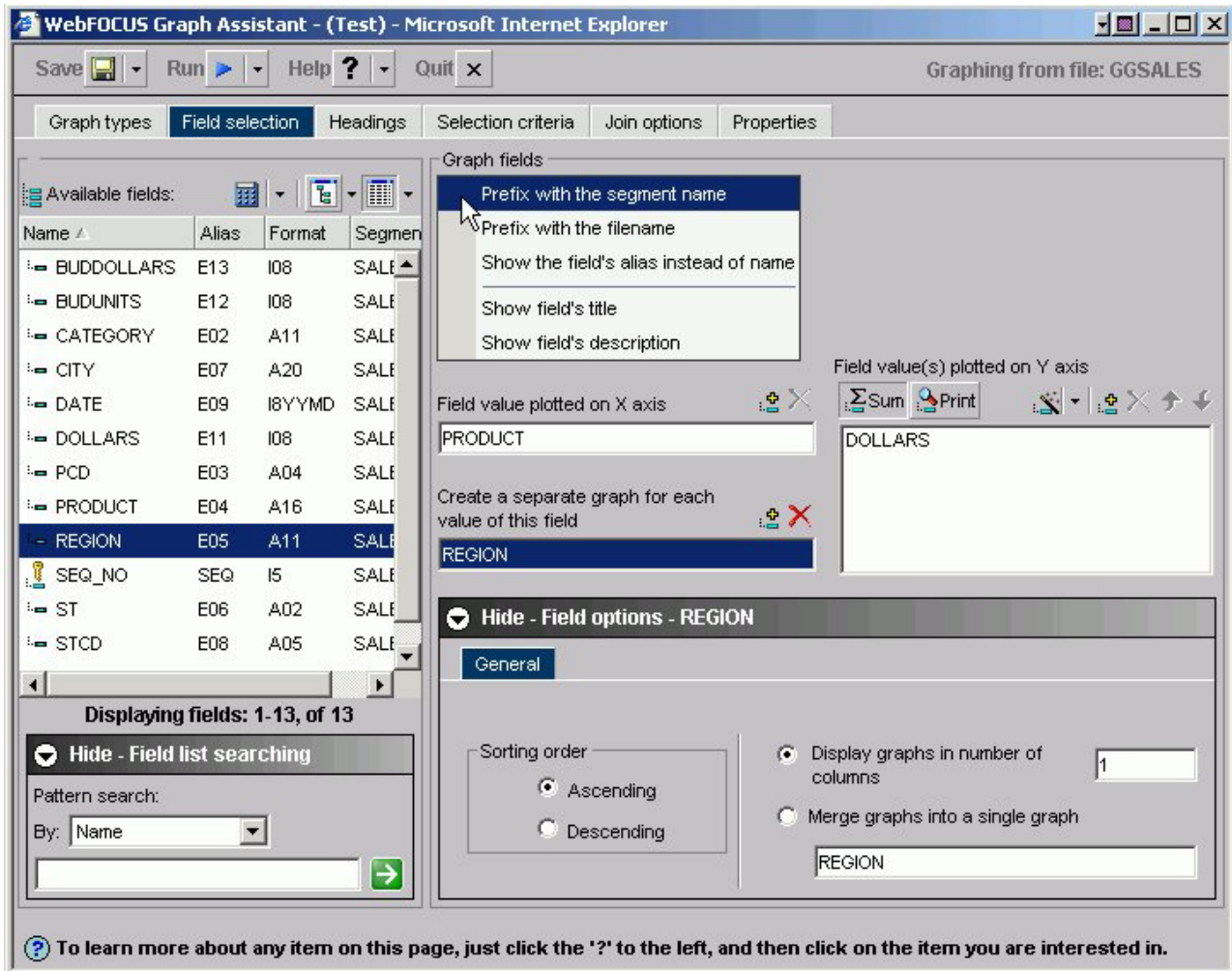
Field Selection Tab: Available Fields

The following image shows the Available fields area of the Field selection tab. The Available fields area in the Field selection tab, as shown in the following image, provides the following functionalities and options for selecting available fields in a graph.
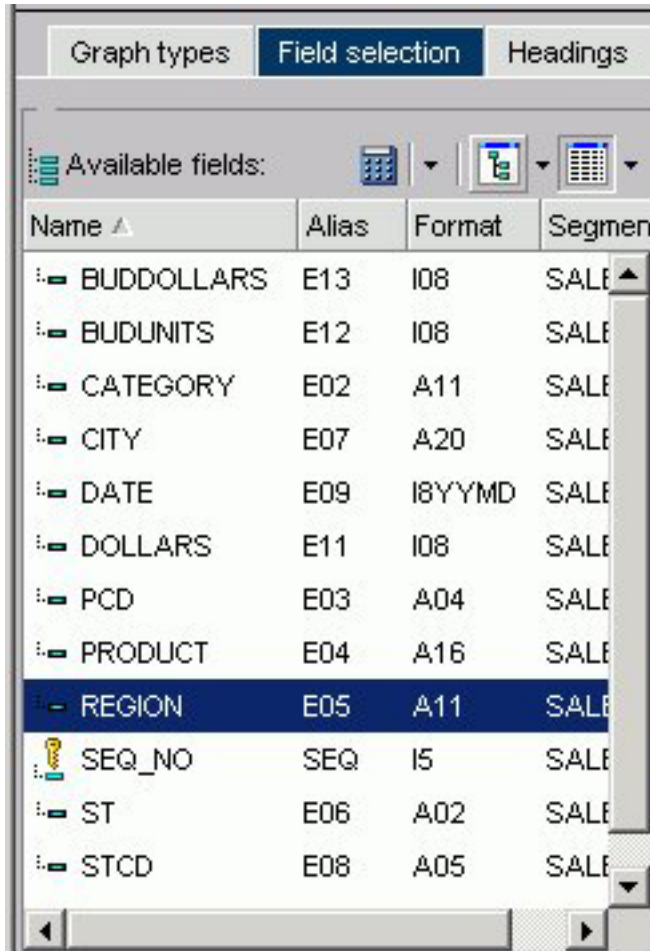
Available fields

Displays a list of field names from the selected data source.

Define button

Click the Define button to select from New define field, Edit define field, or Delete define field. If you select the New define field or Edit define field, the Define field creator dialog box opens where you can create or edit a defined field. For more information, see Field Selection Tab: Field Creator Dialog Box.

Tree button

When this button is selected, field names are displayed in a tree structure. Optionally, data can be displayed in different ways by clicking the adjacent down-arrow to open a drop-down list that provides options to Show name, Show alias, Show title, or Show description. The Show Field Tree display type is the default value, and for some database structures, the Show Dimension Tree option can be selected.

List icon

This button is selected by default to display a list of all field names. Additional related information can be displayed by clicking the adjacent down-arrow to open a drop-down list that provides options to display Name, Alias, Title, Format, Description, Segment, and Filename. You can also choose the option to Sort by file order.

Field Selection Tab: Graph Fields

The Graph fields area in the Field selection tab, as shown in the following image, provides the following functionalities, including many options, for selecting and displaying fields in a graph.
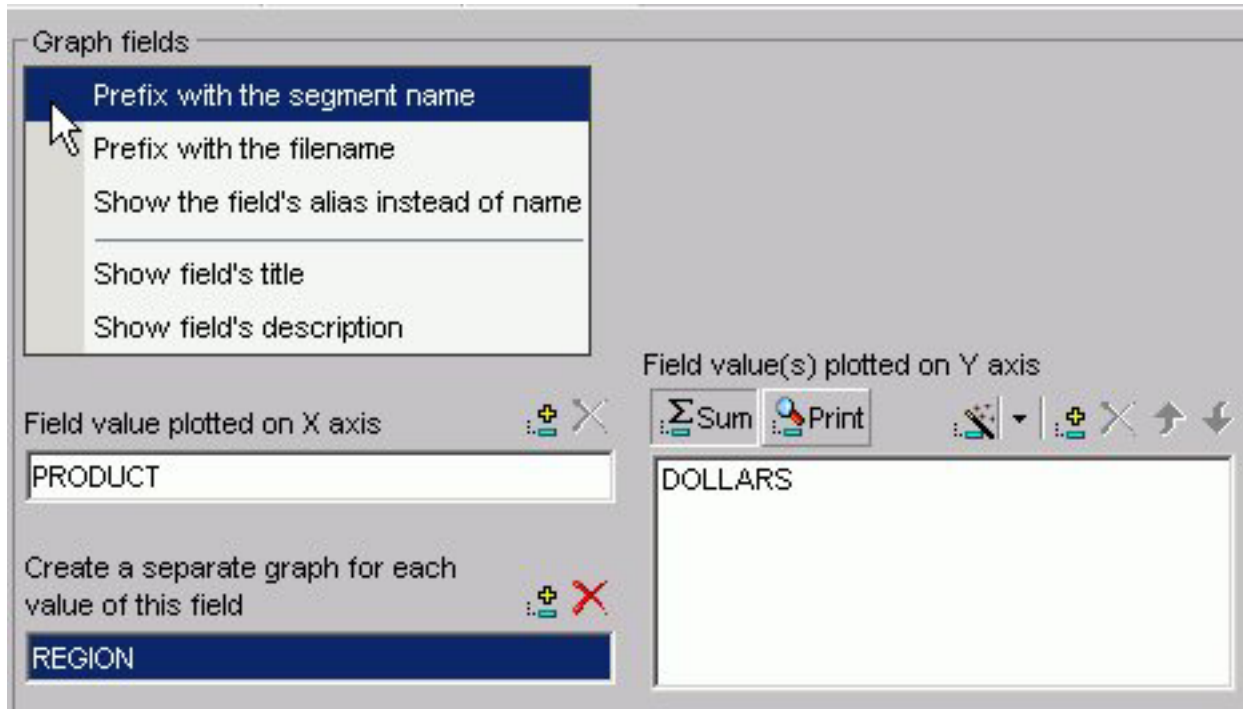
You can view the selected fields with the following options:

> **Prefix with the segment name** prefixes the selected fields with the segment name.

> **Prefix with the filename** prefixes the selected fields with the name of the data source.

> **Show the field's alias instead of name** displays the aliases defined in the Master File instead of the field names.

> **Show field's title** displays the titles defined in the Master File instead of the field names.

> **Show field's description** displays the descriptions defined in the Master File instead of the field names.

Field Value Plotted on X axis

> Displays the horizontal (X-axis) field that was selected from the Available fields area.

> **Add button**

> > Adds a field to the X-axis.

> **Remove button**

> > Removes a field from the X-axis.

Create a separate graph for each value of this field

Displays the second horizontal (X-axis) field (formerly referred to as an Across field) that was selected from the Available fields area.

Add button

Adds a second field to the X-axis.

Remove button

Removes a field from the X-axis.

Field value(s) plotted on Y axis

Displays the vertical (Y-axis) field that has been selected from the Available fields area.

Sum

Creates a graph that aggregates data by the X-axis field. This is the default.

Print

Creates a graph that plots each Y-axis field value on a record-by-record basis. If the selected X-axis value is numeric, a scatter graph is generated; if the selected X-axis value is alphanumeric, a line graph is generated.

New computed field button

Opens the Computed Field Creator dialog box where you can define the parameters for a new computed field. You can also click the adjacent down-arrow to open a drop-down list to select the Edit computed field option. For more information, see Field Selection Tab: Field Creator Dialog Box.

Add button

Adds a field to the Y-axis.

Remove button

Removes a field from the Y-axis.

Move Up or Move Down buttons

Moves the position of the field within the graph.

Field Selection Tab: Field Options

Below the Graph fields area in the Field selection tab is the Field options panel you can display by clicking **Show - Field options** or hide by clicking **Hide - Field options**. A different Field options panel is displayed depending on whether you have a Y-axis, X-axis, or secondary X-axis field selected in the Graph fields area.

For the Y-axis Field options Display tab:

The Display tab appears by default when the Field options panel is displayed for Y-axis fields and provides the buttons, fields, and options described after the following image.
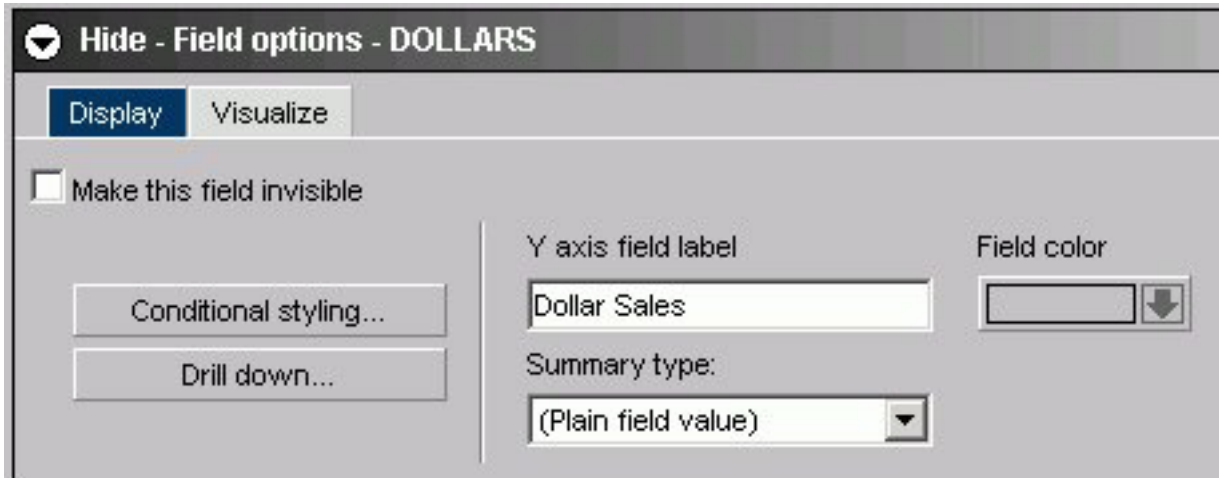
Make this field invisible

> Check to temporarily hide the display of the selected Y-axis field.

Y axis field label

> The selected Y-axis field name is automatically displayed. You can edit this field to display a different label for the Y-axis field.

Summary Type

> Select a summary type for the selected Y-axis field. The choices are: Plain field value, Average Square, Average, Count, Distinct, Show first in group, Show last in group, Maximum, Minimum, Percentage, Count Percentage, Row Percentage, Sum, and Total.

Field Color

> Select to open a pop-up color palette, shown in the following image, where you can select the display color for the selected Y-axis field.
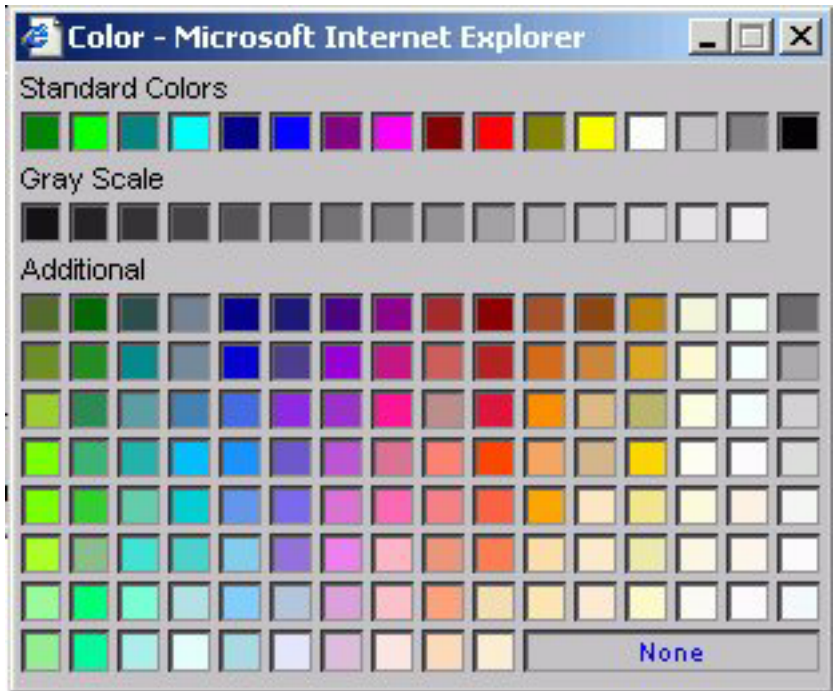
Conditional Styling button

Click to open the conditional styling dialog box, as shown in the following image, which you can use to add conditional styling in a graph.
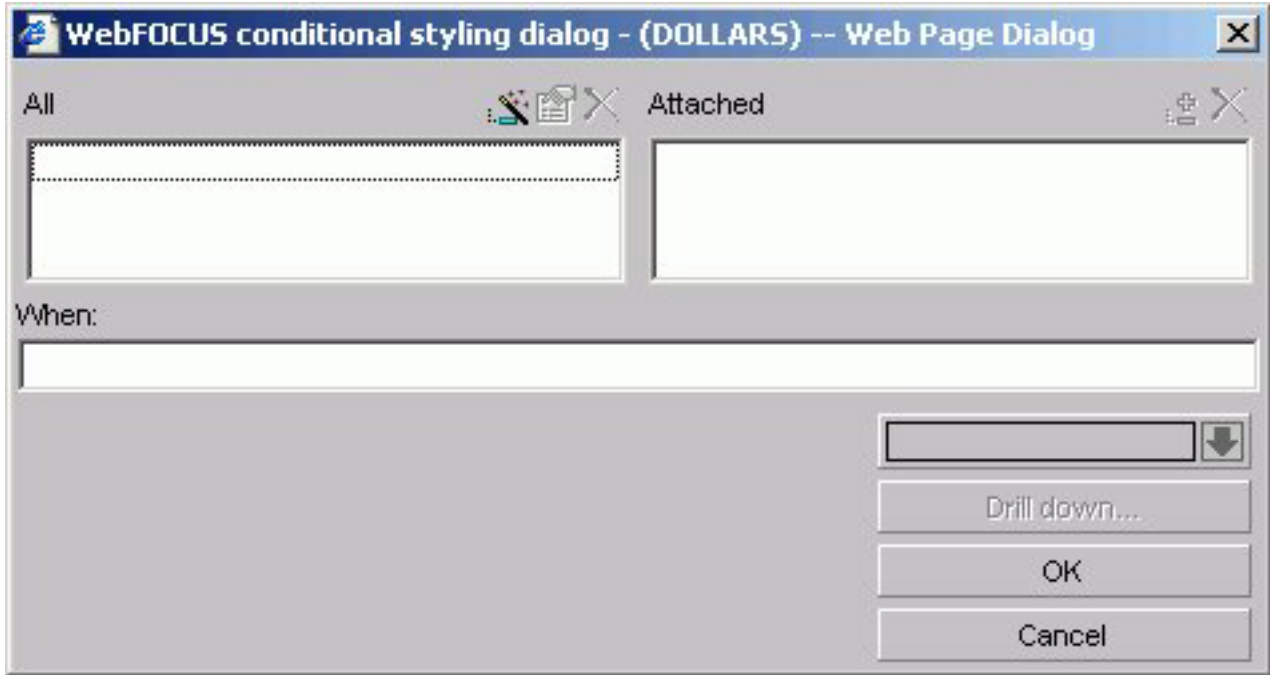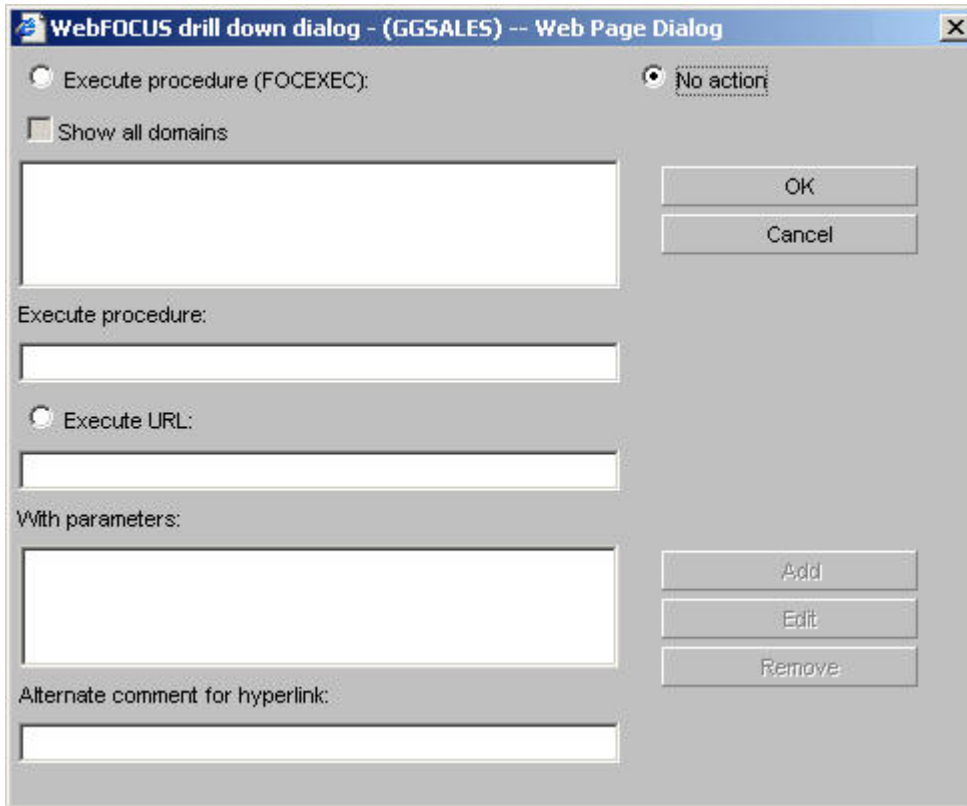
OK

Returns to the Field selection tab after the conditional styling is added to the graph.

Drill Down

Click to open the drill-down dialog box, which provides the fields and options listed after the following image, that you can use to add drill down capability to a graph.

No action

      This option is selected by default. (No procedure will be executed.)

Execute procedure (FOCEXEC)

      Select this option to access procedures in the Standard Reports folder of the default domain. Click the procedure you want to execute.

      Show all domains

            When Execute procedure (FOCEXEC) is selected, click this check box to gain access to procedures in the Standard Reports folders of all available domains.

Execute procedure

      The procedure you selected appears in this text box.

Execute URL

      Select this option to enter the URL you want to execute.

With parameters

      When a parameter is created, it is automatically added to this list box.

Alternate comment for hyperlink

Type text here to display in place of the hyperlink specified in the Execute URL field.

Add button

Click to open the drill-down parameter dialog box, which provides the fields and options listed after the following image.

Parameter name

Enter the parameter name you created in the drill down procedure.

Parameter value: Field

The parameter will be set to the corresponding value of the object the user drills down on in the specified field.

Parameter value: Constant value

The parameter is set to the specified value.

OK button

Returns to the drill down dialog box. The parameter is added to the With parameters list box.

For the Y-axis Field options Visualize tab:

The Visualize tab appears when the Field options panel is displayed for an X-axis field and provides the options described after the following image.

Force the values of this field to be plotted as:

Choose Bar, Line, or Area.

Place this field on the specified axis

You can choose 1st axis or 2nd axis. (Available for dual axis graph types only.)

For the X-axis Field options:

The General tab appears when the Field options panel is displayed for an X-axis field and provides the fields and options described after the following image.
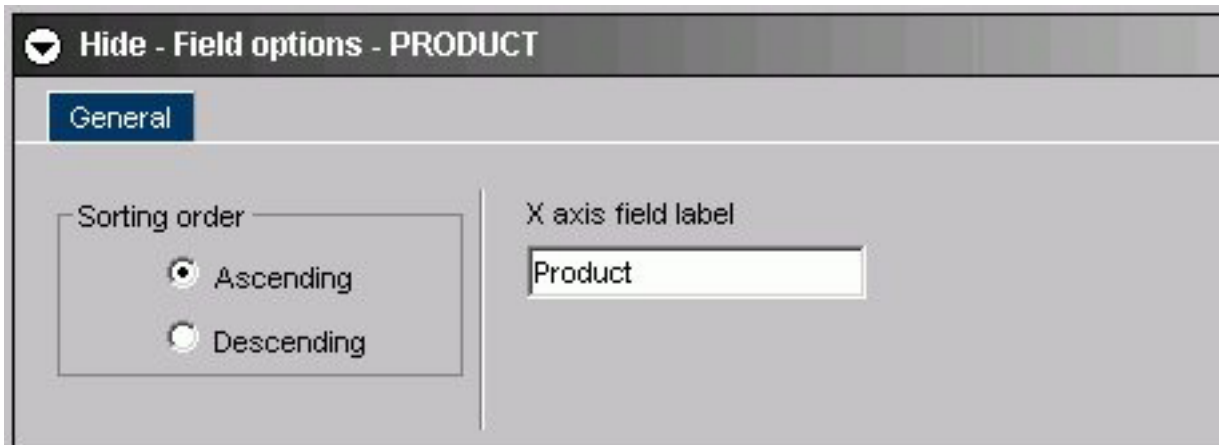
Sorting order

Click Ascending or Descending.

X-axis field label

The X-axis field name is automatically displayed. You can edit this field to display a different label for the X-axis field.

For the Secondary X-axis (Across) Field options:

The General tab appears when the Field options panel is displayed for a second X-axis (Across) field and provides the fields and options described after the following image.
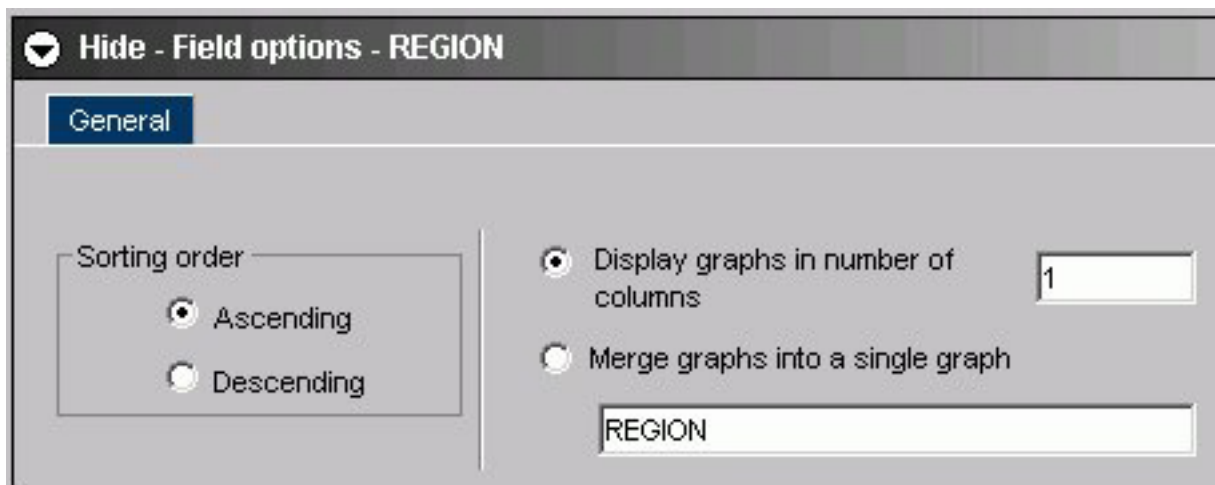
Sorting order

Click Ascending or Descending.

Display graphs in number of columns

Enter a number in the text box.

Merge graphs into a single graph

Click this option button to activate merge functionality.

Axis label

The selected field name is is automatically displayed. You can edit this field to display a different label for this Across field.

Field Selection Tab: Field Creator Dialog Box

Use the Field Creator Dialog Box to create or edit Defined and Computed fields, as shown in the following image.
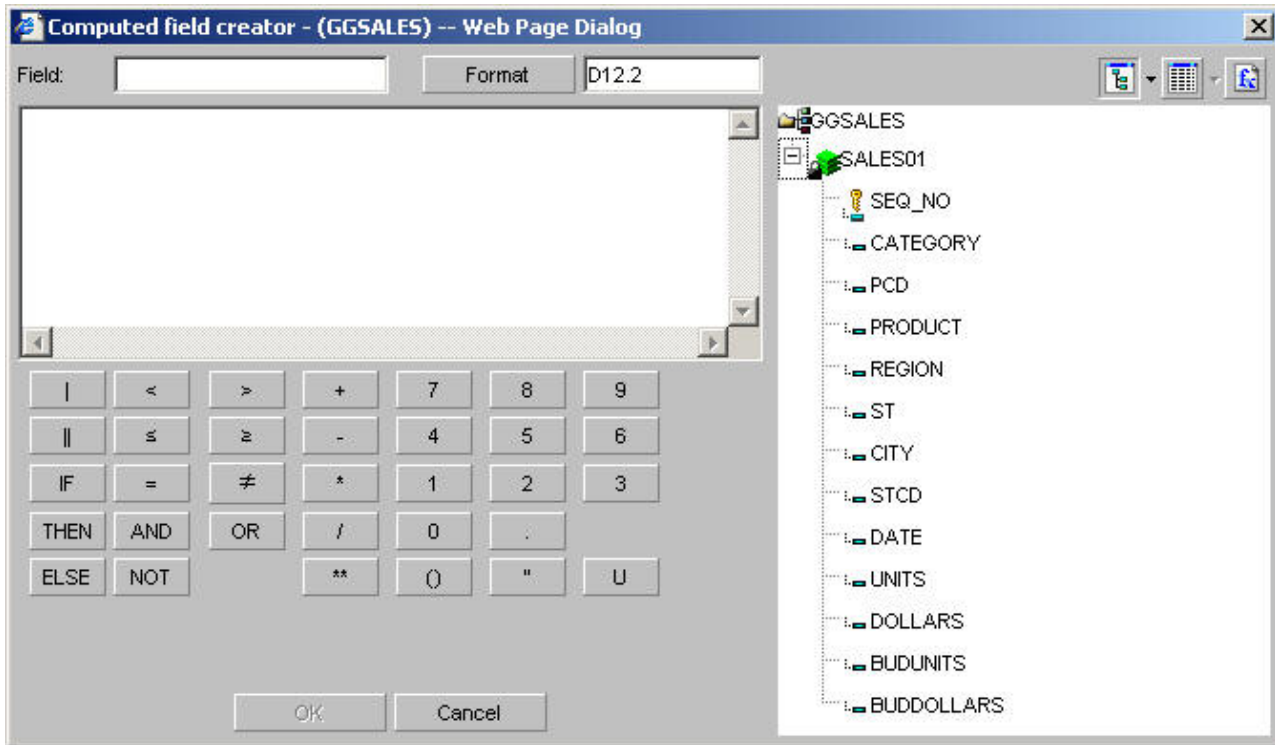
Field

> Enter a name for the computed or defined field.

Format

> Click to display the Change Format Dialog box, as shown in the following image.
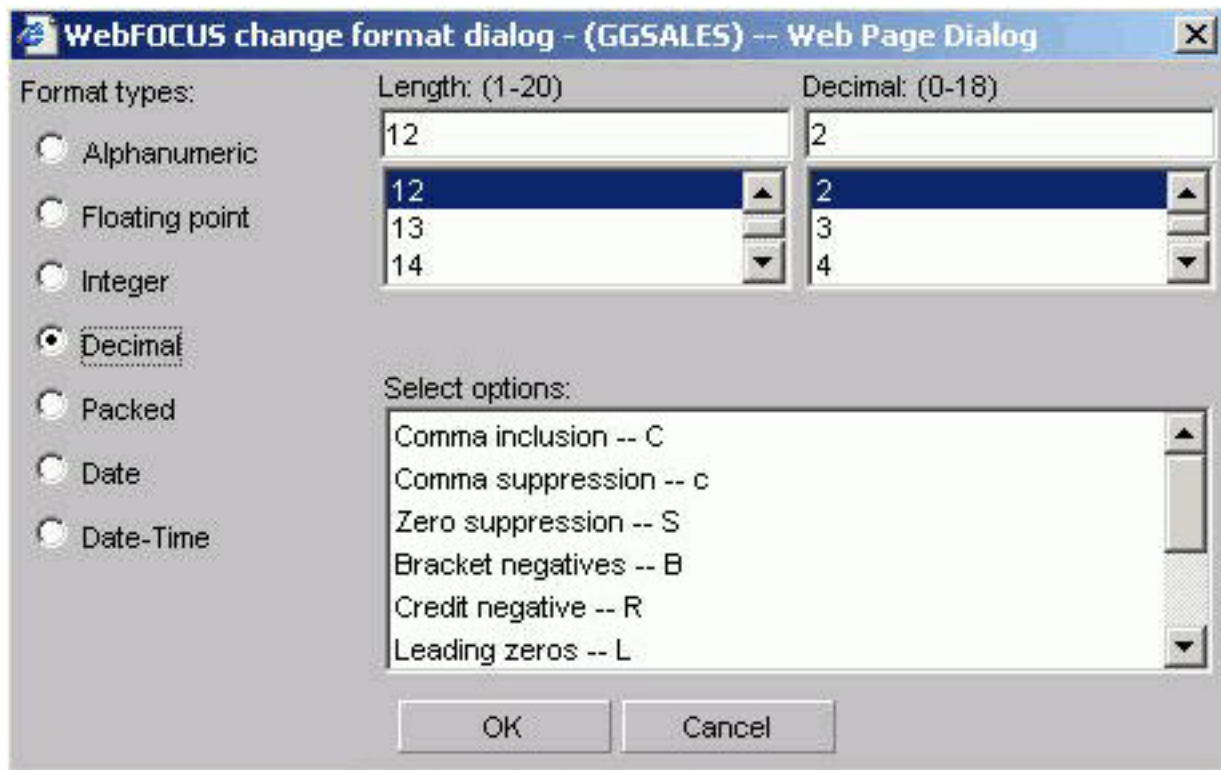
Format types

> Choose:
> - Alphanumeric
> - Floating point
> - Integer
> - Decimal
> - Packed
> - Date (opens additional date related fields in the dialog box)
> - Date-Time (opens additional date related fields in the dialog box)

Length

> Choose an available number based on the format type you chose.

Select options

> Choose options based on the format type you chose.

Tree or List mode buttons

> Opens the field list in either tree or list mode.

Fields window

> Double-click a field to include a field name in an expression.

Functions

> Displays a list of pre-defined functions shown in the following image.
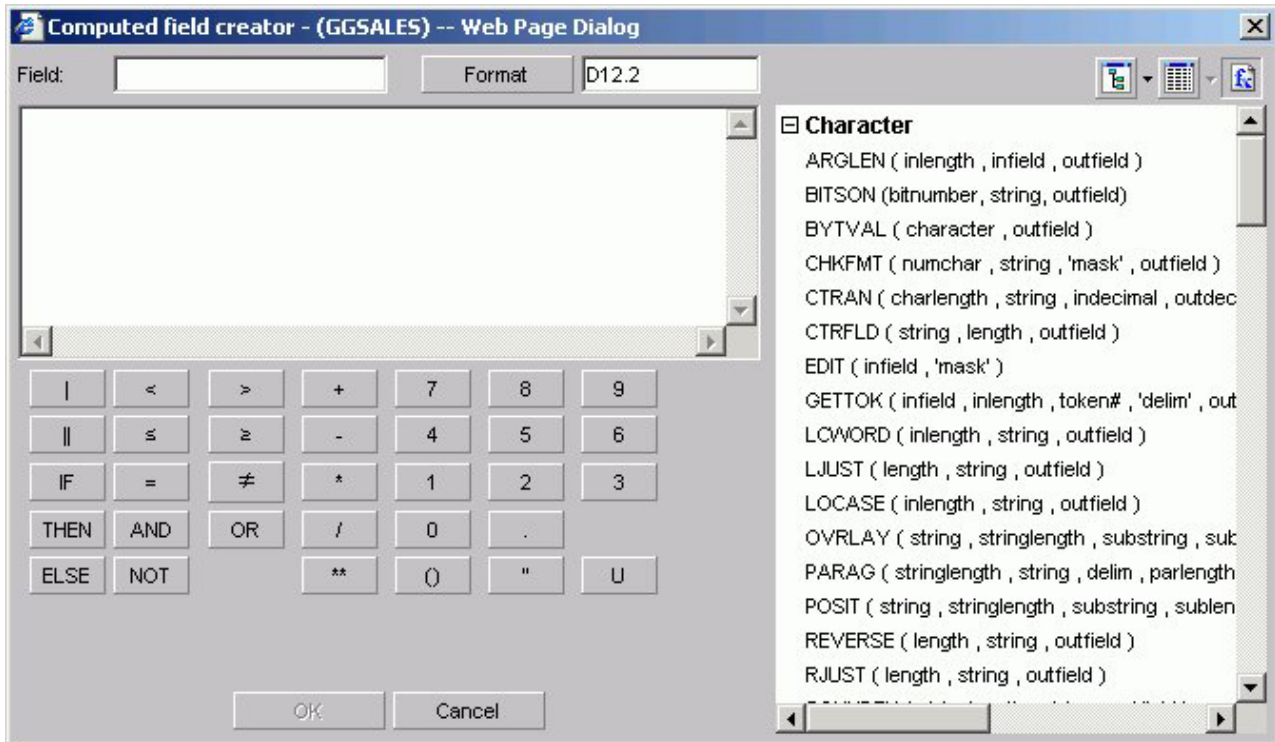
Operator on the computer keypad

Click to include arithmetic operations in an expression.

Boolean operator on the computer keypad

Click to include a Boolean operator in an expression.

Conditional operator on the computer keypad

Click to include a conditional operator in an expression.

Headings Tab

The Headings tab is used to create headings and footings. It has Page and Graph subtabs.

Headings Tab: Page

The Page subtab of the Headings tab shown in the following image is used to create report headings and footings.
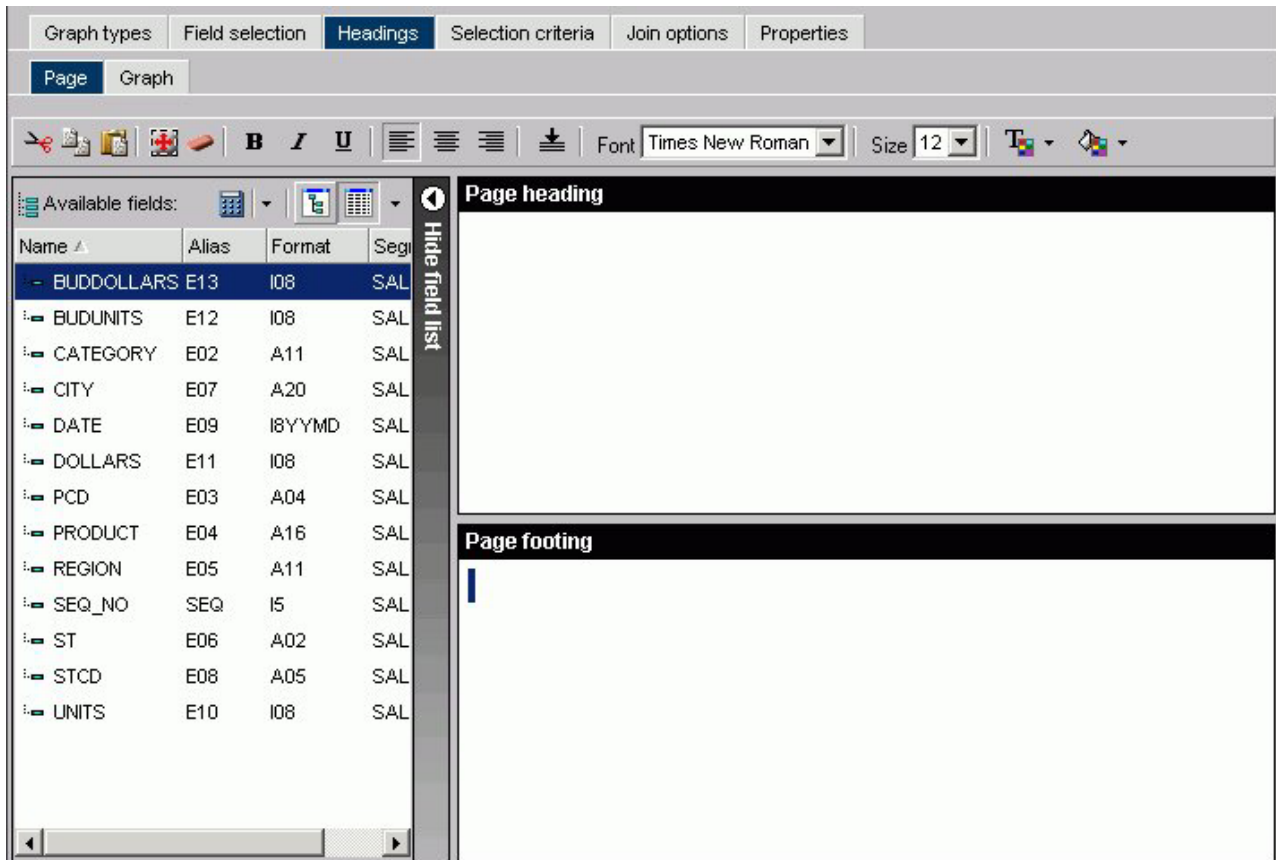
Formatting tool bar

- **Cut, copy, paste.** Cut or copy and paste a selection.
- **Select all.** Select all highlighted text.
- **Remove formatting.** Remove formatting from selection.
- **Font styles.** Select from bold, italic, and underline.
- **Alignment.** Select left, center, right, or bottom alignment.
- **Font.** Select a different font from the drop-down list.
- **Font size.** Select a font size from the drop-down list.
- **Text color.** Select the text color from the color menu.
- **Background color.** Select the background color from the color menu.

Show/Hide Fields list

Insert a field in your heading or footing from the fields list.

Page Heading

Enter the text for the heading in the Page Heading box.

Page Footing

Enter the text for the footing in the Page Footing box.

Headings Tab: Graph

The Graph subtab of the Headings tab shown in the following image is used to create chart titles, subtitles, and footings.
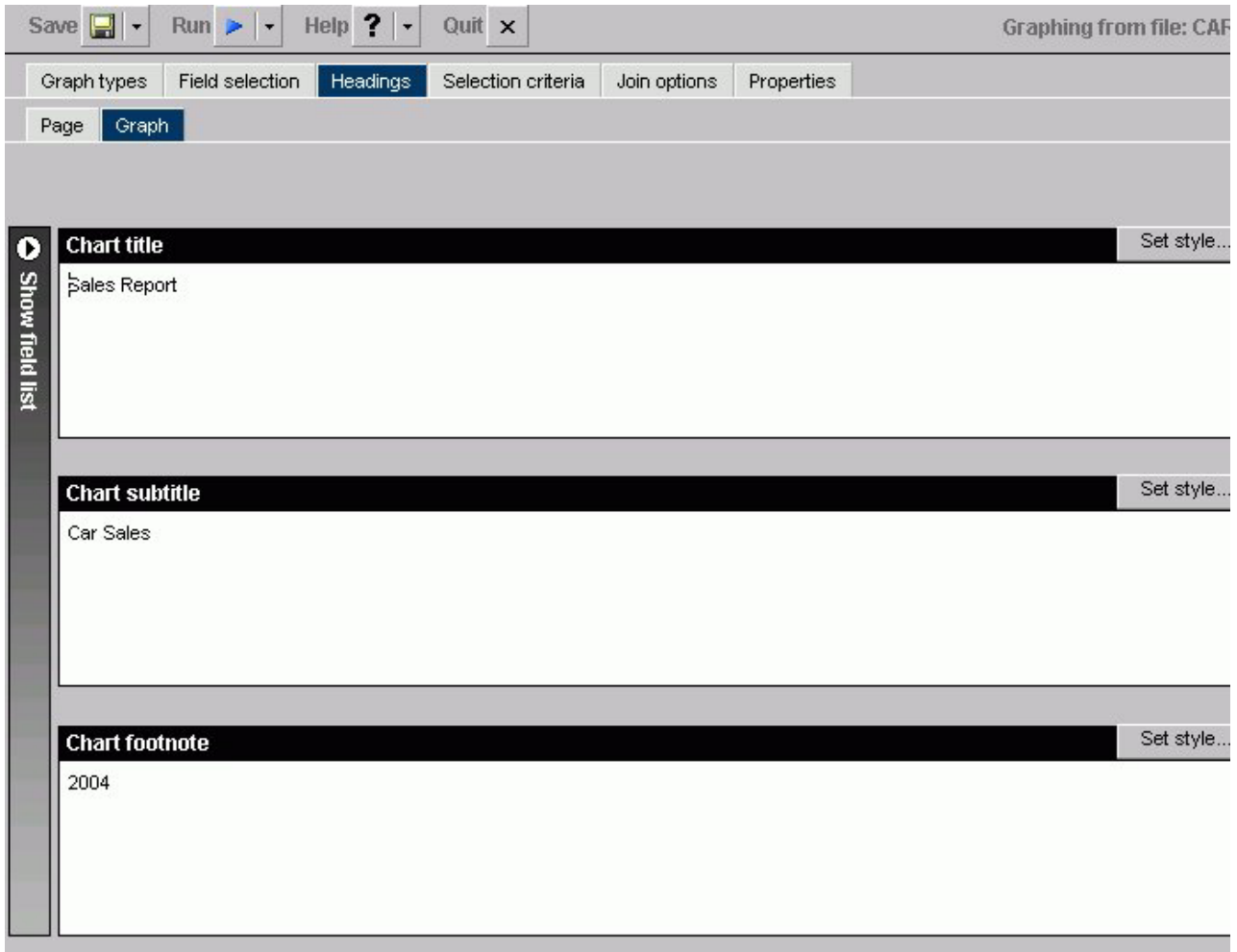
Show/Hide Fields list

> Insert a field in your heading or footing from the fields list.

Chart Title

> Enter text for the chart title here. The chart title will appear in the graph output as well as in your browser's title bar.

Chart Subtitle

> Enter text for the chart subtitle here.

Chart Footnote

> Enter text for the chart footnote here.

Style Title, Style Subtitle, Style Footnote

> To change the properties, click **Style Title**, **Style Subtitle**, or **Style Footnote** button for the respective text box (Chart Title, Chart Subtitle, Chart Footnote). For details, see Chart Title, Subtitle, and Footnote Properties Dialog Box.

Chart Title, Subtitle, and Footnote Properties Dialog Box

In the Headings tab, click Set style button in the Graph subtab in the Chart Title, Chart Subtitle, and Footnote areas to open the Heading properties dialog box as shown in the following image.
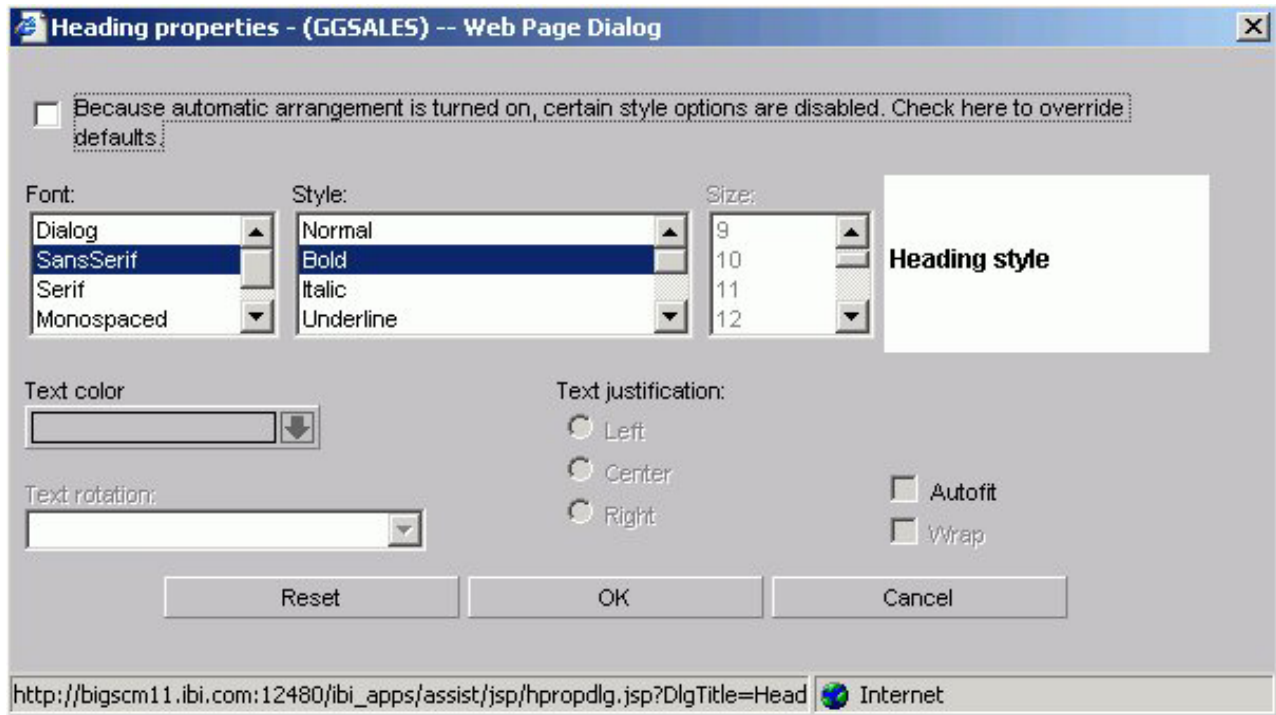
Font

Select the font type.

Style

Select the font style.

Size

Select the text size.

Text Color

Select a text color from the drop-down list.

Text Rotation

Horizontal. Displays text in a horizontal orientation that reads left to right.

Vertical - Bottom to top. Displays text in a vertical orientation that reads from bottom to top.

Vertical - Top to bottom. Displays text in a vertical orientation that reads from top to bottom.

Text Justification

Indicate whether the text will be left, center, or right justified.

Autofit

Checked. Automatically adjusts the heading font size to fit the graph and disables the Size drop-down list.

Unchecked. This is the default. To select a different heading font size, select the font size in the Size drop-down list.

Wrap

Check this box to wrap the text.

Selection Criteria Tab in HTML Graph Assistant

You can create WHERE and WHERE TOTAL statements from the Selection criteria tab in the Graph Assistant.

Selection Criteria Tab - Screen Conditions Window

The Screening conditions window shown in the following image allows you to drag and drop a field from the Available fields window.
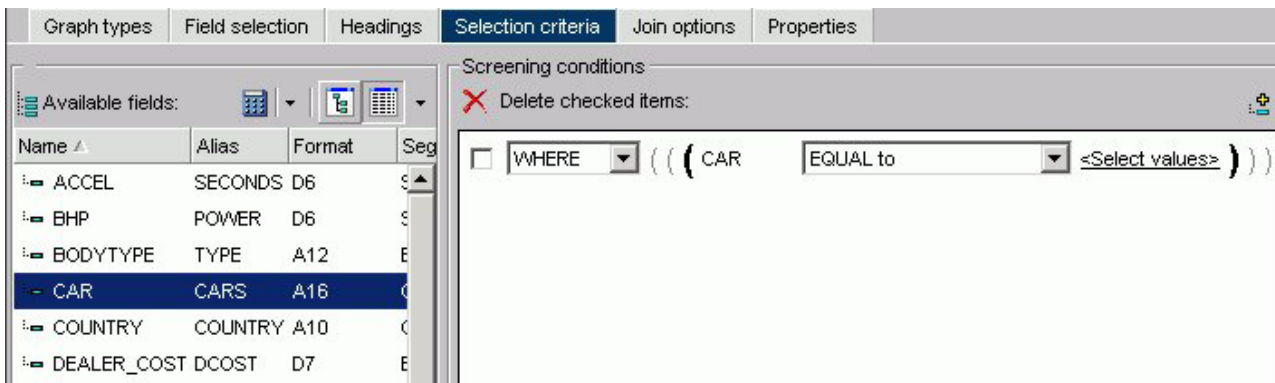
WHERE/WHERE TOTAL

Use WHERE to select records based on the values of an individual field.

Use WHERE TOTAL to select records based on the aggregate value of a field, for example, the sum or average of a field's values.

Fieldname

Is the fieldname you initially chose.

Expression

Select an expression from the drop-down list, for example, equals.

Select Values

Click to open the Values dialog box.
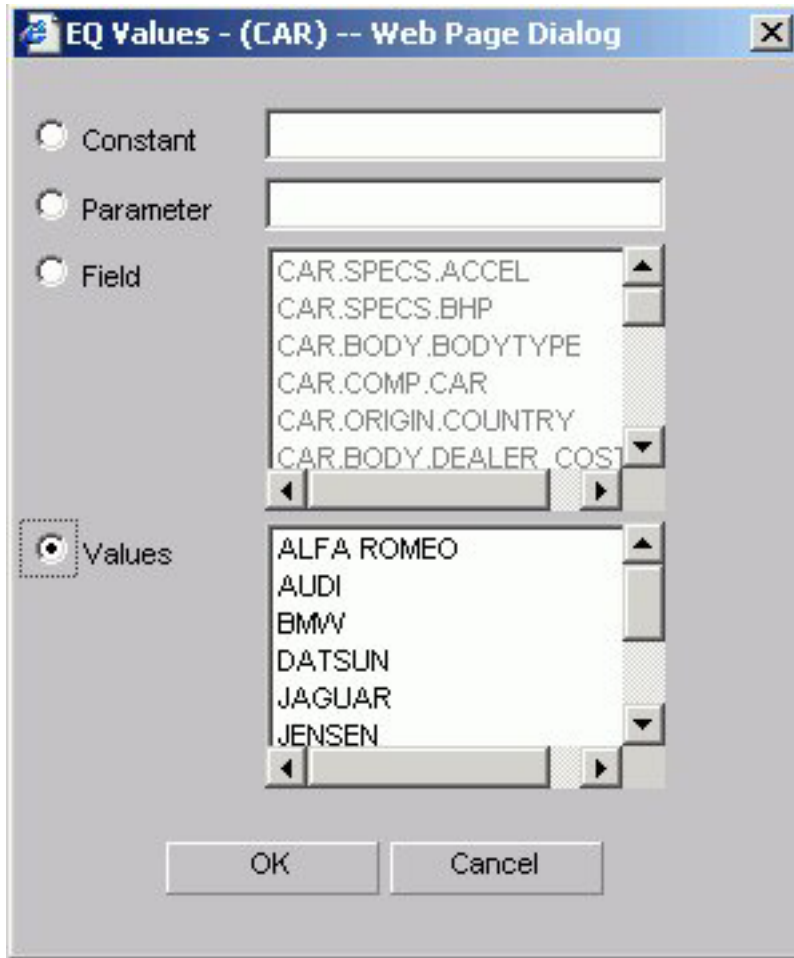
Constant

Enter a constant.

Parameter

Enter the name of the &variable.

Field

Select a field from the Field list box.

Values

Select a field from the Values list box.

Join Options Tab

The Join options tab is used to create joins.

Join Options Tab: Add or New

The following image shows the Join options tab where you create joins for a graph.
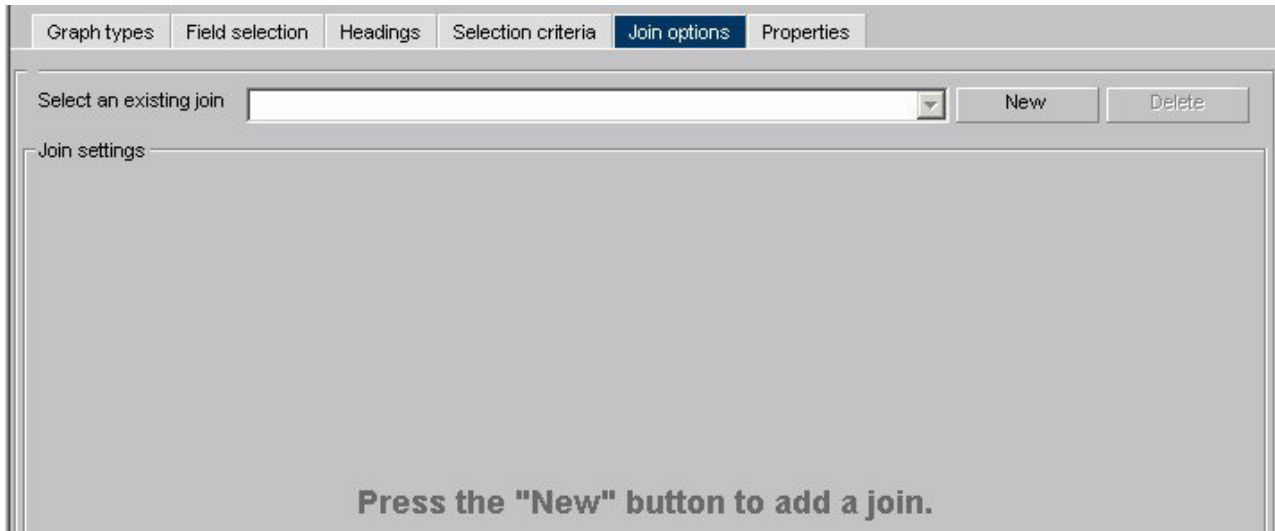
Select an existing join

Enter the name of a join you have previously created.

New

Press the New button to create a new join.

Delete

Click to delete the join.

OK button

Click to select a Master File.

Cancel

Click to return to the Add/New Join window.

When you select a target data source you have multiple join related settings and options, as shown in the following image.

The Join Settings section provides the following fields and buttons for creating joins:

Description

> Enter a description for your join.

Internal name

> DB2 Web Query assigns an alphanumeric name for the join, for example J001.

Save & Create

> Once you have selected your host and target fields, click Save & Create to create the join.

View

> Click to view the syntax of your join, which displays the code sent to the DB2 Web Query Reporting Server to validate the join, but may not be the same code required to run the report. This is the case for a DEFINE based JOIN where only the JOIN statement is displayed for validation, but both the JOIN and DEFINE statements are needed when the report is run.

The Join Type drop-down list provides the following join type options:

Unspecified

> Select from the Join Types drop-down list to indicate neither an inner or outer join.

Inner Join

Select from the Join Types drop-down list for a join that results when a report omits host rows that lack corresponding cross-referenced rows.

Left Outer Join

Select from the Join Types drop-down list to extend the results of an Inner Join and retrieve records from both host and cross-referenced tables, including all records from the left table (host) and any records from the right table (cross-referenced) where the condition values match. If there are no matching values in the cross-referenced table, the join still retrieves records from the host table.

The Instances drop-down list provides the following join instance options:

Multiple Instance (formerly Non Unique)

Select from the Instances drop-down list for a one-to-many join structure that matches one value in the host data source to multiple values in the cross-referenced field. Joining employee ID in a company's employee data source to employee ID in a data source that lists all the training classes offered by that company would result in a listing of all courses taken by each employee, or a joining of the one instance of each ID in the host file to the multiple instances of that ID in the cross-referenced file.

Single Instance (formerly Unique)

Select from the Instances drop-down list for a one-to-one join structure that matches one value in the host data source to one value in the cross-referenced data source. Joining an employee ID in an employee data source to an employee ID in a salary data source is an example of a unique join.

Host Field List:

Search (binoculars) icon

Opens the Field List Searching dialog box. See Searching Field Lists for details.

Define icon

Click the Define icon to select: New define field, Edit define field, or Delete define field. If you select the New define field or Edit define field, the Define field creator dialog box opens where you can create or edit a defined field.

Tree icon

Displays a tree structure of field names by segment with segment, alias, title, description, and format information below the tree. Individual field information appears when you click on the field.

List icon

Displays a list of all field names with information that you select from the adjacent arrow that opens a drop-down list: name, alias, title, remarks, format, description, segment, and file name. You can also choose to sort by file order.

Add button

Adds a field.

Remove button

Removes a field.

Move Up or Move Down buttons

Moves the position of the field within the join.

Target Field List:

Select a target button

Click to select a target Master File.

Tree icon

Displays a tree structure of field names by segment with segment, alias, title, description, and format information below the tree. Individual field information appears when you click the field.

List icon

Displays a list of all field names with information that you select from the adjacent arrow that opens a drop-down list: name, alias, title, remarks, format, description, segment, and file name. You can also choose to sort by file order.

Add button

Adds a field.

Remove button

Removes a field.

Move Up or Move Down buttons

Moves the position of the field within the join.

Properties Tab

The Properties tab is used to set properties options for one or more components of a graph.

Legend Properties Dialog Box

Click the **Style Legend Text** button on the Settings subtab in the Properties tab to open the Legend properties dialog box as shown in the following image.
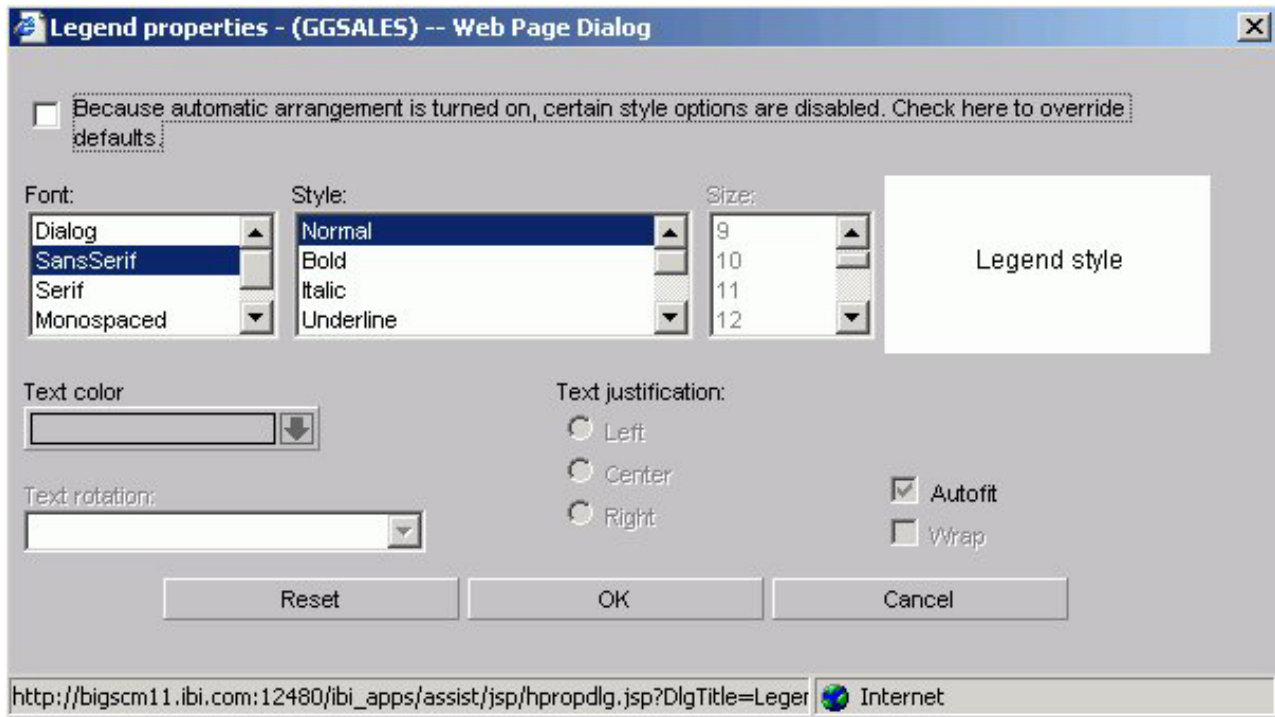
Font

Select the font type.

Style

Select the font style.

Size

Select the text size.

Text Color

Select a text color from the drop-down list.

Text Rotation

Horizontal. Displays legend text in a horizontal orientation that reads left to right.

Vertical - Bottom to top. Displays legend text in a vertical orientation that reads from bottom to top.

Vertical - Top to bottom. Displays legend text in a vertical orientation that reads from top to bottom.

Text Justification

Indicate whether the text will be left, center, or right justified.

Autofit

Checked. This is the default. Automatically adjusts the legend font size to fit the graph and disables the Size drop-down list.

Unchecked. To select a different legend font size, uncheck this box and select the font size in the Size drop-down list.

Wrap

Checked to enable wrapping the text.

X-axis Label Properties Dialog Box

Click the **Style Labels** button on the X axis subtab of the Properties tab to open the X-axis properties dialog box as shown in the following image.
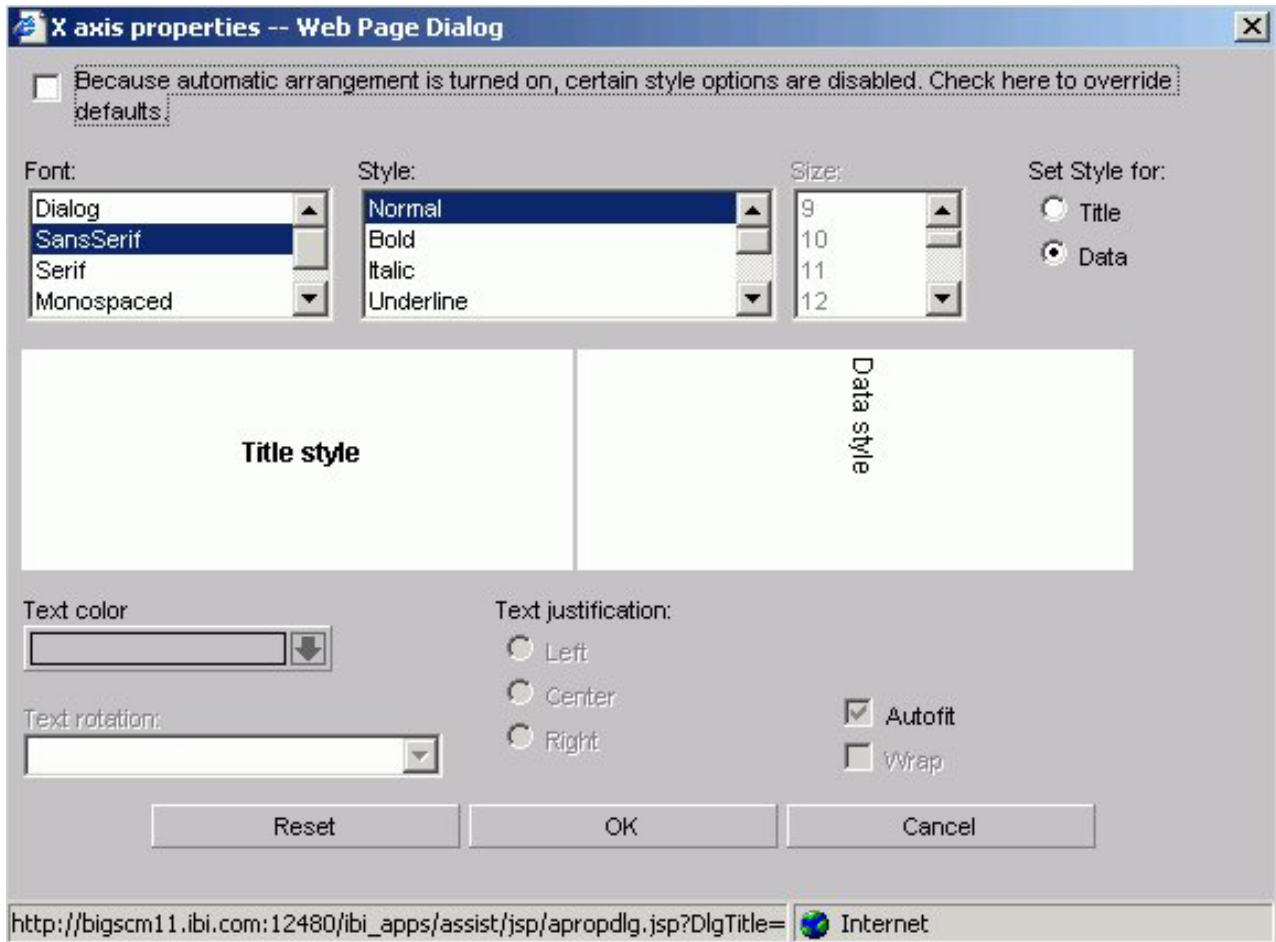
Font

Select the font type.

Style

Select the font style.

Size

Select the text size.

Set Style For

Select to style Title or Data.

Text Color

Select a text color from the drop-down list.

Text Rotation

Horizontal. Displays text in a horizontal orientation that reads left to right.

Vertical - Bottom to top. Displays text in a vertical orientation that reads from bottom to top.

Vertical - Top to bottom. Displays text in a vertical orientation that reads from top to bottom.

45-degree bottom to top. Displays text in a diagonal that reads from bottom to top.

Text Justification

Indicate whether the text will be left, center, or right justified.

Autofit

Checked. This is the default. Automatically adjusts the X-axis font size to fit the graph and disables the Size drop-down list.

Unchecked. To select a different X-axis font size, uncheck this box and select the font size in the Size drop-down list.

Wrap

Check this box to wrap the text.

Y-axis Label Properties Dialog Box

Click the **Style Labels** button on the Y axis subtab of the Properties tab to open the Y-axis properties dialog box as shown in the following image.
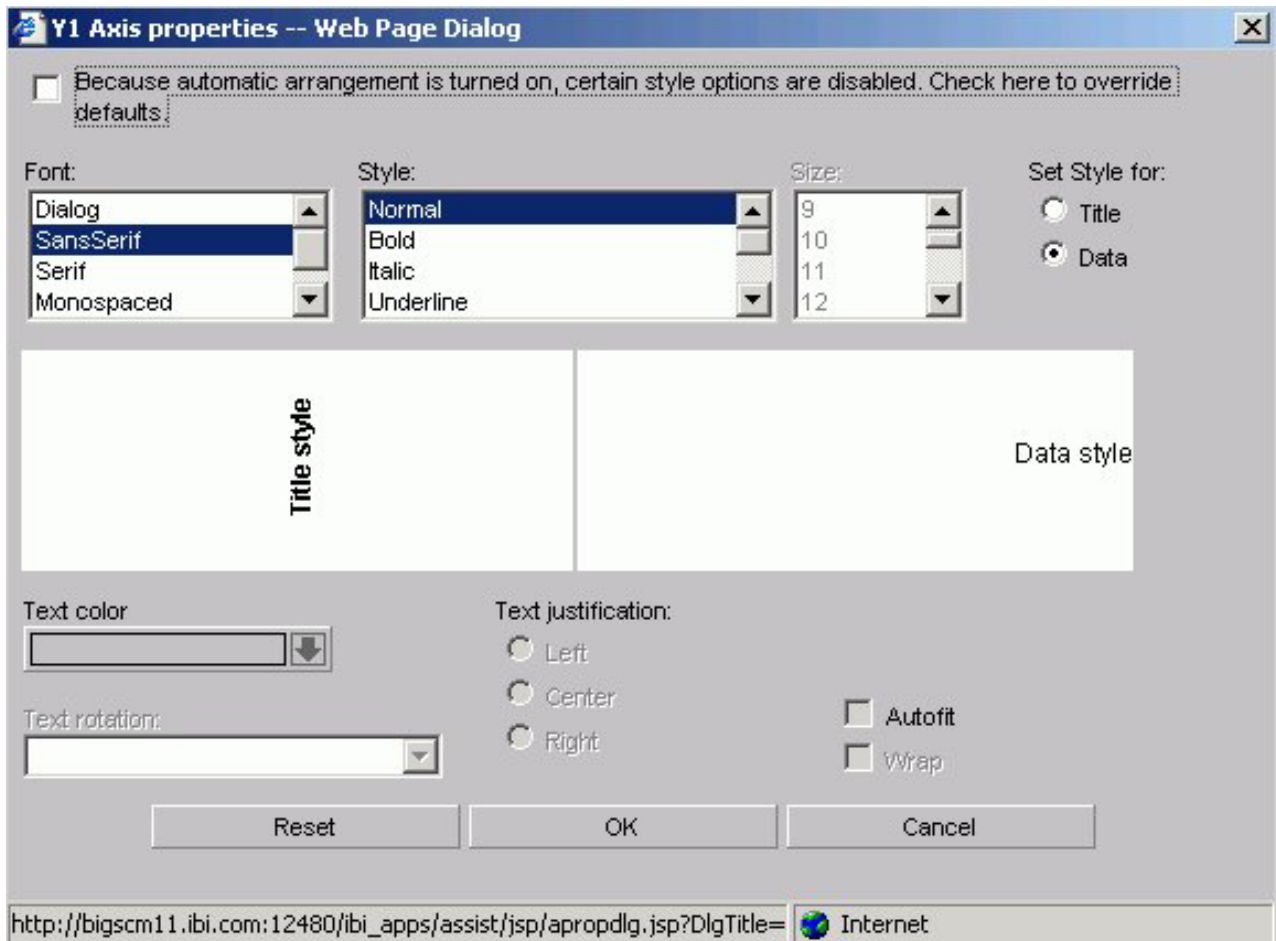
Font

> Select the font type.

Style

> Select the font style.

Size

> Select the text size.

Set Style For

> Select to style Title or Data.
>
> If you have selected more than one Y-axis field, the Title option will not be available in the Label Properties dialog box. The Data option will be selected by default.

Text Color

> Select a text color from the drop-down list.

Text Rotation

> **Horizontal.** Displays text in a horizontal orientation that reads left to right.
>
> **Vertical - Bottom to top.** Displays text in a vertical orientation that reads from bottom to top.

> **Vertical - Top to bottom.** Displays text in a vertical orientation that reads from top to bottom.
>
> **45-degree bottom to top.** Displays text in a diagonal that reads from bottom to top.

Text Justification

> Indicate whether the text will be left, center, or right justified.

Autofit

> **Checked.** Automatically adjusts the Y-axis font size to fit the graph and disables the Size drop-down list.
>
> **Unchecked.** This is the default. To select a different Y-axis font size, select the font size in the Size drop-down list.

Wrap

> Check this box to wrap the text.

Graph Assistant Properties Tab: Options

The following image shows the Options subtab of the Properties tab.

Graph Size

> Select the DB2 Web Query default size, let the applet application choose the size, or set a custom size for horizontal and vertical, both in pixels. For more information about graph size, see Set the Graph Height and Width.

Output

Select **Screen** or **Printer** from the list box. If you select:

Screen, the output appears in the browser. Choose: PNG, SVG, GIF, JPEG, PDF/SVG, PDF/GIF.

Printer, graph output is sent directly to your browser's default printer. Choose: PNG, SVG, GIF, JPEG.

Note: To include a graph in the PDF Layout Painter, the graph must be saved in .SVG format.

Background Color

Select a background color from the color picker for the current graph.

Frame Color

Select a frame color from the color picker for the current graph.

Stop retrieving records after the specified number have been included in the report

Record limits are used to limit the amount of data that appears or is used in your graph.

Limit the number of read operations performed during report generation

Read limits are used to limit the amount of records retrieved (or read) from the data source. The Read Limit option is not available with FOCUS data sources.

Use SQL Engine Joins. Multiplicative effect may result. Web Query protection is suppressed.

Enables the database to use its internal optimization techniques which reduce the volume of database-to-server communications and improve response time.

Summary Text for Graph

Provides a SUMMARY attribute for reports and graphs that maps to the HTML <TABLE SUMMARY> tag. A description of the graph is placed into a summary object inside the HTML table in the source code. This complies with section 508 accessibility.

The description of the graph can be no longer than 500 characters.

Graph Assistant Properties Tab: Settings

The following image shows the Settings subtab of the Properties tab.

Graph Settings

Show Data Values

> Check this box to display data values in your graph. If this box is unchecked, data values will not be included.

FOCUS Grid

> Grids are parallel lines drawn across a graph at the vertical and horizontal class marks on the axes. Check this box if you wish grid lines displayed. This option is only available on connected point plots, histograms, bar charts, and scatter diagrams.

Graph on Server

> Runs the graph on a server (rather than your local machine) and produces the graph as an HTML file with a GIF image. Graph on Server cannot be used when the Enable Zoom/Pan box is checked.
>
> **Note:** This option is only available if the server was configured for server side graphics.

Enable Zoom/Pan

Enables you to zoom on a graph. This feature is available for all graphs except radar, polar, 3D, pie, spectral, and stock graphs. Enable Zoom/Pan cannot be used when the Graph on Server box is checked.

Legend Settings

Show Legend

Check this to display the legend in your graph.

Legend Style

Select the position of the legend's text. You may select to have the legend's text on the right or left side, above or below the legend marker, or inside the legend marker.

Reverse Legend Order

Enables the display of the graph legend in reverse order. This is generally most helpful when Reverse Order of Groups is selected.

Style Legend Text

Opens the Legend properties dialog box where you can select and style the font for the legend text.

Markers

Select how you want markers (shapes that represent data points) to appear on your graph.

Show Markers

Will show marker shapes at the data points on line graphs. You must have this option set to on in order for values to appear when you hover your cursor over an area or line graph.

Use on Legend

Will show marker shapes on the graph legend.

Show Lines

Is the default in the Markers group. When this check box is deselected, marker shapes show on a line graph, but connecting lines are not drawn.

Graph Assistant Properties Tab: X axis

The following image shows the X axis subtab of the Properties tab.

Show axis labels

> **Top** - X-axis labels appear above vertically oriented graphs and to the right of horizontally oriented graphs.

> **Bottom** - X-axis labels appear below vertically oriented graphs and to the left of horizontally oriented graphs. (This is the default.)

> **Top and Bottom** - X-axis labels appear both above and below vertically oriented graphs and to both the left and right of horizontally oriented graphs.

> **Stagger Labels** - When this box is checked, X-axis labels will be staggered.

> **Hide first label** - When this box is checked, the first X-axis label will not appear.

> **Hide last label** - When this box is checked, the last X-axis label will not appear.

Value Format

> Enables you to select a format for the display of X-axis values. You can only change the display format if your x-axis values are numeric and the graph style is bubble, histogram (bar), or scatter.

Style Labels

> Opens the X-axis properties dialog box where you can select and style the fonts for the X-axis.

Reverse order of groups

> When checked, enables the display of the groups (X-axis) data in reverse order. When unchecked, the graph will display groups in natural order.

Major grid lines

> Divide a graph at regular intervals according to the scale. Major grid lines cross their axis line at the exact location of the scale label. Styles include:

Regular Grids - Normal grid lines that are the height of the frame.

Grids and Ticks - Normal grid lines that extend outside the frame.

Inner Ticks - Small tick marks from the frame edge inward.

Outer Ticks - Small tick marks from the frame edge outward.

Spanning Ticks - Small tick marks that span the frame edge.

Minor grid lines

Divide a graph at regular intervals according to the scale. Minor grid lines cross their axis line at equal intervals between major grids/ticks; they never align directly with a scale label. Styles include:

Regular Grids - Normal grid lines that are the height of the frame.

Grids and Ticks - Normal grid lines that extend outside the frame.

Inner Ticks - Small tick marks from the frame edge inward.

Outer Ticks - Small tick marks from the frame edge outward.

Spanning Ticks - Small tick marks that span the frame edge.

Graph Assistant Properties Tab: Y axis

The following image shows the Y axis subtab of the Properties tab.

Axis Selection List Box

If you have more than one Y-axis, select which axis you would like to apply properties to here, either Y1 Axis or Y2 Axis.

Show axis labels

Left Side - Y-axis labels will appear on the low (bottom or left) side of the graph.

Right Side - Y-axis labels will appear on the high (top or right) side of the graph.

Both Sides - Y-axis labels will appear on both the low and high sides of the graph.

Hide Maximum Value - When checked, the largest Y-axis label will not appear.

Hide Minimum Value - When checked, the smallest Y-axis label will not appear.

Always include Zero on Scale - Includes zero on the Y-axis scale when checked. If you have manually set your scale range to exclude zero, this property will be ignored.

Log Scale - When checked, the Y-axis scale will use logarithmic scaling. When unchecked, the Y-axis scale will use linear scaling.

Value Format

Enables you to select a format for the display of Y-axis series values. Value Format cannot be used in combination with Slice Label.

Style Labels

Opens the Y-axis properties dialog box where you can select and style the fonts for the Y-axis.

Reverse order of series

When checked, enables the display of the series (Y-axis) data in reverse order. When unchecked, the graph will display its series in natural order.

Descending Axis

Inverts the direction of the Y-axis so values of zero or less appear at the top of the graph, while values greater than zero appear at the bottom of the graph.

Axis Line

Enables the display of the Y-axis base line.

Zero Line

Enables the display of the Y-axis Zero line when zero is within the Y-axis range.

Missing Values Display

Displays missing data values (in a bar graph, line graph, area graph, or any variation of these graph types) in one of the following formats:
- Graph as zero.
- Graph as gap.
- Dotted line to zero.
- Interpolated dotted line.

Major Grid Lines

Divide a graph at regular intervals according to the scale. Major grid lines cross their axis line at the exact location of the scale label. Styles include:

Regular Grids - Normal grid lines that are the height of the frame.

Grids and Ticks - Normal grid lines that extend outside the frame.

Inner Ticks - Small tick marks from the frame edge inward.

Outer Ticks - Small tick marks from the frame edge outward.

Spanning Ticks - Small tick marks that span the frame edge.

Minor Grid Lines

Divide a graph at regular intervals according to the scale. Minor grid lines cross their axis line at equal intervals between major grids/ticks; they never align directly with a scale label. Styles include:

Regular Grids - Normal grid lines that are the height of the frame.

Grids and Ticks - Normal grid lines that extend outside the frame.

Inner Ticks - Small tick marks from the frame edge inward.

Outer Ticks - Small tick marks from the frame edge outward.

Spanning Ticks - Small tick marks that span the frame edge.

Graph Assistant Properties Tab: Pie

The following image shows the Pie subtab of the Properties tab.

Show Axis Labels

When checked, enables the display of feelers and data text in a pie graph.

Feeler Lines and Labels - Shows both feelers and data text.

Labels Only - Shows only data text.

Labels on Slices - Shows data text on slices.

A feeler is a line or set of lines that stretch from a pie slice label to its pie slice. Feelers are visual aids that help you know that a particular number belongs to a particular pie slice.

Slice Label

When this option button is selected, the Y-axis label will appear at the end of the feeler.

Percent Value

> When this option button is selected, the Y-axis value expressed as a percentage appears at the end of the feeler.

Slice Label and % Value

> When this option button is selected, the Y-axis label, and its value expressed as a percentage, appear at the end of the feeler.

True Value of Slice

> When this option button is selected, the Y-axis value appears at the end of the feeler.

Value Format

> Enables you to select a format for the display of Y-axis series values. Value Format cannot be used in combination with Slice Label.

Chapter 7. Using Functions

Introduces functions and explains the different types of available functions.

The following topics offer an introduction to functions and explain how to use them. Functions provide a convenient way to perform certain calculations and manipulations.

In order to use any of the specified functions described in this reference manual, you need to create a temporary field which is discussed later.

Types of Functions

You can access any of the following types of functions:

- **Character functions.** Manipulate alphanumeric fields or character strings.
- **Data source and decoding functions.** Retrieve data source values and assign values based on the value of an input field.
- **Date and time functions.** Manipulate dates and times.
- **Format conversion functions.** Convert fields from one format to another.
- **Numeric functions.** Perform calculations on numeric constants and fields.
- **System functions.** Call the operating system to obtain information about the operating environment.

Character Functions

The following functions manipulate alphanumeric fields or character strings.

ARGLEN

Measures the length of a character string within a field, excluding trailing blanks.

BITSON

Evaluates an individual bit within a character string to determine whether it is on or off.

BYTVAL

Translates a character to its corresponding ASCII or EBCDIC decimal value.

CHKFMT

Checks a character string for incorrect characters or character types.

CTRAN

Translates a character within a character string to another character based on its decimal value.

CTRFLD

Centers a character string within a field.

EDIT

Extracts characters from or adds characters to a character string.

GETTOK

Divides a character string into substrings, called tokens, where a specific character, called a delimiter, occurs in the string.

LCWORD

Converts the letters in a character string to mixed case.

LJUST

Left-justifies a character string within a field.

LOCASE

Converts alphanumeric text to lowercase.

OVRLAY

Overlays a base character string with a substring.

PARAG

Divides a line of text into smaller lines by marking them with a delimiter.

POSIT

Finds the starting position of a substring within a larger string.

REVERSE

Reverses the characters in a character string.

RJUST

Right-justifies a character string.

SOUNDEX

Searches for a character string phonetically without regard to spelling.

SPELLNUM

Takes an alphanumeric string or a numeric value with two decimal places and spells it out with dollars and cents.

SUBSTR

Extracts a substring based on where it begins and its length in the parent string.

UPCASE

Converts a character string to uppercase.

Data Source and Decoding Functions

The following functions retrieve data source values and assign values.

DECODE

Assigns values based on the coded value of an input field.

LAST

Retrieves the preceding value for a field.

Date and Time Functions

The following functions manipulate dates and times.

Standard Date and Time Functions

AYM

Adds or subtracts months from dates that are in year-month format.

AYMD

Adds or subtracts days from dates that are in year-month-day format.

CHGDAT

Rearranges the year, month, and day portions of alphanumeric dates, and converts dates between long and short date formats.

DA

Converts dates to the corresponding number of days elapsed since December 31, 1899.

DADMY converts dates in day-month-year format.

DADYM converts dates in day-year-month format.

DAMDY converts dates in month-day-year format.

DAMYD converts dates in month-year-day format.

DAYDM converts dates in year-day-month format.

DAYMD converts dates in year-month-day format.

DATEADD

Adds a unit to or subtracts a unit from a date format.

DATECVT

Converts date formats.

DATEDIF

Returns the difference between two dates in units.

DATEMOV

Moves a date to a significant point on the calendar.

DATETRAN

Formats dates in international formats.

DMY, MDY, and YMD

Calculates the difference between two dates.

DOWK and DOWKL

Finds the day of the week that corresponds to a date.

DT

Converts the number of days elapsed since December 31, 1899 to the corresponding date.

DTDMY converts numbers to day-month-year dates.

DTDYM converts numbers to day-year-month dates.

DTMDY converts numbers to month-day-year dates.

DTMYD converts numbers to month-year-day dates.

DTYDM converts numbers to year-day-month dates.

DTYMD converts numbers to year-month-day dates.

GREGDT

Converts dates in Julian format to year-month-day format.

HADD

Increments a date-time field by a given number of units.

HCNVRT

Converts a date-time field to a character string.

HDATE

Extracts the date portion of a date-time field, converts it to a date format, and returns the result in the format YYMD.

HDIFF

Calculates the number of units between two date-time values.

HDTTM

Converts a date field to a date-time field. The time portion is set to midnight.

HGETC

Stores the current date and time in a date-time field.

HHMMSS

Retrieves the current time from the system.

HINPUT

Converts an alphanumeric string to a date-time value.

HMIDNT

Changes the time portion of a date-time field to midnight (all zeros).

HNAME

Extracts a specified component from a date-time field and returns it in alphanumeric format.

HPART

Extracts a specified component from a date-time field and returns it in numeric format.

HSETPT

Inserts the numeric value of a specified component into a date-time field.

HTIME

Converts the time portion of a date-time field to the number of milliseconds or microseconds.

JULDAT

Converts dates from year-month-day format to Julian (year-day format).

TIMETOTS

Converts a time to a timestamp.

TODAY

Retrieves the current date from the system.

YM

Calculates the number of months that elapse between two dates. The dates must be in year-month format.

Format Conversion Functions

The following functions convert fields from one format to another.

ATODBL

Converts a number in alphanumeric format to double-precision format.

EDIT

Converts an alphanumeric field that contains numeric characters to numeric format or converts a numeric field to alphanumeric format.

FTOA

Converts a number in a numeric format to alphanumeric format.

HEXBYT

Obtains the ASCII or EBCDIC character equivalent of a decimal integer value.

ITONUM

Converts a large binary integer in a data source to double-precision format.

ITOPACK

Converts a large binary integer in a data source to packed-decimal format.

ITOZ

Converts a number in numeric format to zoned format.

PCKOUT

Writes a packed number of variable length to an extract file.

Numeric Functions

The following functions perform calculations on numeric constants or fields.

ABS

Returns the absolute value of a number.

BAR

Produces a horizontal bar chart.

CHKPCK

Validates the data in a field described as packed format.

DMOD, FMOD, and IMOD

Calculates the remainder from a division.

EXP

Raises the number ″e″ to a specified power.

INT

Returns the integer component of a number.

LOG

Returns the natural logarithm of a number.

MAX and MIN

Returns the maximum or minimum value, respectively, from a list of values.

PRDNOR and PRDUNI

Generates reproducible random numbers.

RDNORM and RDUNIF

Generates random numbers.

SQRT

Calculates the square root of a number.

System Functions

The following functions call the operating system to obtain information about the operating environment.

FGETENV

Retrieves the value of an environment variable and returns it as an alphanumeric string.

GETUSER

Retrieves the ID of the connected user.

HHMMSS

Retrieves the current time from the system.

TODAY

Retrieves the current date from the system.

Supplying an Argument in a Function

When supplying an argument in a function, you must understand which types of arguments are acceptable, the formats and lengths for these arguments, and the number and order of these arguments.

Argument Types

The following are acceptable arguments for a function:

- Numeric constant, such as 6 or 15.
- Date constant, such as 022802.
- Date in alphanumeric, numeric, or date format.
- Alphanumeric literal, such as STEVENS or NEW YORK NY. A literal must be enclosed in single quotation marks.
- Number in alphanumeric format.
- Field name, such as FIRST_NAME or HIRE_DATE. A field can be a data source field or temporary field. The field name can be up to 66 characters long or a qualified field name, unique truncation, or alias.

- Expression, such as a numeric, date, or alphanumeric expression. An expression can use arithmetic operators and the concatenation sign (|). For example, the following are valid expressions:

  `CURR_SAL * 1.03 and FN || LN`
- Format of the output value enclosed in single quotation marks.
- Another function.

Argument Formats

Depending on the function, an argument can be in alphanumeric, numeric, or date format. If you supply an argument in the wrong format, you will cause an error or the function will not return correct data. To obtain the valid formats, click the Format button on the tool for a list of possible types and lengths. The following are the types of argument formats:

- **Alphanumeric argument.** An alphanumeric argument is stored internally as one character per byte. An alphanumeric argument can be a literal, an alphanumeric field, a number or date stored in alphanumeric format, an alphanumeric expression, or the format of an alphanumeric field.

- **Numeric argument.** A numeric argument is stored internally as a binary or packed number. A numeric argument includes integer (I), floating-point single-precision (F), floating-point double-precision (D), and packed-decimal (P) formats. A numeric argument can be a numeric constant, field, or expression, or the format of a numeric field. All numeric arguments are converted to floating-point double-precision format when used with a function, but results are returned in the format specified for the output field.

- **Date argument.** A date argument can be in either alphanumeric, numeric, or date format. The list of arguments for the individual function will specify what type of format the function accepts. A date argument can be a date in alphanumeric, numeric, or date format; a date field or expression; or the format of a date field. If you supply an argument with a two-digit year, the function assigns a century based on the DATEFNS, YRTHRESH, and DEFCENT parameter settings.

Using a Function in a Temporary Field

In order to use any of the specified functions described in this reference manual, you need to create a temporary field.

A temporary field is a field whose value is not stored in the data source, but can be calculated from the data that is there, or assigned an absolute value. When you create a temporary field, you determine its value by writing an expression. You can combine fields, constants, and operators in an expression to produce a single value. For example, if your data contains salary and deduction amounts, you can calculate the ratio of deductions to salaries using the following expression: deduction / salary.

You can specify the expression yourself, or you can use one of the many supplied functions that perform specific calculations or manipulations. In addition, you can use expressions and functions as building blocks for more complex expressions, as well as use one temporary field to evaluate another. You can create temporary fields from the Field selection tab by selecting the Define or Compute tool.

All examples in this reference manual show the complete syntax for Computed fields, as seen in a text editor. For example, `COMPUTE NAME_LEN/I3 = ARGLEN(15,` `LAST_NAME, 'I3');`

When creating a temporary field using the Compute tool, you do *not* need to:

- Include COMPUTE in the expression, which is implicit.
- End the expression with a semicolon, which is implicit.
- Include AND when creating more than one expression; it is implicit. Each COMPUTE expression must be created separately as a new temporary field.

To illustrate, the following image in Report Assistant shows the function, ARGLEN in its full syntax. In the Compute tool, the Field name is NAME_LEN, the Format is I3, and the expression window contains ARGLEN(15, LAST_NAME, 'I3').

Character Functions

Character functions manipulate alphanumeric fields and character strings.

ARGLEN: Measuring the Length of a String

The ARGLEN function measures the length of a character string within a field, excluding trailing spaces. The field format in a Master File specifies the length of a field, including trailing spaces.

Measure the Length of a Character String

```
ARGLEN(inlength, infield, 'outfield')
```

where:

inlength

> Integer
>
> Is the length of the field containing the character string, or a field that contains the length.

infield

Alphanumeric

Is the name of the field containing the character string.

outfield

Integer

Is the format of the output value enclosed in single quotation marks.

Measuring the Length of a Character String

ARGLEN determines the length of the character string in LAST_NAME and stores the result in NAME_LEN:

```
COMPUTE NAME_LEN/I3 = ARGLEN(15, LAST_NAME, 'I3');
```

BITSON: Determining If a Bit Is On or Off

The BITSON function evaluates an individual bit within a character string to determine whether it is on or off. If the bit is on, BITSON returns a value of 1; if the bit is off, it returns a value of 0. This function is useful in interpreting multi-punch data, where each punch conveys an item of information.

Determine If a Bit Is On or Off

```
BITSON(bitnumber, string, 'outfield')
```

where:

bitnumber

Integer

Is the number of the bit to be evaluated, counted from the left-most bit in the character string.

string

Alphanumeric

Is the character string to be evaluated, enclosed in single quotation marks, or a field that contains the character string. The character string is in multiple eight-bit blocks.

outfield

Integer

Is the format of the output value enclosed in single quotation marks.

Evaluating a Bit in a Field

BITSON evaluates the 24th bit of LAST_NAME and stores the result in BIT_24:

```
COMPUTE BIT_24/I1 = BITSON(24, LAST_NAME, 'I1');
```

BYTVAL: Translating a Character to a Decimal Value

The BYTVAL function translates a character to the ASCII, EBCDIC, or Unicode decimal value that represents it, depending on the operating system.

Translate a Character

```
BYTVAL(character, 'outfield')
```

where:

character

>Alphanumeric

>Is the character to be translated. You can specify a field that contains the character, or the character itself enclosed in single quotation marks. If you supply more than one character, the function evaluates the first.

outfield

>Integer

>Is the format of the output value enclosed in single quotation marks.

Translating the First Character of a Field

BYTVAL translates the first character of LAST_NAME into its ASCII or EBCDIC decimal value and stores the result in LAST_INIT_CODE. Since the input string has more than one character, BYTVAL evaluates the first one.

```
COMPUTE LAST_INIT_CODE/I3 = BYTVAL(LAST_NAME, 'I3');
```

CHKFMT: Checking the Format of a String

The CHKFMT function checks a character string for incorrect characters or character types. It compares each character string to a second string, called a mask, by comparing each character in the first string to the corresponding character in the mask. If all characters in the character string match the characters or character types in the mask, CHKFMT returns the value 0. Otherwise, CHKFMT returns a value equal to the position of the first character in the character string not matching the mask.

If the mask is shorter than the character string, the function checks only the portion of the character string corresponding to the mask. For example, if you are using a four-character mask to test a nine-character string, only the first four characters in the string are checked; the rest are returned as a no match with CHKFMT giving the first non-matching position as the result.

Check the Format of a Character String

```
CHKFMT(numchar, string, 'mask', 'outfield')
```

where:

numchar

>Integer

>Is the number of characters being compared to the mask.

string

>Alphanumeric

>Is the character string to be checked enclosed in single quotation marks, or a field that contains the character string.

mask

>Alphanumeric

>Is the mask, which contains the comparison characters enclosed in single quotation marks.

Some characters in the mask are generic and represent character types. If a character in the string is compared to one of these characters and is the same type, it matches. Generic characters are:

A is any letter between A and Z (uppercase or lowercase).

9 is any digit between 0-9.

X is any letter between A-Z or any digit between 0-9.

$ is any character.

Any other character in the mask represents only that character. For example, if the third character in the mask is B, the third character in the string must be B to match.

outfield

> Integer
>
> Is the format of the output value enclosed in single quotation marks.

Checking the Format of a Field

CHKFMT examines EMP_ID for nine numeric characters starting with 11 and stores the result in CHK_ID:

```
COMPUTE CHK_ID/I3 = CHKFMT(9, EMP_ID, '119999999', 'I3');
```

CTRAN: Translating One Character to Another

The CTRAN function translates a character within a character string to another character based on its decimal value. This function is especially useful for changing replacement characters to unavailable characters, or to characters that are difficult to input or unavailable on your keyboard.

To use CTRAN, you must know the decimal equivalent of the characters in internal machine representation.

In Unicode configurations, this function uses values in the range:
- 0 to 255 for 1-byte characters.
- 256 to 65535 for 2-byte characters.
- 65536 to 16777215 for 3-byte characters.
- 16777216 to 4294967295 for 4-byte characters (primarily for EBCDIC).

Translate One Character to Another

```
CTRAN(charlen, string, decimal, decvalue, 'outfield')
```

where:

charlen

> Integer
>
> Is the number of characters in the string, or a field that contains the length.

string

> Alphanumeric
>
> Is the character string to be translated enclosed in single quotation marks, or the field that contains the character string.

decimal

Integer

Is the ASCII or EBCDIC decimal value of the character to be translated.

decvalue

Integer

Is the ASCII or EBCDIC decimal value of the character to be used as a substitute for *decimal*.

outfield

Alphanumeric

Is the format of the output value enclosed in single quotation marks.

Translating Spaces to Underscores on an EBCDIC Platform

CTRAN translates the spaces in ADDRESS_LN3 (EBCDIC decimal value 64) to underscores (EBCDIC decimal value 109) and stores the result in ALT_ADDR:

```
COMPUTE ALT_ADDR/A20 = CTRAN(20, ADDRESS_LN3, 64, 109, 'A20');
```

CTRFLD: Centering a Character String

The CTRFLD function centers a character string within a field. The number of leading spaces is equal to or one less than the number of trailing spaces.

CTRFLD is useful for centering the contents of a field and its report column, or a heading that consists only of an embedded field. HEADING CENTER centers each field value including trailing spaces. To center the field value without the trailing spaces, first center the value within the field using CTRFLD.

Limit: Using CTRFLD in a styled report (StyleSheets feature) generally negates the effect of CTRFLD unless the item is also styled as a centered element. Also, if you are using CTRFLD on a platform for which the default font is proportional, either use a non-proportional font, or issue SET STYLE=OFF before running the request.

Center a Character String

```
CTRFLD(string, length, 'outfield')
```

where:

string

Alphanumeric

Is the character string enclosed in single quotation marks, or a field that contains the character string.

length

Integer

Is the number of characters in *string* and *outfield*, or a field that contains the length. This argument must be greater than 0. A length less than 0 can cause unpredictable results.

outfield

Alphanumeric

Is the format of the output value enclosed in single quotation marks.

Centering a Field

CTRFLD centers LAST_NAME and stores the result in CENTER_NAME:

```
COMPUTE CENTER_NAME/A12 = CTRFLD(LAST_NAME, 12, 'A12');
```

EDIT: Extracting or Adding Characters

The EDIT function extracts characters from or adds characters to an alphanumeric string. It can extract a substring from different parts of the parent string, and can also insert characters from a parent string into another substring. For example, it can extract the first two characters and the last two characters of a string to form a single substring.

EDIT works by comparing the characters in a mask to the characters in a source field. When it encounters a nine in the mask, EDIT copies the corresponding character from the source field to the new field. When it encounters a dollar sign in the mask, EDIT ignores the corresponding character in the source field. When it encounters any other character in the mask, EDIT copies that character to the corresponding position in the new field. EDIT does not require an outfield argument because the result is obviously alphanumeric and its size is determined from the mask value.

EDIT can also convert the format of a field. For information on converting a field with EDIT, see EDIT: Converting the Format of a Field.

Extract or Add Characters

```
EDIT(fieldname, 'mask');
```

where:

fieldname

> Alphanumeric

> Is the source field.

> Is the string to extract characters from. It should be at least as long as the mask.

mask

> Alphanumeric

> Is a character string enclosed in single quotation marks. The length of the mask, excluding any characters other than nine and $, determines the length of the output field.

Extracting and Adding a Character to a Field

EDIT extracts the first initial from the FIRST_NAME field and stores the result in FIRST_INIT. EDIT also adds dashes to the EMP_ID field and stores the result in EMPIDEDIT:

```
COMPUTE FIRST_INIT/A1 = EDIT(FIRST_NAME, '9$$$$$$$$$'); AND
COMPUTE EMPIDEDIT/A11 = EDIT(EMP_ID, '999-99-9999');
```

GETTOK: Extracting a Substring (Token)

The GETTOK function divides a character string into substrings, called tokens. A specific character, called a delimiter, occurs in the string and separates the string

into tokens. GETTOK returns the token specified by the token_number. GETTOK ignores leading and trailing blanks in the parent character string.

For example, suppose you want to extract the fourth word from a sentence. Use the space character for a delimiter and four for the token_number. GETTOK divides the sentence into words using this delimiter, then extracts the fourth word. If the string is not divided by the delimiter, use the PARAG function for this purpose.

Extract a Substring (Token)

```
GETTOK(infield, inlen, token_number, 'delim', outlen, 'outfield')
```

where:

infield

> Alphanumeric

> Is the field containing the parent character string.

inlen

> Integer

> Is the length of the parent string in characters. If this argument is less than or equal to 0, the function returns spaces.

token_number

> Integer

> Is the number of the token to extract. If this argument is positive, the tokens are counted from left to right. If this argument is negative, the tokens are counted from right to left. For example, -2 extracts the second token from the right. If this argument is 0, the function returns spaces. Leading and trailing null tokens are ignored.

delim

> Alphanumeric

> Is the delimiter in the parent string enclosed in single quotation marks. If you specify more than one character, only the first character is used.

outlen

> Integer

> Is the maximum size of the token. If this argument is less than or equal to 0, the function returns spaces. If the token is longer than this argument, it is truncated; if it is shorter, it is padded with trailing spaces.

outfield

> Alphanumeric

> Is the format of the output value enclosed in single quotation marks. The delimiter is not included in the token.

Extracting a Token From a Field

GETTOK extracts the last token from ADDRESS_LN3 and stores the result in LAST_TOKEN:

```
COMPUTE LAST_TOKEN/A10 = GETTOK(ADDRESS_LN3, 20, -1, ' ', 10, 'A10');
```

LCWORD: Converting a Character String to Mixed Case

The LCWORD function converts the letters in a character string to mixed case. It converts every alphanumeric character to lowercase except the first letter of each new word and the first letter after a single or double quotation mark. For example, O'CONNOR is converted to O'Connor and JACK'S to Jack'S.

If LCWORD encounters a number in the character string, it treats it as an uppercase character and continues to convert the following alphabetic characters to lowercase. The result of LCWORD is a word with an initial uppercase character followed by lowercase characters.

Convert a Character String to Mixed Case

```
LCWORD(length, string, 'outfield')
```

where:

length

> Integer
>
> Is the length in characters of the character string or field to be converted, or a field that contains the length.

string

> Alphanumeric
>
> Is the character string to be converted enclosed in single quotation marks, or a field containing the character string.

outfield

> Alphanumeric
>
> Is the format of the output value enclosed in single quotation marks. The length must be greater than or equal to the length of *length*.

Converting a Character String to Mixed-Case

LCWORD converts the LAST_NAME field to mixed-case and stores the result in MIXED_CASE:

```
COMPUTE MIXED_CASE/A15 = LCWORD(15, LAST_NAME, 'A15');
```

LJUST: Left-Justifying a Character String

The LJUST function left-justifies a character string within a field. All leading spaces become trailing spaces.

LJUST will not have any visible effect in a report that uses StyleSheets (SET STYLE=ON) unless you center the item.

Left-Justify a Character String

```
LJUST(length, string, 'outfield')
```

where:

length

> Integer
>
> Is the length in characters of *string* and *outfield*, or a field that contains the length.

string

> Alphanumeric
>
> Is the character string to be justified, or a field that contains the string.

outfield

> Alphanumeric
>
> Is the format of the output value enclosed in single quotation marks.

Left-Justifying a Field

LJUST left-justifies the XNAME field and stores the result in YNAME:

```
COMPUTE YNAME/A25 = LJUST(15, XNAME, 'A25');
```

LOCASE: Converting Text to Lowercase

The LOCASE function converts alphanumeric text to lowercase.

Convert Text to Lowercase

```
LOCASE(length, string, 'outfield')
```

where:

length

> Integer
>
> Is the length in characters of *string* and *outfield*, or a field that contains the length. The length must be greater than 0 and the same for both arguments; otherwise, an error occurs.

string

> Alphanumeric
>
> Is the character string to be converted in single quotation marks, or a field that contains the string.

outfield

> Alphanumeric
>
> Is the format of the output value enclosed in single quotation marks. The field name can be the same as *string*.

Converting a Field to Lowercase

LOCASE converts the LAST_NAME field to lowercase and stores the result in LOWER_NAME:

```
COMPUTE LOWER_NAME/A15 = LOCASE(15, LAST_NAME, 'A15');
```

OVRLAY: Overlaying a Character String

The OVRLAY function overlays a base character string with a substring.

Overlay a Character String

```
OVRLAY(string1, stringlen, string2, sublen, position, 'outfield')
```

where:

string1

Alphanumeric

Is the base character string.

stringlen

Integer

Is the length in characters of *string1* and *outfield*, or a field that contains the length. If this argument is less than or equal to 0, unpredictable results occur.

string2

Alphanumeric

Is the substring that will overlay string1.

sublen

Integer

Is the length of string2, or a field that contains the length. If this argument is less than or equal to 0, the function returns spaces.

position

Integer

Is the position in the base string at which the overlay begins. If this argument is less than or equal to 0, the function returns spaces. If this argument is larger than *stringlen*, the function returns the base string.

outfield

Alphanumeric

Is the format of the output value enclosed in single quotation marks. If the overlaid string is longer than the output field, the string is truncated to fit the field.

Replacing Characters in a Character String

OVRLAY replaces the last three characters of EMP_ID with CURR_JOBCODE to create a new security identification code and stores the result in NEW_ID:

```
COMPUTE NEW_ID/A9 = OVRLAY(EMP_ID, 9, CURR_JOBCODE, 3, 7, 'A9');
```

PARAG: Dividing Text Into Smaller Lines

The PARAG function divides a line of text into smaller lines by marking them with a delimiter. It scans a specific number of characters from the beginning of the line and replaces the last space in the group scanned with the delimiter. It then scans the next group of characters in the line, starting from the delimiter, and replaces the last space in this group with a second delimiter. It repeats this process until reaching the end of the line.

Each group of characters marked off by the delimiter becomes a sub-line. The GETTOK function can then place the sub-lines into different fields. If the function does not find any spaces in the group it scans, it replaces the first character after the group with the delimiter. Therefore, make sure that no word of text is longer than the number of characters scanned (the maximum sub-line length).

If the input lines of text are roughly equal in length, you can keep the sub-lines equal by specifying a sub-line length that evenly divides into the length of the text

lines. For example, if the text lines are 120 characters long, divide each of them into two sub-lines of 60 characters or three sub-lines of 40 characters. This technique enables you to print lines of text in paragraph form.

However, if you divide the lines evenly, you may create more sub-lines than you intend. For example, suppose you divide 120-character text lines into two lines of 60 characters maximum, but one line is divided so that the first sub-line is 50 characters and the second is 55. This leaves room for a third sub-line of 15 characters. To correct this, insert a space (using weak concatenation) at the beginning of the extra sub-line, then append this sub-line (using strong concatenation) to the end of the one before it.

Divide Text Into Smaller Lines

```
PARAG(length, string, 'delim', subsize, 'outfield')
```

where:

length

> Integer

> Is the length in characters of *string* and *outfield*, or a field that contains the length.

string

> Alphanumeric

> Is the text enclosed in single quotation marks, or a field that contains the text.

delim

> Alphanumeric

> Is the delimiter enclosed in single quotation marks. Choose a character that does not appear in the text.

subsize

> Integer

> Is the maximum length of each sub-line.

outfield

> Alphanumeric

> Is the format of the output value enclosed in single quotation marks.

Dividing Text Into Smaller Lines

PARAG divides ADDRESS_LN2 into smaller lines of not more than ten characters using a comma as the delimiter. It then stores the result in PARA_ADDR:

```
COMPUTE PARA_ADDR/A20 = PARAG(20, ADDRESS_LN2, ',', 10, 'A20');
```

POSIT: Finding the Beginning of a Substring

The POSIT function finds the starting position of a substring within a larger string. For example, the starting position of the substring DUCT in the string PRODUCTION is four. If the substring is not in the parent string, the function returns the value 0.

Find the Beginning of a Substring

```
POSIT(parent, inlength, substring, sublength, 'outfield')
```

where:

parent

> Alphanumeric
>
> Is the parent character string enclosed in single quotation marks, or a field that contains the parent character string.

inlength

> Integer
>
> Is the length of the parent character string in characters, or a field that contains the length. If this argument is less than or equal to 0, the function returns a 0.

substring

> Alphanumeric
>
> Is the substring whose position you want to find. This can be the substring enclosed in single quotation marks, or the field that contains the string.

sublength

> Integer
>
> Is the length of *substring*. If this argument is less than or equal to 0, or if it is greater than *inlength*, the function returns a 0.

outfield

> Integer
>
> Is the format of the output value enclosed in single quotation marks.

Finding the Position of a Letter

POSIT determines the position of the first capital letter I in LAST_NAME and stores the result in I_IN_NAME:

```
COMPUTE I_IN_NAME/I2 = POSIT(LAST_NAME, 15, 'I', 1, 'I2');
```

REVERSE: Reversing Characters in a Character String

The REVERSE function reverses the characters in a character string.

Reverse Characters in a Character String

```
REVERSE(length, string, 'outfield')
```

where:

length

> Integer
>
> Is the length in characters of *string* and *outfield*, or a field that contains the length.

string

> Alphanumeric

> Is the text enclosed in single quotation marks, or a field that contains the text.

outfield

> Alphanumeric

> Is the format of the output value enclosed in single quotation marks.

Reversing the Characters in a String

REVERSE reverses the characters in PRODCAT and stores the result in REVERSE_NAME:

```
COMPUTE REVERSE_NAME/A15 = REVERSE(15, PRODCAT, 'A15');
```

RJUST: Right-Justifying a Character String

The RJUST function right-justifies a character string. All trailing blacks become leading blanks. This is useful when you display alphanumeric fields containing numbers.

RJUST does not have any visible effect in a report that uses StyleSheets (SET STYLE=ON) unless you center the item. Also, if you use RJUST on a platform on which StyleSheets are turned on by default, issue SET STYLE=OFF before running the request.

Right-Justify a Character String

```
RJUST(length, string, 'outfield')
```

where:

length

> Integer

> Is the length in characters of *string* and *outfield*, or a field that contains the length. The lengths must be the same to avoid justification problems.

string

> Alphanumeric

> Is the character string, or a field that contains the character string enclosed in single quotation marks.

outfield

> Alphanumeric

> Is the format of the output value enclosed in single quotation marks.

Right-Justifying a Field

RJUST right-justifies the LAST_NAME field and stores the result in RIGHT_NAME:

```
COMPUTE RIGHT_NAME/A15 = RJUST(15, LAST_NAME, 'A15');
```

SOUNDEX: Comparing Character Strings Phonetically

The SOUNDEX function searches for a character string phonetically without regard to spelling. It converts character strings to four character codes. The first character must be the first character in the string. The last three characters represent the next three significant sounds in the character string.

To conduct a phonetic search, do the following:

1. Use SOUNDEX to translate data values from the field you are searching for to the phonetic codes.

2. Use SOUNDEX to translate your best guess target string to a phonetic code. Remember that the spelling of your target string need be only approximate; however, the first letter must be correct.

3. Use WHERE or IF criteria to compare the temporary fields created in Step 1 to the temporary field created in Step 2.

Compare Character Strings Phonetically

```
SOUNDEX(inlength, string, 'outfield')
```

where:

inlength

> 2-byte Alphanumeric
>
> Is the length, in characters, of *string*, or a field that contains the length. It can be a number enclosed in single quotation marks, or a field containing the number. The number must be from 01 to 99, expressed with two digits (for example '01'); a number larger than 99 causes the function to return asterisks (*) as output.

string

> Alphanumeric
>
> Is the character string enclosed in single quotation marks, or a field that contains the character string.

outfield

> Alphanumeric
>
> Is the format of the output value enclosed in single quotation marks.

Comparing Character Strings Phonetically

The following request creates three fields:

- PHON_NAME contains the phonetic code of employee last names.
- PHON_COY contains the phonetic code of your guess, MICOY.
- PHON_MATCH compares the guess with the phonetic code.

```
COMPUTE PHON_NAME/A4 = SOUNDEX('15', LAST_NAME, 'A4'); AND
COMPUTE PHON_COY/A4 WITH LAST_NAME = SOUNDEX('15', 'MICOY', 'A4'); AND
COMPUTE PHON_MATCH/A3 = IF PHON_NAME IS PHON_COY THEN 'YES' ELSE 'NO';
```

SPELLNM: Spelling Out a Dollar Amount

The SPELLNM function spells out an alphanumeric string or numeric value containing two decimal places as dollars and cents. For example, the value 32.50 is THIRTY TWO DOLLARS AND FIFTY CENTS.

Spell Out a Dollar Amount

```
SPELLNM(outlength, number, 'outfield')
```

where:

outlength

> Integer

Is the length of *outfield* in characters, or a field that contains the length.

If you know the maximum value of *number*, use the following table to determine the value of *outlength*:

If number is less than...	...outlength should be
$10	37
$100	45
$1,000	59
$10,000	74
$100,000	82
$1,000,000	96

number

> Alphanumeric or Decimal (9.2)
>
> Is the number to be spelled out.

outfield

> Alphanumeric
>
> Is the format of the output value enclosed in single quotation marks.

Spelling Out a Dollar Amount

SPELLNM spells out the values in CURR_SAL and stores the result in AMT_IN_WORDS:

```
COMPUTE AMT_IN_WORDS/A82 = SPELLNM(82, CURR_SAL, 'A82');
```

SUBSTR: Extracting a Substring

The SUBSTR function extracts a substring based on where it begins and its length in the parent string. SUBSTR can vary the position of the substring depending on the values of other fields.

Extract a Substring

```
SUBSTR(inlength, parent, start, end, sublength, 'outfield')
```

where:

inlength

> Integer
>
> Is the length of the parent string in characters, or a field that contains the length.

parent

> Alphanumeric
>
> Is the parent string enclosed in single quotation marks, or the field containing the parent string.

start

> Integer
>
> Is the starting position of the substring in the parent string. If this argument is less than one, the function returns spaces.

end

> Integer

> Is the ending position of the substring. If this argument is less than *start* or greater than *inlength*, the function returns spaces.

sublength

> Integer

> Is the length of the substring (normally end - start + 1). If *sublength* is longer than *end* - *start* +1, the substring is padded with trailing spaces. If it is shorter, the substring is truncated. This value should be the declared length of *outfield*. Only *sublength* characters will be processed.

outfield

> Alphanumeric

> Is the format of the output value enclosed in single quotation marks.

Extracting a String

POSIT determines the position of the first letter I in LAST_NAME and stores the result in I_IN_NAME. SUBSTR then extracts three characters beginning with the letter I from LAST_NAME, and stores the results in I_SUBSTR.

```
COMPUTE I_IN_NAME/I2 = POSIT(LAST_NAME, 15, 'I', 1, 'I2'); AND
COMPUTE I_SUBSTR/A3 = SUBSTR(15, LAST_NAME, I_IN_NAME, I_IN_NAME+2, 3, 'A3');
```

UPCASE: Converting Text to Uppercase

The UPCASE function converts a character string to uppercase. It is useful for sorting on a field that contains both mixed-case and uppercase values. Sorting on a mixed-case field produces incorrect results because the sorting sequence in EBCDIC always places lowercase letters before uppercase letters, while the ASCII sorting sequence always places uppercase letters before lowercase. To obtain correct results, define a new field with all of the values in uppercase, and sort on that.

Convert Text to Uppercase

```
UPCASE(length, input, 'outfield')
```

where:

length

> Integer

> Is the length in characters of *input* and *outfield*.

input

> Alphanumeric

> Is the character string enclosed in single quotation marks, or the field containing the character string.

outfield

> Alphanumeric

> Is the format of the output value enclosed in single quotation marks.

Converting a Mixed-Case Field to Uppercase

UPCASE converts the LAST_NAME_MIXED field to uppercase and stores the result in LAST_NAME_UPPER:

```
COMPUTE LAST_NAME_UPPER/A15 = UPCASE(15, LAST_NAME_MIXED, 'A15') ;
```

Data Source and Decoding Functions

Data source and decoding functions retrieve data source values and assign values based on the value of an input field.

DECODE: Decoding Values

The DECODE function assigns values based on the coded value of an input field. DECODE is useful for giving a more meaningful value to a coded value in a field. For example, the field GENDER may have the code F for female employees and M for male employees for efficient storage (for example, one character instead of six for female). DECODE expands (decodes) these values to ensure correct interpretation on a report.

You can use DECODE by supplying values directly in the function or by reading values from a separate file.

Supply Values in the Function

```
DECODE fieldname(code1 result1 code2 result2...[ELSE default]);
```

where:

fieldname

> Alphanumeric or Numeric

> Is the name of the input field.

code

> Alphanumeric or Numeric

> Is the coded value for which DECODE searches. If the value has embedded blanks, commas, or other special characters, enclose it in single quotation marks. When DECODE finds the specified value, it assigns the corresponding result.

result

> Alphanumeric or Numeric

> Is the value assigned to a code. If the value has embedded blanks or commas or contains a negative number, enclose it in single quotation marks.

default

> Alphanumeric or Numeric

> Is the value assigned if the code is not found. If you omit a default value, DECODE assigns a blank or zero to non-matching codes.

You can use up to 40 lines to define the code and result pairs for any given DECODE function, or 39 lines if you also use an ELSE phrase. Use either a comma or blank to separate the code from the result, or one pair from another.

Supplying Values in the Function

EDIT extracts the first character of the CURR_JOBCODE field, then DECODE returns either ADMINISTRATIVE or DATA PROCESSING depending on the value extracted.

```
COMPUTE DEPX_CODE/A1 = EDIT(CURR_JOBCODE, '9$$'); AND
COMPUTE JOB_CATEGORY/A15 = DECODE DEPX_CODE(A 'ADMINISTRATIVE'
B 'DATA PROCESSING') ;
```

Read Values From a File

```
DECODE fieldname(ddname [ELSE default]);
```

where:

fieldname

>Alphanumeric or Numeric

>Is the name of the input field.

ddname

>Alphanumeric

>Is a logical name or a shorthand name that points to the physical file containing the decoded values.

default

>Alphanumeric or Numeric

>Is the value assigned if the code is not found. If you omit a default, DECODE assigns a blank or zero to non-matching codes.

Guidelines for Reading Values From a File

- Each record in the file is expected to contain pairs of elements separated by a comma or blank.
- If each record in the file consists of only one element, this element is interpreted as the code, and the result becomes either a blank or zero, as needed.

  This makes it possible to use the file to hold screening literals referenced in the screening condition

  ```
  IF field IS (filename)
  ```

  and as a file of literals for an IF criteria specified in a computational expression. For example:

  ```
  TAKE = DECODE SELECT (filename ELSE 1);
  VALUE = IF TAKE IS 0 THEN... ELSE...;
  ```

  TAKE is 0 for SELECT values found in the literal file and 1 in all other cases. The VALUE computation is carried out as if the expression had been:

  ```
  IF SELECT (filename) THEN... ELSE...;
  ```

- The file can contain up to 32,767 characters in the file.
- Leading and trailing blanks are ignored.
- The remainder of each record is ignored and can be used for comments or other data. This convention applies in all cases, except when the file name is HOLD. In that case, the file is presumed to have been created by the HOLD command, which writes fields in the internal format, and the DECODE pairs are interpreted accordingly. In this case, extraneous data in the record is ignored.

Reading Values From a File

DECODE assigns the value 0 to an employee whose EMP_ID appears in the HOLD file and 1 when EMP_ID does not appear in the file.

```
COMPUTE NOT_IN_LIST/I1 = DECODE EMP_ID(HOLD ELSE 1);
```

LAST: Retrieving the Preceding Value

The LAST function retrieves the preceding value for a field.

The effect of LAST depends on whether it appears in a DEFINE or COMPUTE command:

- In a DEFINE command, the LAST value applies to the previous record retrieved from the data source before sorting takes place.
- In a COMPUTE command, the LAST value applies to the record in the previous line of the internal matrix.

Retrieve the Preceding Value

```
LAST fieldname
```

where:

fieldname

> Alphanumeric or Numeric
>
> Is the field name.

Retrieving the Preceding Value

LAST retrieves the previous value of the DEPARTMENT field to determine whether to restart the running total of salaries by department. If the previous value equals the current value, CURR_SAL is added to RUN_TOT to generate a running total of salaries within each department.

```
COMPUTE RUN_TOT/D12.2M = IF DEPARTMENT EQ LAST DEPARTMENT THEN
                 (RUN_TOT + CURR_SAL) ELSE CURR_SAL ;
```

Date and Time Functions

Date and time functions manipulate date and time values.

When using standard date and time functions, you need to understand the settings that alter the behavior of these functions, as well as the acceptable formats and how to supply values in these formats.

You can affect the behavior of date and time functions by defining which days of the week are work days and which are not. Then, when you use a date function involving work days, dates that are not work days are ignored.

AYM: Adding or Subtracting Months to or From Dates

The AYM function adds months to or subtracts months from a date in year-month format. You can convert a date to this format using the CHGDAT or EDIT function.

Add or Subtract Months to or From a Date

```
AYM(indate, months, 'outfield')
```

where:

indate

> Integer (I4, I4YM, I6, or I6YYM)
>
> Is the original date in year-month format, the name of a field that contains the date, or an expression that returns the date. If the date is not valid, the function returns a 0.

months

> Integer
>
> Is the number of months you are adding to or subtracting from the date. To subtract months, use a negative number.

outfield

> Integer (I4YM or I6YYM)
>
> Is the format of the output value enclosed in single quotation marks.
>
> **Tip:** If the input date is in integer year-month-day format (I6YMD or I8YYMD), divide the date by 100 to convert to year-month format and set the result to an integer. This drops the day portion of the date, which is now after the decimal point.

Adding Months to a Date

The COMPUTE command converts the dates in HIRE_DATE from year-month-day to year-month format and stores the result in HIRE_MONTH. AYM then adds six months to HIRE_MONTH and stores the result in AFTER6MONTHS.

```
COMPUTE HIRE_MONTH/I4YM = HIRE_DATE/100; AND
COMPUTE AFTER6MONTHS/I4YM = AYM(HIRE_MONTH, 6, 'I4YM');
```

AYMD: Adding or Subtracting Days to or From a Date

The AYMD function adds days to or subtracts days from a date in year-month-day format. You can convert a date to this format using the CHGDAT or EDIT function.

If the addition or subtraction of days crosses forward or backward into another century, the century digits of the output year are adjusted.

Add or Subtract Days to or From a Date

```
AYMD(indate, days, 'outfield')
```

where:

indate

> Integer (I6, I6YMD, I8, I8YYMD)
>
> If the date is not valid, the function returns a 0.

days

> Integer

Is the number of days you are adding to or subtracting from *indate*. To subtract days, use a negative number.

outfield

Integer (I6, I6YMD, I8, or I8YYMD)

Is the format of the output value enclosed in single quotation marks. If *indate* is a field, *outfield* must have the same format.

Adding Days to a Date

AYMD adds 35 days to each value in the HIRE_DATE field, and stores the result in AFTER35DAYS:

```
COMPUTE AFTER35DAYS/I6YMD = AYMD(HIRE_DATE, 35, 'I6YMD');
```

CHGDAT: Changing How a Date String Displays

The CHGDAT function rearranges the year, month, and day portions of an input character string representing a date. It may also convert the input string from long to short or short to long date representation. Long representation contains all three date components: year, month, and day; short representation omits one or two of the date components, such as year, month, or day. The input and output date strings are described by display options that specify both the order of date components (year, month, day) in the date string and whether two or four digits are used for the year (for example, 97 or 1997). CHGDAT reads an input date character string and creates an output date character string that represents the same date in a different way.

Note: CHGDAT requires a date character string as input, not a date itself. Convert the input to a date character string (using the EDIT or DATECVT functions, for example) before applying CHGDAT.

The order of date components in the date character string is described by display options comprised of the following characters in your chosen order:

Character	Description
D	Day of the month (01 through 31).
M	Month of the year (01 through 12).
Y[Y]	Year. Y indicates a two-digit year (such as 94); YY indicates a four-digit year (such as 1994).

To spell out the month rather than use a number in the resulting string, append one of the following characters to the display options for the resulting string:

Character	Description
T	Displays the month as a three-letter abbreviation.
X	Displays the full name of the month.

Display options can consist of up to five display characters. Characters other than those display options are ignored.

For example: The display options 'DMYY' specify that the date string starts with a two digit day, then two digit month, then four digit year.

Note: Display options are *not* date formats.

Short to Long Conversion

If you are converting a date from short to long representation (for example, from year-month to year-month-day), the function supplies the portion of the date missing in the short representation, as shown in the following table:

Portion of Date Missing	Portion Supplied by Function
Day (for example, from YM to YMD)	Last day of the month.
Month (for example, from Y to YM)	Last month of the year (December).
Year (for example, from MD to YMD)	The year 99.
Converting year from two-digit to four-digit (for example, from YMD to YYMD)	If DATEFNS=ON, the century will be determined by the 100-year window defined by DEFCENT and YRTHRESH. If DATEFNS=OFF, the year 19*xx* is supplied, where *xx* is the last two digits in the year.

Change the Date Display String

```
CHGDAT('in_display_options', 'out_display_options', date_string, 'outfield')
```

where:

in_display_options

> Alphanumeric (A1 to A5)

> Is a series of up to five display options that describe the layout of *date_string*. These options can be stored in an alphanumeric field or supplied as a literal enclosed in single quotation marks.

out_display_options

> Alphanumeric (A1 to A5)

> Is a series of up to five display options that describe the layout of the converted date string. These options can be stored in an alphanumeric field or supplied as a literal enclosed in single quotation marks.

date_string

> Alphanumeric (A2 to A8)

> Is the input date character string with date components in the order specified by *in_display_options*.

> Note that if the original date is in numeric format, you must convert it to a date character string. If *date_string* does not correctly represent the date (the date is invalid), the function returns blank spaces.

outfield

> Alphanumeric

> Is the format of the output value enclosed in single quotation marks.

A*xx*, where *xx* is a number of characters large enough to fit the date string specified by *out_display_options*. A17 is long enough to fit the longest date string.

Usage Notes for CHGDAT

Since CHGDAT uses a date string (as opposed to a date) and returns a date string with up to 17 characters, use the EDIT or DATECVT functions or any other means to convert the date to or from a date character string.

Converting the Date Display From YMD to MDYYX

The EDIT function changes HIRE_DATE from numeric to alphanumeric format. CHGDAT then converts each value in ALPHA_HIRE from displaying the components as YMD to MDYYX and stores the result in HIRE_MDY, which has the format A17. The option X in the output value displays the full name of the month.

```
COMPUTE ALPHA_HIRE/A17 = EDIT(HIRE_DATE); AND
COMPUTE HIRE_MDY/A17 = CHGDAT('YMD', 'MDYYX', ALPHA_HIRE, 'A17');
```

DA Functions: Converting a Date to an Integer

The DA functions convert a date to the number of days between December 31, 1899 and that date. By converting a date to the number of days, you can add and subtract dates and calculate the intervals between them.

There are six DA functions; each one accepts a date in a different format.

Convert a Date to an Integer

```
function(indate, 'outfield')
```

where:

function

> Is one of the following:
>
> DADMY converts a date in day-month-year format.
>
> DADYM converts a date in day-year-month format.
>
> DAMDY converts a date in month-day-year format.
>
> DAMYD converts a date in month-year-day format.
>
> DAYDM converts a date in year-day-month format.
>
> DAYMD converts a date in year-month-day format.

indate

> Integer or Packed
>
> I or P format with date display options.
>
> Is the date to be converted, or the name of a field that contains the date. The date is truncated to an integer before conversion.
>
> To specify the year, enter only the last two digits; the function assumes the century component. If the date is invalid, the function returns a 0.

outfield

> Integer

Is the format of the output value enclosed in single quotation marks. The
format of the date returned depends on the function.

Converting Dates and Calculating the Difference Between Them

DAYMD converts the DAT_INC and HIRE_DATE fields to the number of days
since December 31, 1899, and the smaller number is then subtracted from the
larger number:

```
COMPUTE DAYS_HIRED/I8 = DAYMD(DAT_INC, 'I8') - DAYMD(HIRE_DATE, 'I8');
```

DATEADD: Adding or Subtracting a Date Unit to or From a Date

The DATEADD function adds a unit to or subtracts a unit from a date format. A
unit is one of the following:

- **Year.**
- **Month.** If the calculation using the month unit creates an invalid date,
  DATEADD corrects it to the last day of the month. For example, adding one
  month to October 31 yields November 30, not November 31 since November has
  30 days.
- **Day.**
- **Weekday.** When using the weekday unit, DATEADD does not count Saturday or
  Sunday. For example, if you add one day to Friday, the result is Monday.
- **Business day.** When using the business day unit, DATEADD uses the BUSDAYS
  parameter setting and holiday file to determine which days are working days
  and disregards the rest. If Monday is not a working day, then one business day
  past Sunday is Tuesday.

You add or subtract non day-based dates (for example, YM or YQ) directly without
using DATEADD.

DATEADD works only with full component dates.

Add or Subtract a Date Unit to or From a Date

```
DATEADD(date, 'unit', #units)
```

where:

date

Date

Is a full component date.

unit

Alphanumeric

Is one of the following enclosed in single quotation marks:

Y indicates a year unit.

M indicates a month unit.

D indicates a day unit.

WD indicates a weekday unit.

BD indicates a business day unit.

#units

Integer

Is the number of date units added to or subtracted from *date*. If this number is not a whole unit, it is rounded down to the next largest integer.

Adding Weekdays to a Date

DATEADD adds three weekdays to NEW_DATE. In some cases, it adds more than three days because HIRE_DATE_PLUS_THREE would otherwise be on a weekend.

```
COMPUTE NEW_DATE/YYMD = HIRE_DATE; AND
COMPUTE HIRE_DATE_PLUS_THREE/YYMD = DATEADD(NEW_DATE, 'WD', 3);
```

DATECVT: Converting the Format of a Date

The DATECVT function converts the format of a date in an application without requiring an intermediate calculation. If you supply an invalid format, DATECVT returns a zero or a blank.

DATECVT turns off optimization and compilation.

Note: You can use simple assignment instead of calling this function.

Convert a Date Format

```
DATECVT(date, 'infmt', 'outfmt')
```

where:

date

Date

Is the date to be converted. If you supply an invalid date, DATECVT returns zero. When the conversion is performed, a legacy date obeys any DEFCENT and YRTHRESH parameter settings supplied for that field.

infmt

Alphanumeric

Is the format of the date enclosed in single quotation marks. It is one of the following:

- A non-legacy date format (for example, YYMD, YQ, M, DMY, JUL).
- A legacy date format (for example, I6YMD or A8MDYY).
- A non-date format (such as I8 or A6). A non-date format in *infmt* functions as an offset from the base date of a YYMD field (12/31/1900).

outfmt

Alphanumeric

Is the output format enclosed in single quotation marks. It is one of the following:

- A non-legacy date format (for example, YYMD, YQ, M, DMY, JUL).
- A legacy date format (for example, I6YMD or A8MDYY).
- A non-date format (such as I8 or A6). A non-date format in *infmt* functions as an offset from the base date of a YYMD field (12/31/1900).

Converting a YYMD Date to DMY

DATECVT converts 19991231 to 311299 and stores the result in CONV_FIELD:

```
COMPUTE CONV_FIELD/DMY = DATECVT(19991231, 'I8YYMD', 'DMY');
```

or

```
COMPUTE CONV_FIELD/DMY = DATECVT('19991231', 'A8YYMD', 'DMY');
```

Converting a Legacy Date to Date Format

DATECVT converts HIRE_DATE from I6YMD legacy date format to YYMD date format:

```
COMPUTE NEW_HIRE_DATE/YYMD = DATECVT(HIRE_DATE, 'I6YMD', 'YYMD');
```

DATEDIF: Finding the Difference Between Two Dates

The DATEDIF function returns the difference between two dates in units. A unit is one of the following:

- **Year.** Using the year unit with DATEDIF yields the inverse of DATEADD. If subtracting one year from date X creates date Y, then the count of years between X and Y is one. Subtracting one year from February 29 produces the date February 28.

- **Month.** Using the month unit with DATEDIF yields the inverse of DATEADD. If subtracting one month from date X creates date Y, then the count of months between X and Y is one. If the to-date is the end-of-month, then the month difference may be rounded up (in absolute terms) to guarantee the inverse rule.

  If one or both of the input dates is the end of the month, DATEDIF takes this into account. This means that the difference between January 31 and April 30 is three months, not two months.

- **Day.**

- **Weekday.** With the weekday unit, DATEDIF does not count Saturday or Sunday when calculating days. This means that the difference between Friday and Monday is one day.

- **Business day.** With the business day unit, DATEDIF uses the BUSDAYS parameter setting and holiday file to determine which days are working days and disregards the rest. This means that if Monday is not a working day, the difference between Friday and Tuesday is one day.

DATEDIF returns a whole number. If the difference between two dates is not a whole number, DATEDIF truncates the value to the next largest integer. For example, the number of years between March 2, 2001, and March 1, 2002, is zero. If the end date is before the start date, DATEDIF returns a negative number.

You can find the difference between non-day based dates (for example YM or YQ) directly without using DATEDIF.

Find the Difference Between Two Dates

```
DATEDIF(from_date, to_date, 'unit')
```

where:

from_date

   Date

Is the start date from which to calculate the difference. Is a full component date.

to_date

> Date
>
> Is the end date from which to calculate the difference.

unit

> Alphanumeric
>
> Is one of the following enclosed in single quotation marks:
>
> Y indicates a year unit.
>
> M indicates a month unit.
>
> D indicates a day unit.
>
> WD indicates a weekday unit.
>
> BD indicates a business day unit.

Finding the Number of Weekdays Between Two Dates

DATECVT converts the legacy dates in HIRE_DATE and DAT_INC to the date format YYMD. DATEDIF then uses those date formats to determine the number of weekdays between NEW_HIRE_DATE and NEW_DAT_INC:

```
COMPUTE NEW_HIRE_DATE/YYMD = DATECVT(HIRE_DATE, 'I6YMD', 'YYMD'); AND
COMPUTE NEW_DAT_INC/YYMD = DATECVT(DAT_INC, 'I6YMD', 'YYMD'); AND
COMPUTE WDAYS_HIRED/I8 = DATEDIF(NEW_HIRE_DATE, NEW_DAT_INC, 'WD');
```

DATEMOV: Moving a Date to a Significant Point

The DATEMOV function moves a date to a significant point on the calendar.

DATEMOV works only with full component dates.

Move a Date to a Significant Point

```
DATEMOV(date, 'move-point')
```

where:

date

> Date
>
> Is a full component date. Is the date to be moved.

move-point

> Alphanumeric
>
> Is the significant point the date is moved to enclosed in single quotation marks. An invalid point results in a return code of zero. Valid values are:
>
> EOM is the end of month.
>
> BOM is the beginning of month.
>
> EOQ is the end of quarter.
>
> BOQ is the beginning of quarter.
>
> EOY is the end of year.

BOY is the beginning of year.

EOW is the end of week.

BOW is the beginning of week.

NWD is the next weekday.

NBD is the next business day.

PWD is the prior weekday.

PBD is the prior business day.

WD- is a weekday or earlier.

BD- is a business day or earlier.

WD+ is a weekday or later.

BD+ is a business day or later.

A business day calculation is affected by the BUSDAYS and HDAY parameter settings.

Determining the End of the Week

DATEMOV determines the end of the week for each date in NEW_DATE and stores the result in EOW:

```
COMPUTE NEW_DATE/YYMDWT = DATECVT(HIRE_DATE, 'I6YMD', 'YYMDWT'); AND
COMPUTE EOW/YYMDWT = DATEMOV(NEW_DATE, 'EOW');
```

DATETRAN: Formatting Dates in International Formats

The DATETRAN function formats dates in international formats.

Format Dates in International Formats

```
DATETRAN (indate, '(intype)', '([formatops])', 'lang', outlen, 'outfield')
```

where:

indate

Is the input date to be formatted. Note that the date format cannot be an alphanumeric or numeric format with date display options.

intype

Is one of the following character strings indicating the input date components and the order in which you want them to display, enclosed in single quotation marks and parentheses.

These are the single component input types:

Single Component Input Type	Description
'(W)'	Day of week component only (original format must have only W component).
'(M)'	Month component only (original format must have only M component).

These are the two-component input types:

Two-Component Input Type	Description
'(YYM)'	Four-digit year followed by month.
'(YM)'	Two-digit year followed by month.
'(MYY)'	Month component followed by four-digit year.
'(MY)'	Month component followed by two-digit year.

These are the three-component input types:

Three- Component Input Type	Description
'(YYMD)'	Four-digit year followed by month followed by day.
'(YMD)'	Two-digit year followed by month followed by day.
'(DMYY)'	Day component followed by month followed by four-digit year.
'(DMY)'	Day component followed by month followed by two-digit year.
'(MDYY)'	Month component followed by day followed by four-digit year.
'(MDY)'	Month component followed by day followed by two-digit year.
'(MD)'	Month component followed by day (derived from three-component date by ignoring year component).
'(DM)'	Day component followed by month (derived from three-component date by ignoring year component).

formatops

Is a string of zeros or more formatting options enclosed in parentheses and single quotation marks. The parentheses and quotation marks are required even if you do not specify formatting options. Formatting options are as follows:

- Options for suppressing initial zeros in month or day numbers.
- Options for translating month or day components to full or abbreviated uppercase or default case (mixed case or lowercase depending on the language) names.
- Date delimiter options and options for punctuating a date with commas.

Valid options for suppressing initial zeros in month or day numbers are:

Format Option	Description
m	Zero-suppresses months (displays numeric months before October as 1 through 9 rather than 01 through 09).
d	Displays days before the tenth of the month as 1 through 9 rather than 01 through 09.

Format Option	Description
dp	Displays days before the tenth of the month as 1 through 9 rather than 01 through 09 with a period after the number.
do	Displays days before the tenth of the month as 1 through 9. For English (langcode EN) only, displays an ordinal suffix (st, nd, rd, or th) after the number.

Valid month and day name translation options are:

Format Option	Description
T	Displays month as an abbreviated name with no punctuation, all uppercase.
TR	Displays month as a full name, all uppercase.
Tp	Displays month as an abbreviated name followed by a period, all uppercase.
t	Displays month as an abbreviated name with no punctuation. The name is all lowercase or initial uppercase, depending on language code.
tr	Displays month as a full name. The name is all lowercase or initial uppercase, depending on language code.
tp	Displays month as an abbreviated name followed by a period. The name displays in the default case of the specified language (for example, all lowercase for French and Spanish, initial uppercase for English and German).
W	Includes an abbreviated day of the week name at the start of the displayed date, all uppercase with no punctuation.
WR	Includes a full day of the week name at the start of the displayed date, all uppercase.
Wp	Includes an abbreviated day of the week name at the start of the displayed date, all uppercase, followed by a period.
w	Includes an abbreviated day of the week name at the start of the displayed date with no punctuation. The name displays in the default case of the specified language (for example, all lowercase for French and Spanish, initial uppercase for English and German).
wr	Includes a full day of the week name at the start of the displayed date. The name displays in the default case of the specified language (for example, all lowercase for French and Spanish, initial uppercase for English and German).

Format Option	Description
wp	Includes an abbreviated day of the week name at the start of the displayed date followed by a period. The name displays in the default case of the specified language (for example, all lowercase for French and Spanish, initial uppercase for English and German).
X	Includes an abbreviated day of the week name at the end of the displayed date, all uppercase with no punctuation.
XR	Includes a full day of the week name at the end of the displayed date, all uppercase.
Xp	Includes an abbreviated day of the week name at the end of the displayed date, all uppercase, followed by a period.
x	Includes an abbreviated day of the week name at the end of the displayed date with no punctuation. The name displays in the default case of the specified language (for example, all lowercase for French and Spanish, initial uppercase for English and German).
xr	Includes a full day of the week name at the end of the displayed date. The name displays in the default case of the specified language (for example, all lowercase for French and Spanish, initial uppercase for English and German).
xp	Includes an abbreviated day of the week name at the end of the displayed date followed by a period. The name displays in the default case of the specified language (for example, all lowercase for French and Spanish, initial uppercase for English and German).

Valid date delimiter options are:

Format Option	Description
B	Uses a blank as the component delimiter. This is the default if the month or day of week is translated or if comma is used.
.	Uses a period as the component delimiter.
-	Uses a minus sign as the component delimiter. This is the default when the conditions for a blank default delimiter are not satisfied.
/	Uses a slash as the component delimiter.
|	Omits component delimiters.
K	Uses appropriate Asian characters as component delimiters.

Format Option	Description
c	Places a comma after the month name (following T, Tp, TR, t, tp, or tr).
	Places a comma and blank after the day name (following W, Wp, WR, w, wp, or wr).
	Places a comma and blank before the day name (following X, XR, x, or xr).
e	Displays the Spanish or Portuguese word de or DE between the day and month and between the month and year. The case of the word de is determined by the case of the month name. If the month is displayed in uppercase, DE is displayed; otherwise de is displayed. Useful for formats DMY, DMYY, MY, and MYY.
D	Inserts a comma after the day number and before the general delimiter character specified.
Y	Inserts a comma after the year and before the general delimiter character specified.

lang

Is the two-character standard ISO code for the language into which the date should be translated, enclosed in single quotation marks. Valid language codes are:

'AR' Arabic

'CS' Czech

'DA' Danish

'DE' German

'DU' Dutch

'EN' English

'ES' Spanish

'FI' Finnish

'FR' French

'EL' Greek

'IW' Hebrew

'IT' Italian

'JA' Japanese

'KO' Korean

'LT' Lithuanian

'NO' Norwegian

'PO' Polish

'PT' Portuguese

'RU' Russian

'SV' Swedish

'TH' Thai

'TR' Turkish

'TW' Chinese (Traditional)

'ZH' Chinese (Simplified)

outlen

Numeric

Is the length of the output field in bytes. If the length is insufficient, an all blank result is returned. If the length is greater than required, the field is padded with blanks on the right.

outfield

Alphanumeric

Is the format of the output value enclosed in single quotation marks.

Usage Notes for the DATETRAN Function

- The type A output field may contain variable length information, since the lengths of month names and day names can vary. Also month and day numbers may be either one or two bytes long if a zero-suppression option is chosen. Unused bytes are filled with blanks.

- All invalid and inconsistent inputs result in all blank output strings. Missing data also results in blank output.

- The base dates (1900-12-31 and 1900-12 or 1901-01) are treated as though the DATEDISPLAY setting were ON (that is, not automatically shown as blanks). To suppress the printing of base dates, which have an internal integer value of 0, test for 0 before calling DATETRAN. For example:

```
RESULT/A40 = IF DATE EQ 0 THEN ' ' ELSE
              DATETRAN (DATE, '(YYMD)', '(.t)', 'FR', 40, 'A40');
```

- Valid translated date components are contained in files named DTLNG*lng* where *lng* is a three-character code that specifies the language. These files must be accessible for each language into which you want to translate dates.

Using the DATETRAN Function

DATETRAN prints the day of the week in the default case for French:

```
COMPUTE OUT/A8=DATETRAN(DATEW, '(W)', '(wr)', 'FR', 8 , 'A8') ;
```

DMY, MDY, YMD: Calculating the Difference Between Two Dates

The DMY, MDY, and YMD functions calculate the difference between two dates in integer, alphanumeric, or packed format.

Calculate the Difference Between Two Dates

```
function(begin, end)
```

where:

function

Is one of the following:

DMY calculates the difference between two dates in day-month-year format.

MDY calculates the difference between two dates in month-day-year format.

YMD calculates the difference between two dates in year-month-day format.

begin

> Integer, Packed, or Alphanumeric
>
> I, P, or A format with date display options.
>
> Is the beginning date, or the name of a field that contains the date.

end

> Integer, Packed, or Alphanumeric
>
> I, P, or A format with date display options.
>
> Is the end date, or the name of a field that contains the date.

Calculating the Number of Days Between Two Dates

YMD calculates the number of days between the dates in HIRE_DATE and DAT_INC:

```
COMPUTE DIFF/I4 = YMD(HIRE_DATE, FST.DAT_INC);
```

DOWK and DOWKL: Finding the Day of the Week

The DOWK and DOWKL functions find the day of the week that corresponds to a date. DOWK returns the day as a three letter abbreviation; DOWKL displays the full name of the day.

Find the Day of the Week

```
{DOWK|DOWKL}(indate, 'outfield')
```

where:

indate

> Integer (I6YMD or I8 YMD)
>
> Is the input date in year-month-day format. If the date is not valid, the function returns spaces. If the date specifies a two-digit year and DEFCENT and YRTHRESH values have not been set, the function assumes the 20th century.

outfield

> DOWK: Alphanumeric
>
> DOWKL: Alphanumeric
>
> Is the format of the output value enclosed in single quotation marks.

Finding the Day of the Week

DOWK determines the day of the week that corresponds to the value in the HIRE_DATE field and stores the result in DATED:

```
COMPUTE DATED/A3 = DOWK(HIRE_DATE, 'A3');
```

DT Functions: Converting an Integer to a Date

The DT functions convert an integer representing the number of days elapsed since December 31, 1899 to the corresponding date. They are useful when you are

performing arithmetic on a date converted to the number of days. The DT functions convert the result back to a date.

There are six DT functions; each one converts a number into a date of a different format.

Note: When USERFNS is set to LOCAL, DT functions only display a six-digit date.

Convert an Integer to a Date
```
function(number, 'outfield')
```

where:

function

> Is one of the following:
>
> DTDMY converts a number to a day-month-year date.
>
> DTDYM converts a number to a day-year-month date.
>
> DTMDY converts a number to a month-day-year date.
>
> DTMYD converts a number to a month-year-day date.
>
> DTYDM converts a number to a year-day-month date.
>
> DTYMD converts a number to a year-month-day date.

number

> Integer
>
> Is the number of days since December 31, 1899. The number is truncated to an integer.

outfield

> Integer
>
> I6*xxx*, where *xxx* corresponds to the function DT*xxx* in the above list.
>
> Is the format of the output value enclosed in single quotation marks. The output format depends on the function being used.

Converting an Integer to a Date

DTMDY converts the NEWF field (which was converted to the number of days by DAYMD) to the corresponding date and stores the result in NEW_HIRE_DATE:
```
COMPUTE NEWF/I8 WITH EMP_ID = DAYMD(HIRE_DATE, NEWF); AND
COMPUTE NEW_HIRE_DATE/I8MDYY WITH EMP_ID = DTMDY(NEWF, NEW_HIRE_DATE);
```

GREGDT: Converting From Julian to Gregorian Format

The GREGDT function converts a date in Julian format to Gregorian format (year-month-day).

A date in Julian format is a five- or seven-digit number. The first two or four digits are the year; the last three digits are the number of the day, counting from January 1. For example, January 1, 1999 in Julian format is either 99001 or 1999001.

DATEFNS Settings for GREGDT

GREGDT converts a Julian date to either YMD or YYMD format using the DEFCENT and YRTHRESH parameter settings to determine the century, if required. GREGDT returns a date as follows:

DATEFNS Setting	I6 or I7 Format	I8 Format or Greater
ON	YMD	YYMD
OFF	YMD	YMD

Convert From Julian to Gregorian Format

```
GREGDT(indate, 'outfield')
```

where:

indate

> Integer (I5 or I7)

> Is the Julian date, which is truncated to an integer before conversion. Each value must be a five- or seven-digit number after truncation. If the date is invalid, the function returns a 0.

outfield

> Integer (I6, I8, I6YMD, or I8YYMD)

> Is the format of the output value enclosed in single quotation marks.

Converting From Julian to Gregorian Format

GREGDT converts the JULIAN field to YYMD (Gregorian) format.

```
COMPUTE GREG_DATE/I8 = GREGDT(JULIAN, 'I8');
```

HADD: Incrementing a Date-Time Value

The HADD function increments a date-time value by a given number of units.

Increment a Date-Time Value

```
HADD(value, 'component', increment, length, 'outfield')
```

where:

value

> Date-time

> Is the date-time value to be incremented, the name of a date-time field that contains the value, or an expression that returns the value.

component

> Alphanumeric

> Is the name of the component to be incremented enclosed in single quotation marks.

> **Note:** WEEKDAY is not a valid component for HADD.

increment

> Integer

Is the number of units by which to increment the component, the name of a numeric field that contains the value, or an expression that returns the value.

length

Integer

Is the length of the returned date-time value. Valid values are:

8 indicates a time value that includes milliseconds.

10 indicates a time value that includes microseconds.

outfield

Date-time

Is the format of the output value enclosed in single quotation marks.

Incrementing the Month Component of a Date-Time Field

HADD adds two months to each value in TRANSDATE and stores the result in ADD_MONTH. If necessary, the day is adjusted so that it is valid for the resulting month.

```
COMPUTE ADD_MONTH/HYYMDS = HADD(TRANSDATE, 'MONTH', 2, 8, 'HYYMDS');
```

HCNVRT: Converting a Date-Time Value to Alphanumeric Format

The HCNVRT function converts a date-time value to alphanumeric format for use with operators such as EDIT, CONTAINS, and LIKE.

Convert a Date-Time Value to Alphanumeric Format

```
HCNVRT(value, '(fmt)', length, 'outfield')
```

where:

value

Date-time

Is the date-time value to be converted, the name of a date-time field that contains the value, or an expression that returns the value.

fmt

Alphanumeric

Is the format of the date-time field enclosed in single quotation marks and parentheses.

length

Integer

Is the length of the alphanumeric field that is returned. You can supply the actual value, the name of a numeric field that contains the value, or an expression that returns the value. If *length* is smaller than the number of characters needed to display the alphanumeric field, the function returns a blank.

outfield

Alphanumeric

Is the format of the output value enclosed in single quotation marks.

Converting a Date-Time Field to Alphanumeric Format

HCNVRT converts the TRANSDATE field to alphanumeric format. The first function does not include date-time display options for the field; the second function does for readability. It also specifies the display of seconds in the input field.

```
COMPUTE ALPHA_DATE_TIME1/A20 = HCNVRT(TRANSDATE, '(H17)', 17, 'A20'); AND
COMPUTE ALPHA_DATE_TIME2/A20 = HCNVRT(TRANSDATE, '(HYYMDS)', 20, 'A20');
```

HDATE: Converting the Date Portion of a Date-Time Value to a Date Format

The HDATE function converts the date portion of a date-time value to the date format YYMD. You can then convert the result to other date formats.

Convert the Date Portion of a Date-Time Value to a Date Format

```
HDATE(value, 'YYMD')
```

where:

value

> Date-time
>
> Is the date-time value to be converted, the name of a date-time field that contains the value, or an expression that returns the value.

YYMD

> Date
>
> Is the output format. The value must be YYMD. YYMD is a constant value and cannot be changed in this syntax, although you can change the format in subsequent DEFINEs or COMPUTEs.

Converting the Date Portion of a Date-Time Field to a Date Format

HDATE converts the date portion of the TRANSDATE field to the date format YYMD:

```
COMPUTE TRANSDATE_DATE/YYMD = HDATE(TRANSDATE, 'YYMD');
```

HDIFF: Finding the Number of Units Between Two Date-Time Values

The HDIFF function calculates the number of units between two date-time values.

Find the Number of Units Between Two Date-Time Values

```
HDIFF(value1, value2, 'component', 'outfield')
```

where:

value1

> Date-time
>
> Is the end date-time value, the name of a date-time field that contains the value, or an expression that returns the value.

value2

Date-time

Is the start date-time value, the name of a date-time field that contains the value, or an expression that returns the value.

component

Alphanumeric

Is the name of the component to be used in the calculation enclosed in single quotation marks. If the component is a week, the WEEKFIRST parameter setting is used in the calculation.

outfield

Floating point or Decimal

Is the format of the output value enclosed in single quotation marks.

Finding the Number of Days Between Two Date-Time Fields

HDIFF calculates the number of days between the TRANSDATE and ADD_MONTH fields and stores the result in DIFF_PAYS, which has the format D12.2:

```
COMPUTE ADD_MONTH/HYYMDS = HADD(TRANSDATE, 'MONTH', 2, 8, 'HYYMDS'); AND
COMPUTE DIFF_DAYS/D12.2 = HDIFF(ADD_MONTH, TRANSDATE, 'DAY', 'D12.2');
```

HDTTM: Converting a Date Value to a Date-Time Value

The HDTTM function converts a date value to a date-time field. The time portion is set to midnight.

Convert a Date Value to a Date-Time Value

```
HDTTM(date, length, 'outfield')
```

where:

date

Date

Is the date value to be converted, the name of a date field that contains the value, or an expression that returns the value.

length

Integer

Is the length of the returned date-time value. Valid values are:

8 indicates a time value that includes milliseconds.

10 indicates a time value that includes microseconds.

outfield

Date-time

Is the format of the output value enclosed in single quotation marks.

Converting a Date Field to a Date-Time Field

HDTTM converts the date field TRANSDATE_DATE to a date-time field:

```
COMPUTE TRANSDATE_DATE/YYMD = HDATE(TRANSDATE, 'YYMD'); AND
COMPUTE DT2/HYYMDIA = HDTTM(TRANSDATE_DATE, 8, 'HYYMDIA');
```

HGETC: Storing the Current Date and Time in a Date-Time Field

The HGETC function stores the current date and time in a date-time field. If millisecond or microsecond values are not available in your operating environment, the function retrieves the value zero for these components.

Store the Current Date and Time in a Date-Time Field

```
HGETC(length, 'outfield')
```

where:

length

> Integer
>
> Is the length of the returned date-time value. Valid values are:
>
> 8 indicates a time value that includes milliseconds.
>
> 10 indicates a time value that includes microseconds.

outfield

> Date-time
>
> Is the format of the output value enclosed in single quotation marks.

Storing the Current Date and Time in a Date-Time Field

HGETC stores the current date and time in DT2:

```
COMPUTE DT2/HYYMDm = HGETC(10, 'HYYMDm');
```

HHMMSS: Retrieving the Current Time

The HHMMSS function retrieves the current time from the operating system as an eight character string, separating the hours, minutes, and seconds with periods.

Retrieve the Current Time

```
HHMMSS('outfield')
```

where:

outfield

> Alphanumeric
>
> Is the format of the output value enclosed in single quotation marks. The field format must be at least A8.

Retrieving the Current Time

HHMMSS retrieves the current time:

```
COMPUTE NOWTIME/A8 = HHMMSS('A8');
```

HINPUT: Converting an Alphanumeric String to a Date-Time Value

The HINPUT function converts an alphanumeric string to a date-time value.

Convert an Alphanumeric String to a Date-Time Value

```
HINPUT(inputlength, 'inputstring', length, 'outfield')
```

where:

inputlength

> Integer

> Is the length of the alphanumeric string to be converted. You can supply the actual value, the name of a numeric field that contains the value, or an expression that returns the value.

inputstring

> Alphanumeric

> Is the alphanumeric string to be converted enclosed in single quotation marks, the name of an alphanumeric field that contains the string, or an expression that returns the string. The string can consist of any valid date-time input value.

length

> Integer

> Is the length of the returned date-time value. Valid values are:

> 8 indicates a time value that includes milliseconds.

> 10 indicates a time value that includes microseconds.

outfield

> Date-time

> Is the format of the output value enclosed in single quotation marks.

Converting an Alphanumeric String to a Date-Time Value

HCNVRT converts the TRANSDATE field to alphanumeric format, then HINPUT converts the alphanumeric string to a date-time value:

```
COMPUTE ALPHA_DATE_TIME/A20 = HCNVRT(TRANSDATE, '(H17)', 17, 'A20'); AND
COMPUTE DT_FROM_ALPHA/HYYMDS = HINPUT(14, ALPHA_DATE_TIME, 8, 'HYYMDS');
```

HMIDNT: Setting the Time Portion of a Date-Time Value to Midnight

The HMIDNT function changes the time portion of a date-time value to midnight (all zeros by default). This allows you to compare a date field with a date-time field.

Set the Time Portion of a Date-Time Value to Midnight

```
HMIDNT(value, length, 'outfield')
```

where:

value

> Date-time

> Is the date-time value whose time is to be set to midnight, the name of a date-time field that contains the value, or an expression that returns the value.

length

>Integer

>Is the length of the returned date-time value. Valid values are:

>8 indicates a time value that includes milliseconds.

>10 indicates a time value that includes microseconds.

outfield

>Date-time

>Is the format of the output value enclosed in single quotation marks.

Setting the Time to Midnight

HMIDNT sets the time portion of the TRANSDATE field to midnight first in the 24-hour system and then in the 12-hour system:

```
COMPUTE TRANSDATE_MID_24/HYYMDS  = HMIDNT(TRANSDATE, 8, 'HYYMDS'); AND
COMPUTE TRANSDATE_MID_12/HYYMDSA = HMIDNT(TRANSDATE, 8, 'HYYMDSA');
```

HNAME: Retrieving a Date-Time Component in Alphanumeric Format

The HNAME function extracts a specified component from a date-time value in alphanumeric format.

Retrieve a Date-Time Component in Alphanumeric Format

```
HNAME(value, 'component', 'outfield')
```

where:

value

>Date-time

>Is the date-time value from which a component is to be extracted, the name of a date-time field that contains the value, or an expression that returns the value.

component

>Alphanumeric

>Is the name of the component to be retrieved enclosed in single quotation marks.

outfield

>Alphanumeric

>Is the format of the output value enclosed in single quotation marks. The field format must be at lease A2.

>The function converts all other components to strings of digits only. The year is always four digits, and the hour assumes the 24-hour system.

Retrieving the Week Component in Alphanumeric Format

HNAME returns the week in alphanumeric format from the TRANSDATE field. Changing the WEEKFIRST parameter setting changes the value of the component.

```
COMPUTE WEEK_COMPONENT/A10 = HNAME(TRANSDATE, 'WEEK', 'A10');
```

Retrieving the Day Component in Alphanumeric Format

HNAME retrieves the day in alphanumeric format from the TRANSDATE field:
```
COMPUTE DAY_COMPONENT/A2 = HNAME(TRANSDATE, 'DAY', 'A2');
```

HPART: Retrieving a Date-Time Component in Numeric Format

The HPART function extracts a specified component from a date-time value and returns it in numeric format.

Retrieve a Date-Time Component in Numeric Format
```
HPART(value, 'component', 'outfield')
```

where:

value

Date-time

Is a date-time value, the name of a date-time field that contains the value, or an expression that returns the value.

component

Alphanumeric

Is the name of the component to be retrieved enclosed in single quotation marks.

outfield

Integer

Is the format of the output value enclosed in single quotation marks.

Retrieving the Day Component in Numeric Format

HPART retrieves the day in integer format from the TRANSDATE field:
```
COMPUTE DAY_COMPONENT/I2 = HPART(TRANSDATE, 'DAY', 'I2');
```

HSETPT: Inserting a Component Into a Date-Time Value

The HSETPT function inserts the numeric value of a specified component into a date-time value.

Insert a Component Into a Date-Time Value
```
HSETPT(dtfield, 'component', value, length, 'outfield')
```

where:

dtfield

Date-time

Is a date-time value, the name of a date-time field that contains the value, or an expression that returns the value.

component

Alphanumeric

Is the name of the component to be inserted enclosed in single quotation marks.

value

Integer

Is the numeric value to be inserted for the requested component, the name of a numeric field that contains the value, or an expression that returns the value.

length

Integer

Is the length of the returned date-time value. Valid values are:

8 indicates a time value that includes milliseconds.

10 indicates a time value that includes microseconds.

outfield

Date-time

Is the format of the output value enclosed in single quotation marks.

Inserting the Day Component Into a Date-Time Field

HSETPT inserts the day as 28 into the ADD_MONTH field and stores the result in INSERT_DAY:

```
COMPUTE ADD_MONTH/HYYMDS = HADD(TRANSDATE, 'MONTH', 2, 8, 'HYYMDS'); AND
COMPUTE INSERT_DAY/HYYMDS = HSETPT(ADD_MONTH, 'DAY', 28, 8, 'HYYMDS');
```

HTIME: Converting the Time Portion of a Date-Time Value to a Number

The HTIME function converts the time portion of a date-time value to the number of milliseconds if the first argument is eight, or microseconds if the first argument is ten. To include microseconds, the input date-time value must be 10-bytes.

Convert the Time Portion of a Date-Time Field to a Number

```
HTIME(length, value, 'outfield')
```

where:

length

Integer

Is the length of the input date-time value. Valid values are:

8 indicates a time value that includes milliseconds.

10 indicates a time value that includes microseconds.

value

Date-time

Is the date-time value from which to convert the time, the name of a date-time field that contains the value, or an expression that returns the value.

outfield

Floating point or Decimal

Is the format of the output value enclosed in single quotation marks.

Converting the Time Portion of a Date-Time Field to a Number

HTIME converts the time portion of the TRANSDATE field to the number of milliseconds:

```
COMPUTE MILLISEC/D12.2 = HTIME(8, TRANSDATE, 'D12.2');
```

JULDAT: Converting From Gregorian to Julian Format

The JULDAT function converts a date from Gregorian format (year-month-day) to Julian format (year-day). A date in Julian format is a five- or seven-digit number. The first two or four digits are the year; the last three digits are the number of the day, counting from January 1. For example, January 1, 1999 in Julian format is either 99001 or 1999001.

DATEFNS Settings for JULDAT

JULDAT converts a Gregorian date to either YYNNN or YYYYNNN format, using the DEFCENT and YRTHRESH parameter settings to determine if the century is required.

JULDAT returns dates as follows:

DATEFNS Setting	I6 or I7 Format	I8 Format or Greater
ON	YYNNN	YYYYNNN
OFF	YYNNN	YYNNN

Convert From Gregorian to Julian Format

```
JULDAT(indate, 'outfield')
```

where:

indate

Integer (I6, I8, I6YMD, I8YYMD)

Is the date or the name of the field that contains the date in year-month-day format (YMD or YYMD).

outfield

Integer (I5 or I7)

Is the format of the output value enclosed in single quotation marks.

Converting From Gregorian to Julian Format

JULDAT converts the HIRE_DATE field to Julian format.

```
COMPUTE JULIAN/I7 = JULDAT(HIRE_DATE, 'I7');
```

TIMETOTS: Converting a Time to a Timestamp

The TIMETOTS function converts a time to a timestamp, using the current date to supply the date component of its value. The first argument must be in H (date-time) format. The DATE component will be set to the current date.

Convert a Time to a Timestamp

```
TIMETOTS (time, length, 'outfield')
```

where:

time

> Date-time
>
> Is the time in a date-time format.

length

> Integer
>
> Is the length of the result. This can be one of the following:
>
> 8 for time values including milliseconds.
>
> 10 for input time values including microseconds.

outfield

> Date-time
>
> Is the format of the output value enclosed in single quotation marks.

Converting a Time to a Timestamp

TIMETOTS converts a time argument to a timestamp:

```
COMPUTE TSTMPSEC/HYYMDS = TIMETOTS(TMSEC, 8, 'HYYMDS'); AND
COMPUTE TSTMPMILLI/HYYMDm = TIMETOTS(TMMILLI, 10, 'HYYMDm');
```

TODAY: Returning the Current Date

The TODAY function retrieves the current date from the operating system in the format MM/DD/YY or MM/DD/YYYY. It always returns a date that is current. Therefore, if you are running an application late at night, use TODAY. You can remove the default embedded slashes with the EDIT function.

Retrieve the Current Date

```
TODAY('outfield')
```

where:

outfield

> Alphanumeric
>
> Is the format of the output value enclosed in single quotation marks. The field format must be at least A8. The following apply:
>
> - If DATEFNS=ON and the format is A8 or A9, TODAY returns the 2-digit year.
> - If DATEFNS=ON and the format is A10 or greater, TODAY returns the 4-digit year.
> - If DATEFNS=OFF, TODAY returns the 2-digit year, regardless of the format of *outfield*.

Retrieving the Current Date

TODAY retrieves the current date and stores it in the DATE field.

```
COMPUTE DATE/A10 = TODAY('A10');
```

YM: Calculating Elapsed Months

The YM function calculates the number of months that elapse between two dates. The dates must be in year-month format. You can convert a date to this format by using the CHGDAT or EDIT function.

Calculate Elapsed Months

```
YM(fromdate, todate, 'outfield')
```

where:

fromdate

> Integer (I4YM or I6YYM)

> Is the start date in year-month format (for example, I4YM). If the date is not valid, the function returns a 0.

todate

> Integer (I4YM or I6YYM)

> Is the end date in year-month format. If the date is not valid, the function returns a 0.

outfield

> Integer

> Is the format of the output value enclosed in single quotation marks.

> **Note:** If *fromdate* or *todate* is in integer year-month-day format (I6YMD or I8YYMD), simply divide by 100 to convert to year-month format and set the result to an integer. This drops the day portion of the date, which is now after the decimal point.

Calculating Elapsed Months

The COMPUTE commands convert the dates from year-month-day to year-month format; then YM calculates the difference between the values in the HIRE_DATE/100 and DAT_INC/100 fields:

```
COMPUTE HIRE_MONTH/I4YM = HIRE_DATE/100; AND
COMPUTE MONTH_INC/I4YM = DAT_INC/100; AND
COMPUTE MONTHS_HIRED/I3 = YM(HIRE_MONTH, MONTH_INC, 'I3');
```

Format Conversion Functions

Format conversion functions convert fields from one format to another.

ATODBL: Converting an Alphanumeric String to Double-Precision Format

The ATODBL function converts a number in alphanumeric format to decimal (double-precision) format.

Convert an Alphanumeric String to Double-Precision Format

```
ATODBL(string, length, 'outfield')
```

where:

string

> Alphanumeric

> Is the alphanumeric string to be converted, or a field that contains the string.

length

> Alphanumeric

Is the two-character length of *infield* in bytes. This can be a numeric constant, or a field that contains the value. If you specify a numeric constant, enclose it in single quotation marks. The maximum value is 15.

outfield

Decimal

Is the format of the output value enclosed in single quotation marks.

Converting an Alphanumeric Field to Double-Precision Format

ATODBL converts the EMP_ID field into double-precision format and stores the result in D_EMP_ID:

```
COMPUTE D_EMP_ID/D12.2 = ATODBL(EMP_ID, '09', 'D12.2');
```

EDIT: Converting the Format of a Field

The EDIT function converts an alphanumeric field that contains numeric characters to numeric format or converts a numeric field to alphanumeric format. It is useful when you need to manipulate a field using a command that requires a particular format.

When EDIT assigns a converted value to a new field, the format of the new field must correspond to the format of the returned value. For example, if EDIT converts a numeric field to alphanumeric format, you must give the new field an alphanumeric format:

```
DEFINE ALPHAPRICE/A6 = EDIT(PRICE);
```

EDIT deals with a symbol in the following way:

- When an alphanumeric field is converted to numeric format, a sign or decimal point in the field is acceptable and is stored in the numeric field.
- When converting a floating-point or packed-decimal field to alphanumeric format, EDIT removes the sign, the decimal point, and any number to the right of the decimal point. It then right-justifies the remaining digits and adds leading zeros to achieve the specified field length. Converting a number with more than nine significant digits in floating-point or packed-decimal format may produce an incorrect result.

EDIT also extracts characters from or adds characters to an alphanumeric string. For more information, see EDIT: Extracting or Adding Characters.

Convert the Format of a Field

```
EDIT(fieldname);
```

where:

fieldname

Alphanumeric or Numeric

Is the field name.

Converting From Numeric to Alphanumeric Format

EDIT converts HIRE_DATE (a legacy date format) to alphanumeric format. CHGDAT is then able to use the field, which it expects in alphanumeric format:

```
COMPUTE ALPHA_HIRE/A17 = EDIT(HIRE_DATE); AND
COMPUTE HIRE_MDY/A17 = CHGDAT('YMD', 'MDYYX', ALPHA_HIRE, 'A17');
```

FTOA: Converting a Number to Alphanumeric Format

The FTOA function converts a number up to 16 digits long from numeric format to alphanumeric format. It retains the decimal positions of a number and right-justifies it with leading spaces. You can also add edit options to a number converted by FTOA.

When using FTOA to convert a number containing decimals to a character string, you must specify an alphanumeric format large enough to accommodate both the integer and decimal portions of the number. For example, a D12.2 format is converted to A14. If the output format is not large enough, decimals are truncated.

Convert a Number to Alphanumeric Format

```
FTOA(number, '(format)', 'outfield')
```

where:

number

Numeric F or D (single and double-precision floating-point)

Is the number to be converted, or the name of the field that contains the number.

format

Alphanumeric

Is the output format of the number enclosed in both single quotation marks and parentheses. Only floating point single-precision and double-precision formats are supported. Include any edit options that you want to appear in the output. The D (floating-point double-precision) format automatically supplies commas.

If you use a field name for this argument, specify the name without quotation marks or parentheses. If you specify a format, the format must be enclosed in parentheses.

outfield

Alphanumeric

Is the format of the output value enclosed in single quotation marks. The length of this argument must be greater than the length of *number* and must account for edit options and a possible negative sign.

Converting From Numeric to Alphanumeric Format

FTOA converts the GROSS field from floating point double-precision to alphanumeric format and stores the result in ALPHA_GROSS:

```
COMPUTE ALPHA_GROSS/A15 = FTOA(GROSS, '(D12.2)', 'A15');
```

HEXBYT: Converting a Decimal Integer to a Character

The HEXBYT function obtains the ASCII, EBCDIC, or Unicode character equivalent of a decimal integer, depending on your configuration and operating environment. It returns a single alphanumeric character in the ASCII, EBCDIC, or Unicode character set. You can use this function to produce characters that are not on your keyboard, similar to the CTRAN function.

In Unicode configurations, this function uses values in the range:

- 0 to 255 for 1-byte characters.

- 256 to 65535 for 2-byte characters.
- 65536 to 16777215 for 3-byte characters.
- 16777216 to 4294967295 for 4-byte characters (primarily for EBCDIC).

The display of special characters depends on your software and hardware; not all special characters may appear.

Convert a Decimal Integer to a Character

```
HEXBYT(input, 'outfield')
```

where:

input

> Integer

> Is the decimal integer to be converted to a single character. In non-Unicode environments, a value greater than 255 is treated as the remainder of *input* divided by 256.

output

> Alphanumeric

> Is the format of the output value enclosed in single quotation marks.

Converting a Decimal Integer to a Character

HEXBYT converts LAST_INIT_CODE to its character equivalent and stores the result in LAST_INIT:

```
COMPUTE LAST_INIT_CODE/I3 = BYTVAL(LAST_NAME, 'I3'); AND
COMPUTE LAST_INIT/A1 = HEXBYT(LAST_INIT_CODE, 'A1');
```

ITONUM: Converting a Large Binary Integer to Double-Precision Format

The ITONUM function converts a large binary integer in a data source to double-precision format. Some programming languages and some data storage systems use large binary integer formats. However, large binary integers (more than 4 bytes in length) are not supported in the Master File so they require conversion to double-precision format.

You must specify how many of the right-most bytes in the input field are significant. The result is an 8-byte double-precision field.

Convert a Large Binary Integer to Double-Precision Format

```
ITONUM(maxbytes, infield, 'outfield')
```

where:

maxbytes

> Numeric

> Is the maximum number of bytes in the 8-byte binary input field that have significant numeric data, including the binary sign. Valid values are:

> 5 ignores the left-most 3 bytes.

> 6 ignores the left-most 2 bytes.

> 7 ignores the left-most byte.

infield

> Alphanumeric

> Is the field that contains the binary number. Both the USAGE and ACTUAL formats of the field must be A8.

outfield

> Decimal

> Is the format of the output value enclosed in single quotation marks. The format must be D*n*.

Converting a Large Binary Integer to Double-Precision Format

ITONUM converts the BINARYFLD field to double-precision format:

```
COMPUTE MYFLD/D14 = ITONUM(6, BINARYFLD, 'D14');
```

ITOPACK: Converting a Large Binary Integer to Packed-Decimal Format

The ITOPACK function converts a large binary integer in a data source to packed-decimal format. Some programming languages and some data storage systems use double-word binary integer formats. Large binary integers (more than 4 bytes in length) are not supported in the Master File so they require conversion to packed-decimal format.

You must specify how many of the right-most bytes in the input field are significant. The result is an 8-byte packed-decimal field of up to 15 significant numeric positions (for example, P15 or P16.2).

Limit: For a field defined as 'PIC 9(15) COMP' or the equivalent (15 significant digits), the maximum number that can be converted is 167,744,242,712,576.

Convert a Large Binary Integer to Packed-Decimal Format

```
ITOPACK(maxbytes, infield, 'outfield')
```

where:

maxbytes

> Numeric

> Is the maximum number of bytes in the 8-byte binary input field that have significant numeric data, including the binary sign.

> Valid values are:

> 5 ignores the left-most 3 bytes (up to 11 significant positions).

> 6 ignores the left-most 2 bytes (up to 14 significant positions).

> 7 ignores the left-most byte (up to 15 significant positions).

infield

> Alphanumeric

> Is the field that contains the binary number. Both the USAGE and ACTUAL formats of the field must be A8.

outfield

> Packed

Is the format of the output value enclosed in single quotation marks. The format must be P*n* or P*n.d*.

Converting a Large Binary Integer to Packed-Decimal Format

ITOPACK converts the BINARYFLD field to packed-decimal format:

```
COMPUTE PACKFLD/P14.4 = ITOPACK(6, BINARYFLD, 'P14.4');
```

ITOZ: Converting a Number to Zoned Format

The ITOZ function converts a number in numeric format to zoned format. Although a request cannot process zoned numbers, it can write zoned fields to an extract file for use by an external program.

Convert to Zoned Format

```
ITOZ(outlength, number, 'outfield')
```

where:

outlength

Integer

Is the length of *number* in bytes. The maximum number of bytes is 15. The last byte includes the sign.

number

Numeric

Is the number to be converted, or the field that contains the number. The number is truncated to an integer before it is converted.

outfield

Alphanumeric

Is the format of the output value enclosed in single quotation marks.

Converting a Number to Zoned Format

ITOZ converts the CURR_SAL field to a zoned format:

```
COMPUTE ZONE_SAL/A8 = ITOZ(8, CURR_SAL, 'A8');
```

PCKOUT: Writing a Packed Number of Variable Length

The PCKOUT function writes a packed number of variable length to an extract file. When a request saves a packed number to an extract file, it typically writes it as an 8- or 16-byte field regardless of its format specification. With PCKOUT, you can vary the field's length between 1 to 16 bytes.

Write a Packed Number of Variable Length

```
PCKOUT(infield, outlength, 'outfield')
```

where:

infield

Numeric

Is the input field that contains the values. The field can be in packed, integer, floating-point, or double-precision format. If the field is not in integer format, its values are rounded to the nearest integer.

outlength

> Numeric

> Is the length of outfield from 1 to 16 bytes.

outfield

> Alphanumeric

> Is the format of the output value enclosed in single quotation marks. The function returns the field as alphanumeric although it contains packed data.

Writing a Packed Number of Variable Length

PCKOUT converts the CURR_SAL field to a 5-byte packed field and stores the result in SHORT_SAL:

```
COMPUTE SHORT_SAL/A5 = PCKOUT(CURR_SAL, 5, 'A5');
```

Numeric Functions

Numeric functions perform calculations on numeric constants and fields.

ABS: Calculating Absolute Value

The ABS function returns the absolute value of a number.

Calculate Absolute Value

```
ABS(argument)
```

where:

argument

> Numeric

> Is the value for which the absolute value is returned, the name of a field that contains the value, or an expression that returns the value. If you use an expression, use parentheses as needed to ensure the correct order of evaluation.

Calculating Absolute Value

The first COMPUTE command creates the DIFF field, then ABS calculates the absolute value of DIFF:

```
COMPUTE DIFF/I5 = DELIVER_AMT - UNIT_SOLD; AND
COMPUTE ABS_DIFF/I5 = ABS(DIFF);
```

BAR: Producing a Bar Chart

The BAR function produces a horizontal bar chart using repeating characters to form each bar. Optionally, you can create a scale to clarify the meaning of a bar chart by replacing the title of the column containing the bar with a scale.

Produce a Bar Chart

```
BAR(barlength, infield, maxvalue, 'char', 'outfield')
```

where:

barlength

Numeric

Is the maximum length of the bar in characters. If this value is less than or equal to 0, the function does not return a bar.

infield

Numeric

Is the data field plotted as a bar chart.

maxvalue

Numeric

Is the maximum value of a bar. This value must be greater than the maximum value stored in *infield*. If *infield* is larger than *maxvalue*, the function uses *maxvalue* and returns a bar of maximum length.

char

Alphanumeric

Is the repeating character that creates the bars enclosed in single quotation marks. If you specify more than one character, only the first character is used.

outfield

Alphanumeric

Is the format of the output value enclosed in single quotation marks. The output field must be large enough to contain a bar of maximum length as defined by *barlength*.

Producing a Bar Chart

BAR creates a bar chart for the CURR_SAL field, and stores the output in SAL_BAR. The bar created can be no longer than 30 characters long, and the value it represents can be no greater than 30,000.

```
COMPUTE SAL_BAR/A30 = BAR(30, CURR_SAL, 30000, '=', SAL_BAR);
```

CHKPCK: Validating a Packed Field

The CHKPCK function validates the data in a field described as packed format (if available on your platform). The function prevents a data exception from occurring when a request reads a field that is expected to contain a valid packed number but does not.

To use CHKPCK:

1. Ensure that the Master File (USAGE and ACTUAL attributes) defines the field as alphanumeric, not packed. This does *not* change the field data, which remains packed, but it enables the request to read the data without a data exception.

2. Call CHKPCK to examine the field. The function returns the output to a field defined as packed. If the value it examines is a valid packed number, the function returns the value; if the value is not packed, the function returns an error code.

Validate a Packed Field

```
CHKPCK(inlength, infield, error, 'outfield')
```

where:

inlength

> Numeric

> Is the length of the packed field. It can be between 1 and 16 bytes.

infield

> Alphanumeric

> Is the name of the packed field. The field is described as alphanumeric, not packed.

error

> Numeric

> Is the error code that the function returns if a value is not packed. Choose an error code outside the range of data. The error code is first truncated to an integer, then converted to packed format. However, it may appear on a report with a decimal point because of the format of the output field.

outfield

> Packed

> Is the format of the output value enclosed in single quotation marks.

Validating Packed Data

CHKPCK validates the values in the PACK_SAL field, and stores the result in the GOOD_PACK field. Values not in packed format return the error code -999. Values in packed format appear accurately.

```
COMPUTE GOOD_PACK/P8CM = CHKPCK(8, PACK_SAL, -999, GOOD_PACK);
```

DMOD, FMOD, and IMOD: Calculating the Remainder From a Division

The MOD functions calculate the remainder from a division. Each function returns the remainder in a different format.

The functions use the following formula.

```
remainder = dividend - INT(dividend/divisor) * divisor
```

- *DMOD* returns the remainder as a decimal number.
- *FMOD* returns the remainder as a floating point number.
- *IMOD* returns the remainder as an integer.

For information on the INT function, see "INT: Finding the Greatest Integer" on page 420.

Calculate the Remainder From a Division

```
function(dividend, divisor, 'outfield')
```

where:

function

Is one of the following:

DMOD returns the remainder as a decimal number.

FMOD returns the remainder as a floating point number.

IMOD returns the remainder as an integer.

dividend

> Numeric
>
> Is the number being divided.

divisor

> Numeric
>
> Is the number dividing the dividend.

outfield

> Numeric
>
> Is the format of the output value enclosed in single quotation marks. The format is determined by the result returned by the specific function.

Calculating the Remainder From a Division

IMOD divides ACCTNUMBER by 1000 and returns the remainder to LAST3_ACCT:

```
COMPUTE LAST3_ACCT/I3L = IMOD(ACCTNUMBER, 1000, LAST3_ACCT);
```

EXP: Raising *e* to the Nth Power

The EXP function raises the value ″e″ (approximately 2.72) to a specified power. This function is the inverse of the LOG function, which returns an argument's logarithm.

EXP calculates the result by adding terms of an infinite series. If a term adds less than .000001 percent to the sum, the function ends the calculation and returns the result as a double-precision number.

Raise *e* to the Nth Power

```
EXP(power, 'outfield')
```

where:

power

> Numeric
>
> Is the power to which ″e″ is raised.

outfield

> Floating point or Decimal
>
> Is the format of the output value enclosed in single quotation marks.

Raising *e* to the Nth Power

EXP raises *e* to the 2nd power:

```
COMPUTE E2/D12.2 = EXP(2, 'D12.2');
```

INT: Finding the Greatest Integer

The INT function returns the integer component of a number.

Find the Greatest Integer

```
INT(argument)
```

where:

argument

> Numeric

> Is the value for which the integer component is returned, the name of a field that contains the value, or an expression that returns the value. If you supply an expression, use parentheses as needed to ensure the correct order of evaluation.

Finding the Greatest Integer

INT finds the greatest integer in the DED_AMT field and stores it in INT_DED_AMT:

```
COMPUTE INT_DED_AMT/I9 = INT(DED_AMT);
```

LOG: Calculating the Natural Logarithm

The LOG function returns the natural logarithm of a number.

Calculate the Natural Logarithm

```
LOG(argument)
```

where:

argument

> Numeric

> Is the value for which the natural logarithm is calculated, the name of a field that contains the value, or an expression that returns the value. If you supply an expression, use parentheses as needed to ensure the correct order of evaluation. If *argument* is less than or equal to 0, LOG returns 0.

Calculating the Natural Logarithm

LOG calculates the logarithm of the CURR_SAL field:

```
COMPUTE LOG_CURR_SAL/D12.2 = LOG(CURR_SAL);
```

MAX and MIN: Finding the Maximum or Minimum Value

The MAX and MIN functions return the maximum or minimum value, respectively, from a list of values.

Find the Maximum or Minimum Value

```
{MAX|MIN}(argument1, argument2, ...)
```

where:

MAX

> Returns the maximum value.

MIN

Returns the minimum value.

argument1, argument2

Numeric

Are the values for which the maximum or minimum value is returned, the name of a field that contains the values, or an expression that returns the values. If you supply an expression, use parentheses as needed to ensure the correct order of evaluation.

Determining the Minimum Value

MIN returns either the value of the ED_HRS field or the constant 30, whichever is lower:

```
COMPUTE MIN_EDHRS_30/D12.2 = MIN(ED_HRS, 30);
```

PRDNOR and PRDUNI: Generating Reproducible Random Numbers

The PRDNOR and PRDUNI functions generate reproducible random numbers:

- PRDNOR generates reproducible double-precision random numbers normally distributed with an arithmetic mean of 0 and a standard deviation of 1. If PRDNOR generates a large set of numbers, they have the following properties:

  - The numbers lie roughly on a bell curve, as shown in the following figure. The bell curve is highest at the 0 mark, meaning that there are more numbers closer to 0 than farther away.

  - The average of the numbers is close to 0.
  - The numbers can be any size, but most are between 3 and -3.
- PRDUNI generates reproducible double-precision random numbers uniformly distributed between 0 and 1 (that is, any random number it generates has an equal probability of being anywhere between 0 and 1).

Generate Reproducible Random Numbers

```
{PRDNOR|PRDUNI}(seed, 'outfield')
```

where:

PRDNOR

Generates reproducible double-precision random numbers normally distributed with an arithmetic mean of 0 and a standard deviation of 1.

PRDUNI

Generates reproducible double-precision random numbers uniformly distributed between 0 and 1.

seed

Numeric

Is the seed or the field that contains the seed, up to 9 digits. The seed is truncated to an integer.

outfield

Decimal

Is the format of the output value enclosed in single quotation marks.

Generating Reproducible Random Numbers

PRDNOR assigns random numbers and stores them in RAND. The seed is 40. To produce a different set of numbers, change the seed.

```
COMPUTE RAND/D12.2 = PRDNOR(40, 'D12.2');
```

RDNORM and RDUNIF: Generating Random Numbers

The RDNORM and RDUNIF functions generate random numbers:

- RDNORM generates double-precision random numbers normally distributed with an arithmetic mean of 0 and a standard deviation of 1. If RDNORM generates a large set of numbers (between 1 and 32768), they have the following properties:
  - The numbers lie roughly on a bell curve, as shown in the following figure. The bell curve is highest at the 0 mark, meaning that there are more numbers closer to 0 than farther away.

Frequency of Occurrence vs. Random Number Generated

- – The average of the numbers is close to 0.
- – The numbers can be any size, but most are between 3 and -3.
- RDUNIF generates double-precision random numbers uniformly distributed between 0 and 1 (that is, any random number it generates has an equal probability of being anywhere between 0 and 1).

Generate Random Numbers

```
{RDNORM|RDUNIF}('outfield')
```

where:

RDNORM

> Generates double-precision random numbers normally distributed with an arithmetic mean of 0 and a standard deviation of 1.

RDUNIF

> Generates double-precision random numbers uniformly distributed between 0 and 1.

`outfield`

> Decimal

> Is the format of the output value enclosed in single quotation marks.

Generating Random Numbers

RDNORM assigns random numbers and stores them in RAND.

```
COMPUTE RAND/D12.2 = RDNORM('D12.2');
```

SQRT: Calculating the Square Root

The SQRT function calculates the square root of a number.

Calculate the Square Root

```
SQRT(argument)
```

where:

argument

> Numeric
>
> Is the value for which the square root is calculated, the name of a field that contains the value, or an expression that returns the value. If you supply an expression, use parentheses as needed to ensure the correct order of evaluation. If you supply a negative number, the result is zero.

Calculating the Square Root

SQRT calculates the square root of LISTPR:

```
COMPUTE SQRT_LISTPR/D12.2 = SQRT(LISTPR);
```

System Functions

System functions call the operating system to obtain information about the operating environment.

FGETENV: Retrieving the Value of an Environment Variable

The FGETENV function retrieves the value of an environment variable and returns it as an alphanumeric string.

Retrieve the Value of an Environment Variable

```
FGETENV(varlength, 'varname', outfieldlen, 'outfield')
```

where:

varlength

> Integer
>
> Is the length of the environment variable name.

varname

> Alphanumeric
>
> Is the name of the environment variable.

outfieldlen

> Integer
>
> Is the length of the field in which the environment variable's value is stored.

outfield

> Alphanumeric
>
> Is the format of the output value enclosed in single quotation marks.

Retrieving the Language Locale

Using the LANG environment variable, FGETENV retrieves the object location for the language locale:

```
COMPUTE LANG_LOCALE/A40 = FGETENV(4, 'LANG', 40, 'A40');
```

GETUSER: Retrieving a User ID

The GETUSER function retrieves the ID of the connected user.

Retrieve a User ID

```
GETUSER('outfield')
```

where:

outfield

> Alphanumeric
>
> Is the format (at least A8) of the output value enclosed in single quotation marks. The length depends on the platform on which the function is issued. Provide a length as long as required for your platform; otherwise, the output will be truncated.

Retrieving a User ID

GETUSER retrieves the user ID of the person running the request:

```
COMPUTE USERID/A8 = GETUSER('A8');
```

HHMMSS: Retrieving the Current Time

The HHMMSS function retrieves the current time from the operating system as an eight-character string, separating the hours, minutes, and seconds with periods. For details on how to use this function, see HHMMSS: Retrieving the Current Time.

TODAY: Returning the Current Date

The TODAY function retrieves the current date from the system. For details on how to use this function, see TODAY: Returning the Current Date.

Chapter 8. Visualizing Trends in Reports

Describes how to insert visual representations (in the form of bar graphs) of selected data directly into your report output.

To make your HTML reports more powerful, you can insert visual representations of selected data directly into the report output. These visual representations are in the form of vertical or horizontal bar graphs that make relationships and trends among data more obvious.

Applying Bar Graphs

Vertical or horizontal bar graphs highlight relationships and trends among data.

- **Vertical Bar Graph.** You can apply a vertical bar graph to report columns associated with an ACROSS sort field. The report output displays a vertical bar graph in a new row above the associated data values as shown in the following image.

Bar graphs that project above the zero line represent positive values, while bar graphs that project below the zero line represent negative values.

- **Horizontal Bar Graph.** You can apply a horizontal bar graph to report columns. The report output displays a horizontal bar graph in a new column to the right of the associated data values as shown in the following image.

City	Budget Dollars	Dollar Sales	DIFFERENCE	
Atlanta	$4,247,597.00	$4,100,107.00	$147,490.00	
Boston	$3,818,397.00	$3,707,986.00	$110,411.00	
Chicago	$3,866,856.00	$3,924,401.00	-$57,545.00	
Houston	$3,680,679.00	$3,714,978.00	-$34,299.00	
Los Angeles	$3,669,484.00	$3,772,014.00	-$102,530.00	
Memphis	$3,689,979.00	$3,687,057.00	$2,922.00	
New Haven	$3,832,202.00	$3,782,049.00	$50,153.00	
New York	$3,926,333.00	$3,902,275.00	$24,058.00	
Orlando	$3,870,405.00	$3,923,215.00	-$52,810.00	
San Francisco	$3,916,863.00	$3,870,258.00	$46,605.00	
Seattle	$4,055,166.00	$4,010,685.00	$44,481.00	
St. Louis	$3,646,838.00	$3,761,286.00	-$114,448.00	

Bar graphs that project to the right of the zero line represent positive values, while bar graphs that project to the left of the zero line represent negative values.

The length of each vertical or horizontal bar graph is proportional to the magnitude of its associated data value. The shortest bar graph appears for the

value with the minimum magnitude, the longest bar graph for the value with the maximum magnitude, and bar graphs of varying length appear for each value within the minimum-maximum magnitude range. Notice in the figure that a value of 147,490.00 produces a longer horizontal bar graph than a value of 50,153.00. Therefore, a complete row of vertical bar graphs or a complete column of horizontal bar graphs forms a bar chart.

You can only apply data visualization bar graphs to numeric report columns (integer, decimal, floating point single-precision, floating point double-precision, and packed). Bar graphs applied to alphanumeric, date, or text field formats are ignored.

You can display data visualization bar graphs in OLAP-enabled HTML reports, where bar graphs are applied to Measures. See Associating Bar Graphs With Measures.

Associating Bar Graphs With Measures

You can associate data visualization bar graphs with any numeric measure that appears in the report output.

The type of bar graph that you can apply depends on the placement of the dimensions included in the report:
- If all report dimensions are vertical (By) sort fields (listed in the Drill Down box in the OCP), you can apply a horizontal bar graph to the specified measures.
- If any dimension is a horizontal (Across) sort field (listed in the Drill Across box in the OCP), you can apply a vertical bar graph to the specified measures.

For illustrations, see Applying Bar Graphs.

For more information about OLAP reports, see Analyzing Data in an OLAP Report.

Data Visualization Bar Graph Attributes

The following table outlines the default attributes used to display data visualization bar graphs applied from the OLAP selections pane or the OLAP Control Panel. The first column lists the bar graph attribute, while the second column lists the default value.

Bar graph attribute	Default value
Color	Positive values: Blue Negative values: Red
Length	Vertical bar graph: 60 pixels Horizontal bar graph: 80 pixels
Width	The size of the font in the report output is used to define a default value for the width of the bar graph.

Note: Currently, you cannot modify bar graph attributes from the OLAP selection panel or the OLAP Control Panel.

Applying Bar Graphs to Measures in an OLAP Report

The quickest way to apply data visualization bar graphs to numeric measures is from the report itself.

Apply Bar Graphs to Measures in an OLAP Report

1. Right-click the title of a measure column.
2. Choose **Visualize** from the menu.

The report runs automatically, displaying a column of bar graphs following the selected measures column.

Tip: To remove the bar graphs, right-click the measure column title and choose **Remove Visualization** from the menu.

Applying and Sorting Bar Graphs in a Report

In the following OLAP report:

1. Right-click **Line Cost of Goods Sold** and choose **Visualize** to apply a data visualization bar graph to each value in the column, as shown in the following image.

The display changes instantly, as shown in the following image.

QUARTER	Store Name:	PRODTYPE	Quantity:	Line Cost Of Goods Sold	
Q1	AV VideoTown	Analog	18,449	3,969,296.00	▮
		Digital	22,206	5,109,400.00	▮
	Audio Expert	Analog	78,449	16,467,146.00	▬▬
		Digital	105,983	25,092,678.00	▬▬▬
	City Video	Analog	6,287	1,315,015.00	▏
		Digital	7,196	1,607,513.00	▏
	Consumer Merchandise	Analog	6,980	1,542,036.00	▏
		Digital	14,957	3,251,090.00	▮
	TV City	Analog	19,077	3,772,119.00	▮
		Digital	41,307	10,128,967.00	▬
	Web Sales	Analog	545	124,366.00	
		Digital	829	190,201.00	ǀ
	eMart	Analog	97,128	21,152,262.00	▬▬
		Digital	108,221	24,990,368.00	▬▬▬
Q2	AV VideoTown	Analog	11,781	2,663,655.00	▮
		Digital	27,377	5,928,507.00	▮
	Audio Expert	Analog	57,944	11,868,758.00	▬
		Digital	111,421	28,064,250.00	▬▬▬

2. Sort the data by highest value. You can either right-click **Line Cost of Goods Sold** and choose **Sort by Highest**, or click the **Up** arrow (the tool tip reads **Sort LINE_COG highest to lowest**).

The following image shows the results of sorting the data by the highest value.

QUARTER	Store Name:	PRODTYPE	Quantity:	Line Cost Of Goods Sold	
Q2	Audio Expert	Digital	111,421	28,064,250.00	
Q1	Audio Expert	Digital	105,983	25,092,678.00	
Q1	eMart	Digital	108,221	24,990,368.00	
Q2	eMart	Digital	115,102	24,971,512.00	
Q1	eMart	Analog	97,128	21,152,262.00	
Q2	eMart	Analog	74,737	16,789,403.00	
Q1	Audio Expert	Analog	78,449	16,467,146.00	
Q4	eMart	Digital	72,126	14,000,951.00	
Q3	eMart	Digital	66,156	13,867,709.00	
Q2	Audio Expert	Analog	57,944	11,868,758.00	
Q3	Audio Expert	Digital	50,076	11,210,406.00	
Q4	Audio Expert	Digital	53,275	11,190,923.00	
Q1	TV City	Digital	41,307	10,128,967.00	
Q4	eMart	Analog	39,515	9,383,389.00	
Q3	eMart	Analog	36,306	8,308,647.00	
Q2	TV City	Digital	29,627	6,732,303.00	
Q2	AV VideoTown	Digital	27,377	5,928,507.00	
Q4	Audio Expert	Analog	25,897	5,916,936.00	

Applying Bar Graphs to Measures Using the Selections Pane or Control Panel

You can apply data visualization bar graphs to any numeric measure.

To indicate the measures for which you want to display bar graphs, you click the check box located to the left of each measure. This check box has three states that control the display modes for the measure.

In the following table the first column shows the three check box states and the second column provides descriptions for the display modes.

Check Box State	Display Mode for the Measure
Check mark	Displays the measure.
Graph icon	Applies a bar graph to the measure and displays both the measure and its associated bar graph.
Blank box	Does not display the measure or an associated bar graph.

You click the check box next to a measure until it reflects the display mode you want.

If an OLAP report contains a measure that does not appear in the report, the Measure control shows a blank check box. To display the measure, click the check box once. To display the associated bar graph, click the check box again.

Note: The three-state check box is *not* active when you apply Stack Measures to your report. These features are mutually exclusive.

Apply Bar Graphs to Measures Using the Selections Pane

1. From the OLAP selections pane, click the arrow to the left of the Measures control.
2. Click the check box beside each numeric measure to which you want to add a bar graph. The check mark in the box is replaced with the Graph icon.
3. Click **Run**. The new report appears with the associated bar graphs.

Apply Bar Graphs to Measures Using the Control Panel

1. Click the **OLAP** button in the OLAP selections pane to open the OLAP Control Panel. The Measures box appears in the lower right corner.
2. If Stack Measures is applied to the report, click the **Stack Measures** check box to turn off this feature.
3. To apply data visualization bar graphs to a measure, click the check box to the left of the measure.

   To apply data visualization graphs to a non-displaying measure, click the check box twice.

   The check mark in the box is replaced with the Graph icon. This icon indicates that data visualization bar graphs are applied to the measure. (If you have not done so in step two, this also deactivates the Stack Measures feature.)

   You can apply data visualization bar graphs to as many numeric measures as you want.
4. After you select all the measures for which you want to display bar graphs, click **Run**.

   The new report output appears with the associated bar graphs.
5. To continue to modify the report (either data visualization or another OLAP configuration), click the **OLAP** button again.

Remove Bar Graphs Using the Selections Pane or Control Panel

1. From the Measures drop-down list in the OLAP selections pane or the Measures box in the OLAP Control Panel, click the check box for any measure to which you have applied data visualization bar graphs.

   This removes the Graph icon and displays a blank check box indicating that the measure will not appear in the report output when you run the report.
2. To display the measure, click the same check box again. A check mark appears in the box.
3. Click **Run** to display the new report output, where the measure appears without its associated bar graph.

Applying Data Visualization Bar Graphs to Measures Using the Selections Pane

Suppose that you want to associate data visualization bar graphs with the Profit column in the following report in order to represent visually the differences between the Costs for and the Prices of your various Products.

You have created the following OLAP report, as shown in the following image, which displays the report data by Product Name.

PAGE 1

Product ■Name:	Our ⬦ Cost:	⬦ Price:	⬦ Profit
110 VHS-C Camcorder 20 X	744,759.00	1,043,859.00	299100
120 VHS-C Camcorder 40 X	796,943.00	1,227,723.00	430780
150 8MM Camcorder 20 X	1,303,440.00	1,732,489.00	429049
2 Hd VCR LCD Menu	185,889.00	257,939.00	72050
250 8MM Camcorder 40 X	251,520.00	313,614.00	62094
330DX Digital Camera 1024K P	32,437.00	45,477.00	13040
650DL Digital Camcorder 150 X	940,750.00	1,191,175.00	250425
750SL Digital Camcorder 300 X	105,750.00	140,859.00	35109
AR2 35MM Camera 8 X	64,385.00	88,835.00	24450
AR3 35MM Camera 10 X	13,395.00	18,189.00	4794
Combo Player - 4 Hd VCR + DVD	1,166,404.00	1,610,364.00	443960
DVD Upgrade Unit for Cent. VCR	754,909.00	1,080,769.00	325860
QX Portable CD Player	245,916.00	419,796.00	173880
R5 Micro Digital Tape Recorder	297,459.00	383,679.00	86220
ZC Digital PDA - Standard	233,313.00	280,163.00	46850
ZT Digital PDA - Commercial	1,376,805.00	1,968,555.00	591750

To associate data visualization bar graphs with the Profit column:

1. Click the **Measures** drop-down list in the report (or open the OLAP Control Panel (OCP) by clicking the **OLAP** button), as shown in the following image.

The check marks indicate that the measures will appear in the report output.

2. Click the **Profit** check box again. The following images shows the Measures drop-down list in the OCP with the Profit check box selected as a Graph icon.

The Graph icon replaces the check mark. This icon indicates that the measure will appear with its associated bar graph.

3. Click the **Run** button to display the new report output, as shown in the following image, visualizing the Profit values in a bar graph representation.

Product Name:	Our Cost:	Price:	Profit	
110 VHS-C Camcorder 20 X	744,759.00	1,043,859.00	299100	
120 VHS-C Camcorder 40 X	796,943.00	1,227,723.00	430780	
150 8MM Camcorder 20 X	1,303,440.00	1,732,489.00	429049	
2 Hd VCR LCD Menu	185,889.00	257,939.00	72050	
250 8MM Camcorder 40 X	251,520.00	313,614.00	62094	
330DX Digital Camera 1024K P	32,437.00	45,477.00	13040	
650DL Digital Camcorder 150 X	940,750.00	1,191,175.00	250425	
750SL Digital Camcorder 300 X	105,750.00	140,859.00	35109	
AR2 35MM Camera 8 X	64,385.00	88,835.00	24450	
AR3 35MM Camera 10 X	13,395.00	18,189.00	4794	
Combo Player - 4 Hd VCR + DVD	1,166,404.00	1,610,364.00	443960	
DVD Upgrade Unit for Cent. VCR	754,909.00	1,080,769.00	325860	
QX Portable CD Player	245,916.00	419,796.00	173880	
R5 Micro Digital Tape Recorder	297,459.00	383,679.00	86220	
ZC Digital PDA - Standard	233,313.00	280,163.00	46850	
ZT Digital PDA - Commercial	1,376,805.00	1,968,555.00	591750	

Notice that the report now contains a new column to the right of the Profit measure. This column displays a horizontal bar chart comprised of bar graphs that represent the individual data values for the Profit measure.

Display Modes in the OLAP Control Panel

The Measures box, from which you select a display mode, is located in the lower right corner of the Control Panel, as shown in the following image.

The state of each measure's check box determines how the measure appears in the report output. In this illustration:

- The COST and PRICE measures will appear in the report output (check mark in the boxes).
- The Profit measure and its associated bar graph will appear in the report output (Graph icon in the box).

Note that the Stack Measures option is inactive when a bar graph is applied to a measure.

Chapter 9. Using the DB2 Web Query Viewer

Describes how to use the DB2 Web Query Viewer to view long reports.

The DB2 Web Query Viewer uses the On-demand Paging facility. When On-demand paging is enabled, DB2 Web Query saves the bulk of your report to your Web server and delivers one page of report output at a time, decreasing the amount of time you wait for your report to process. The bulk of your report remains on the Web server until you request it or close the Viewer.

The DB2 Web Query Viewer improves your ability to handle long reports by allowing you to view a single page of report output. You can use the Viewer to:

- View single pages of long reports.
- Search for specific pages in a report.
- Search for specific strings of information.
- Deliver a full report to your Web server.

Navigating a Report With the DB2 Web Query Viewer

When you run a report designated for On-Demand Paging, the DB2 Web Query Viewer opens automatically and displays the first page of the report. The Viewer consists of two panes: the Report Pane and the Viewer Control Panel as shown in the following image.

```
PAGE 1


Product  Unit              Order  Order    Ordered
Code     Price  Product   Number  Date       Units

B141     58.00  Hazelnut       1  01/01/96     300
                               2  01/01/96     117
                              16  01/01/96     285
                              17  01/01/96     140
                              31  01/01/96     349
                              46  01/01/96     240
                              61  01/01/96     509
                              76  01/01/96     358
                              91  01/01/96     179
                             106  01/01/96     583
                             121  01/01/96      37
                             136  01/01/96     448
                             151  01/01/96     267
                             166  01/01/96     363
```

Page [1] of 87 🔲🔲 |◀ ◀ ▶ ▶| [_____] 🔍 A=a → ❓ ✕

The Report Pane is the larger pane and contains one page of report output. When you first run a report, the Report Pane contains the first page of report output. The Viewer Control Panel contains the controls that allow you to display specific pages, deliver the entire report to your Web server, and search your document for particular strings of information.

Using the Viewer Control Panel

The Viewer Control Panel, as shown in the following image, (located at the bottom of the window) contains the controls you use to navigate through the report and to search for a string in the report. The Viewer Control Panel's navigational controls allow you to display the next or previous page, the first or last page, or a specific page. You use the searching function to have the Viewer locate a search string you specify within all report pages.

Page [1] of 87 🔲🔲 |◀ ◀ ▶ ▶| [_____] 🔍 A=a → ❓ ✕

Note: When specifying a search string, you must specify the actual number of spaces between characters because HTML displays a single space, even when multiple spaces are used between characters.

Navigate Through a Report

The Viewer Control Panel offers several ways to view pages in your report:

- To display a specific page:

    1. Enter a page number in the Page input box as shown in the following image.

    2. Click Go to Page as shown in the following image.

- To display the previous or the next page in sequence, click **Previous** or **Next** as shown in the following image.

- To display the first or last page of the report, click **First Page** or **Last Page** as shown in the following image.

- To download the entire report to the browser as a single document, click **All Pages** as shown in the following image.

- To close the DB2 Web Query Viewer, click **Close** as shown in the following image.

Searching a Report

The Viewer Control Panel contains controls that offer several ways to search your report. Using the Viewer's search controls, you can select a string of information, such as a phrase that occurs in your report or a group of numbers, and search for each occurrence of that string. You can further customize your search by matching capitalization of words exactly (a case-sensitive search) or by controlling the direction of your search (either forward or backward from your starting point in the report). Use these controls to search your report:

- To perform a case-sensitive search, click **Match Case** as shown in the following image.

- To search backward in a report, click **Search Backward** as shown in the following image.

- To locate a specific string, type the string you want to search for and click **Find** as shown in the following image.

Search the Report

1. Required: Enter the string in the Search input box.
2. Required: Click **Match Case** if you want to perform a case-sensitive search.

   Notice that the DB2 Web Query Viewer displays the Match Case button with a red line across it to indicate that it is active.
3. To begin your search, click:
   a. **Search Backward** to search for the string from the current page back to the first page, or
   b. **Find** to search from the current page to the end of your report.

   The DB2 Web Query Viewer searches the report and underlines the first occurrence of the string.
4. Click **Find** again to search for another occurrence of the string.

Using the Viewer Control Panel to Search

You want to use the Viewer Control Panel to navigate a long report called Coffee Sales to find occurrences of the string "Kona," a type of coffee that you sell. After you run the report, DB2 Web Query displays the first page of the report in the Viewer.

1. To search for sales of Kona, type Kona in the input box and click **Find** as shown in the following image.

PAGE 1

Product Code	Unit Price	Product	Order Number	Order Date	Ordered Units
B141	58.00	Hazelnut	1	01/01/96	300
			2	01/01/96	117
			16	01/01/96	285
			17	01/01/96	140
			31	01/01/96	349
			46	01/01/96	240
			61	01/01/96	509
			76	01/01/96	358
			91	01/01/96	179
			106	01/01/96	583
			121	01/01/96	37
			136	01/01/96	448
			151	01/01/96	267
			166	01/01/96	363
			167	01/01/96	535

Page 1 of 87   Kona

The DB2 Web Query Viewer returns your report with the first occurrence of your search string underlined as shown in the following image.

<table>
<tr><td></td><td></td><td></td><td>4260</td><td>12/01/97</td><td>230</td></tr>
</table>

```
                                4260 12/01/97    230
                                4261 12/01/97    175
                                4274 12/01/97    254
                                4275 12/01/97    194
                                4276 12/01/97    380
                                4289 12/01/97    307
                                4290 12/01/97    427
                                4291 12/01/97    306
                                4304 12/01/97    221
                                4305 12/01/97    396
                                4306 12/01/97    149
                                4307 12/01/97    376
B144    76.00 Kona                 35 01/01/96    471
                                  50 01/01/96    377
                                  80 01/01/96     47
                                 125 01/01/96    510
                                 140 01/01/96     38
                                 155 01/01/96    347
                                 215 02/01/96    472
```

Page 25 of 87   Kona

2. Click **Find** again to locate the next occurrence of Kona.

Creating On-demand Paging Reports

You can create your own On-demand Paging report from Report Assistant. Note that On-demand Paging is not currently available with OLAP-enabled reports.

Create an On-demand Paging Report

1. Required: From the Domains list window, select the Reports folder.
2. Required: Select a report from the list.
3. Click **Report Assistant**.

   Report Assistant opens.
4. Add fields and customize your report. For more information on creating reports with Report Assistant, see Chapter 4, "Creating a Report With Report Assistant," on page 71.
5. Click the **Report Options** tab and check the **On-demand Paging** check box.
6. Click **Run** to view your report.

   DB2 Web Query displays your report within the Viewer.

   Note: The Viewer is launched over a blank browser window. This window is necessary for the operation of the Viewer. It must be closed separately from the Viewer.

7. To return to Report Assistant, close the DB2 Web Query Viewer and close the blank browser page.

8. Close Report Assistant.

   DB2 Web Query prompts you to save your report before exiting.

Creating an On-demand Paging Report

You want to select On-demand Paging for a report on the sale of coffee products called Coffee Sales.

1. Navigate to the Domains view.

2. Open the **Regional Sales** domain.

3. From the Reports folder, expand the **Product Sales** group folder and select **Coffee Sales**.

4. Open **Report Assistant**.

   Report Assistant displays the Fields dialog box with the fields already selected for the report.

5. Select the **Report Options** tab. The Report Options window opens.

6. Check the **On-demand Paging** check box to enable On-demand Paging.

7. Click **Run** to run Coffee Sales immediately.

   DB2 Web Query displays the first page of Coffee Sales in the Viewer as shown in the following image.

PAGE 1

Product Code	Unit Price	Product	Order Number	Order Date	Ordered Units
B141	58.00	Hazelnut	1	01/01/96	300
			2	01/01/96	117
			16	01/01/96	285
			17	01/01/96	140
			31	01/01/96	349
			46	01/01/96	240
			61	01/01/96	509
			76	01/01/96	358
			91	01/01/96	179
			106	01/01/96	583
			121	01/01/96	37
			136	01/01/96	448
			151	01/01/96	267
			166	01/01/96	363

Page 1 of 87

8. Click **Close** on the DB2 Web Query Viewer.

   DB2 Web Query notifies you that the report has been deleted from the server.

Chapter 10. Understanding User Interface Basics

Introduces user interface design in DB2 Web Query and describes the components of a typical user interface.

A user interface for a Web application is a collection of HTML files that display pages in a browser.

You can design a user interface by using the HTML Layout Painter to create HTML forms and procedures in one integrated process. For details, see Designing a User Interface for a Web Application With the HTML Layout Painter.

Note: The HTML Layout Painter is accessible only through the DB2 Web Query Developer Workbench.

User Interface Basics

A user interface is the means by which you enable communication between a user and your application.

A DB2 Web Query user interface allows a user to supply information, choose options, and run reports. A simple one typically consists of:

- **A launch page,** which enables a user to run one or more reports.
- **An HTML display page,** on which a report appears.

User interface design is one of the most creative phases of application development. Developer Workbench provides many tools that support it.

Modify WFSIGNON.HTML

1. Modify WFSIGNON.HTML by using the Developer Studio text editor (TED).
   Note: For details on this editor, see Editing Application Components as Text in Developer Studio.
2. Modify the lines with letters on the left, as explained in the notes following the code.

   ```
       <!DOCTYPE HTML PUBLIC "-//IETF//DTD HTML//EN">
       <html>
       <head>
       <title>WebFOCUS signon screen</title>
       </head>
       <body bgcolor="#FFFFFF" alink="#0000FF">
       <font size=4>
   a. <form action="/ib_apps/WFServlet" method="POST">
       <a href="http://www.informationbuilders.com"><img border="0"
       src="ib_logo_home.gif"   align="left" placement="break"
       id="image_AED3B8DFBD244B27A511729EB6D221CA"  />
   ```

   If a manager selects **Midwest** and clicks **Run**, the report appears.

```
Coffee Sales for Midwest Region
Unit Sales        Dollar Sales
  332777              4178513
```

At minimum, a launch page is a simple form that contains a Run and Clear button, allowing the user to run a report. More often, it is a form, like the one shown, containing controls that prompt for user input.

Controls include text boxes, check boxes, radio buttons, drop-down menus, push buttons, and hyperlinks. A launch page usually displays default values or valid values to help the user provide input. The Reporting Server uses the input to process the procedure and return the desired output.

When a launch page is complete, deploy it to the Web server using the Deploy Wizard. Supporting files such as procedures are deployed to the Reporting Server. Once the files are stored on the appropriate servers, users can access the launch page by typing the URL in the browser.

Adding a Control to a Form

A control prompts a user for a value required by a procedure. You can add a single control, or a combination of controls, to a form on a launch page.

You must add a Submit button. Submitting a form calls DB2 Web Query and passes the supplied values to the procedure.

DB2 Web Query Controls

The following table describes types of controls.

Control	Enables a user to...
One-Line Text Box	Enter a single value for a variable, such as user name or password on a custom logon page.
Radio Button	Choose one value from a group.
Check Box	Choose multiple values.
Drop-Down List	Choose a value from a drop-down list. You can add text to describe the values.
Multi-Select Drop-Down List	Choose multiple values from a drop-down list. Allows a user to report on several field values at one time.
Dynamic Multi-Select Drop-Down List	Choose multiple values from a drop-down list populated by field values retrieved live from a data source.
Text Area	Enter an ad hoc report request.

Control	Enables a user to...
Submit Button	Run a report. You must include a Submit button on a form. You may optionally include a button that clears the form of current values and restores the defaults.
Push Button	Trigger an action.
Hyperlink	Run a procedure using fixed values.

Types of Launch Pages

A launch page can be a single page or a frameset. A single page is a one-page display that is replaced by report output when a user submits a request. A frameset is a multiple-page display, and any page (frame) can be replaced by report output.

Ways to Create a Launch Page

A launch page calls the DB2 Web Query Client to pass the name of a report to run, with any required values.

You can create a launch page using the HTML Layout Painter. The HTML Layout Painter allows you to create a launch and display page in an integrated process within Developer Workbench. See Designing a User Interface for a Web Application With the HTML Layout Painter.

About an HTML Display Page

Report output from a Reporting Server appears on an HTML page in a browser. You have many design options for report display. For example, you can display a report on a page by itself or in a frame on a page, or you can design a custom page with multiple reports.

Your application will probably include different types of reports, which may influence the display pages you design. Some types of reports include:

- **Dynamic.** A report created and formatted at the time a user requests it. This type of report displays the most current information from live data retrieval.
- **Drill-Down.** A dynamic, multilevel report consisting of a parent (summary) component and a child (detailed) component.
- **Static.** HTML file created and formatted before a user requests it. Ready for access without live data retrieval. Called by HTML code (A HREF).

You can create a report using the Report Assistant or Power Painter.

Chapter 11. Creating a Reporting Procedure With SQL Report Wizard

Describes how to use the SQL Report Wizard.

The SQL Report Wizard assists you with SQL passthru, which allows you to execute SQL code that retrieves data from an RDBMS. You can use the resulting extract file in the Report Assistant or the Graph Assistant.

Note: The SQL Report Wizard is accessible only through the DB2 Web Query Developer Workbench.

Using the SQL Report Wizard

The SQL Report Wizard supports DB2 cli engines.

Include SQL Commands From an External .sql File

1. Access the SQL Report Wizard by doing one of the following:

   In the Explorer, right-click a group folder under the Reports folder and select **New**, **Procedure**. In the New Report dialog box, enter a file name and then select **SQL Report Wizard** from the Create with drop-down list. Click **Create**.

   The SQL Report Wizard - Welcome window opens.
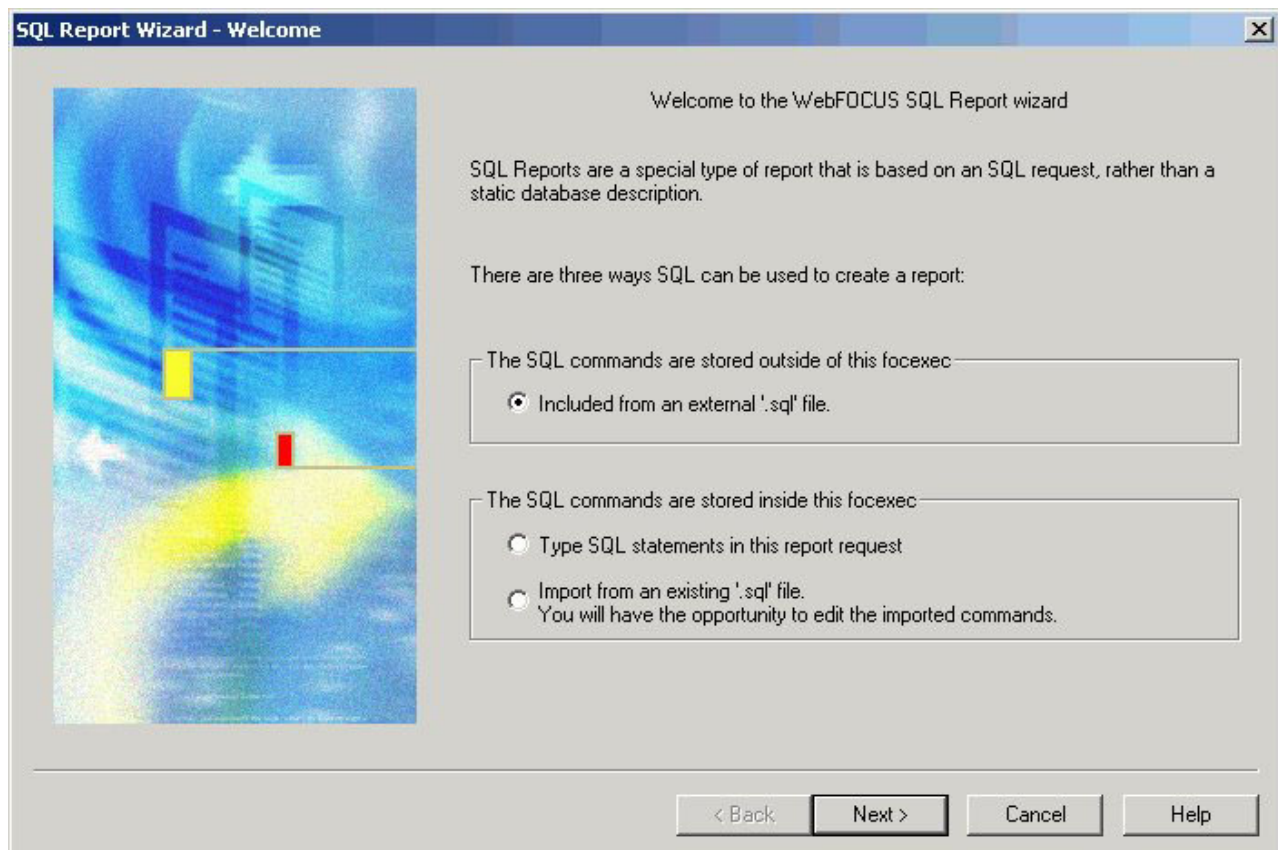
2. Click the option button next to **Included from an external '.sql' file**. This enables you to browse and select external procedures that exist in the DB2 Web Query repository. This enables sites to leverage pre-existing SQL procedures.

3. Required: Click **Next** to see the SQL Report Wizard - Data access information window.
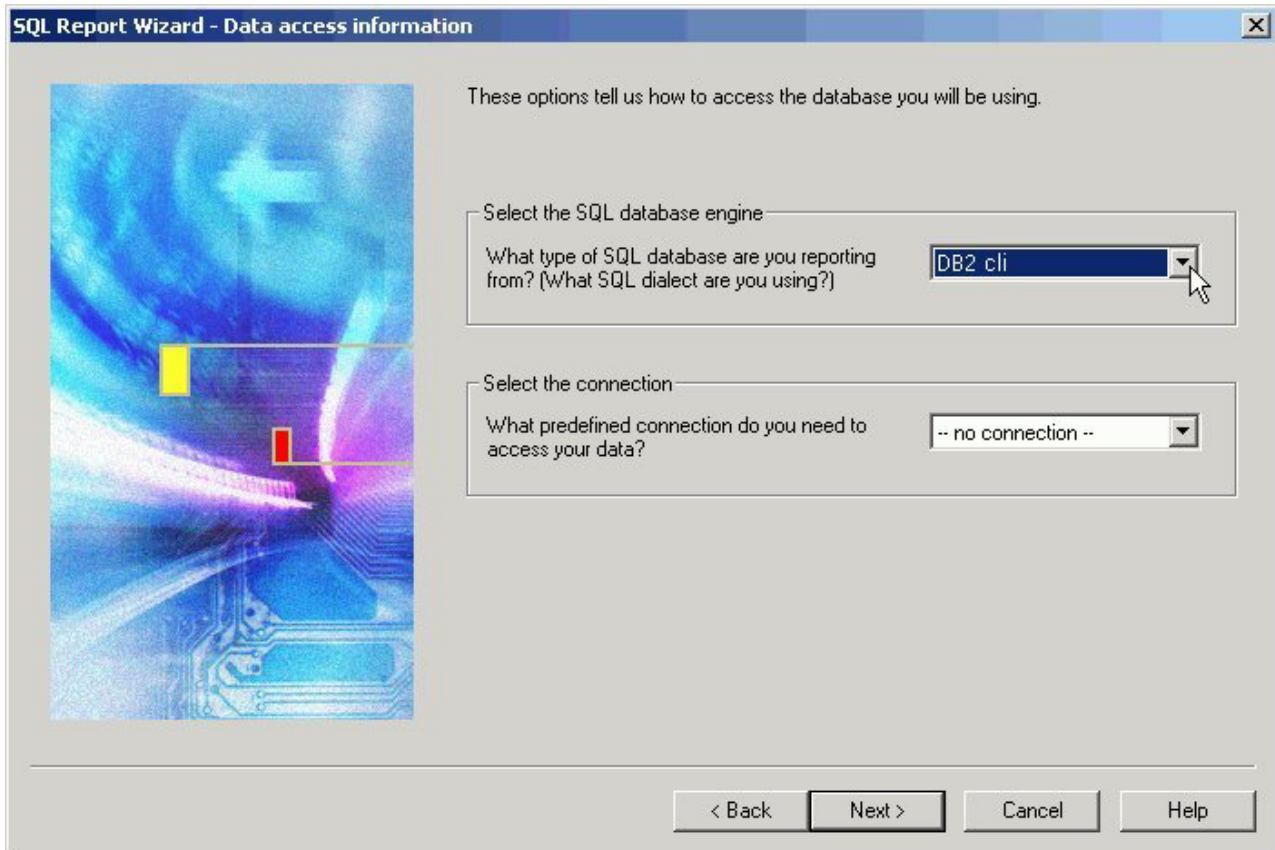
4. In the Select the SQL database engine area, select a database engine from the drop-down list. The list consists of available engines in the edasprof.prf file.

5. In the Select the connection area, choose a connection from the drop-down list generated from the engine that you selected. You can choose the default value, which is the first connection in the edasprof.prf file for the selected engine, or choose another connection defined in edasprof.prf.

6. Click **Next** to see the SQL Report Wizard - Include external SQL file window.
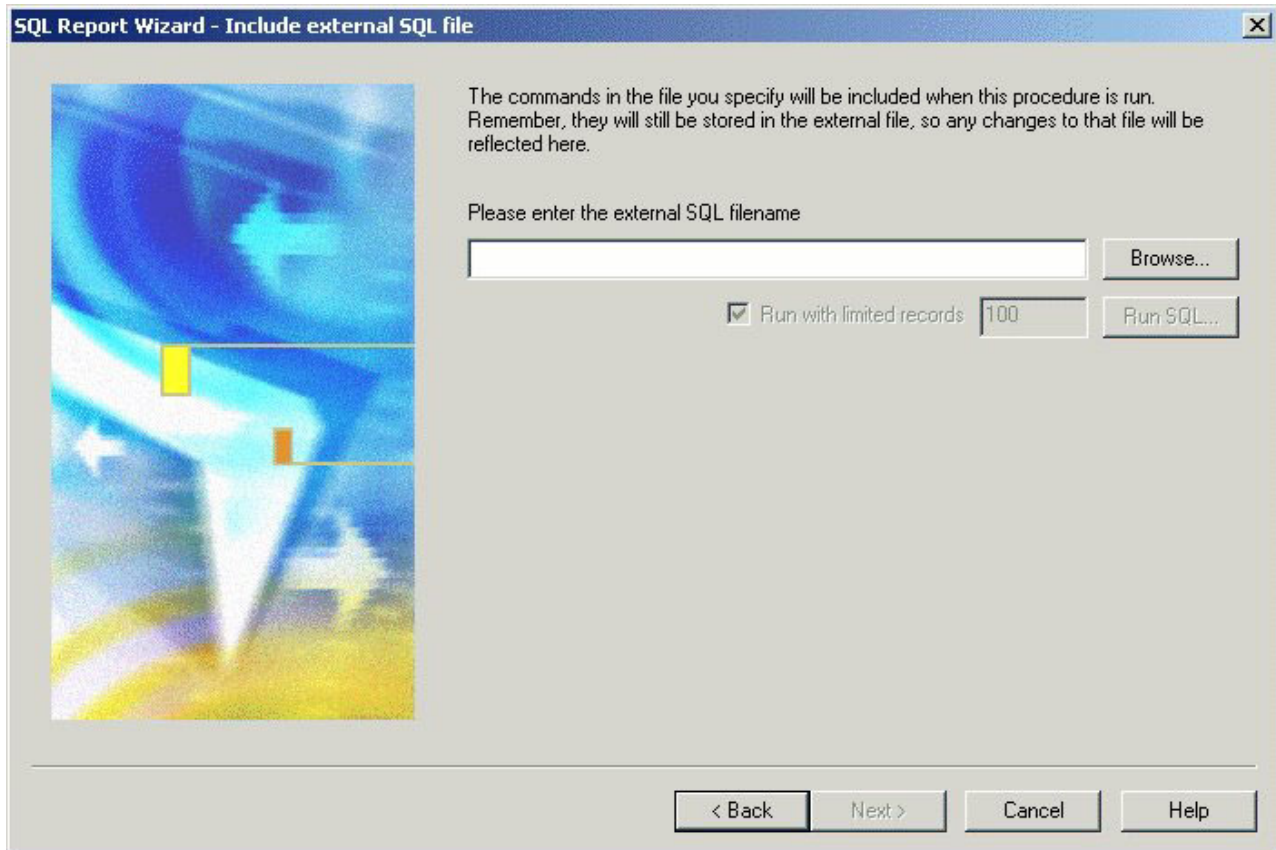
7. Enter the external SQL file name in the field, or click **Browse** to select it.

8. Optionally, you can run with limited records by clicking **Run SQL**. By default, the **Run with limited records** check box is selected so you can test your procedure with a readlimit if the engine supports it. There is a field box next to the check box in which you can enter the number of records to be read. 100 is the default limit.

9. Click **Next** to see the SQL Report Wizard - Summary of SQL options window.

   Do one of the following:

   • To create a report, select the **Create Report** option button. This option is selected by default.

   • To create a graph, select the **Create Graph** option button.

10. Click **Finish** to run the SQL procedure. The Report Assistant or Graph Assistant opens. When you have completed the procedure, you can run it from the group folder in the Explorer view.

Pass SQL Commands to the RDBMS Using SQL Passthru

1. Required: Access the SQL Report Wizard by doing one of the following:

   In the Explorer, right-click a group folder under the Reports folder and select **New**, **Procedure**. In the New Report dialog box, enter a file name and then select **SQL Report Wizard** from the Create with drop-down list. Click **Create**.

   The SQL Report Wizard - Welcome window opens.

2. Click the option button next to **Type SQL statements in the report request**. This enables you to enter SQL commands that will be passed on to the RDBMS with the SQL Passthru feature.

3. Click **Next** to see the SQL Report Wizard - Data access information window.

4. In the Select the SQL database engine area, select a database engine from the drop-down list. The list consists of available engines in the edasprof.prf file.

5. In the Select the connection area, choose a connection from the drop-down list generated from the engine that you selected. You can choose the default value, which is the first connection in the edasprof.prf file for the selected engine, or choose another connection defined in edasprof.prf.

6. Click **Next** to see the SQL Report Wizard - Enter SQL statements window.
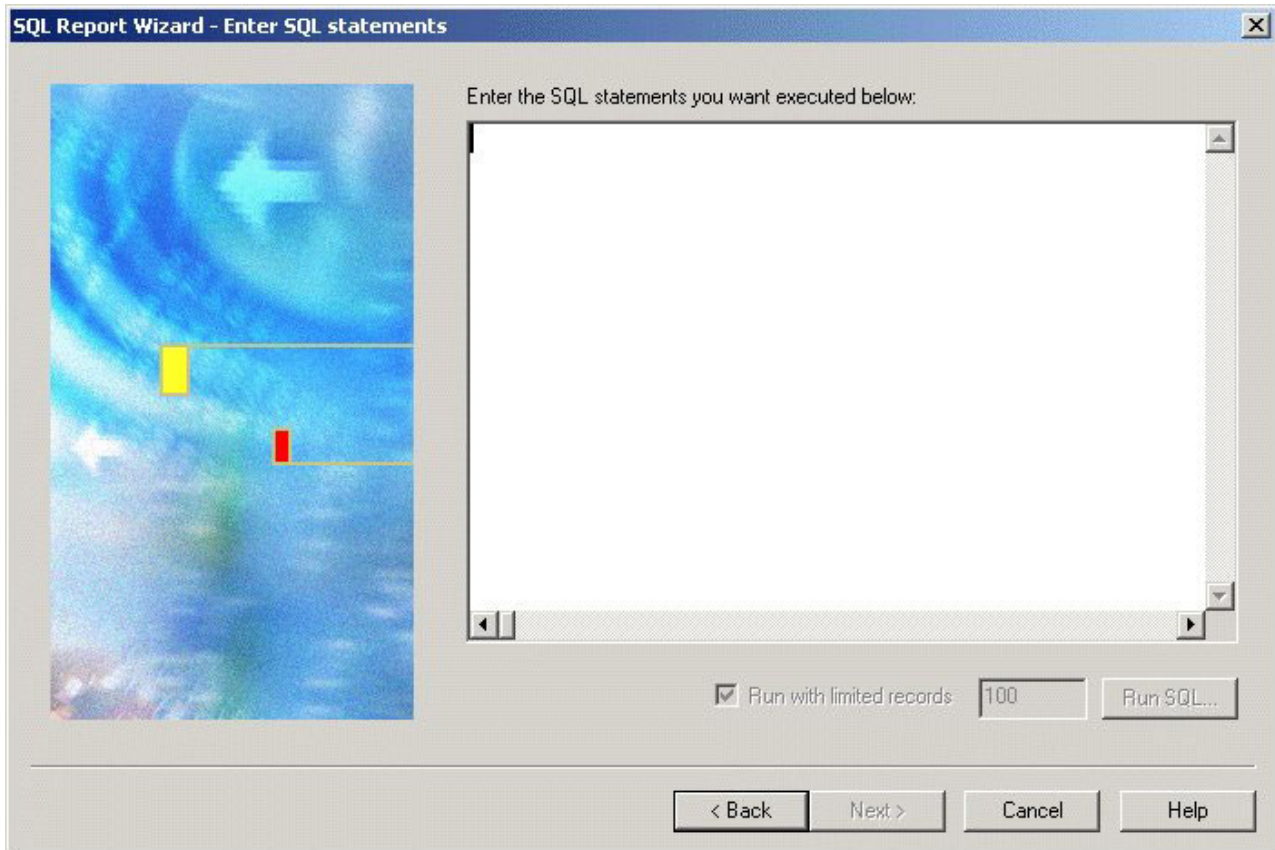
7. In the field box, type the SQL statements you want to pass to the RDBMS.

8. Optionally, you can run with limited records by clicking **Run SQL**. By default, the **Run with limited records** check box is selected so you can test your procedure with a read limit if the engine supports it. There is a field box next to the check box in which you can enter the number of records to be read. 100 is the default limit.

9. Click **Run SQL** to run your report.

10. Click **Next** to see the SQL Report Wizard - Summary of SQL options window.

    Do one of the following:

    - To create a report, select the **Create Report** option button. This option is selected by default.

    - To create a graph, select the **Create Graph** option button.

11. Click **Finish** to run the SQL procedure.The Report Assistant or Graph Assistant opens. When you have completed the procedure, you can run it from the group folder in the Explorer view.

Import SQL Commands From an Existing .sql File

1. Access the SQL Report Wizard by doing one of the following:

   In the Explorer, right-click a group folder under the Reports folder and select **New**, **Procedure**. In the New Report dialog box enter a file name and then select **SQL Report Wizard** from the Create with drop-down list. Click **Open**.

   The SQL Report Wizard - Welcome window opens.

2. Click the option button next to **Import from an existing .sql file**. This enables you to modify SQL code after importing it from an external file to the procedure being built. It enables you to modify the request using bits of code.

3. Click **Next** to see the SQL Report Wizard - Data access information window.

4. In the Select the SQL database engine area, select a database engine from the drop-down list. The list consists of available engines in the edasprof.prf file.

5. In the Select the connection area, choose a connection from the drop-down list generated from the engine that you selected. You can choose the default value, which is the first connection in the edasprof.prf file for the selected engine, or choose other connections defined in edasprof.prf.

6. Click **Next** to see the SQL Report Wizard - Import external SQL file window.
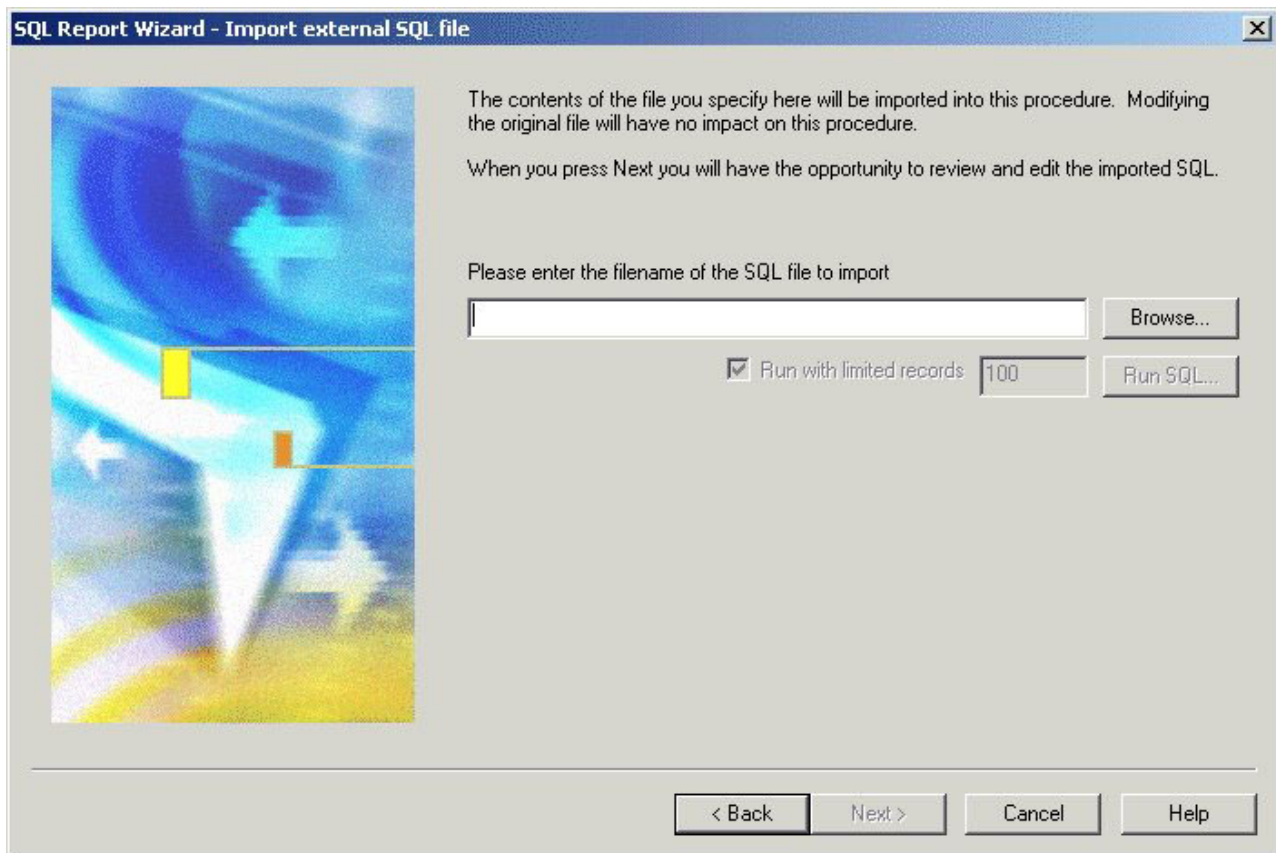
7. Type the SQL file name that you want to import or click **Browse** to select it.

8. Optionally, you can run with limited records by clicking **Run SQL**. By default, the **Run with limited records** check box is selected so you can test your procedure with a readlimit if the engine supports it. There is a field box next to the check box in which you can enter the number of records to be read. 100 is the default limit.

9. Click **Run SQL** to run your report.

10. Click **Next** to see the SQL Report Wizard - Enter SQL statements window. You can edit the imported SQL code if necessary.

11. Click **Run SQL** to run your report.

12. Click **Next** to see the SQL Report Wizard - Summary of SQL options window.

    Do one of the following:

    - To create a report, select the **Create Report** option button. This option is selected by default.
    - To create a graph, select the **Create Graph** option button.

13. Click **Finish** to run the SQL procedure. The Report Assistant or Graph Assistant opens. When you have completed the procedure, you can run it from the group folder in the Explorer view.

Chapter 12. Designing a User Interface for a Web Application With the HTML Layout Painter

Describes how to create a user interface in the local development environment using the HTML Layout Painter.

The HTML Layout Painter allows you to graphically create an HTML page that incorporates forms, reports, graphs, and Web objects. You can also directly edit HTML code in the HTML Editor or a third party editor. The HTML Layout Painter is fully integrated with JavaScript and Cascading Style Sheets (CSS).

Note: The HTML Layout Painter does not support OLAP-enabled reports.The HTML Layout Painter is accessible only through the DB2 Web Query Developer Workbench.

Uses for the HTML Layout Painter

You can do the following when creating an HTML layout:

- Build an HTML launch page. The HTML Layout Painter allows you to add push buttons, hyperlinks, and other objects that launch other DB2 Web Query reports in your application.
- Create a launch page for one or more reports that contain parameters.
- Create a complete report by adding multiple reports and graphs into a single HTML formatted report.
- Create an advanced report layout by including images, frames, and other Web elements. You can change the location, size, and properties of all objects in your layout.
- Directly edit an HTML page in the HTML Editor or in a third party editor.
- Set background, font, and other properties in the Style Composer tool.

Getting Started With the HTML Layout Painter

Note: The HTML Layout Painter does not support OLAP-enabled reports. If you execute a report from the HTML Layout Painter with this option, the output window will not display the OLAP controls and you will receive a scripting error. In order to execute this type of report, you must use a frame. For more information, see Adding a Frame to the Layout.

Access the HTML Layout Painter

1. Required: Create a new HTML file by completing one of these actions:
   - With the HTML folder highlighted, select **New** from the File menu, then select **HTML file**.

     or

   - Click **Layout Reports and Graphs** in the QuickLinks pane.

   The Add HTML File dialog box opens.
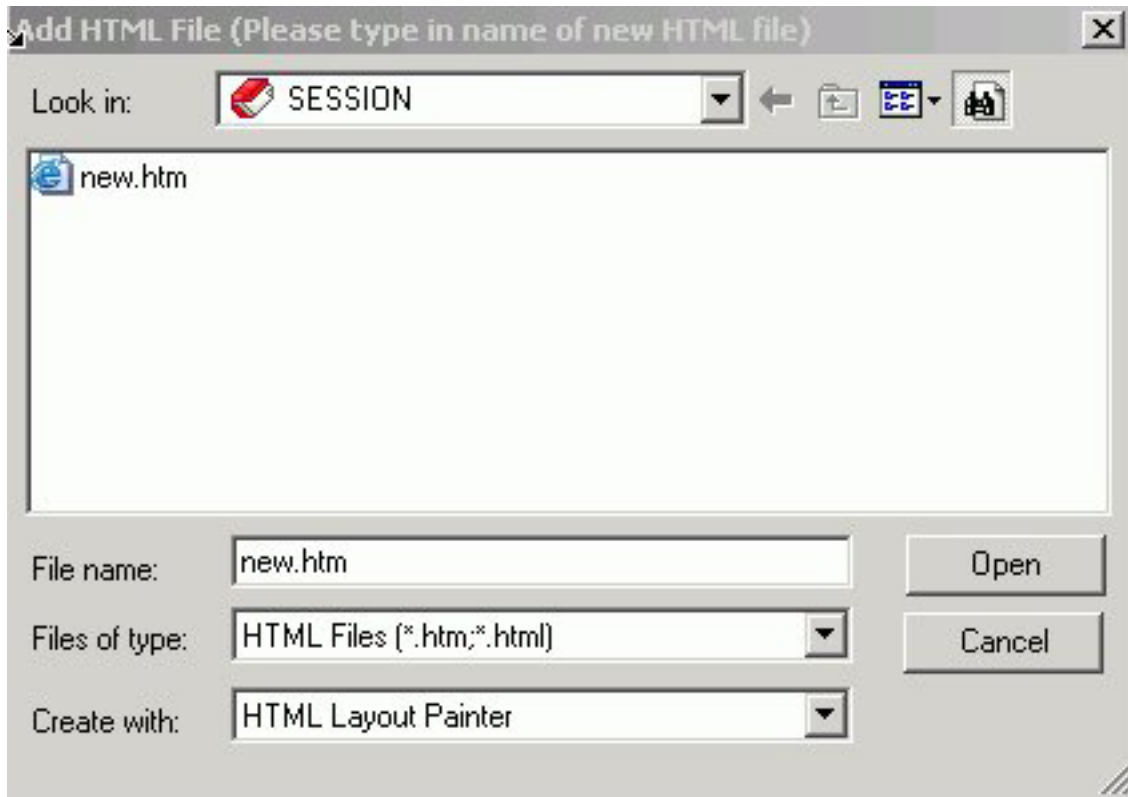
2. Enter a name for the new procedure in the File name field.
3. Select **HTML Layout Painter** from the Create with drop-down list.
4. Click **Open**.
   The HTML Layout Painter opens.

HTML Layout Painter Windows and Toolbars

The following image is an example of the HTML Layout Painter.
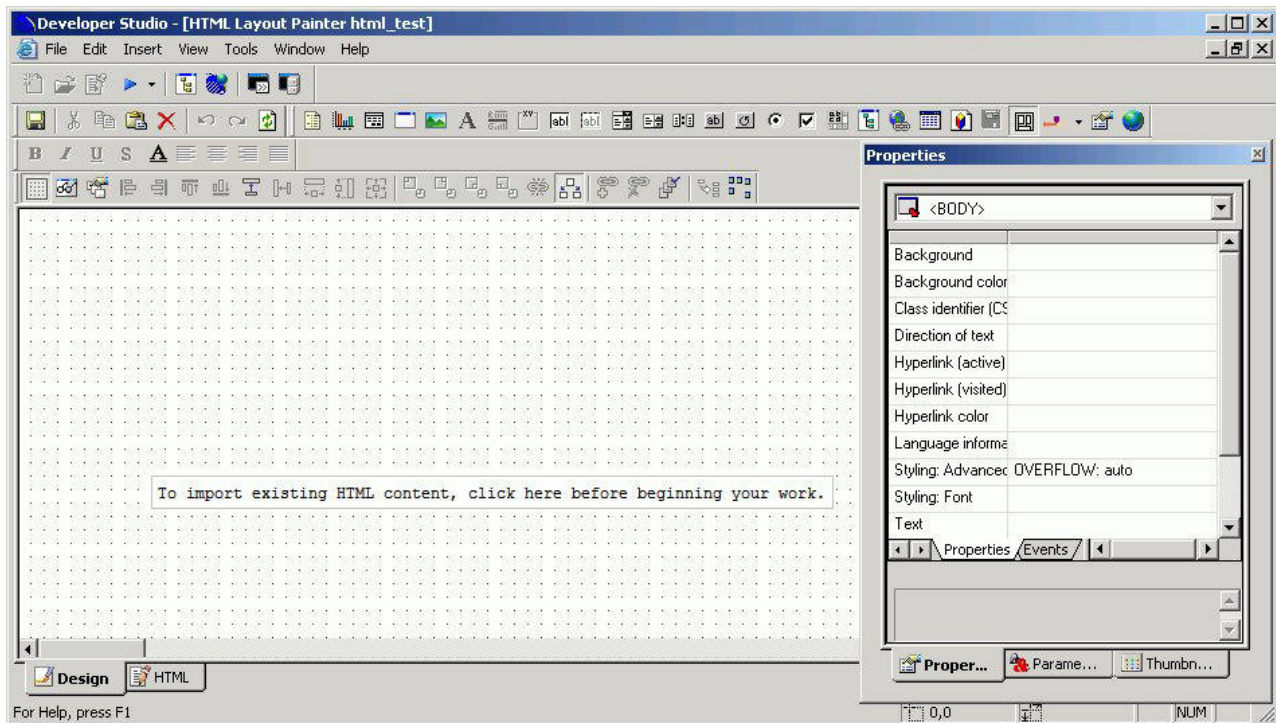
The main elements of the HTML Layout Painter are:

Menu Bar

Displays pull-down menus for the HTML Layout Painter.

Developer Workbench

Displays tool buttons such as Open and Run.

Standard Toolbar

Displays buttons such as Cut and Paste that allow you edit the layout.

Components Toolbar

Contains buttons that add objects and form controls to the layout.

Positioning Toolbar

Contains buttons that control the appearance of the layout. For details, see Positioning Toolbar.

Formatting Toolbar

Displays buttons that format and align text when using a text element in the layout.

Properties Window

Displays the properties and events of objects in the layout. For details, see Working With the Properties Window.

Events Tab

Displays the events associated with objects in the layout. For details, see Working With the Properties Window.

QuickLinks Window

Displays links to information on Help, configuration options, metadata creation, and reports and procedures.

Parameters Tab

Displays information about the parameters in your report or graph. For details, see Working With the Parameters Tab in the Properties Window.

Thumbnails Tab

Select an image from the Thumbnails tab to jump to the selected object in the HTML layout.

Standard Toolbar

The Standard toolbar contains the following buttons:

Button	Description
	Saves the existing procedure (.fex) and HTML files to the current project. This button is grayed out once the layout is saved and no new changes have been made. When a change has been made to the layout, the Save button is active until the report is saved again.
	Removes the highlighted object(s) and saves it to the clipboard.
	Copies the highlighted object(s) to the clipboard.
	Pastes the object(s) to the specified location.
	Deletes the highlighted object(s).
	Resets the layout by reversing the last action performed.
	Repeats the last action performed.
	The Refresh All option enables you view any edits or changes that you made to your document. Refresh All reloads all pages and reruns the reports and graphs in Design View. **Note:** Refresh also shows changes made to reports and graphs that are referenced in your document.

Components Toolbar

The Components toolbar contains the following buttons:

Button	Description
	Inserts a report object to the layout.

Button	Description
	Inserts a graph object to the layout. **Tip:** You may right-click a graph object to open the Advanced Graph Assistant.
	Inserts a placeholder for a form control.
	Inserts a placeholder for an IFRAME. An IFRAME can have its own URL and contain HTML content, and it can be a placeholder for a drill-down report.
	Inserts a placeholder for an image.
	Inserts text.
	Inserts a line.
	Adds a group box.
	Adds a text box.
	Adds a hidden text box.
	Adds a drop-down list.
	Adds a list box (a drop-down list that allows multiple selections).
	Inserts a placeholder for a double list parameter control.
	Adds a push button.
	Adds a reset button.
	Adds a radio button.
	Adds a check box list.
	Inserts a text area.
	Inserts a tree control.
	Inserts a hyperlink.

Button	Description
	Adds a calendar.
	Adds an ActiveX control.
	Saves a selection.
	Inserts a panel to group objects together. The panel is invisible at run time.
	Inserts a slider parameter control bar.
	Inserts a tab control.

Formatting Toolbar

The Formatting toolbar contains options that can be applied to individual strings of text as well as to the entire text element; with the exception of the alignment options. The alignment options can only be applied to the text element.

For more information about using text elements, see Adding Text to the Layout.

Button	Description
B	Applies bold formatting to the text.
I	Applies italic formatting to the text.
U	Applies underline formatting to the text
S	Applies superscript typography to the text.
A	Font style opens the Font dialog box where you can set the Font, Font Style, Size, Color, and Effect of the text.
	Aligns the text element to the left.
	Aligns the text element to the center.
	Aligns the text element to the right.
	Aligns the text to fill the width of the text element.

Creating a Report Page Layout

You can use the HTML Layout Painter to create an HTML page that launches and displays your report or graph. You can add elements to the HTML page including a report or graph, text, form controls, and so on.

You can also set properties for the HTML page in the HTML Layout Painter. For details, see Setting HTML Page Properties.

Adding a Report or Graph to the HTML Layout

You can add a report or graph to the HTML Layout that will display when you run the procedure. You can add an existing report or graph that resides on an available server.

You can also include parameters in a report whose values can be assigned with forms that are added with the HTML Layout Painter. For information, see Using Form Controls to Supply Parameter Values.

You can set the graphic used as a placeholder for a report or graph in the layout using the Settings dialog box. For details, see Set Page Properties.

Add an Existing Report or Graph to a Layout

1. Insert a report or graph object by doing one of the following:
   - Click the **Report** or **Graph** button from the Components toolbar.

     The cursor changes into a crosshair. Click and drag the crosshair to create a report or graph object and adjust it to the size you want.

     A report or graph object is created in the layout and assigned the name report(*n*) or graph(*n*), where (*n*) is a number. The object will appear in gray and white to indicate that the placeholder does not have a report or graph associated with it. Once a report or graph is associated with the object, the object displays the contents of the report or graph.
   - From the Insert menu, select **Import Existing Procedure**.

     Note: If you want to access the Manage Layout dialog box, choose this method.
   - Right-click in the layout and select **New Report** or **New Graph** from the pop-up menu.
2. Do one of the following:

   For a report:

   Right-click the report object and select **Import existing report**.

   For a graph:

   Right-click the graph object and select **Import existing graph**.

   The Get source file dialog box opens.

   Enter the name of the procedure you want to add to the layout.
3. Click **Open**.

   The report or graph object appears in the Design View of the HTML Layout Painter.
4. Optionally, change the properties by adjusting the properties displayed in the Properties tab of the Properties window. For details, see Report Properties in the Properties Window.

Report Properties in the Properties Window

When a report is selected, the Properties tab in the Properties window contains options that control the properties of your report.
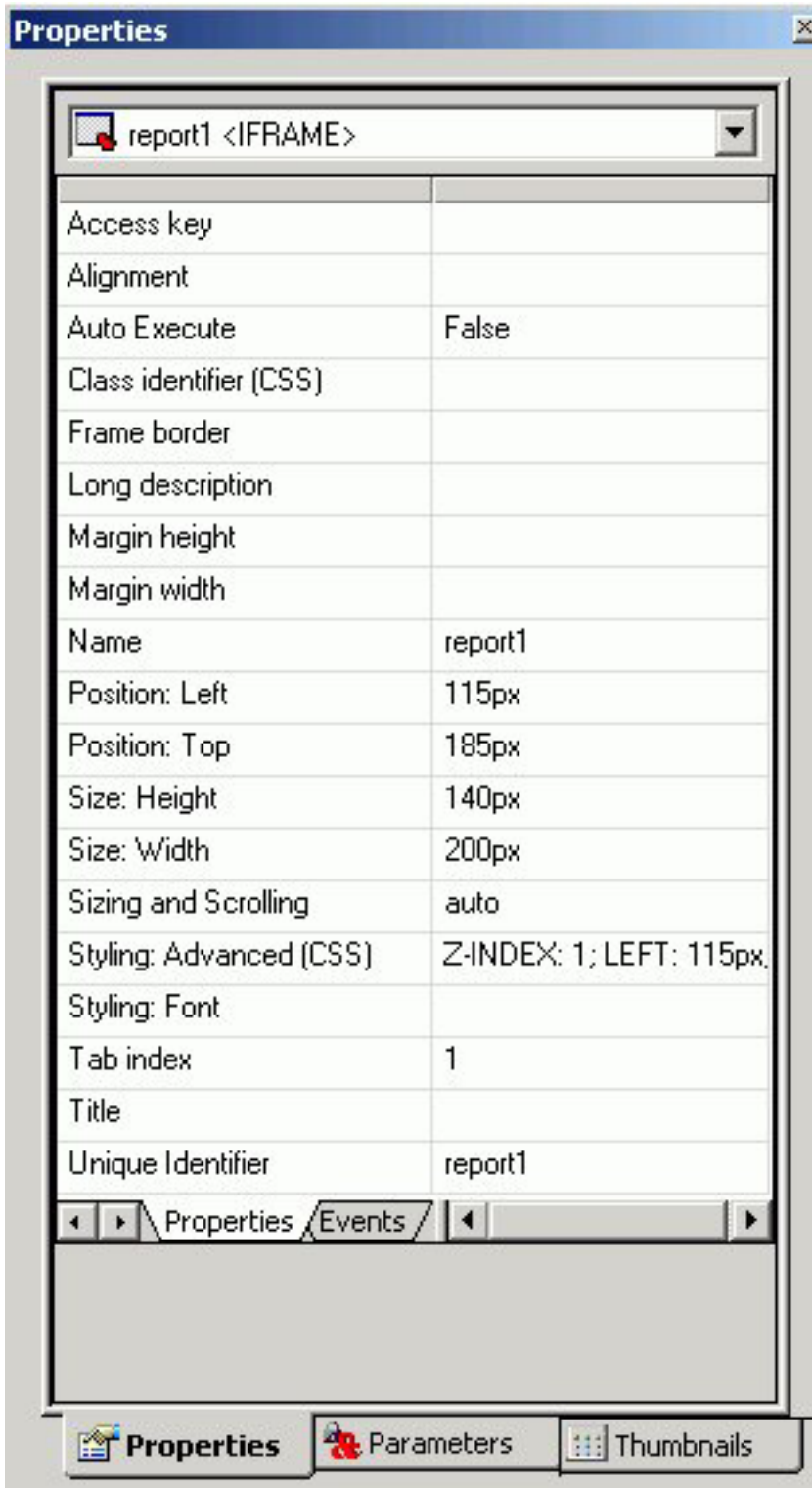
Properties ⊠

report1 <IFRAME> ▼

Access key	
Alignment	
Auto Execute	False
Class identifier (CSS)	
Frame border	
Long description	
Margin height	
Margin width	
Name	report1
Position: Left	115px
Position: Top	185px
Size: Height	140px
Size: Width	200px
Sizing and Scrolling	auto
Styling: Advanced (CSS)	Z-INDEX: 1; LEFT: 115px.
Styling: Font	
Tab index	1
Title	
Unique Identifier	report1

◄ ► Properties / Events / | ◄ ► |

📝 **Properties**   👥 Parameters   ⊞ Thumbnails

Click a property to display a description of the selected property at the bottom of the Properties window.

Graph Properties in the Properties Window

When a graph is selected, the Properties tab in the Properties window contains options that control the properties of your graph.
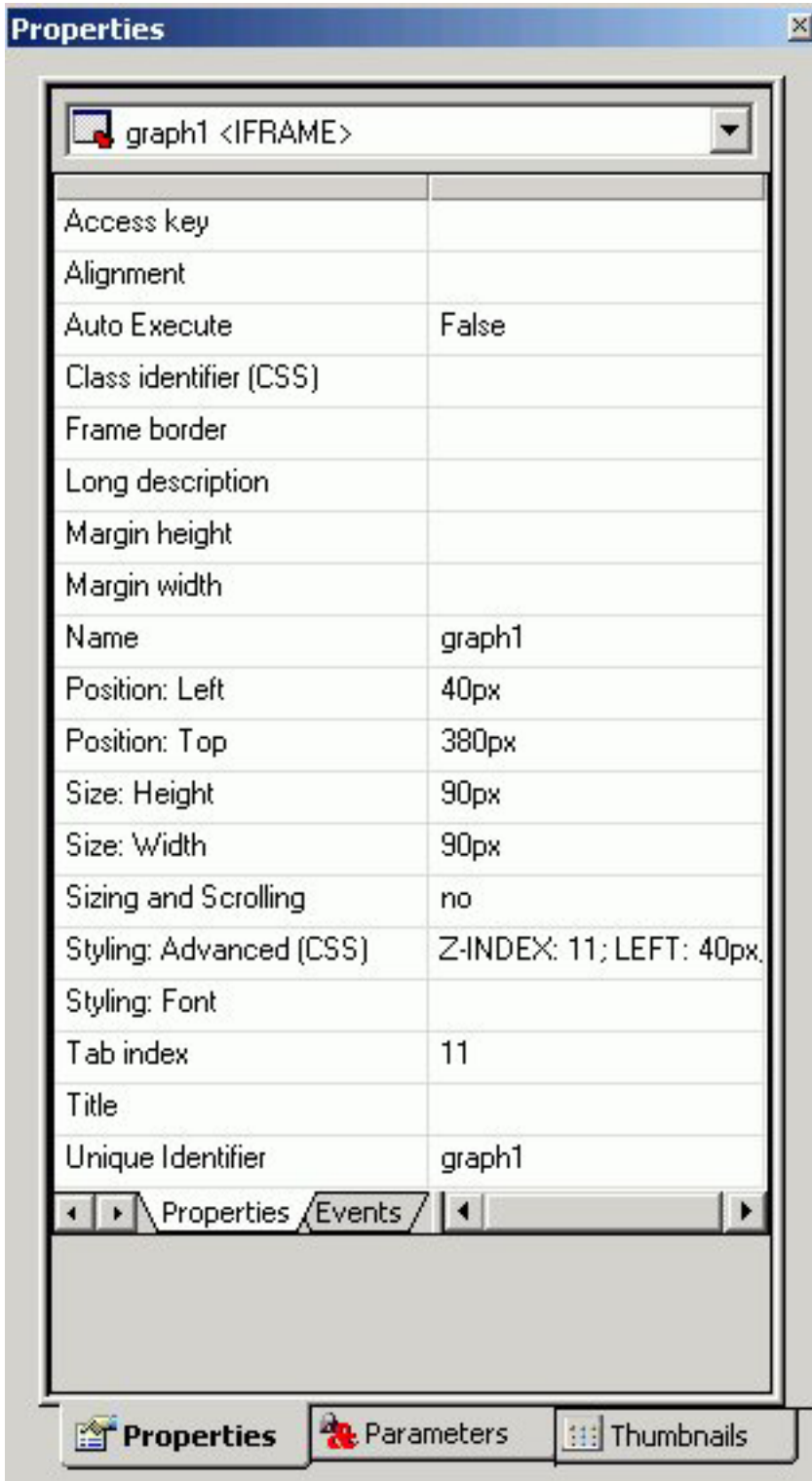
Properties	
graph1 <IFRAME>	
Access key	
Alignment	
Auto Execute	False
Class identifier (CSS)	
Frame border	
Long description	
Margin height	
Margin width	
Name	graph1
Position: Left	40px
Position: Top	380px
Size: Height	90px
Size: Width	90px
Sizing and Scrolling	no
Styling: Advanced (CSS)	Z-INDEX: 11; LEFT: 40px.
Styling: Font	
Tab index	11
Title	
Unique Identifier	graph1

Properties / Events /

Properties   Parameters   Thumbnails

Click a property to display a description of the selected property at the bottom of the Properties window.

Selecting Components to Import Into a Procedure

You can select to include or exclude components to be imported into your procedure with the Manage Layout dialog box. The Manage Layout dialog box allows you to select from components of your request such as TABLE, SET, and GRAPH.

If you have several requests in one procedure, you can use the Manage Layout dialog box to suppress the display of individual TABLE, SET, and GRAPH components (for example, temporary HOLD files). By default, the last TABLE is set to display.

Select Components to Import Into a Procedure

1. Select **Import Existing Procedure** from the Insert menu.

   The Get source file dialog box opens.
2. Select a File name and click **Open**.

   The Manage Layout dialog box opens. For details, see Manage Layout Dialog Box.

Manage Layout Dialog Box

The following image is the Manage Layout dialog box.
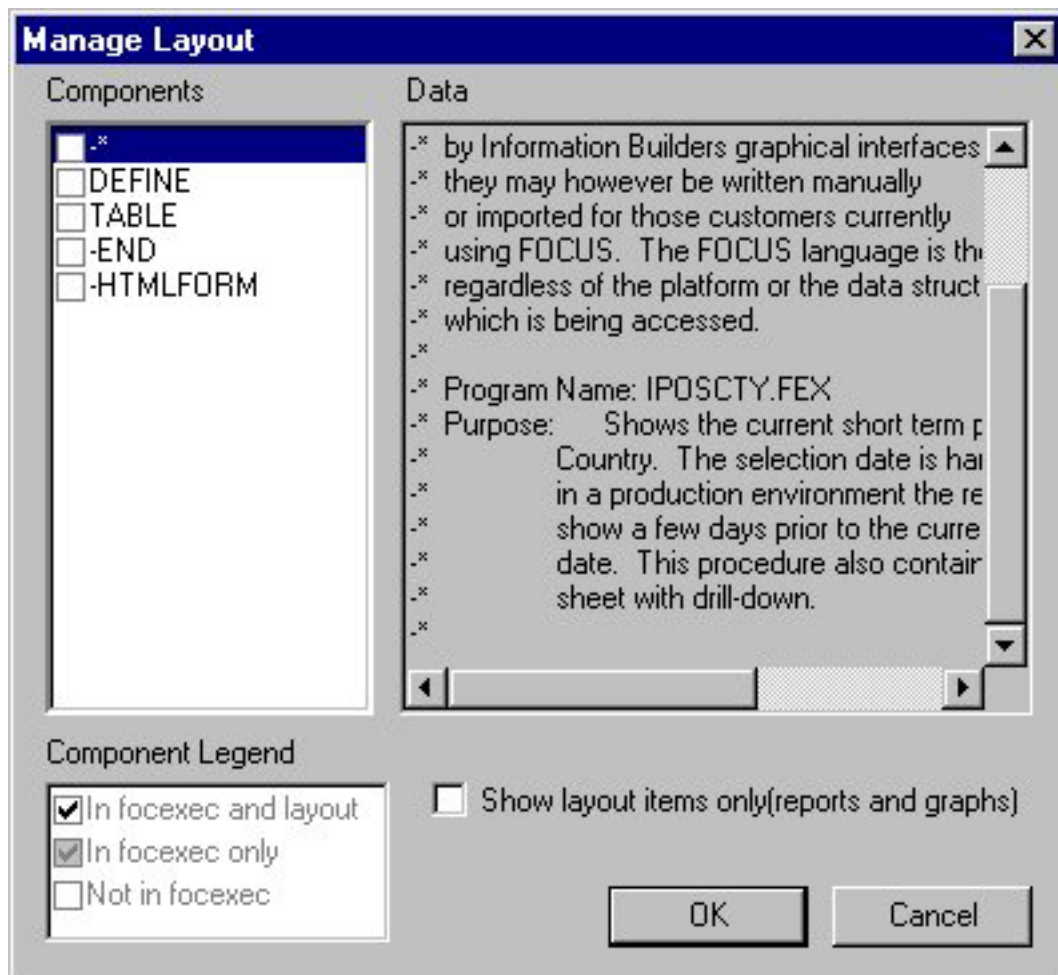
The Manage Layout dialog box contains the following fields/options:

Components

> Displays all the components (SET, DEFINE, etc.) in the request.

Data

> Displays the contents of the selected component.

Component Legend

> Determines whether the request is included in the procedure, outside of the procedure, or in the procedure and layout.

> **In focexec and layout**

>> Activates the component and displays the component in the layout.

> **In focexec only**

>> Deactivates the component and does not display the component in the layout. This is a good method for hiding report requests that produce temporary files.

> **Not in focexec**

>> Removes the component from the procedure.

Show layout items only (reports and graphs)

> Displays the layout's reports and graphs in the Data field.

> This check box only displays when you access the Manage Layout dialog box from the Insert menu. When accessing it from the pop-up menu, only the report or graph component displays, and the check box does not apply.

Adding a Frame to the Layout

You can use a frame to embed additional Web sources or run reports. You can also use a frame as the output location or target for a drill-down report. You can also use a frame to run a table of contents report, an OLAP report, a PDF report, or an Excel report.

Add a Frame to the Layout

1. Required: Insert a frame by doing one of the following:
   - Click the **Frame** button.
   - From the Insert menu, select **Frame**.

   The cursor changes into a crosshair.

2. Click and drag the crosshair to create a frame and adjust it to the size you want.

   A frame is created in the layout and assigned the name iframe*n*, where *n* is a number.

3. Optionally, change the frame's properties by adjusting the properties displayed in the Properties tab of the Properties window. For details, see Frame Properties in the Properties Window.

Assign a URL, HTML Form, or Report to a Frame

1. Insert a frame by doing one of the following:
   - Click the **Frame** button.
   - From the Insert menu, select **Frame**.

   The cursor changes into a crosshair.

2. Click and drag the crosshair to create a frame and adjust it to the size you want.

   A frame is created in the layout and assigned the name iframe*(n)*, where *(n)* is a number.
3. Right-click the frame, and select **Frame Properties** from the pop-up menu.

   The Hyperlink Properties dialog box opens.
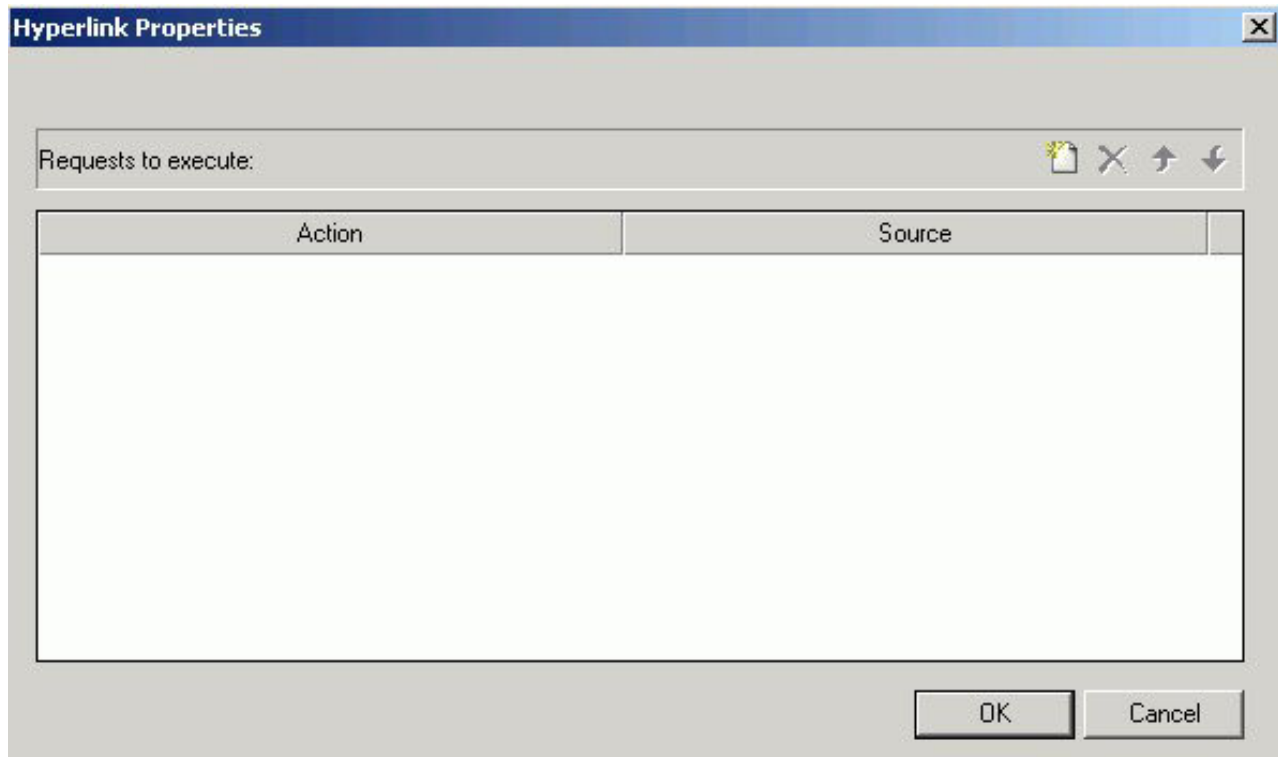
4. Click the **New** icon to create a new action. From the drop down list in the Action field you can select:
   - **URL:** To assign a URL address to the frame, select **URL** and enter the fully qualified URL in the Source field.
   - **HTML File:** To assign an HTML file to the frame, click select **HTML File** and enter the file's location in the Source field, or click the browse **(...)** button to browse to the location. An HTML file refers to the HTML files in your application.
   - **External Procedure:** To assign an external procedure to the frame, select **External Procedure** and enter the procedure's location in the Source field, or click the browse **(...)** button to browse to the location.
5. Click **OK**.
6. Optionally, change the frame's properties by adjusting the properties displayed in the Properties tab of the Properties window. For details, see Frame Properties in the Properties Window.

Show/Hide a Frame in the Layout

1. Required: From the HTML Layout Painter, use form controls to supply parameter values for a report.

   Note: A report with parameters requires that you to select values (at run time) in order to generate the output.

2. Click the frame (report object) and select **False** from the Auto Execute drop-down list in the Properties tab of the Properties window.
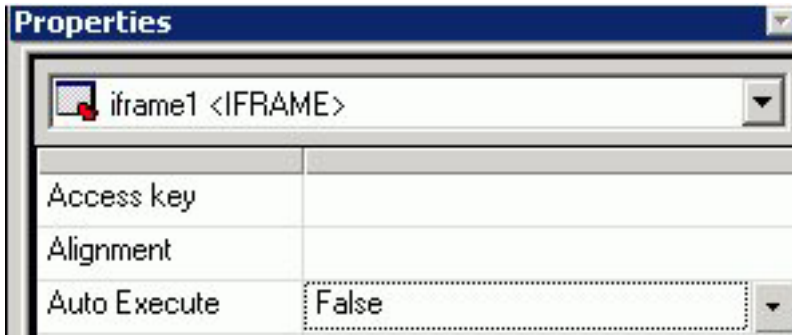
Note: False is the default Auto Execute option for reports with parameters.

3. Right-click the frame (report object) in the Design View and select **Style** from the context menu.
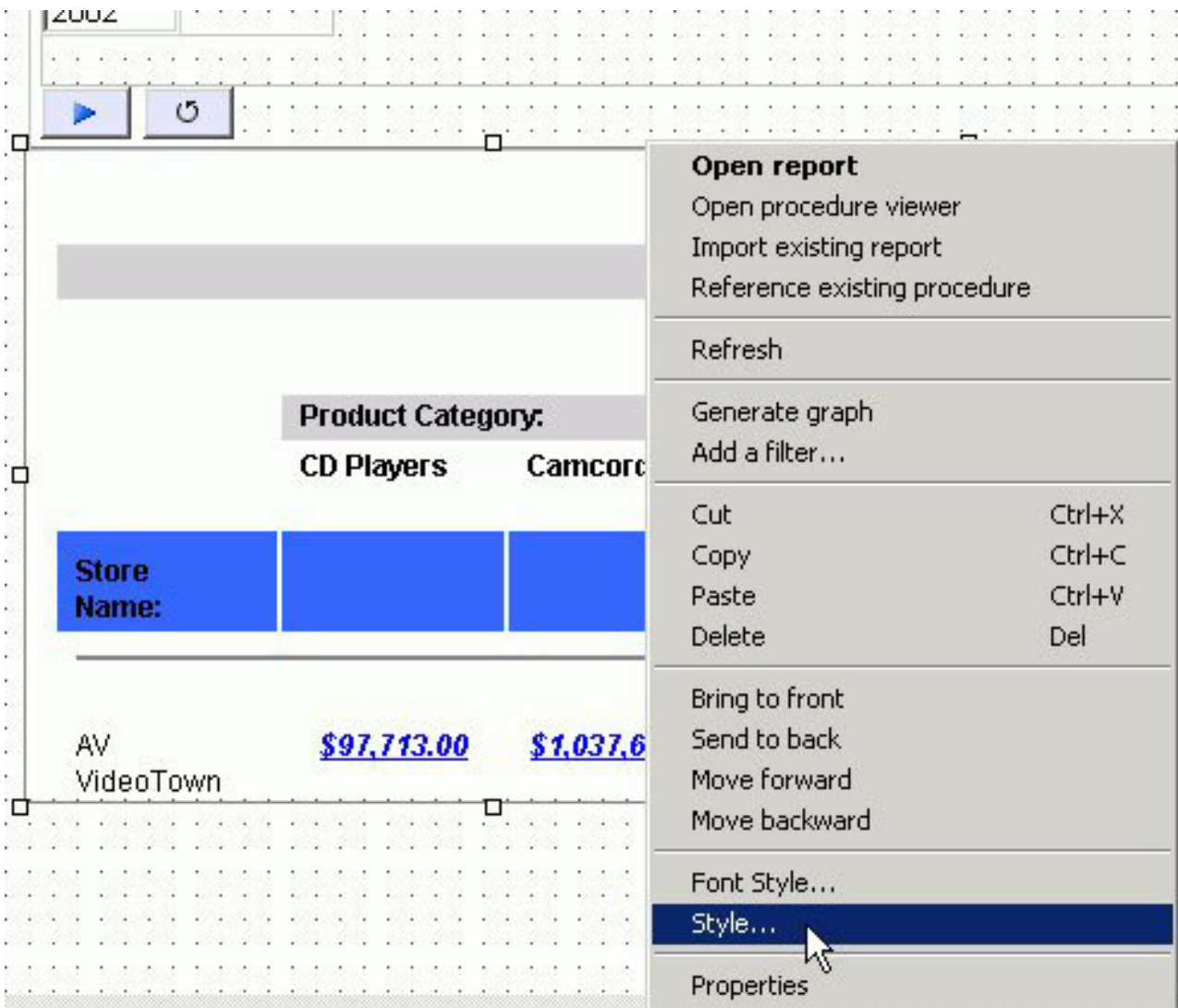
The Style Composer opens.

4. Select **Layout** from the left-side of the Style Composer to view the layout options for the selected frame.

5. From the Flow control area, select **Do not display** from the Display drop-down list.
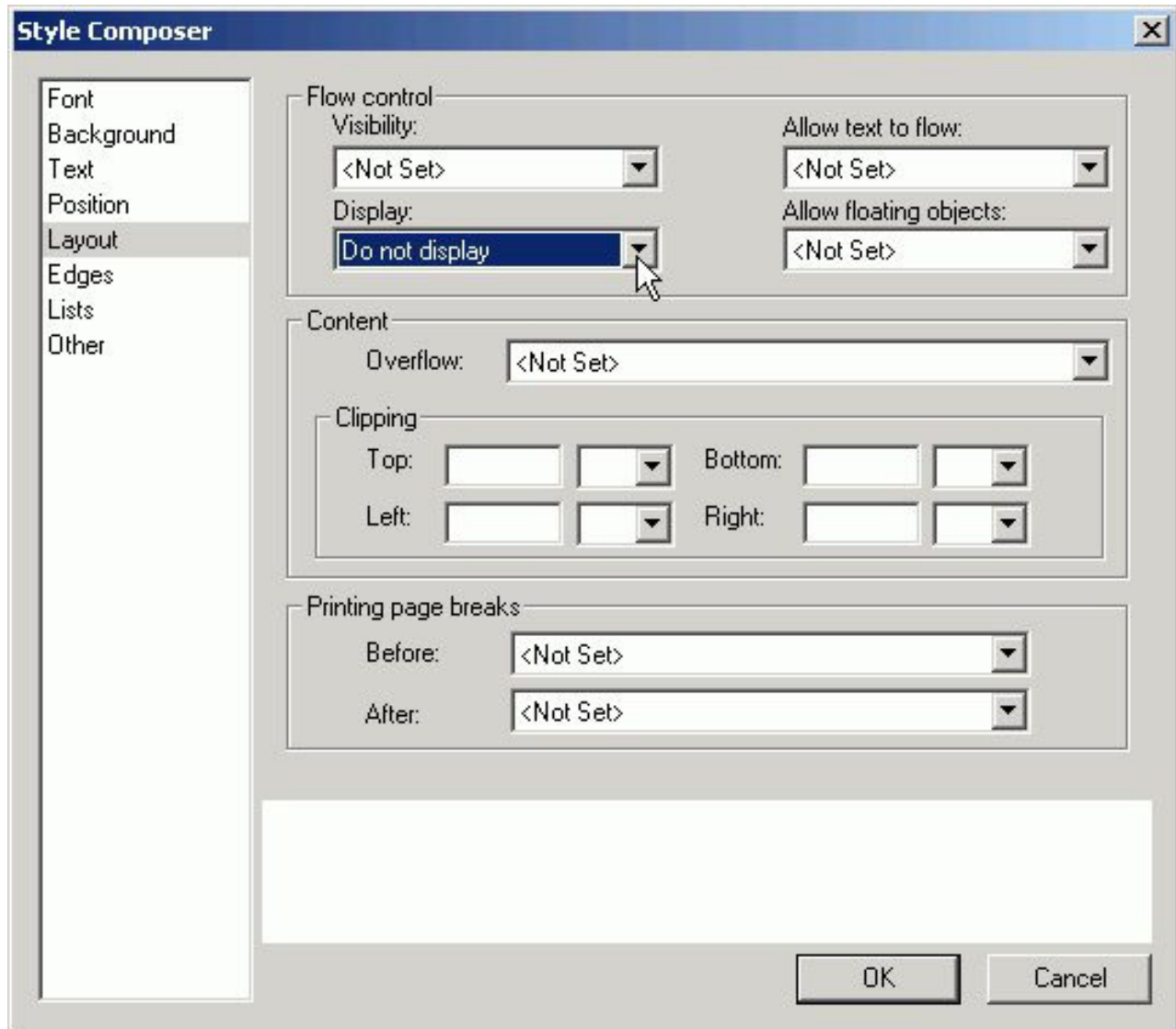
This option hides the frame at run time until the values are selected.

6. Click **OK** to close the Style Composer.

7. Run the HTML page.

In the example below, the frame is not shown before the values are selected.
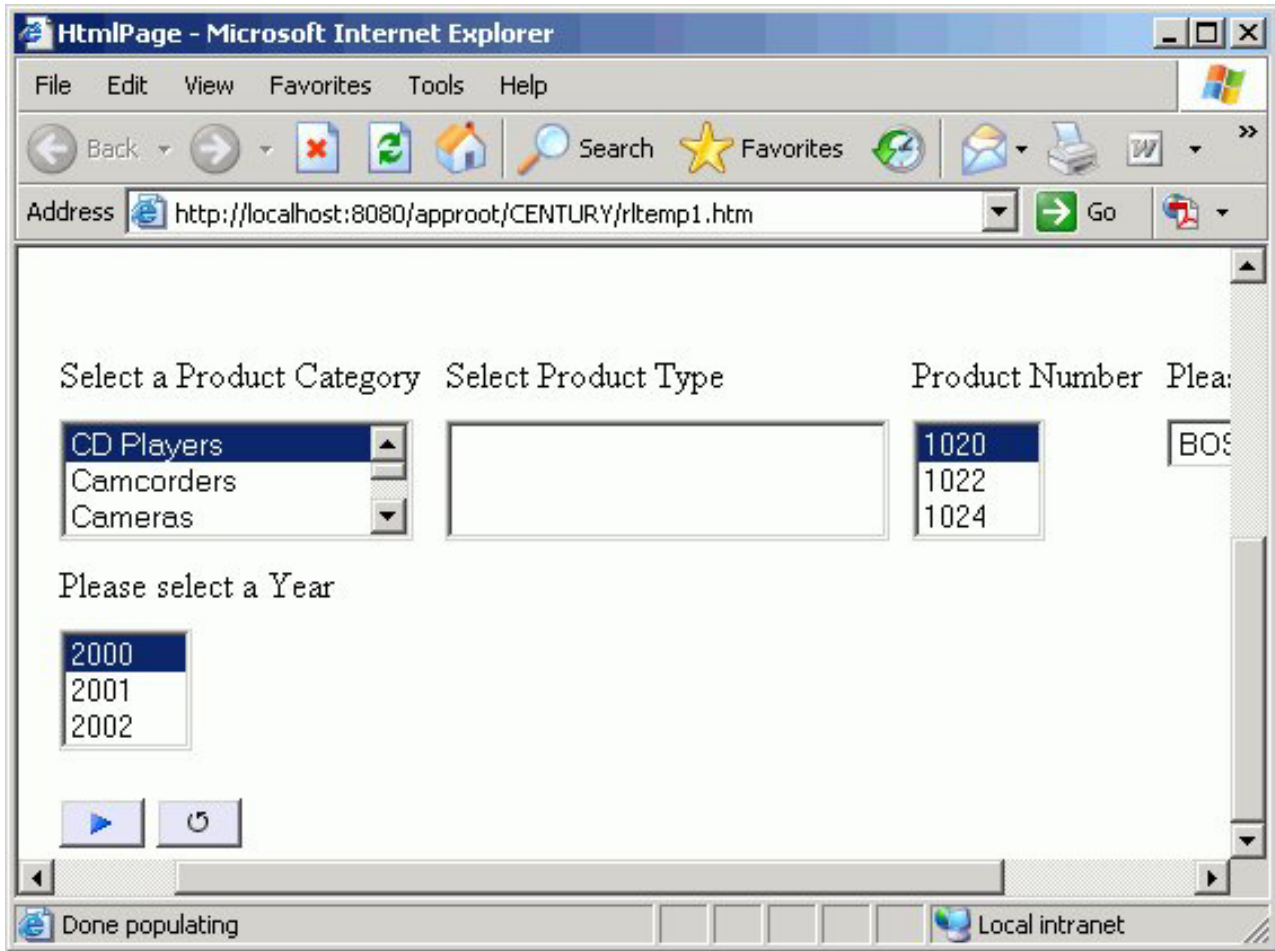
In the same example below, parameter values have been selected and the frame appears showing the output results.
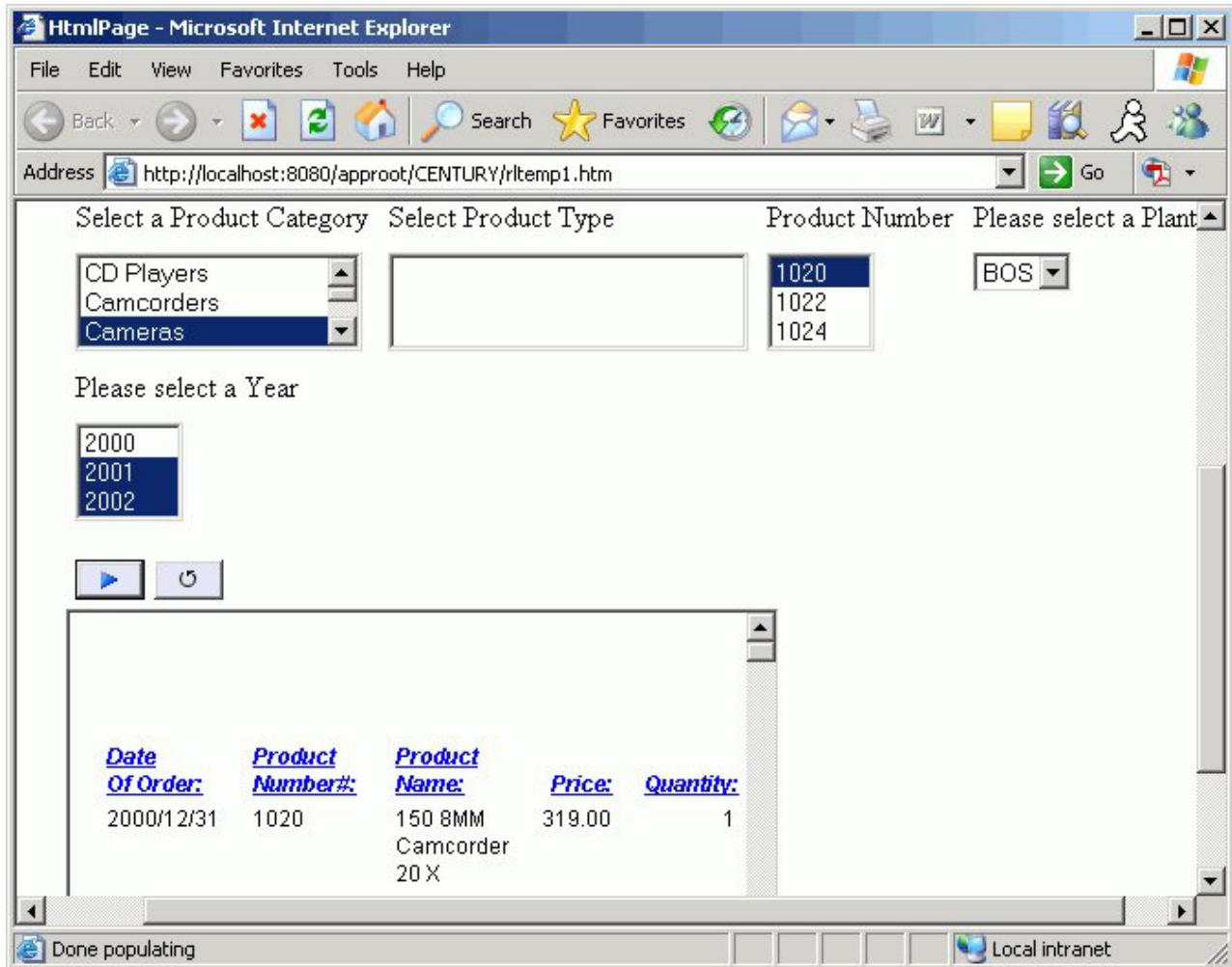
Frame Properties in the Properties Window

When a frame is selected, the Properties tab in the Properties window contains options that control the properties of your frame.
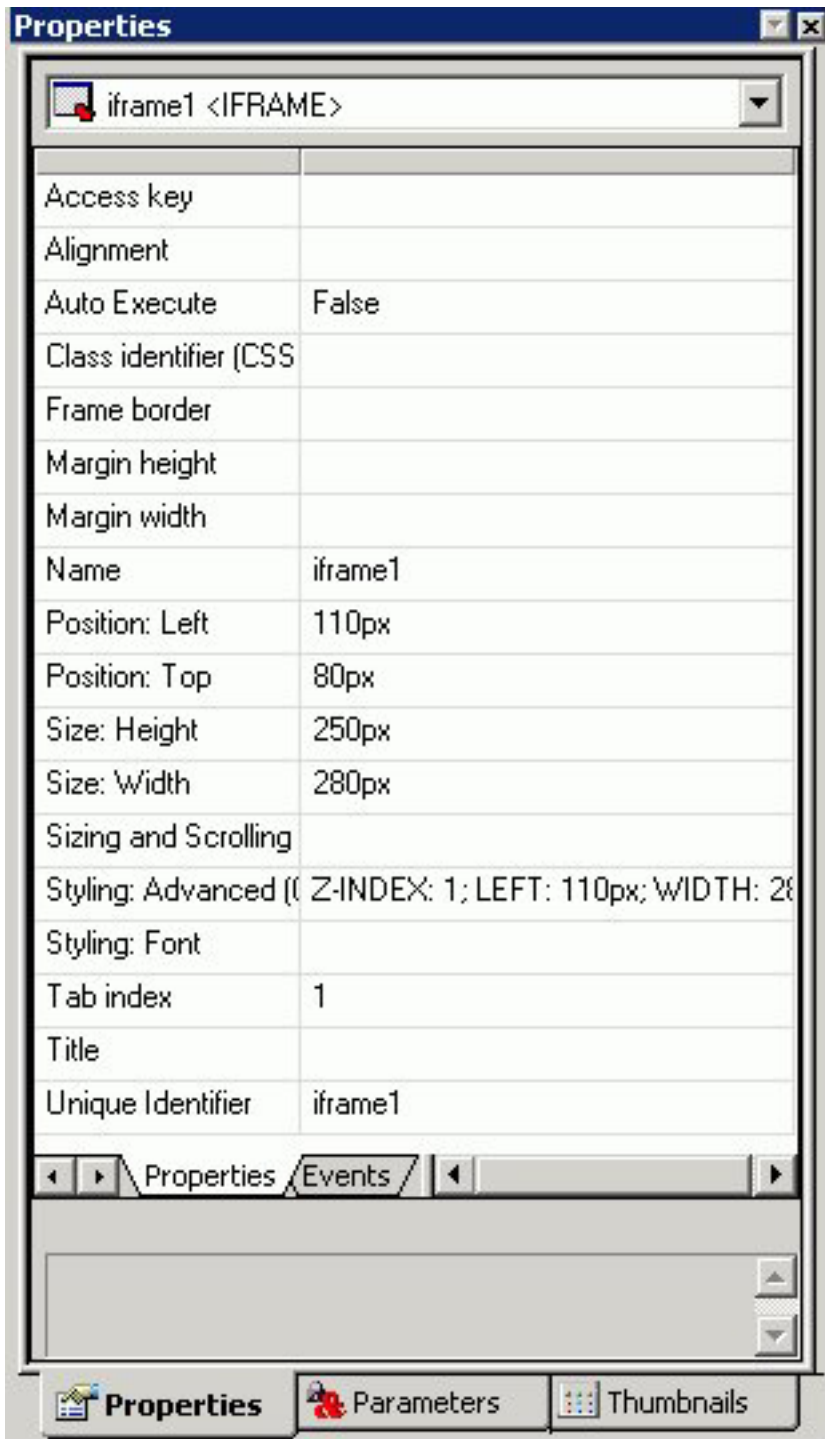
Properties	
iframe1 <IFRAME>	▼

Access key	
Alignment	
Auto Execute	False
Class identifier (CSS	
Frame border	
Margin height	
Margin width	
Name	iframe1
Position: Left	110px
Position: Top	80px
Size: Height	250px
Size: Width	280px
Sizing and Scrolling	
Styling: Advanced ((	Z-INDEX: 1; LEFT: 110px; WIDTH: 28
Styling: Font	
Tab index	1
Title	
Unique Identifier	iframe1

◄ ► \ Properties ⁄ Events ⁄ ◄

Properties | **Parameters** | **Thumbnails**

Click a property to display a description of the selected property at the bottom of the Properties window.

Adding a Line to the Layout

You can add a horizontal or vertical line to the layout. This is useful for distinguishing between sections of your launch or display page.

Add a Line to the Layout

1. Do one of the following:
   - Click the **Line** button.
   - From the Insert menu, select **Line**.

   The cursor changes into a crosshair.
2. Click and drag the crosshair to create a horizontal or vertical line.

   A line is created in the layout.
3. Optionally, change the line's properties by adjusting the properties displayed in the Properties tab of the Properties window. For details, see Line Properties in the Properties Window.

Line Properties in the Properties Window

When a line is selected, the Properties tab in the Properties window contains options that control the properties of lines.

Properties	
line1 <SPAN>	
Access key	
Class identifier (CSS)	
Direction of text	
Language information	
Position: Left	20px
Position: Top	250px
Size: Height	8px
Size: Width	65px
Styling: Advanced (CSS)	FONT-SIZE: 1pt; Z-INDEX
Tab index	13
Title	
Unique Identifier	line1

Properties / Events /

Properties · Parameters · Thumbnails
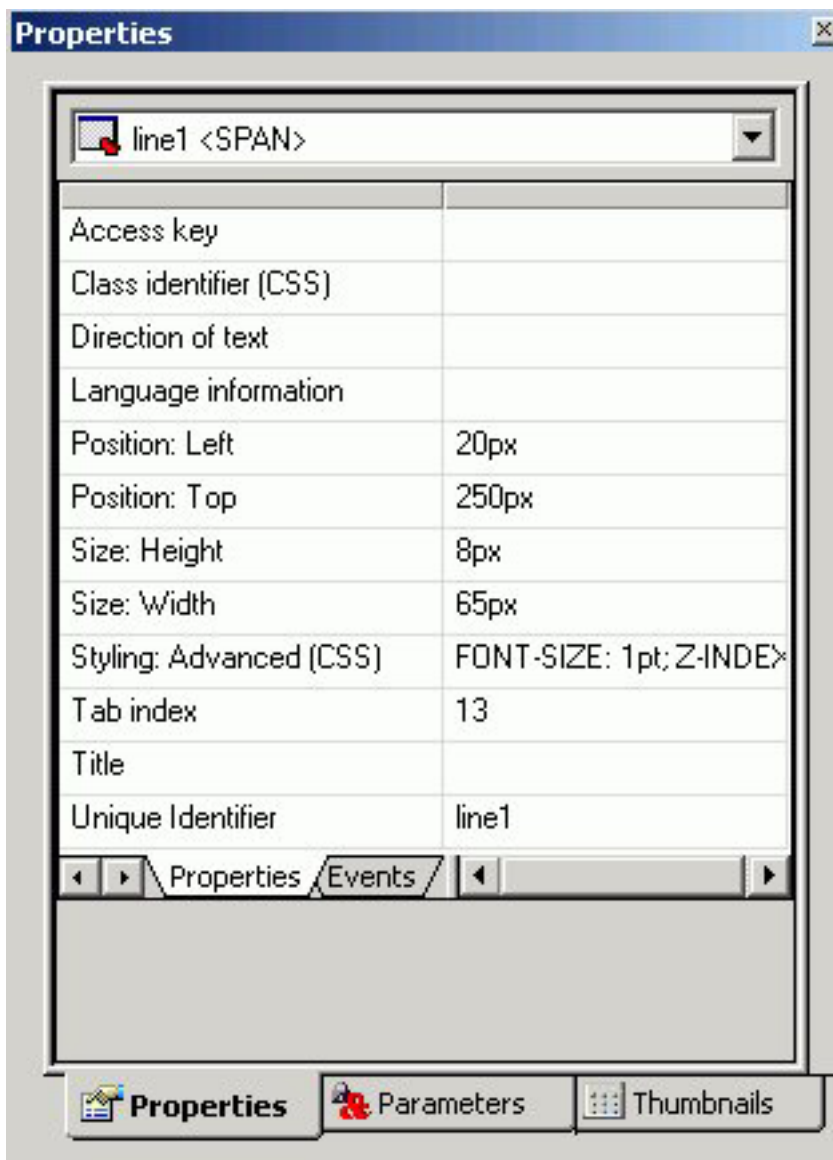
To view a description of a property at the bottom of the Properties window, click the desired property.

Adding an Image to the Layout

You can add an image to the layout. This is useful for including graphics such as a company logo.

You can insert an image into your report layout and add a hyperlink to it. After you run your report and click the image you can launch a URL or run a report the same way you can by clicking a hyperlink or push button. For more information, see Add a Hyperlink to a Push Button or an Image.

Note: When inserting images, images must be referenced from a specific directory location. Links to images are not supported.

Add an Image to a Layout

1. Do one of the following:
   - Click the **Image** button.
   - From the Insert menu, select **Image**.

   The Get source file dialog box opens.
2. Navigate to the directory where the image is located using the Look in menu, then select the image you want to add to the layout.
3. Click **Open**.
4. Optionally, change the image's properties by adjusting the properties displayed in the Properties tab of the Properties window. For details, see Image Properties in the Properties Window.

   Note: You can always return an image to its original size by right-clicking the image and selecting **Image properties** from the pop-up menu, then selecting **Restore size**.

Image Properties in the Properties Window

When an image is selected, the Properties tab in the Properties contains options that control the properties of images.
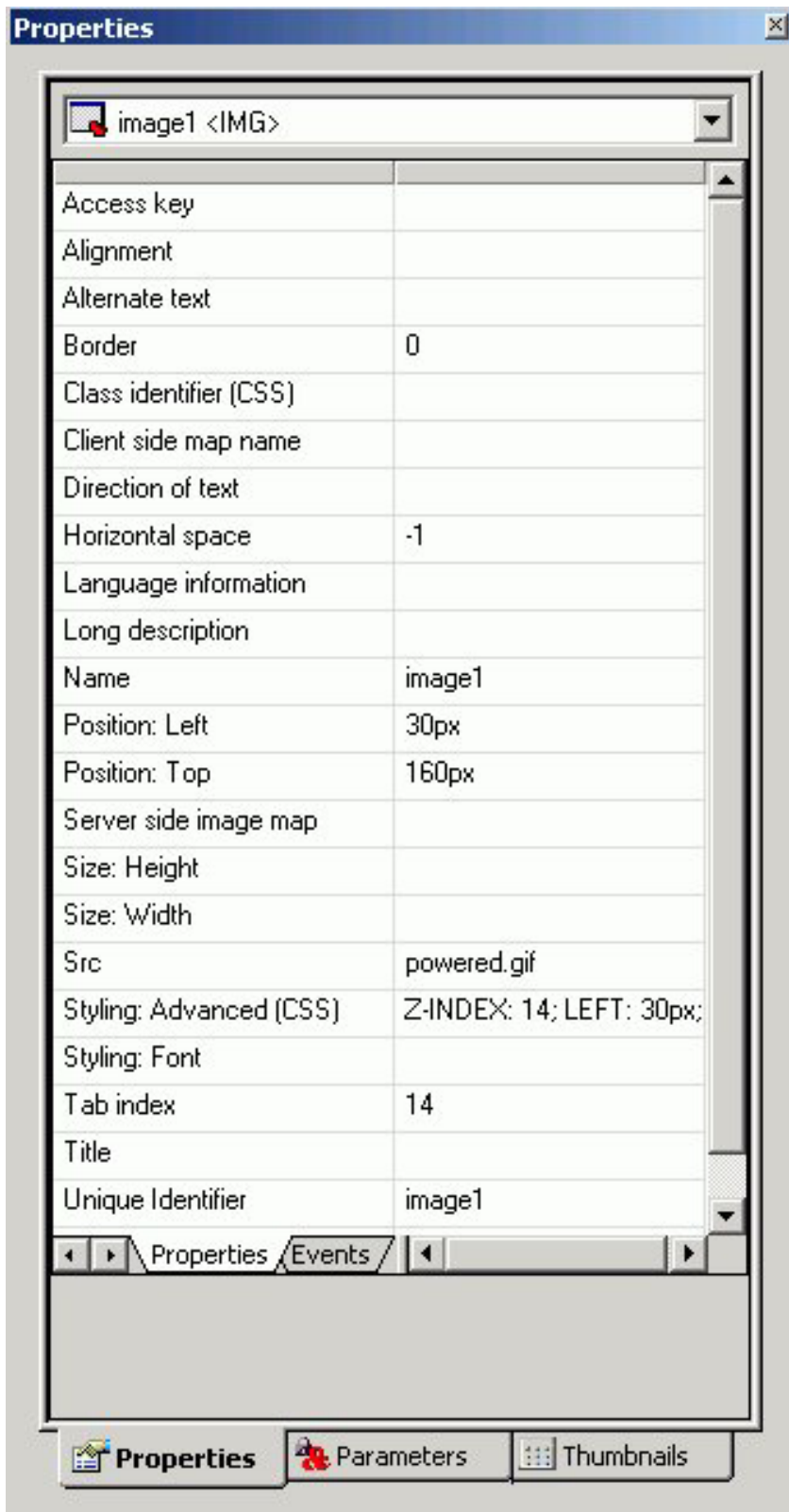
Click a property to display a description of the selected property at the bottom of the Properties window.

Adding Text to the Layout

You can add text to the layout. This is useful for including headings for your Web page, or adding directions or explanation for your report or graph.

Add Text to a Layout

1. Insert text to the layout by doing one of the following:
   - Click the **Text** button.
   - From the Insert menu, select **Text**.

   A text object appears.
2. Replace the text with the text you want to appear in the layout.
3. Optionally, change the text's properties by adjusting the properties displayed in the Properties tab of the Properties window. For details, see Text Properties in the Properties Window.

Format Text in the Layout

You may apply various formatting and style options to words and individual text characters within the text element. The formatting options are available from the Formatting toolbar in the HTML Layout Painter.

Note: Any formatting and styling that you may have applied to individual text strings within the text element will remain unchanged. Changes made to the entire text element are only applied to part of the text string that has not been formatted.

1. Required: Insert a text element into the layout and type text in the text element.
2. Required: Select the text that you wish to format:
   - To format the entire text element, single click the text object in the layout.
   - To format an individual word or text character, highlight part of the text within the text element.

   The Formatting toolbar is activated.

   Note: The Bold, Italic, Underline, Superscript, and Font Style options are available when formatting individual words or text characters. The Font Style and Alignment options are available when the entire text element is selected.
3. Select from the formatting options available from the Formatting Toolbar.
4. Select **Font Style** from the Formatting toolbar to open the Font dialog box, from which you can change the type, style, color, size, and effect of the Font.

   Tip: You may also access the Font Style dialog box from the Font ellipse button of the **Styling Font** field in the Properties window.
5. Click **OK** to close the Font dialog box.

The format options are applied to the text selected.

Text Properties in the Properties Window

When text is selected, the Properties tab in the Properties window contains options that control the properties of text in your report.
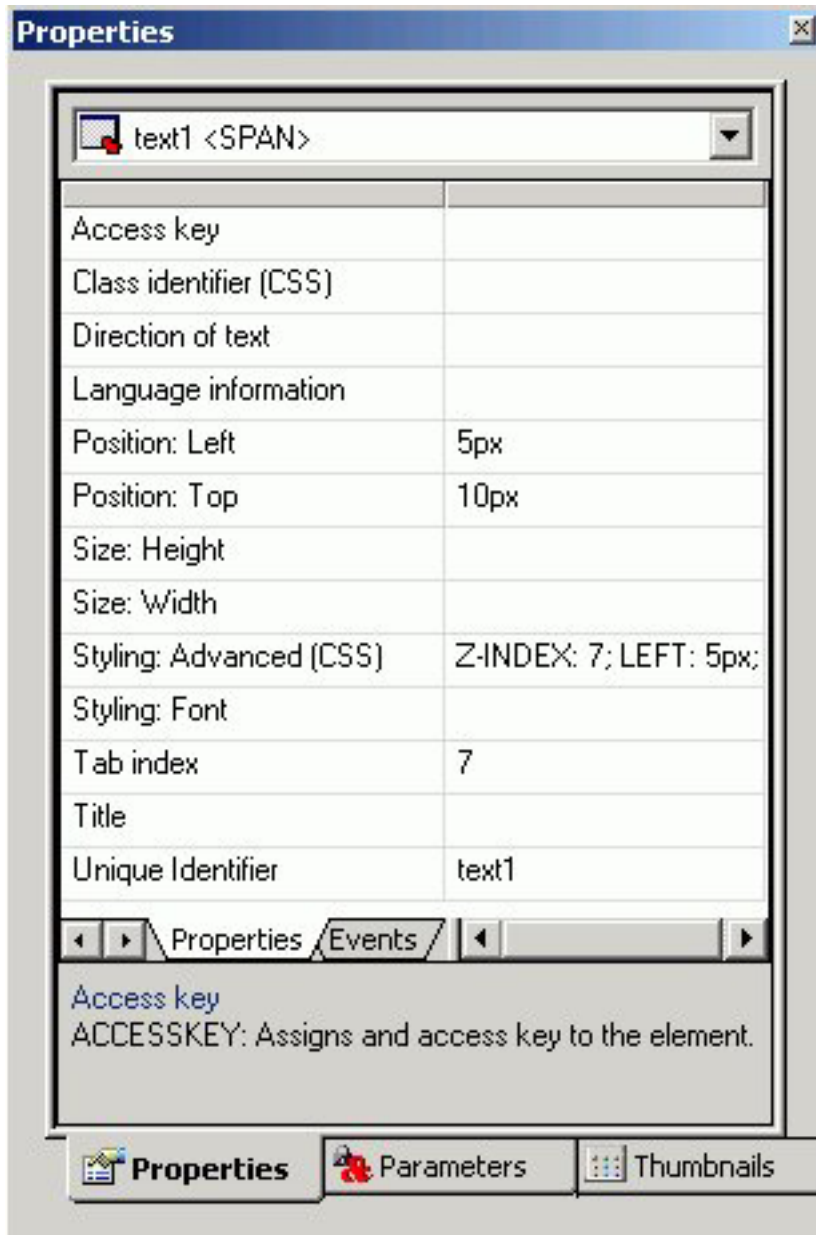
Adding a Hyperlink to the Layout

You can create a hyperlink for your HTML page. A hyperlink can execute a report, link to a URL, or open an HTML page. You can create a hyperlink in two ways:

- Insert a hyperlink. For details, see Create a Hyperlink.
- Add a hyperlink to a push button or image. For details, see Add a Hyperlink to a Push Button or an Image.

Create a Hyperlink

1. Do one of the following:
   - Click the **Hyperlink** button.
   - Select **Hyperlink** from the Insert menu.

   The Hyperlink Properties dialog box opens.
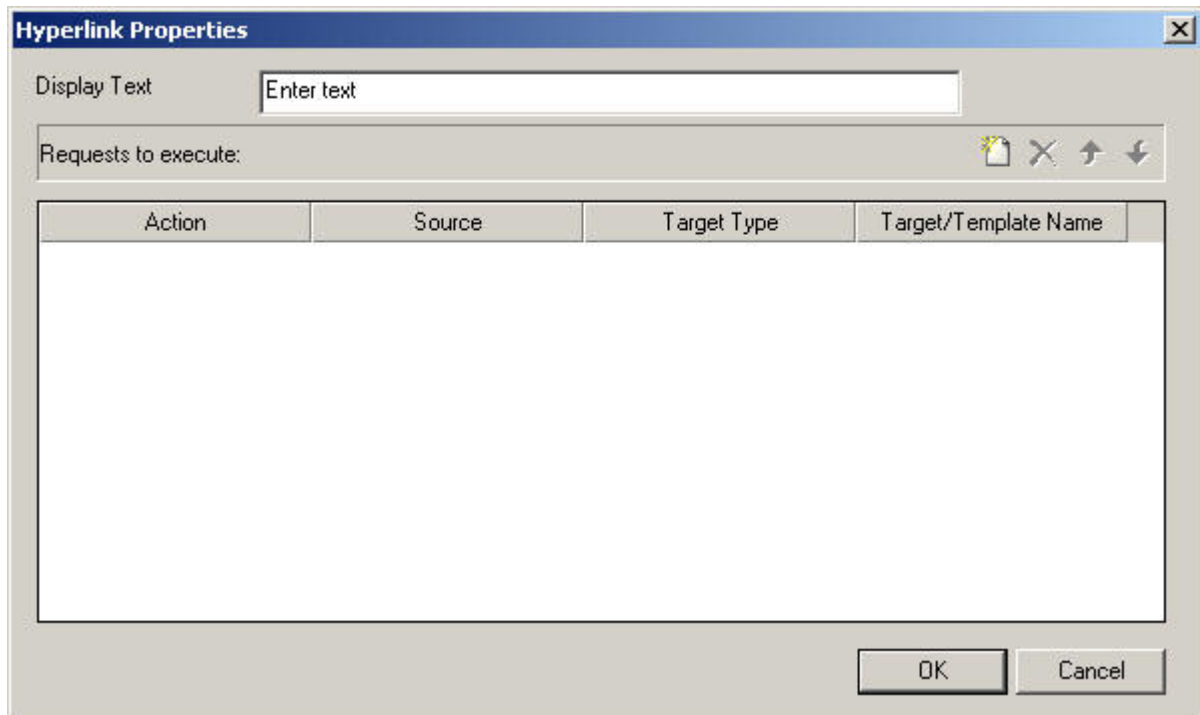
2. Enter the text you want to display as the hyperlink in the Display Text field.

3. Required: Set the action of the hyperlink:

   - To link to a URL, select **URL** in the Action section, and enter the URL in the Source field.

   - To open an HTML page, click select **HTML** in the Action section, and enter the HTML page in the Source field.

   - To execute an embedded procedure, select **Embedded procedure** and enter the procedure name in the Source field, or click the browse **(...)** button to browse to the procedure.

   - To execute an external procedure, select **External procedure** and enter the procedure name in the Source field, or click the browse **(...)** button to browse to the procedure.

   - To refresh or repopulate Active Reports based on selected values in Active Report controls, select **Refresh Active Reports** and specify the source or which Active Report(s) should be refreshed.

     For more information about Active Report controls, see Creating Active Dashboards.

4. Optionally, direct the output to a specific location by selecting **Window** or **Frame** in the Target Type field.

5. Specify a name for the target window by selecting one of the default values from the Target/Template Name drop-down list or by typing the name of a new or existing window in the Target/Template Name field.

6. Click **OK**.

7. Execute the request and click the image to launch the web page of the URL Address you typed in the Source field.

Add a Hyperlink to a Push Button or an Image

To add a hyperlink to a push button or image, complete the following steps.

1. Right-click the push button or image, and select **Create hyperlink**. The Hyperlink Properties dialog box opens.
2. Click the **New** icon to generate a new request and select **URL** from the Action drop-down list.
3. Type a URL in the Source field.
4. Optionally, in the Target Type field, direct the output to a specific location by selecting **Window** or **Frame** from the drop-down list.
5. In the Target/Template Name field, specify a target window by selecting one of the default values from the drop-down list or by typing the name of a new or existing window.
6. Click **OK**.
7. Execute the request and click the image to launch the web page of the URL Address you typed in the Source field.

Adding a Group Box to the Layout

A group box can be used to create a border around a group of objects, for example, forms or reports and graphs.

Add a Group Box

1. Insert a group box by doing one of the following:
   - Click the **Group box** button.
   - From the Insert menu, select **Group box**.

   The cursor changes into a crosshair.
2. Click and drag the crosshair to create a group box and adjust it to the size you want.

   A group box is created in the layout and assigned the name groupbox*(n)*, where *(n)* is a number.
3. Optionally, you may change the default name of the group box and format the text, as shown in the image below.
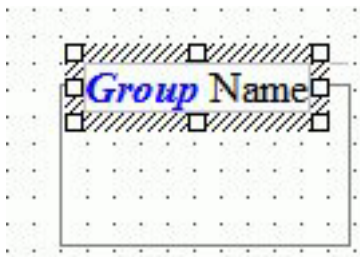
   For more information about formatting text, see Format Text in the Layout.
4. Optionally, change the group box's properties by adjusting the properties displayed in the Properties tab of the Properties window. For details, see Group Box Properties in the Properties Window.

Group Box Properties in the Properties Window

When a group box is selected, the Properties tab in the Properties window contains options that control the properties of a group box.
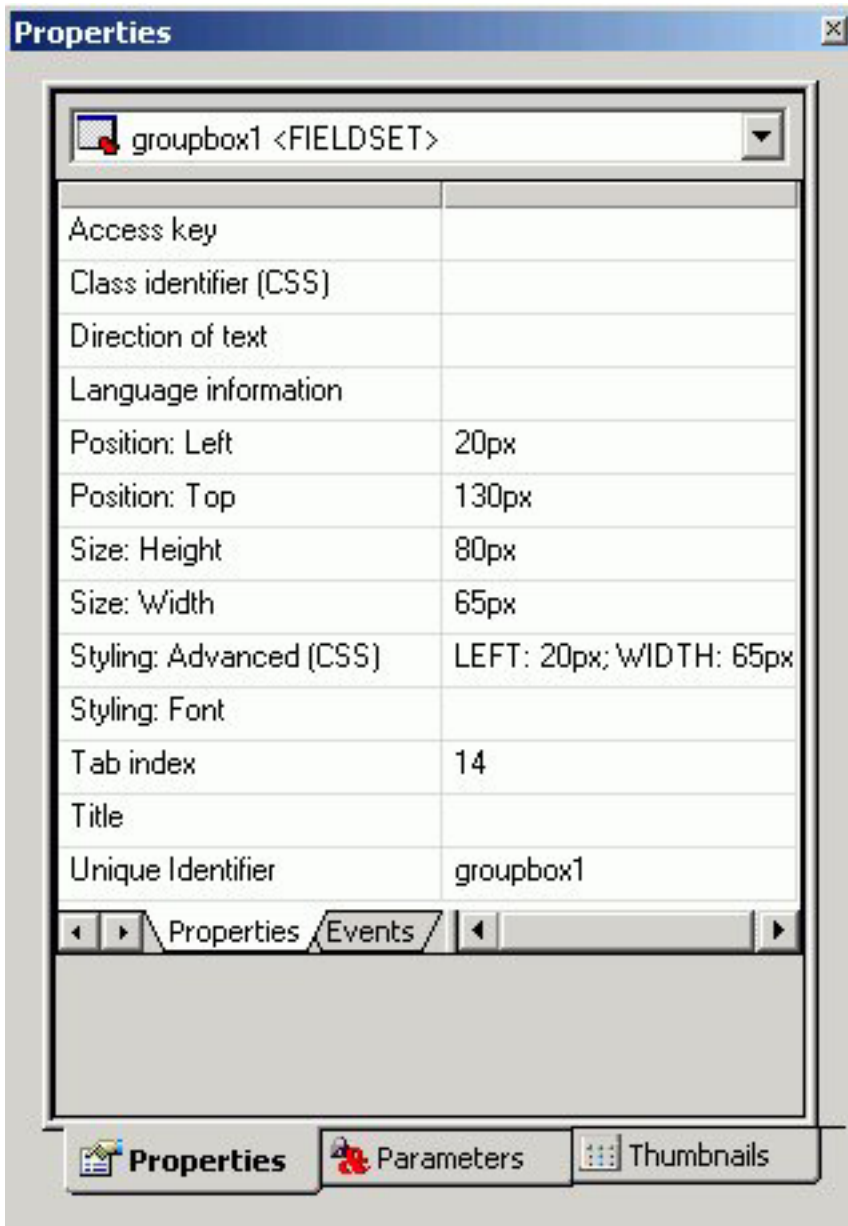
Click a property to display a description of the selected property at the bottom of the Properties window.

Adding a Push Button to the Layout

You can add a push button to the layout. A push button enables you to execute a report, or link to a URL or HTML form. This behavior is similar to a hyperlink.

For information on adding a hyperlink to a push button, see Add a Hyperlink to a Push Button or an Image.

Create a Submit Button

1. If the submit button is deleted from the layout, a push button can be used to replace the button. Insert a push button by doing one of the following:

   • Click the **Push** button from the Components toolbar.

- From the Insert menu, select **Push Button**.

  The cursor changes into a crosshair.

2. Click and drag the crosshair to create a push button and adjust it to the size you want.

   A push button is created in the layout and assigned the name button*(n)*, where *(n)* is a number.

3. Assign an action to the push button by using the Hyperlink Properties dialog box. Right click on the push button and select **Hyperlink properties** from the context menu.

   The Hyperlink Properties dialog box opens. Use the Hyperlink Properties dialog box to assign a target and action to the push button.

4. Click **OK** to close the Hyperlink Properties dialog box.

5. Run the HTML page.

6. Click the push button to submit your request.

Run Multiple Reports with One Submit Button

1. Right click on the submit button, or the frame for the whole control, and select **Hyperlink properties** from the context menu.

   The following image is an example of the submit button selected.
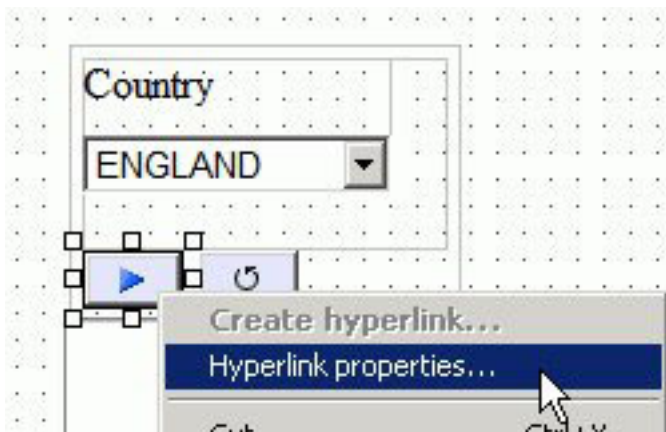
The following image is an example of the whole control selected.
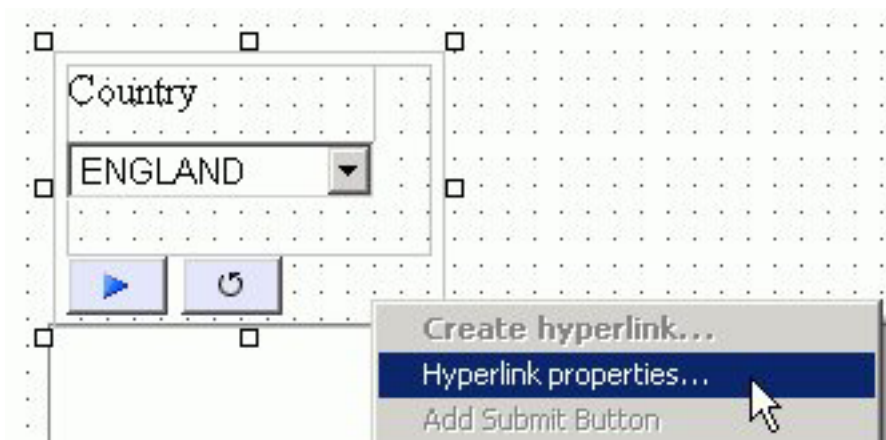
The Hyperlink Properties dialog box opens.
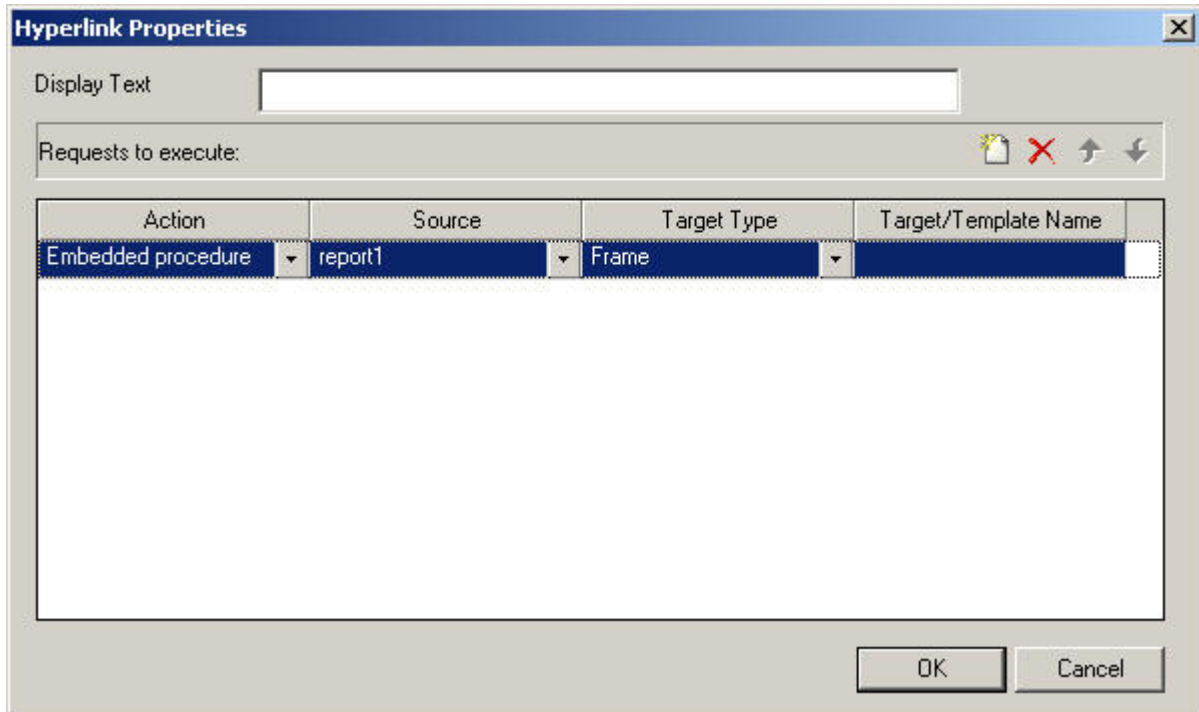
2. Required: Click the **New** icon and use the drop-down lists to add the second report request to be executed with the submit button.
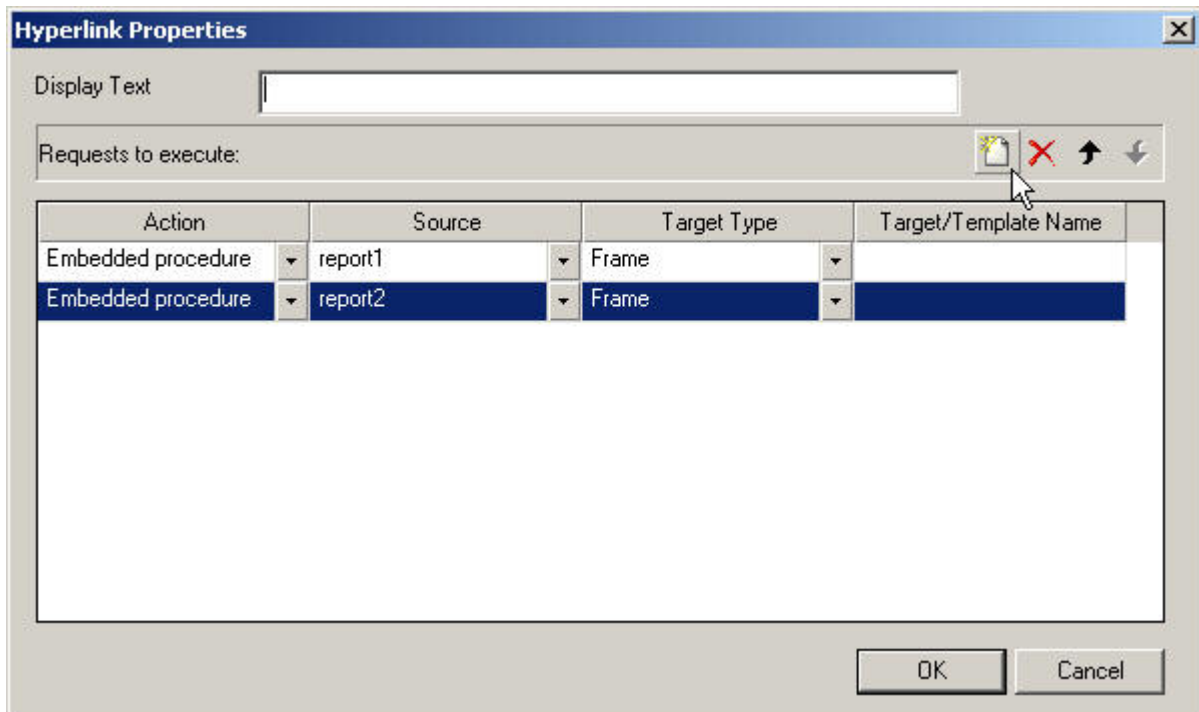
Repeat these steps for multiple procedures.

Push Button Properties in the Properties Window

When a push button is selected, the Properties tab in the Properties window contains options that control the properties of your buttons.
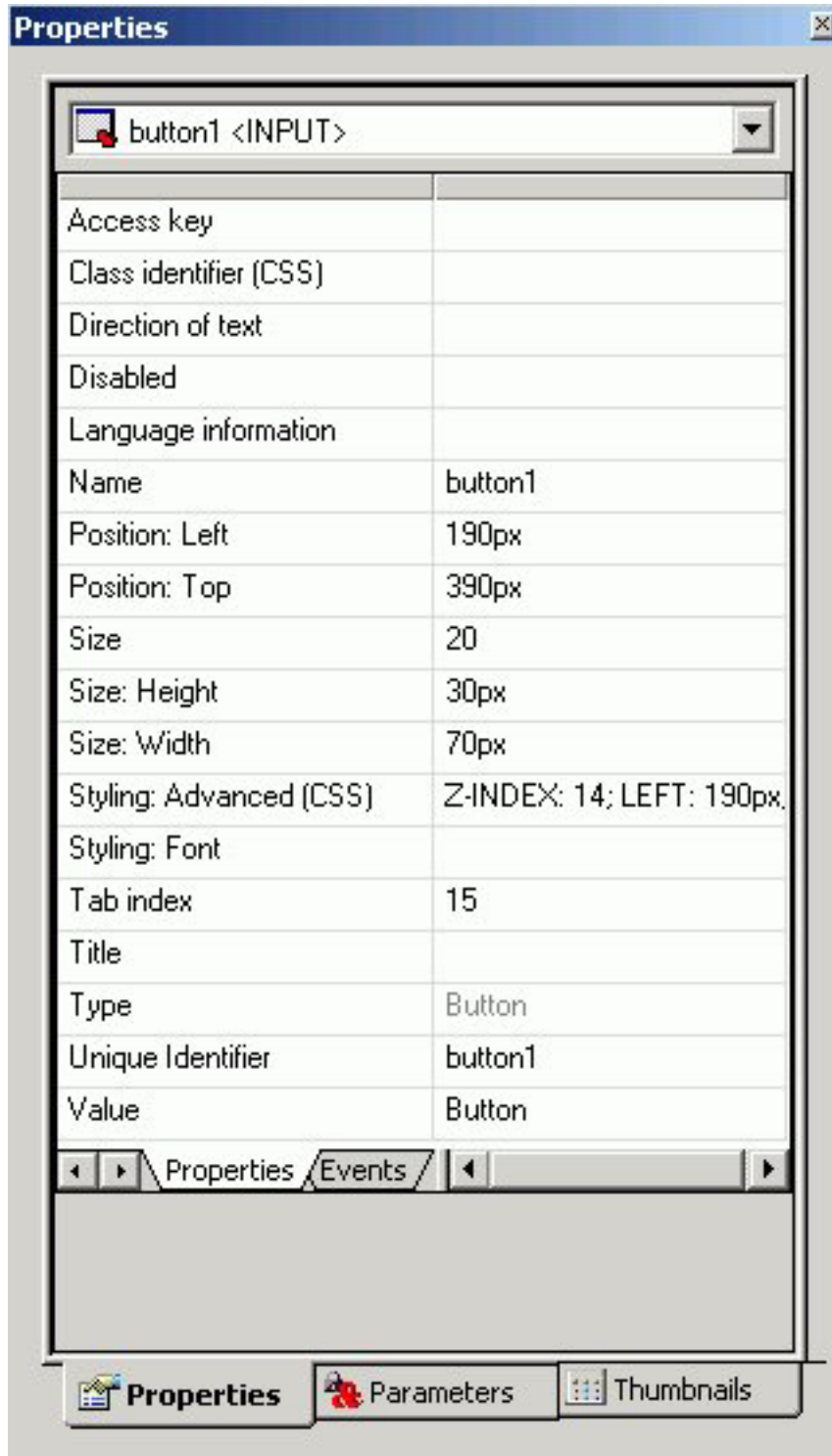
Click a property to display a description of the selected property at the bottom of the Properties window.

Adding a Reset Button to the Layout

You can add a reset button to the layout. A reset button enables you to reset the form back to its initial settings.

Create a Reset Button

1. If the reset button is deleted from the layout, you may create a new reset button. Insert a reset button by doing one of the following:

   - Click the **Reset** button from the Components toolbar.
   - From the Insert menu, select **Reset Button**.

   The cursor changes into a crosshair.

2. Click and drag the crosshair to create a reset button and adjust it to the size you want.

   A reset button is created in the layout and assigned the name reset*(n)*, where *(n)* is a number.

3. Run the HTML page.

4. When selecting criteria to submit a report, click the reset button to reset the form back to its initial settings.

Reset Button Properties in the Properties Window

When a Reset button is selected, the Properties tab in the Properties window contains options that control the properties of your buttons.
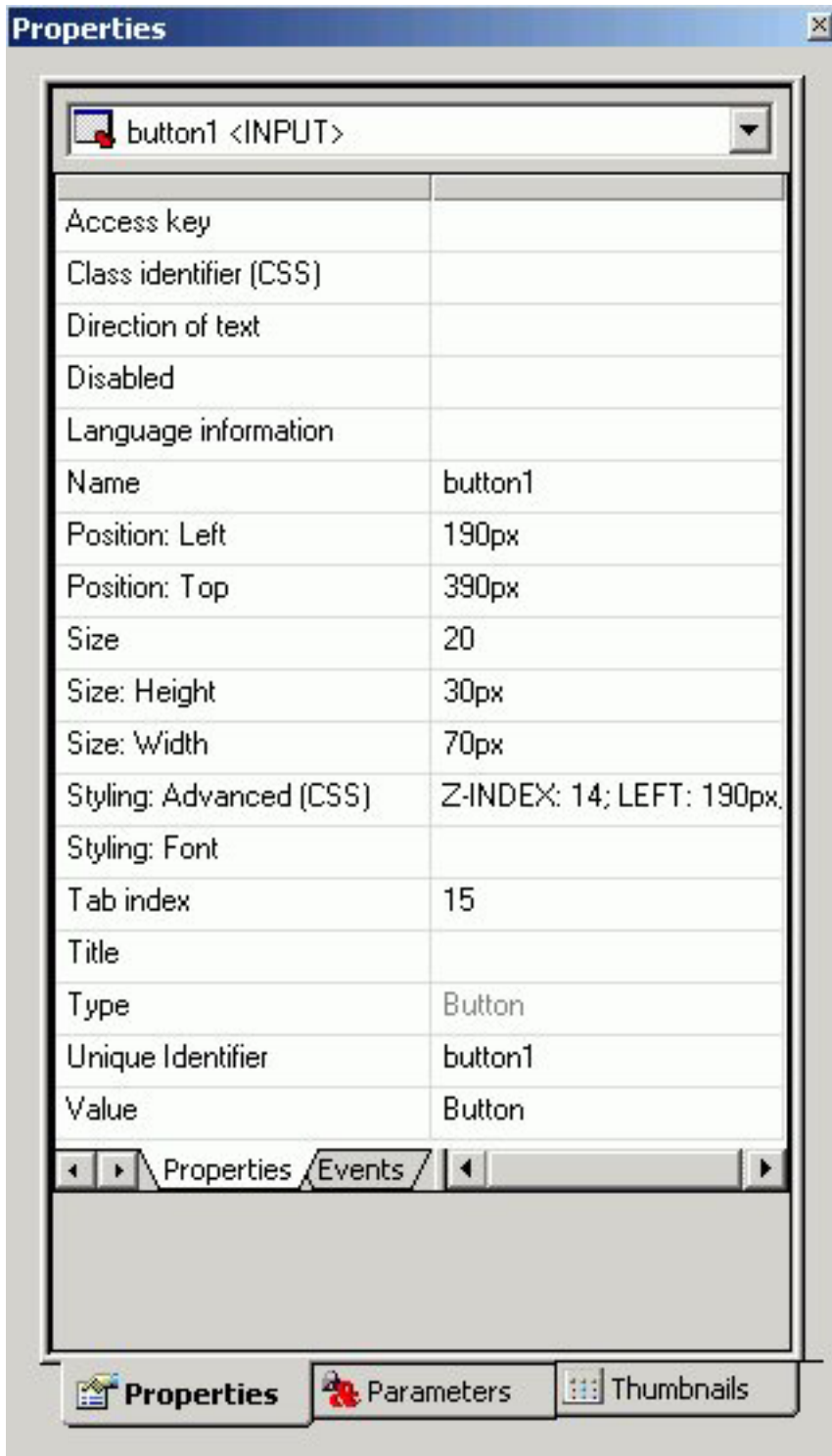
Click a property to display a description of the selected property at the bottom of the Properties window.

Adding a Tab Control to the Layout

You can add a tab control to the layout. Tab controls enable you to create multiple pages in one HTML form and present a better display of viewing secondary information.

When a tab control object is added to the layout, each tab control consists of:

- A tab item.

  A tab item is the tab label. You may edit the name of the tab item, style the tab item, and add multiple tab items. Each tab item is associated with a tab body.

- A tab body.

  A tab body is the tab page where you associate your components such as report/graph objects, images, and lines.

The Tab control can be displayed as a full screen or part of an HTML page.

In the example below, the selected tab shows a report and a second tab that contains a graph.
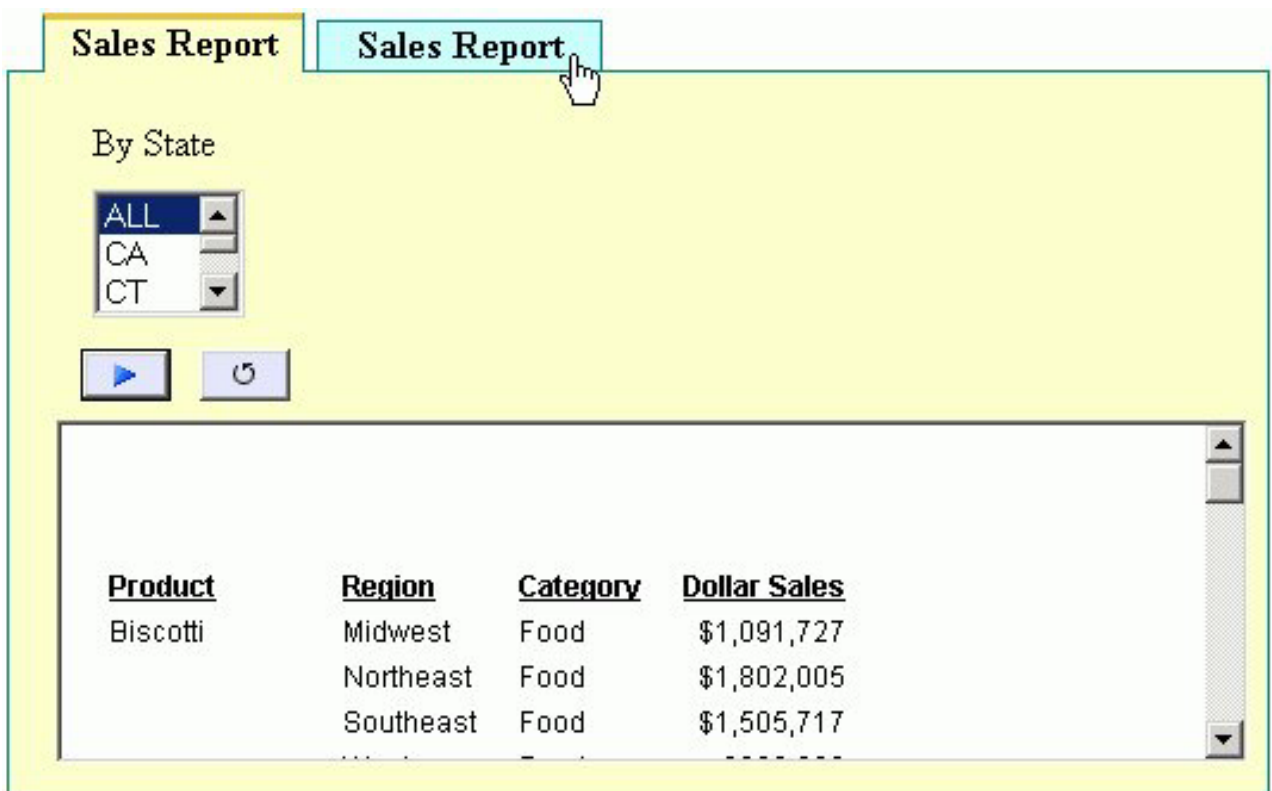
Create a Tab Control

1. Insert a tab control to the layout by doing one of the following:

   - Click the **Tab control** button from the Components toolbar.
   - Select **Tab Control** from the Insert menu.

   The cursor changes into a crosshair.

2. Click and drag the crosshair to create a tab control object and adjust it to the size you want.

   Tip: You should make the object large enough to associate report/graph components within the tab control.

   A tab control object is created in the layout and assigned the Properties name tab*(n)*, where*(n)* is a number of the tab. By default, the tab control has one tab page. Each tab page consists of a tab item (tabitem*(n)*) and tab body (tabitembody*(n)*).

3. Optionally, change the tab control's properties by adjusting the properties displayed in the Properties tab of the Properties window.

Enable Full Screen Mode for the Tab Control

You may resize the tab control to fit the full screen of your layout, making the tab control the full background of your browser window at run time. When set to full screen mode, scroll bars will not be applied to the output window. Therefore, you may have to adjust the tab control (and any items on the tab) in the layout to ensure that they appear appropriately for display at run time.

Note: It is recommended to set the tab control to full screen mode at the beginning of the development process. If there are existing components on the layout that are not part of the tab control, these components will become inaccessible if the tab control is changed to full screen mode.

1. Select **tab control** from the Insert menu.

   The cursor changes into a crosshair.

2. Click and drag the crosshair in the layout to create the tab control object.

3. Right-click the tab control and select **Full screen mode** from the context menu, as shown in the image below.
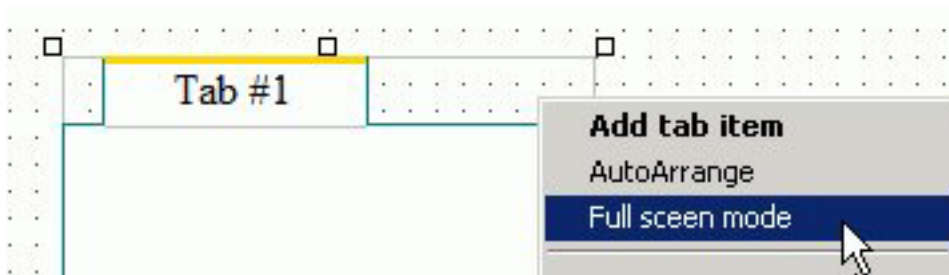
The tab control displays as a full screen in the layout, as shown in the image below.
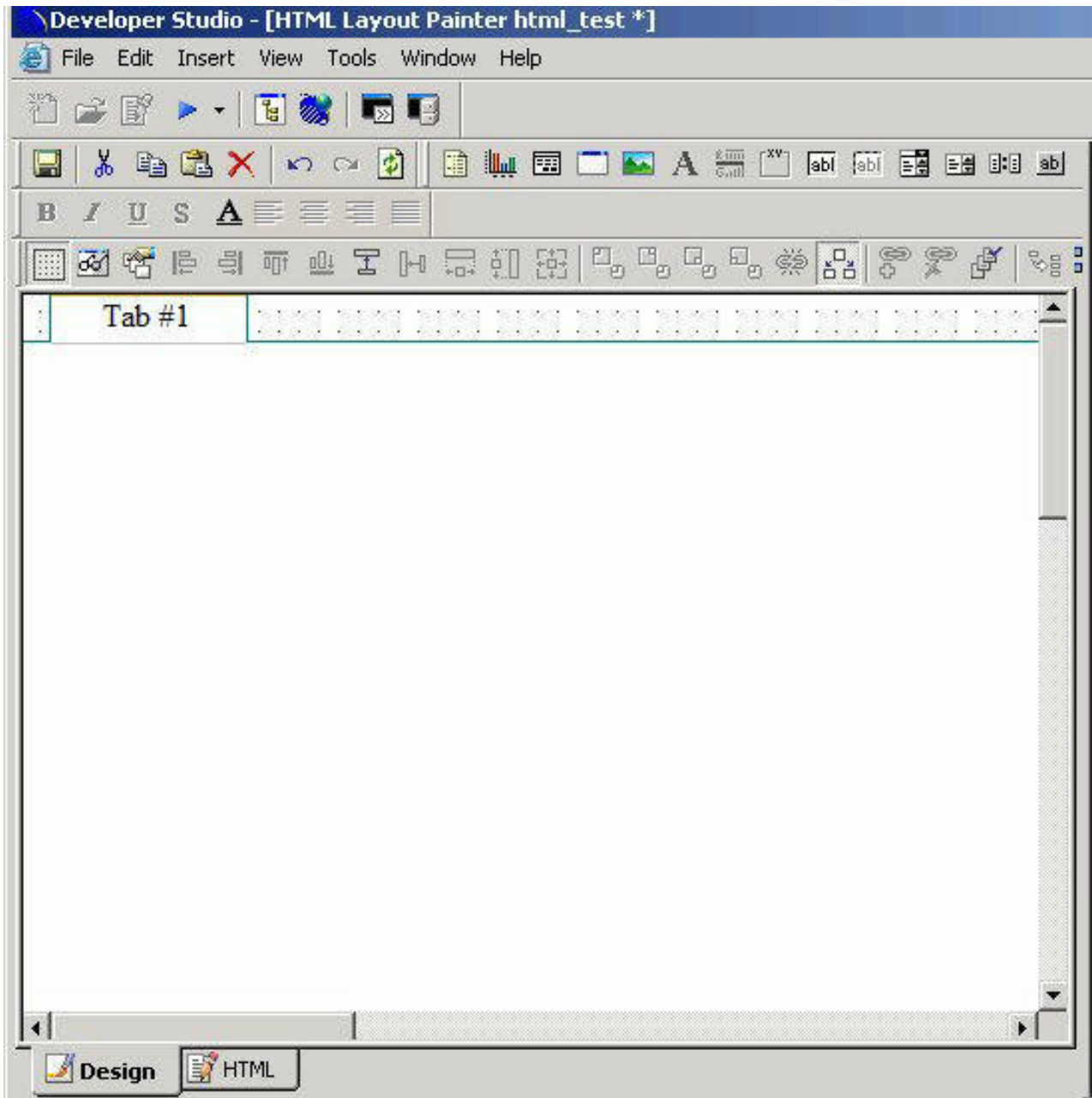
Note: This setting can be applied only to one tab control in your application. If one tab control is set to full screen mode, the full screen mode item will be greyed out for any additional tab controls.

4. To resize the tab control, right-click and uncheck the **Full screen mode** option, as shown in the image below.
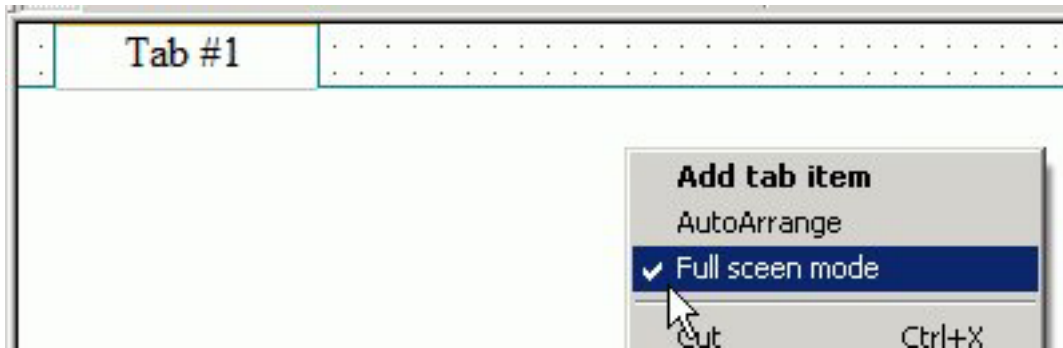
Add Additional Tabs

1. Required: Select the tab control object in the layout.
2. Required: Right-click and select **Add tab item** from the context menu.
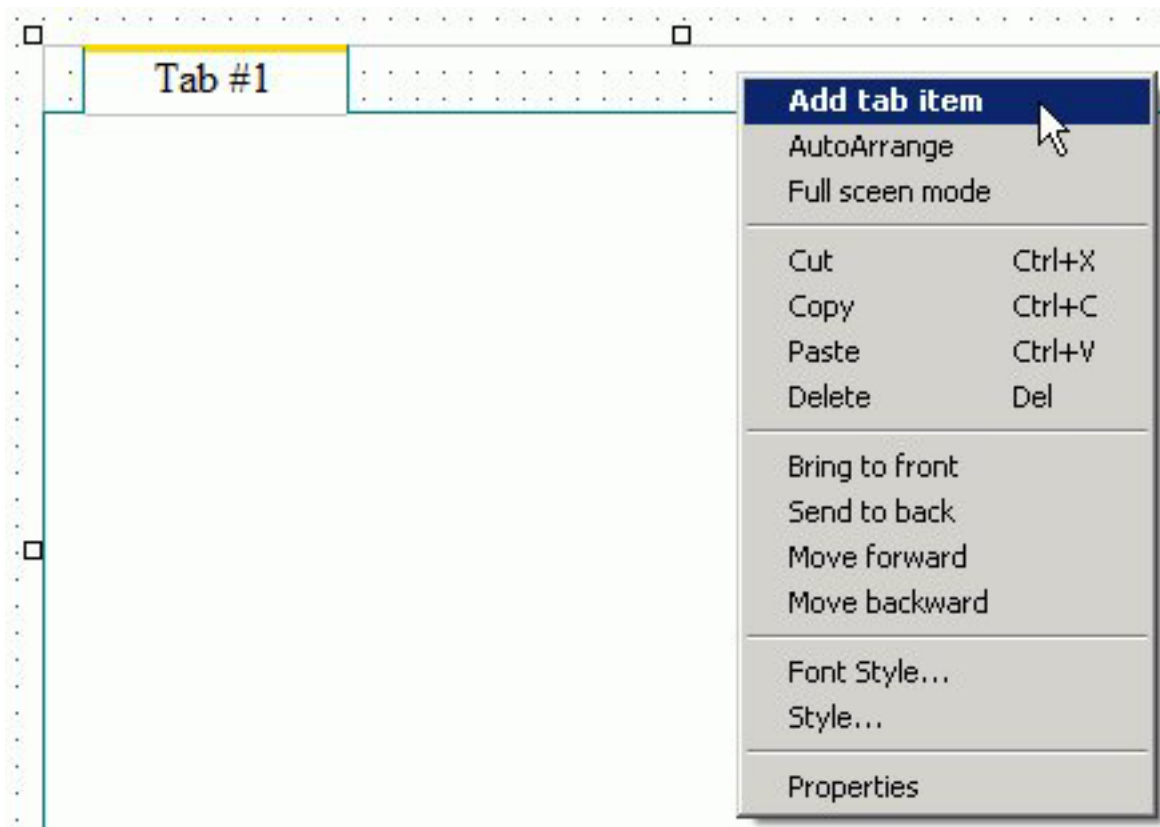
A tab is added to the tab control object.
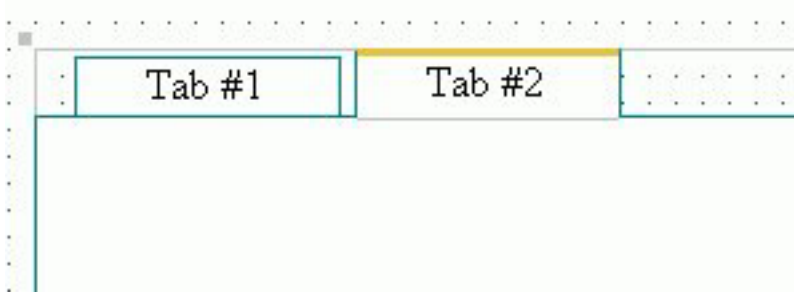
3. To align multiple tab items, select the tab control object and click **AutoArrange** from the context menu,
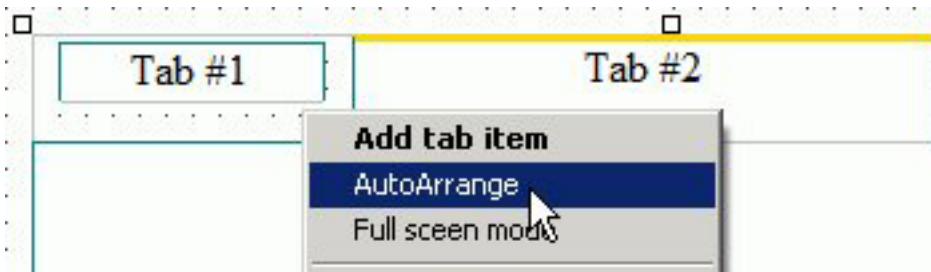
The tab items are resized to the size of the widest tab item and evenly spaced.

Tab Control Properties in the Properties Window

When a Tab control is selected, the Properties tab in the Properties window contains options that control the properties of your tabs.

Note: You may set options for the Tab control, individual tab items, and the tab body.
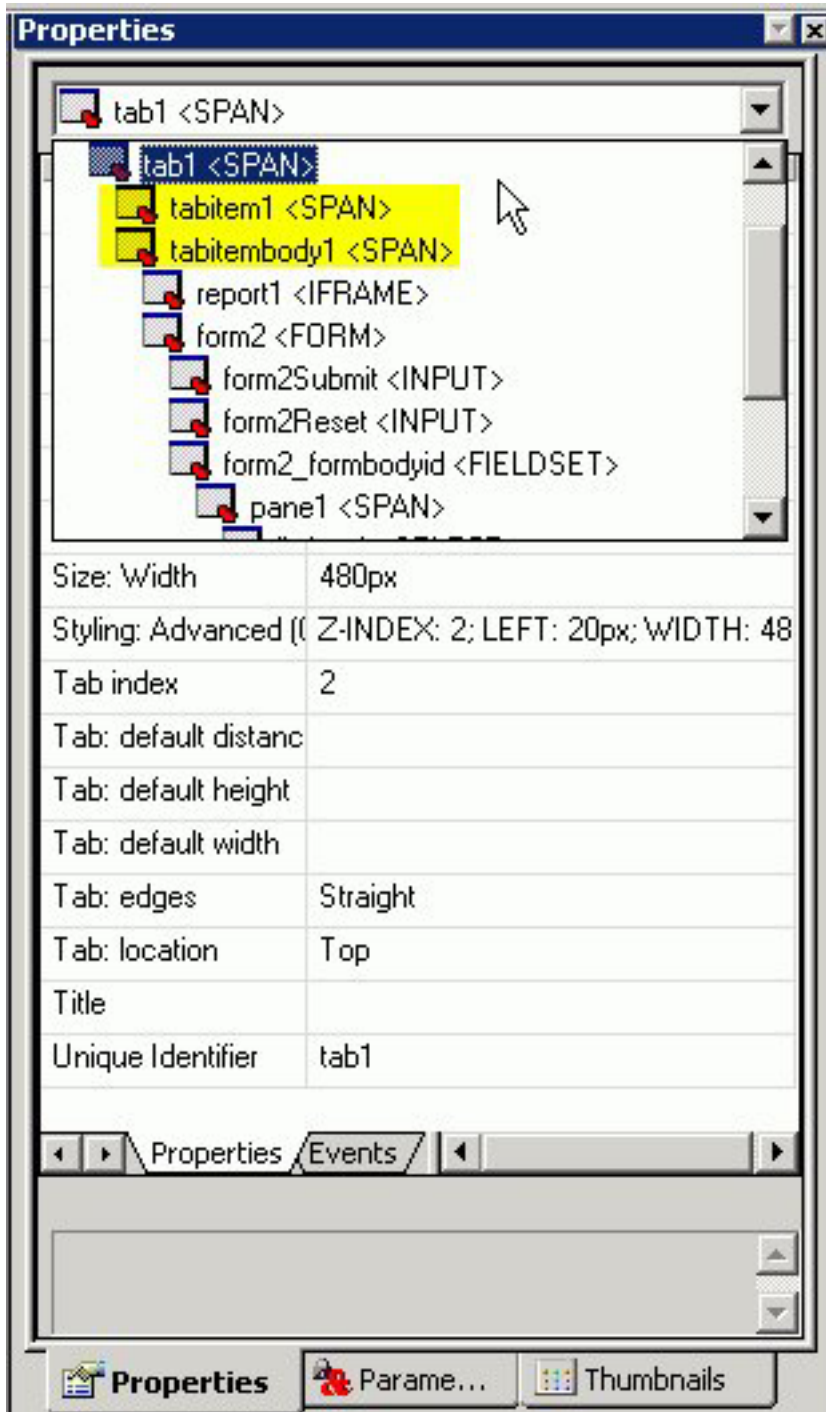
Click a property to display a description of the selected property at the bottom of the Properties window.

Modify and Style the Tabs

You may modify and style the tab item(s) and tab body properties.

1. Use the Formatting toolbar to format the text in the tab item.
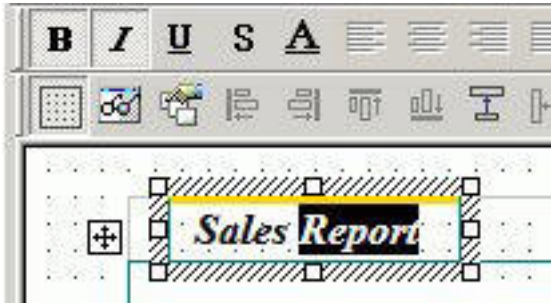
2. Click the tab item/tab body and use the right-click context menu to edit the text (if applicable), Font Style, Style, and Properties.
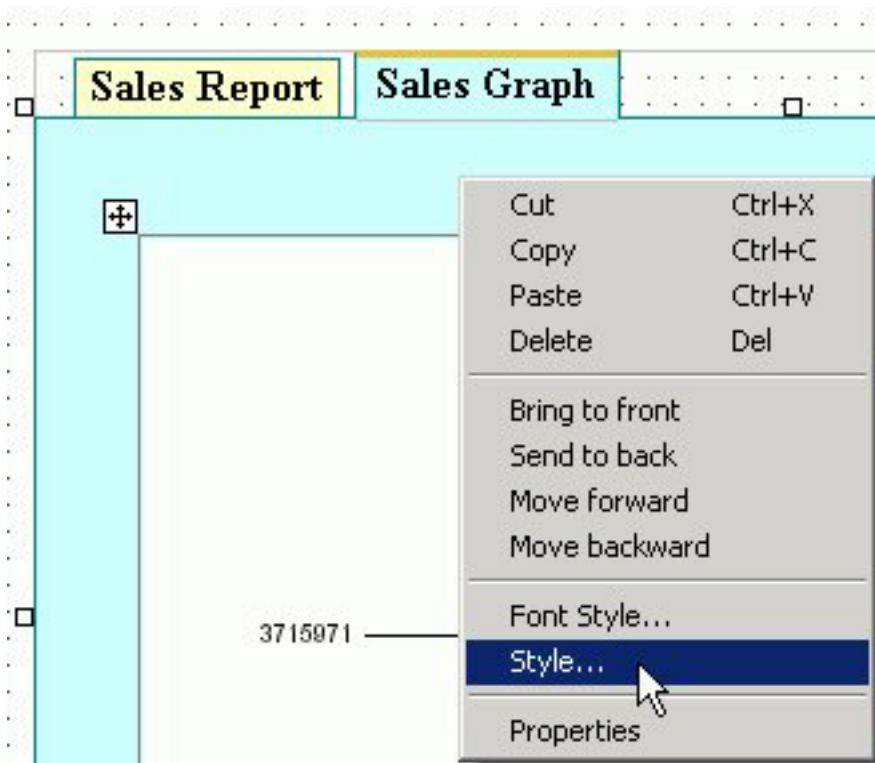
Modify the Size, Appearance, and Location of the Tabs

When the tab control is selected, you may change the default size of the tab labels, the appearance of the tabs, and the location of the tab items on the tab control.

Note: These properties are available from the Properties tab of the Properties window when the tab control is selected. The tab control appears as tab(n) in the Properties window drop-down list.

1. To change the default size of the tab labels, adjust the **Tab: default distance**, **Tab: default height**, and **Tab: default width** properties.

2. To change the appearance of the tabs, select Straight or Round from the **Tab: edges** properties field.

   Note: The default tab edge is Straight.

3. To change the location of the tab items on the tab control, select Top, Bottom, Left, or Right from the **Tab: location** properties field.

The default tab location is Top.

Note: If the tab location is changed, any background images applied to the tabs will not be rotated. You will have to reinsert a different image that is rotated appropriately.

Use the Tab Item Background Properties Field

In addition to using the Style Composer, you can add background images to tab items by using the Background properties field.

The Background properties field is available from the Properties tab of the Properties window when the tab item is selected. The tab item appears as tabitem*(n)* in the Properties window drop-down list.

1. Select the tab item in the layout, or click the tabitem*(n)* property from the Properties window drop-down list.
2. Click the **Background** ellipse button from the tab item properties window.

   The Get source file dialog box opens, as shown in the image below.
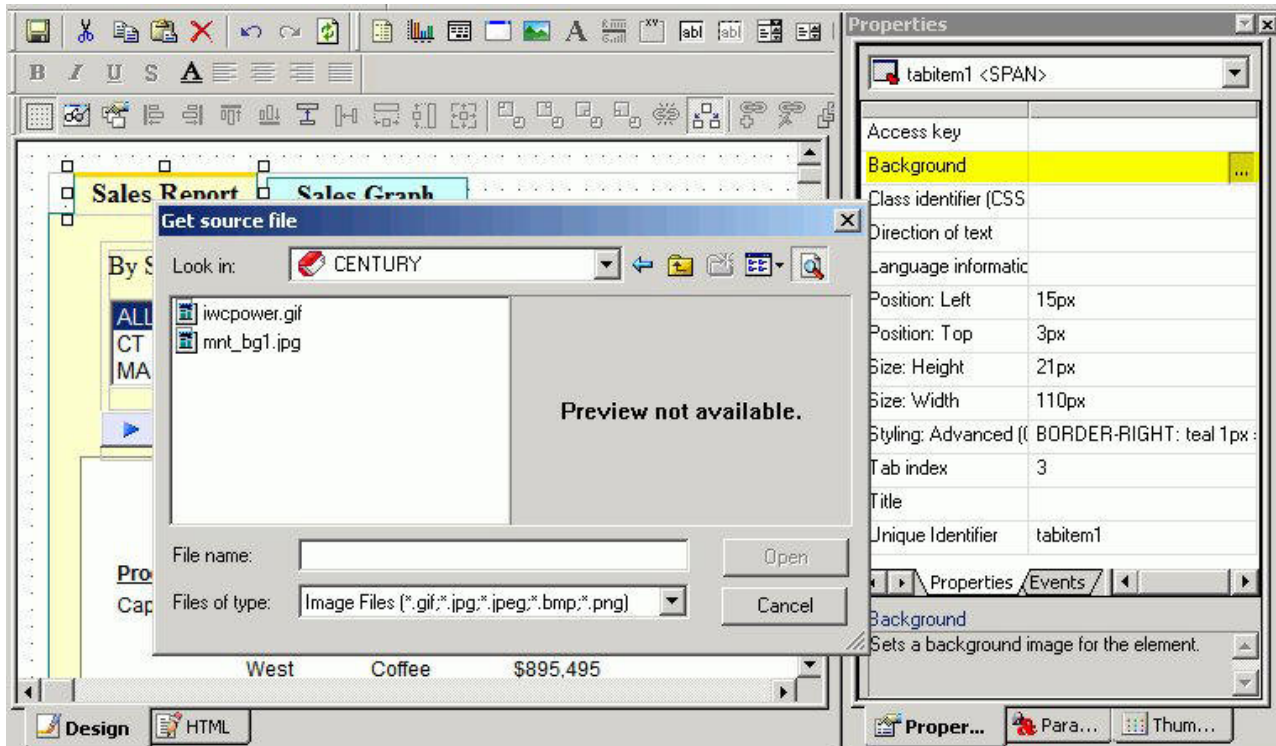
3. Select a File name and click **Open**.

   The background image is added to the tab item.

   Note: You may have to manually resize the tab item to fit the image.

   Tip: If the tab location is changed (from Top to Left for example), any background images applied to the tabs will not be rotated. You will have to reinsert a different image that is rotated appropriately.

Add Background Images to Tabs

You may add background images to a tab item or tab body using the Style Composer. For example, you may add a small icon with text to a tab item; or your company logo as the background image for a tab body.

Note: Background images can also be applied to tab items by using the Background Property setting. For more information, see Use the Tab Item Background Properties Field.

1. Select **Style** from the right-click context menu of the tab item/tab body.

   The Style Composer dialog box appears.

2. Select **Background** to show the Background image options.

3. Select a source file in the **Image** field and adjust the Tiling, Scrolling, and Position options.

   Tip: If adding an icon to a tab item, select a small image and do not tile the image.
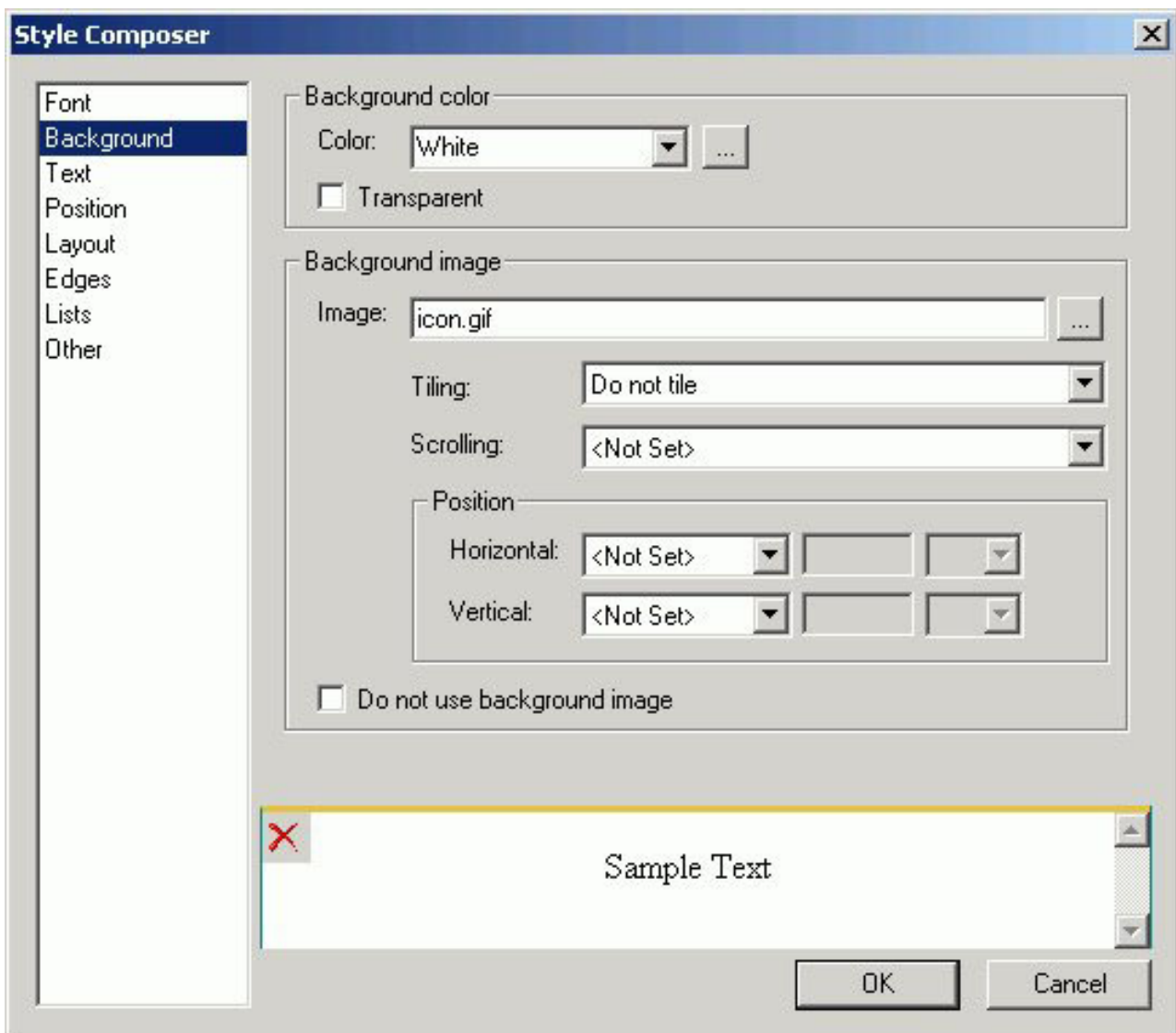
4. If using text in addition to a background image for a tab item, select **Text** from the Style Composer to view and change the Alignment options.

5. Click **OK** to close the Style Composer.

   The background image is added to the tab item/tab body.

Associate Components to the Tab Body

You may associate any component from the Insert menu (such as an image, line, and so on) to the tab body. This procedure details how to add a report/graph object component to the tab body.

1. To add a new report/graph object to the tab body:

   - Select **New Report** or **New Graph** from the Insert menu.

     The cursor changes into a crosshair.

   - In the tab control body, click and drag the crosshair to create the report/graph object and adjust it to the size you want.

   - Open, import, or reference a report or graph procedure.

2. To associate an existing component in the layout to a tab body:

   - Select the component in the layout.

   - Hold down the **Alt** key on your keyboard and drag the component into the tab body.

   The component is associated to the tab body.

Using Form Controls to Supply Parameter Values

Form controls allow you to prompt users for a parameter value. When you create a parameter as part of a report or graph, the HTML Layout Painter automatically adds a form control, Submit button, and Reset button for the parameter to your layout, and the parameter appears in the parameters box. You can also add a form control and then associate it with a parameter.

When you delete a parameter in the Report Assistant that was assigned a form control in the HTML Layout Painter, you must delete the associated controls individually.

Form controls, with the exception of a text box which does not supply a list of possible values, can supply values with a dynamic or static list of values:

- A dynamic list retrieves values from a specified data source when the request is run.

- A static list consists of a list of values you supply. These values do not change unless you change them.

- An active report control lists Active Report values that mimic Active Report menu items.

  Note: Active Report Controls cannot be associated to any parameters in the layout. This type of control can only be associated with an Active Report in the Layout.

The properties of a form control, as well as the parameters associated with each form control, can be controlled with the Properties tab and Parameters tab of the Properties window. For details, see Working With the Properties Window and Working With the Parameters Tab in the Properties Window, respectively.

For details about static and dynamic form controls, see Creating a Static List of Parameters and Creating a Dynamic List of Parameters.

For details about Active Report form controls, see Creating Active Dashboards.

Types of Form Controls

Form controls can be single or multi-select.

Examples of single-select form controls are:
- **A text box.** For details, see Using a Text Box.
- **A drop-down list.** For details, see Using a Drop-Down List.
- **Radio buttons.** For details, see Using Radio Buttons.
- **A text area.** For details, see Using a Text Area.

Examples of multi-select form controls are:
- **A list box.** For details, see Using a List Box.
- **Check boxes.** For details, see Add a List Box.
- **A tree control.** For details, see Using Tree Controls.

The default control type for all single select parameters is a drop down list. The default control type for all multi-select parameters is a list box.

You can change the type of form control using the Parameters tab. For details, see Change the Type of Form Control Associated With a Parameter.

Note: Individual multi-select parameters must be designated as multi-select in the Report Assistant in order to process multiple values.

Using a Text Box

A text box enables you to enter a parameter's value in a text entry field.

Create a Text Box
1. Add a text box by doing one of the following:
   - Click the **Text Box** button.
   - From the Insert menu, select **Text box**.

   The cursor changes to a crosshair.
2. Click and drag the crosshair to create a text box and adjust it to the size you want.

   A text box is created in the layout and assigned the name edit*n*, where *n* is a number.
3. To associate an existing parameter with the text box, click the **Parameters** tab of the Properties window, find the row with the name of the text box (edit*n*), click the Parameter field in that row, click the down arrow to the right of Unassigned at the edge of the Parameter field, and select the parameter you want to associate with the text box.
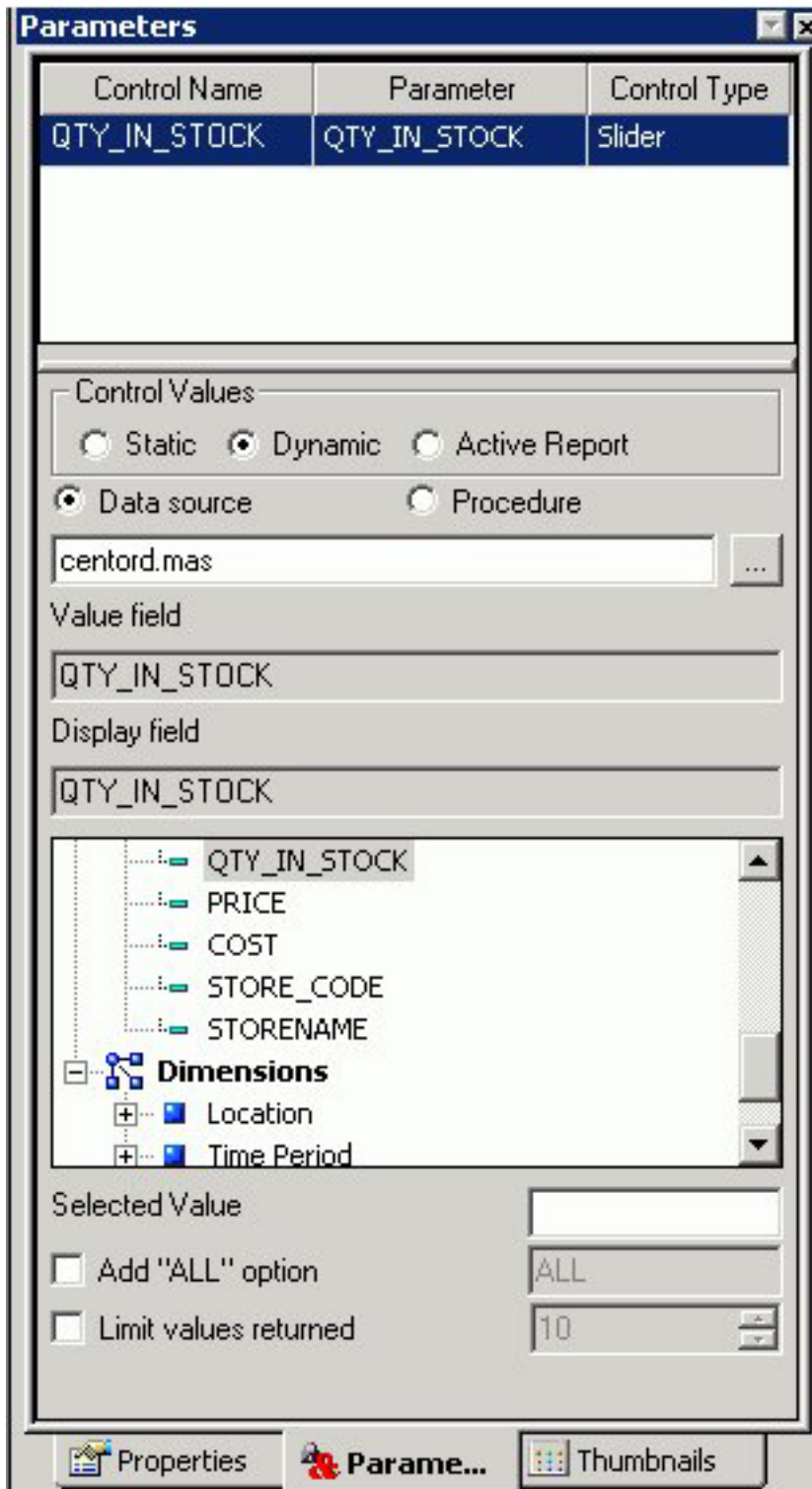4. Optionally, enter a value for the text box.
5. Optionally, change the text box's properties by adjusting the properties displayed in the Properties tab of the Properties window. For details, see Text Box Properties in the Properties Window.

Enter Masked Text in a Text Box

When entering a value in a text box at run time, you may set the mask text property so that the text is not displayed as text, but masked by default characters. This is recommended when using passwords or other sensitive information.

1. Select the Text Box object to view the associated properties.

   You may also select the text box properties from the drop-down list of the Properties window. The properties for the text box appears as edit*n* <INPUT>.

2. Required: From the Mask text property field, select **Yes**.
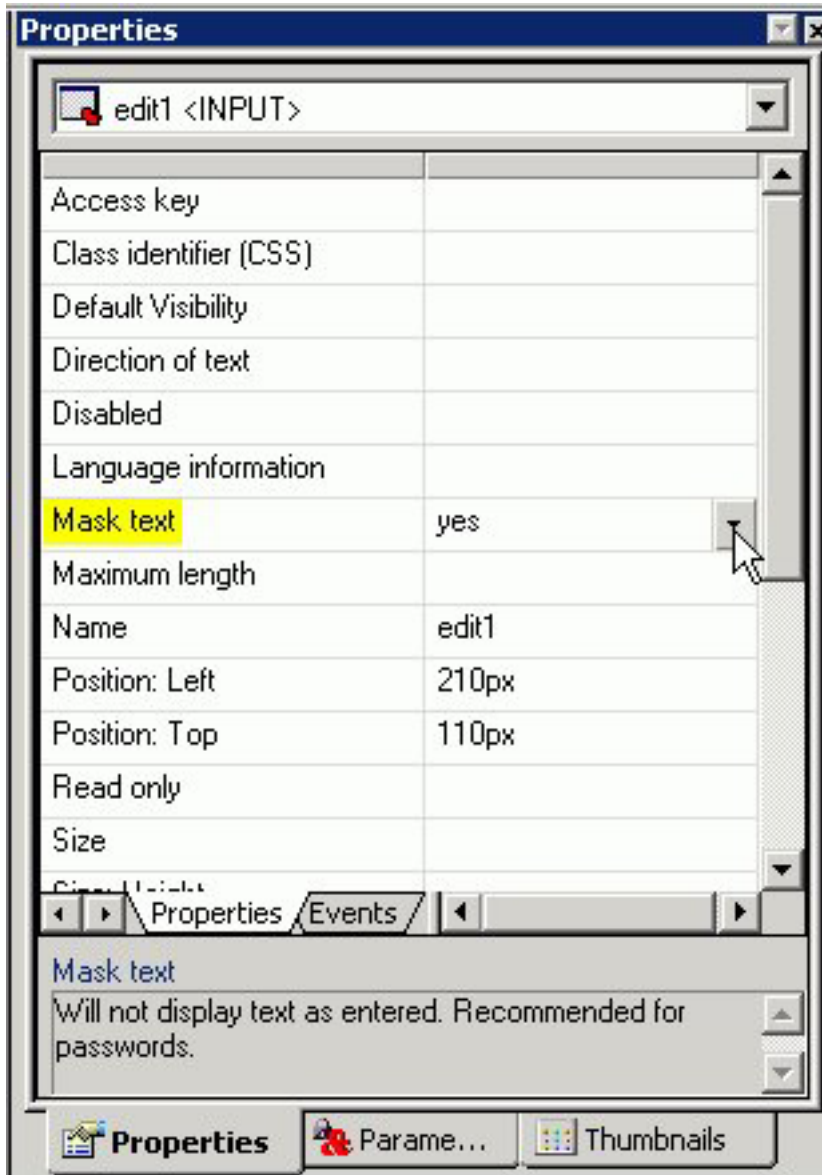
3. Run the report and enter a value in the text box.

The value being entered appears as masked text, as shown in the image below.

Enter Salary: ●●●●●●●

Salary Increase: Production Department

LAST_NAME	FIRST_NAME	SALARY	PCT_INC
BANNING	JOHN	$29,700.00	.00

Text Box Properties in the Properties Window

When a text box is selected, the Properties tab in the Properties window contains options that control the properties of your text box.

edit1 <INPUT>	▼
Access key	
Class identifier (CSS)	
Default Visibility	
Direction of text	
Disabled	
Language information	
Mask text	yes
Maximum length	
Name	edit1
Position: Left	210px
Position: Top	110px
Read only	
Size	
Size: Height	
Size: Width	190px
Styling: Advanced (CSS)	Z-INDEX: 2; LEFT: 210px; WIDT
Styling: Font	
Tab index	2
Title	
Type	
Unique Identifier	edit1
Value	

◄ ► \ Properties / Events / ◄ ►

Access key
ACCESSKEY: Assigns and access key to the element.

📄 **Properties** | 🔧 Parame... | ▦ Thumbnails

Click a property to display a description of the selected property at the bottom of the Properties window.

Using a Text Area

A text area is a single-select form control that enables you to enter multiple lines of text that can be assigned to a single variable. The behavior is similar to a text box, but you are not restricted to entering just one line of text. For example, if you want to assign a paragraph (multiple lines of text) to a variable that can be referenced by a procedure, you can add the paragraph to a text area from the Parameters tab of the Properties window.

The following image shows the text area component in the HTML Layout Painter.
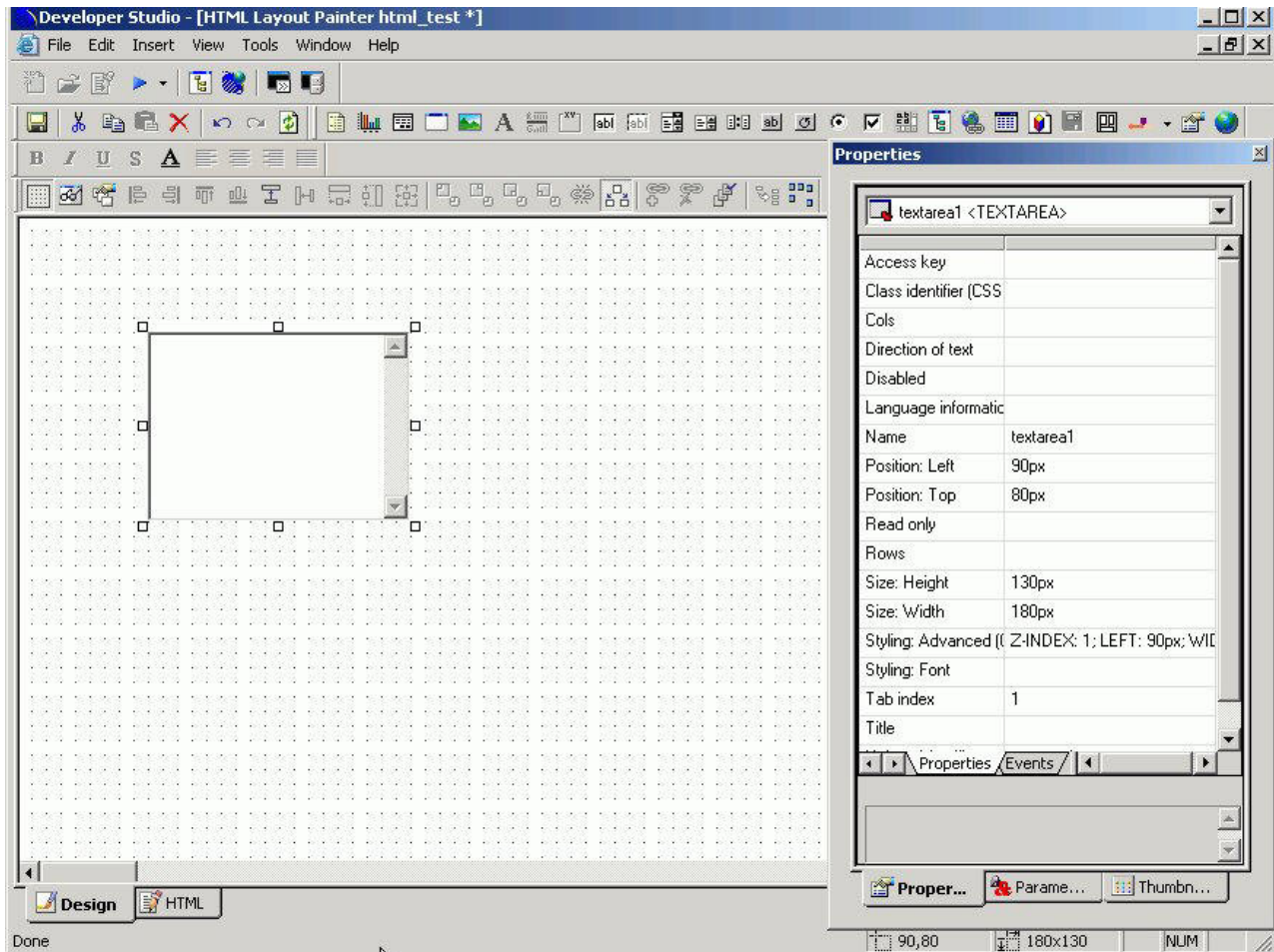
Create a Text Area

1. Required: Click the **Text Area** button.

   or

   From the Insert menu, select **Text Area**.

   The cursor changes to a crosshair.

2. Click a location on the layout page, drag the crosshair to create a rectangular text area, release the left mouse button, and adjust the text area to the desired size.

   A text area is created in the layout and assigned the name textarea*(n)*, where *n* is a number.

3. To associate values from an existing parameter with the text area, click the **Parameters** tab in the Properties window, find the row with the name of the text area (textarea*n*), click the Parameter field in that row, click the down arrow to the right of Unassigned at the edge of the Parameter field, and select the parameter you want to associate with the text area.

4. To associate static text values or dynamically retrieved data source values, select the **Static** or **Dynamic** option in the Control Values section of the Parameters tab.
   - **Static** - To add one or more lines of text as static values, type or paste text in the area below the Static values label in the Parameters tab.
   - **Dynamic** - To add dynamic values, select the **Data source** option to retrieve values from a data source, or select the **Procedure** option to retrieve values from a procedure. Click the browse **(...)** button, select a source file from the Get source file dialog box, and click **Open**. For the Data source option only, select a data field from the tree structure displayed at the bottom of the Parameters tab.

5. Optionally, you can change the text area's properties in the Properties tab of the Properties window.

Text Area Properties and Events in the Properties Window

When a text area is selected on the layout page and the Properties tab is selected in the Properties window, properties are listed in the left column that can be controlled from the right column. Selecting an option in the left column displays a description at the bottom of the Properties window.

A text area is associated with the following properties:

Access key, Class identifier, Columns, Direction of text, Disabled, break style, Language information, Name, Read only, Rows, Tabs Index, Title Unique Identifier

When a text area is selected on the layout page and the Events tab is selected within the Properties tab in the Properties window, events are listed in the left column that can be assigned an action from the right column.

A text area is associated with the following events:

onblur, onchange, onclick, ondblclick, onfocus, onkeydown, onkeypress, onkeyup, onmousedown, onmousemove, onmouseout, onmouseover, onmouseup, onselect

Optionally, you can change the text area's properties and parameters. For details, see Working With the Properties Window and Working With the Parameters Tab in the Properties Window.

Using a Drop-Down List

A drop-down list enables you to select a single value from a list of supplied values. You can use either a dynamic drop-down list, or a static drop-down list.

Add a Drop-Down List

1. Required: Add a drop-down list by doing one of the following:
   - Click the **drop-down list** button.
   - From the Insert menu, select **drop-down list**.

The cursor changes to a crosshair.

2. Required: Click and drag the crosshair to create a drop-down list, and adjust it to the size you want.

   A drop-down is created in the layout and assigned the name combobox*(n)*, where *n* is a number.

3. Required: To associate an existing parameter with the drop-down list, click the **Parameters** tab of the Properties window, find the row with the name of the drop-down list (combobox*(n)*), click the Parameter field in that row, click the down arrow to the right of Unassigned at the edge of the Parameter field, and select the parameter you want to associate with the drop-down list.

4. In the Control Values section of the Parameters tab, select static or dynamic values for the drop-down list. For details, see Associating Form Controls With Parameters.

5. Optionally, change the drop-down list's properties by adjusting the properties displayed in the Properties tab of the Properties window. For details, see Drop-Down List Properties in the Properties Window.

Drop-Down List Properties in the Properties Window

When a drop-down List is selected, the Properties tab in the Properties window contains options that control the properties of your drop-down list.
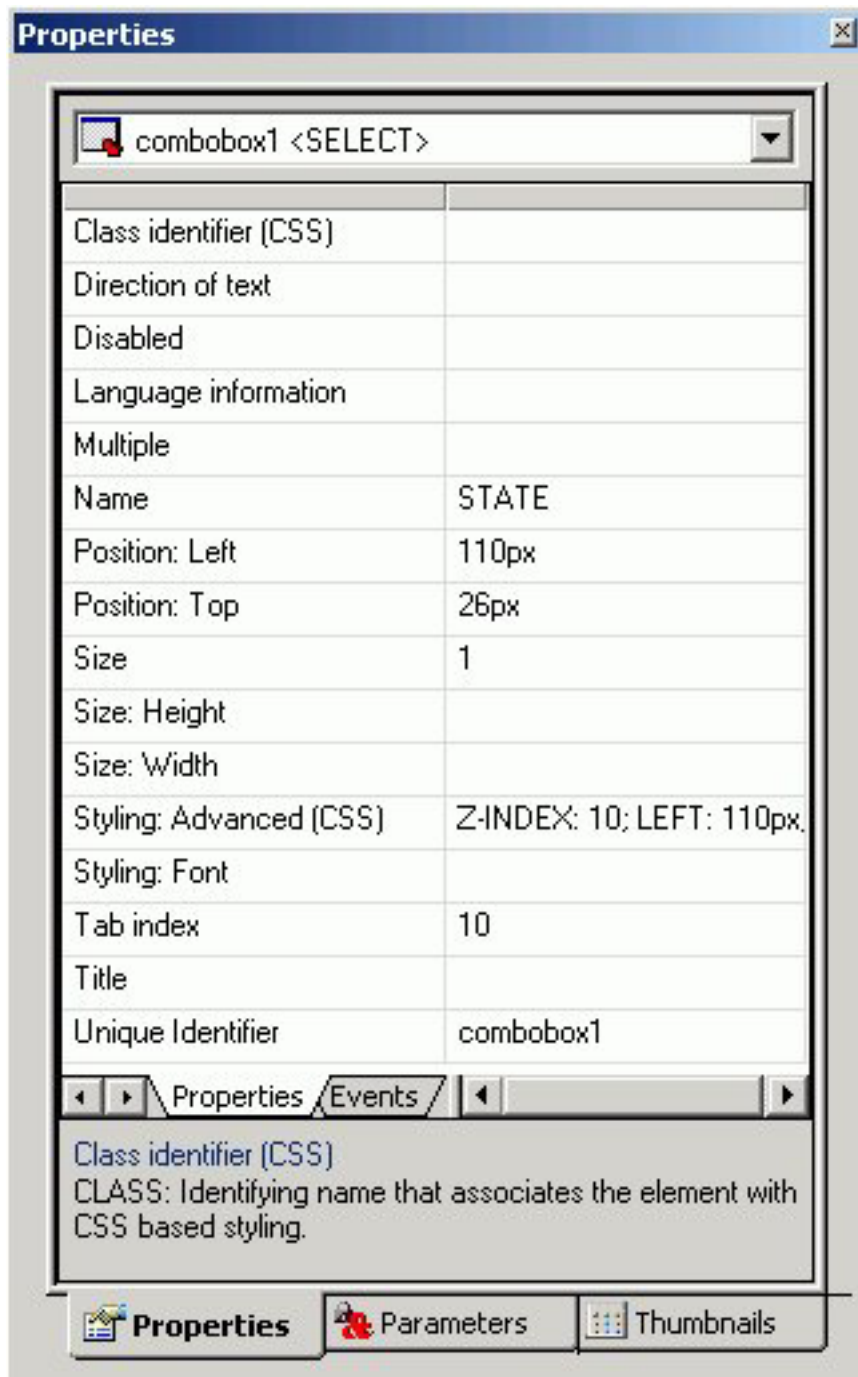
Click a property to display a description of the selected property at the bottom of the Properties window.

Using a List Box

A list box enables you to select single or multiple values at one time:

- A single select list enables you to select only one value for each time a request is run.
- A multi-select list enables you to select multiple values by using the Ctrl key while selecting values. In order to provide multiple values, the procedure must be set up to accept multiple values.

List box values can be dynamic or static.

Add a List Box

1. Add a list box by doing one of the following:
   - Click the **list box** button.
   - From the Insert menu, select **List box**.

   The cursor changes to a crosshair.

2. Click and drag the crosshair to create a list box, and adjust it to the size you want.

   A list box is created in the layout and assigned the name listbox(*n*), where *n* is a number.

3. To associate an existing parameter with the list box, click the **Parameters** tab of the Properties window, find the row with the name of the list box (listbox(*n*)), click the Parameter field in that row, click the down arrow to the right of Unassigned at the edge of the Parameter field, and select the parameter you want to associate with the list box.

4. In the Control Values section of the Parameters tab, select static or dynamic values for the list box. For details, see Associating Form Controls With Parameters.

5. Optionally, change the list box's properties by adjusting the properties displayed in the Properties tab of the Properties window. For details, see List Box Properties in the Properties Window.

List Box Properties in the Properties Window

When a list box is selected, the Properties tab in the Properties window contains properties of your list box.
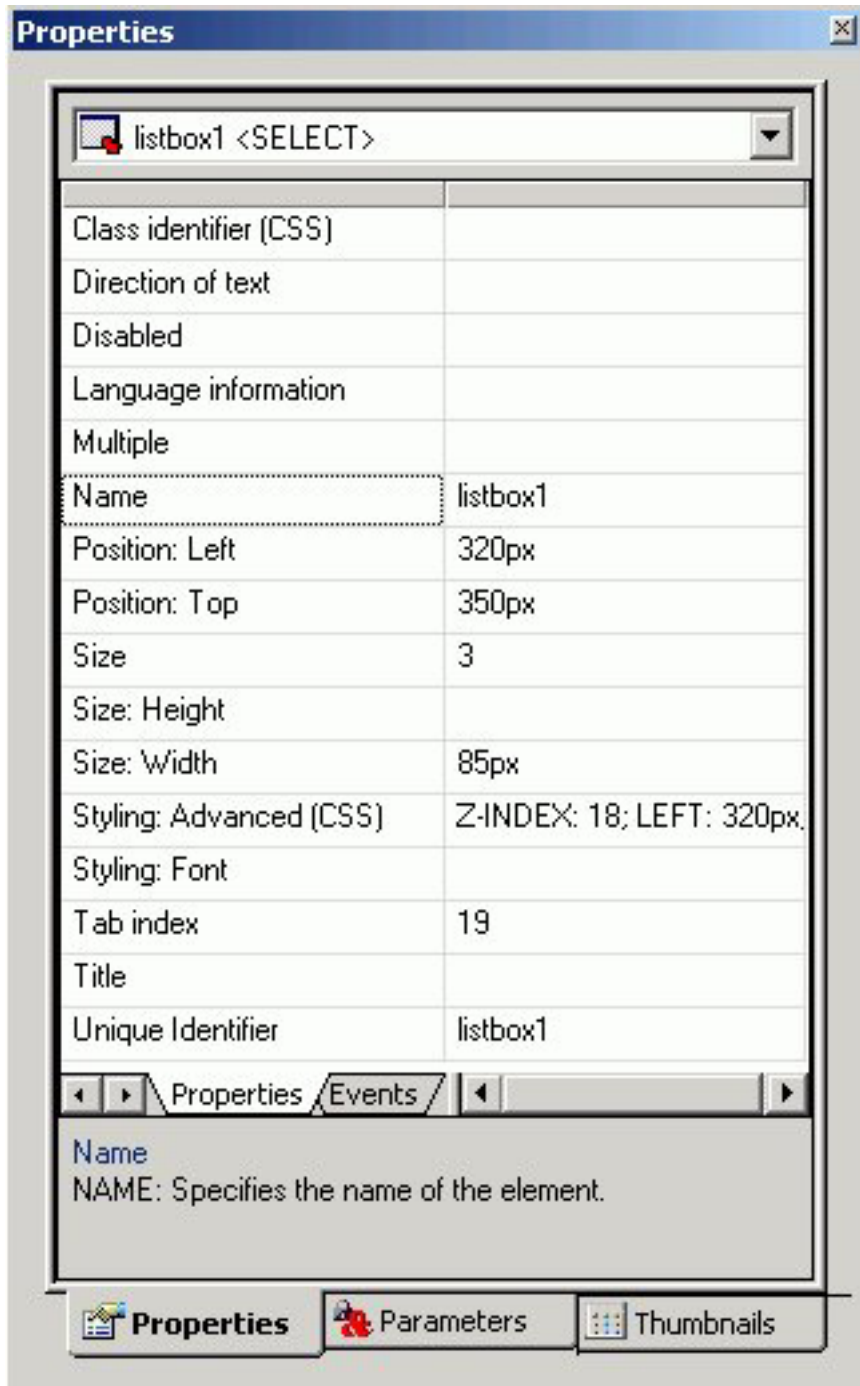
Click a property to display a description of the selected property at the bottom of the Properties window.

Using a Double List Control

You may add a double list form control for displaying multi-select parameter values. This enables you to view a list of the available values and add or remove them from one list to another. At run time, a report is generated based on the values that are added.

Add a Double List Form Control

1. Add a double list control by doing one of the following.
   - Click the **Double List control** button from the Components toolbar.
   - From the Insert menu, select **Double List Control**.

   The cursor changes to a crosshair.

2. Click and drag the crosshair to create a double list control, and adjust it to the size you want.

   A double list control is created in the layout and assigned the name customselect*(n)*_selectfrom and customselect*(n)*_selectto, where *n* is a number.

3. To associate an existing parameter with the double list control, click the **Parameters** tab of the Properties window, find the row with the name of the double list control (listbox*(n)*), click the Parameter field in that row, click the down arrow to the right of Unassigned at the edge of the Parameter field, and select the parameter you want to associate with the double list control.

   Note: You may also add a double list control from the New Parameters dialog box that appears when creating a report with multi-select parameters. For more information about the New Parameters dialog box, see Setting Form Controls for New Parameters.

4. In the Control Values section of the Parameters tab, select **Static** or **Dynamic** values for the double list control. For details, see Associating Form Controls With Parameters.

5. Optionally, change the double list control's properties by adjusting the properties displayed in the Properties tab of the Properties window. For details, see Double List Box Properties in the Properties Window.

6. Run the HTML page and select values by using the arrows to add or remove values to the selected column.

   Note: The selected values appear in the second column. The output is generated based on the selected values in the second column. In the example below, Action and Classic are the selected values for the report.
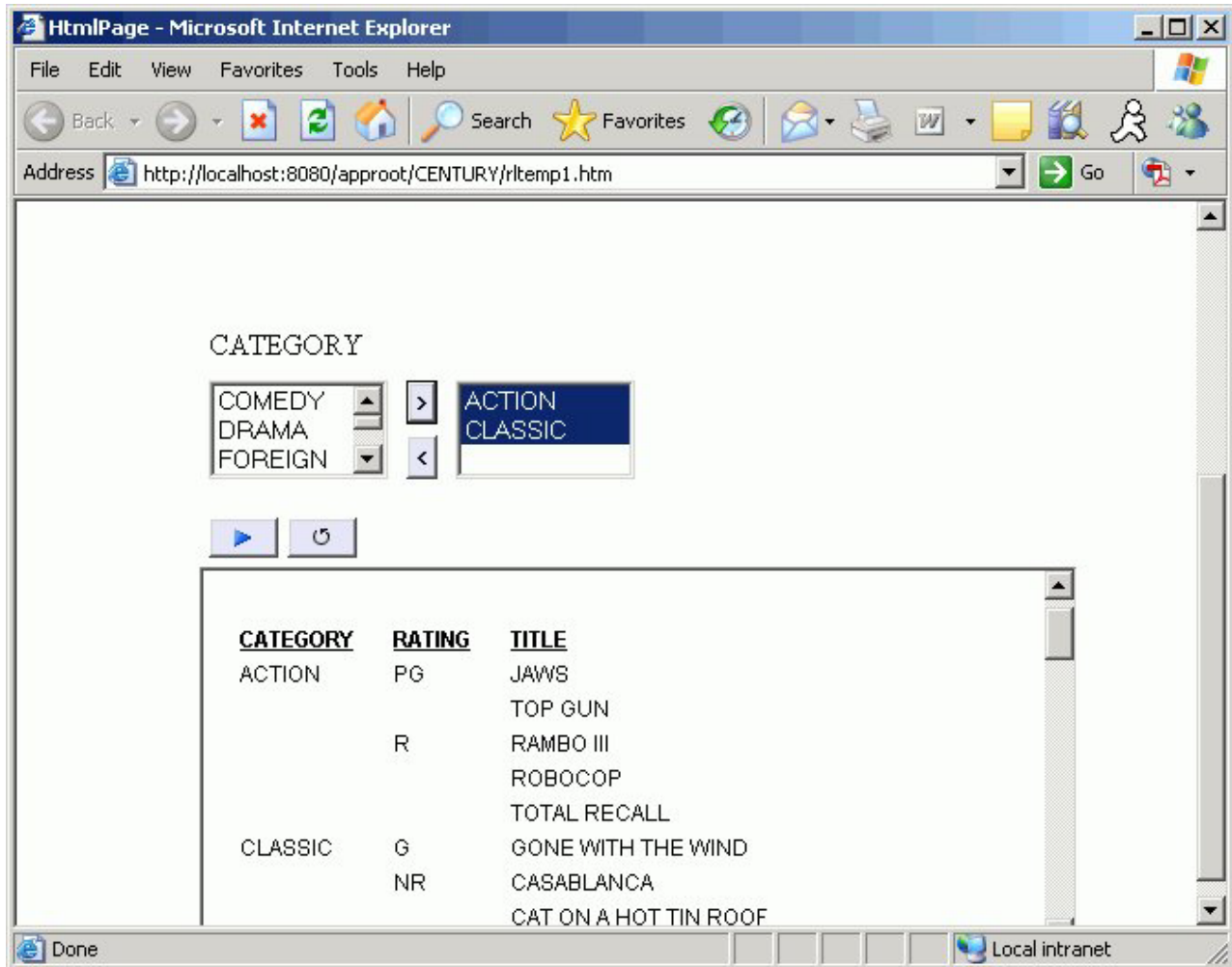
Double List Box Properties in the Properties Window

When a double list box is selected, the Properties tab in the Properties window contains properties of your double list control.
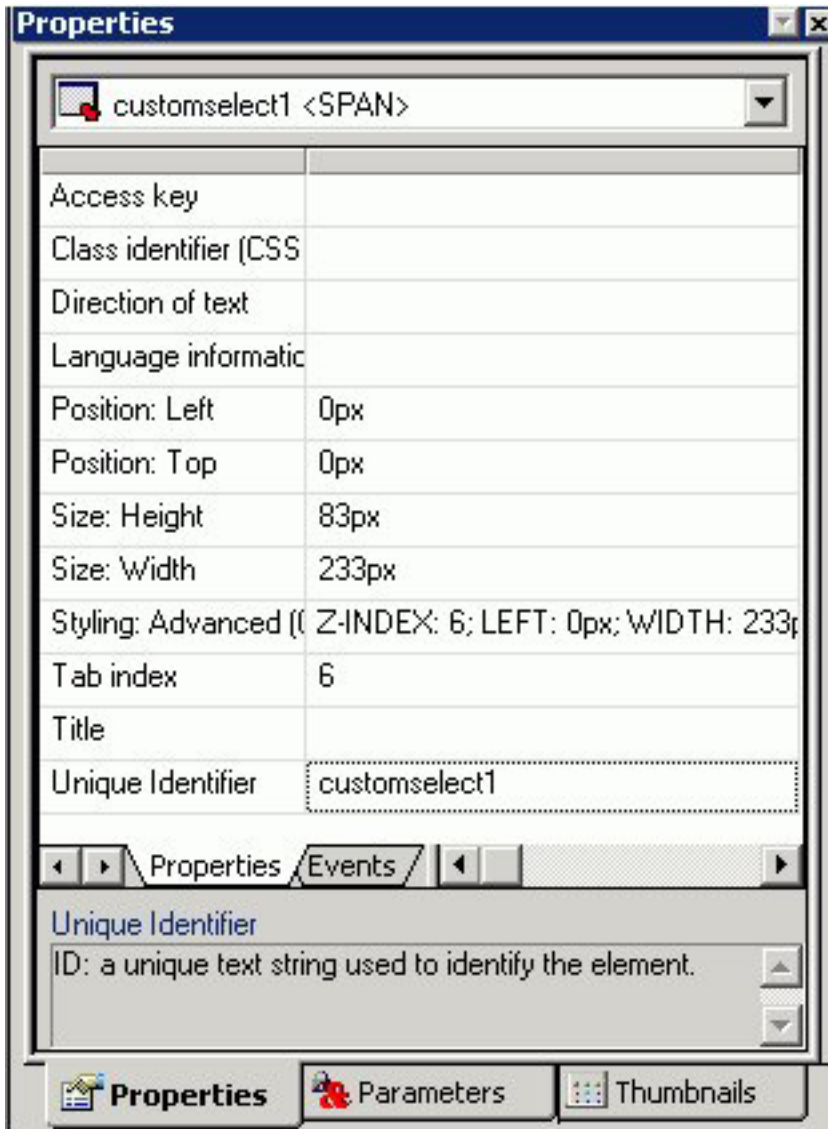
Click a property to display a description of the selected property at the bottom of the Properties window.

Using Check Boxes

Check boxes enable you to select values from a list of values. You can select single or multiple values at one time:

- A **single select** list enables you to select only one value each time a request is run.
- A **multi-select** list enables you to select multiple values. In order to provide multiple values, the procedure must be set up to accept multiple values.

Check box list values can be dynamic or static.

Add Check Boxes

1. Add check boxes by doing one of the following:
   - Click the **Check box** button.
   - From the Insert menu, select **Check box**.

The cursor changes to a crosshair.

2. Click and drag the crosshair to create a check box list and adjust it to the size you want.

   A check box is created in the layout and assigned the name checkbox*(n)*, where *n* is a number.

3. To associate an existing parameter with the check box, click the **Parameters** tab of the Properties window, find the row with the name of the check box (checkbox*(n)*), click the Parameter field in that row, click the down arrow to the right of Unassigned at the edge of the Parameter field, and select the parameter you want to associate with the check box.

4. In the Control Values section of the Parameters tab, select static or dynamic values for the check boxes. For details, see Associating Form Controls With Parameters.

5. Optionally, check box list's properties by adjusting the properties displayed in the Properties tab of the Properties window. For details, see Check Box Properties in the Properties Window.

Check Box Properties in the Properties Window

When a check box is selected, the Properties tab in the Properties window contains properties of your check box.

Properties	
checkbox1 <SPAN>	
Access key	
Class identifier (CSS)	
columns	1
Direction of text	
Language information	
Position: Left	300px
Position: Top	450px
Size: Height	70px
Size: Width	110px
Styling: Advanced (CSS)	Z-INDEX: 19; LEFT: 300px.
Tab index	20
Title	
Unique Identifier	checkbox1

Properties / Events

Position: Left
Defines the 'left' positioning property (CSS) by setting the distance of the element's left edge from the left of the document or of the parent element.

Properties   Parameters   Thumbnails

Click a property to display a description of the selected property at the bottom of the Properties window.

Using Radio Buttons

Radio buttons enable you to select a single value from a list of supplied values. Radio button values can be static or dynamic.

Add Radio Buttons

1. Add radio buttons by doing one of the following:
   - Click the **Radio** button.
   - From the Insert menu, select **Radio button**.

   The cursor changes to a crosshair.
2. Click and drag the crosshair to create a radio buttons list and adjust it to the size you want.

A radio button placeholder is created in the layout and assigned the name radio*(n)*, where *n* is a number.

3. To associate an existing parameter with the radio buttons, click the **Parameters** tab of the Properties window, find the row with the name of the radio buttons (radio*(n)*), click the Parameter field in that row, click the down arrow that appears to the right of Unassigned at the edge of the Parameter field, and select the parameter you want to associate with the radio buttons.

4. In the Control Values section of the Parameters tab, select static or dynamic values for the radio buttons. For details, see Associating Form Controls With Parameters.

5. Optionally, change the radio buttons' properties by adjusting the properties displayed in the Properties tab of the Properties window. For details, see Radio Buttons Properties in the Properties Window.

Radio Buttons Properties in the Properties Window

When a radio button is selected, the Properties tab in the Properties window contains options that control the properties of your radio buttons.
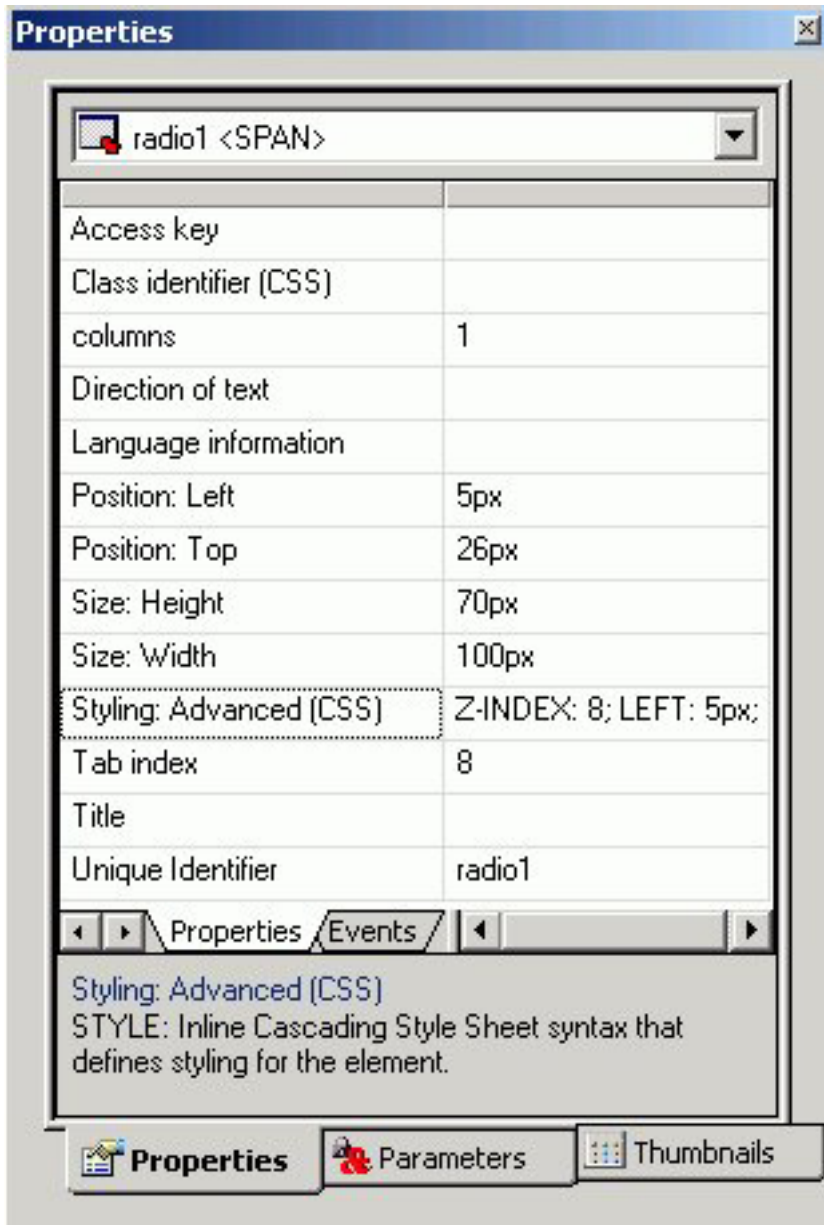
Click a property to display a description of the selected property at the bottom of the Properties window.

Using Tree Controls

You can show hierarchical data from a multi-dimensional data source (for example, SAP BW), by using a tree structure in an HTML report. This feature is only available for hierarchies that use the parent/child model; level hierarchies are not supported.

The behavior of the tree control is integrated with the parameter definition. If a parameter is defined as a single value and that parameter is assigned to a tree control, the control uses option buttons for each node in the hierarchy. If the parameter is defined as MULTIPLE OR then the control uses check boxes for each node in the hierarchy, enabling you to select multiple nodes.

The following image shows the Tree Control component in the HTML Layout Painter.
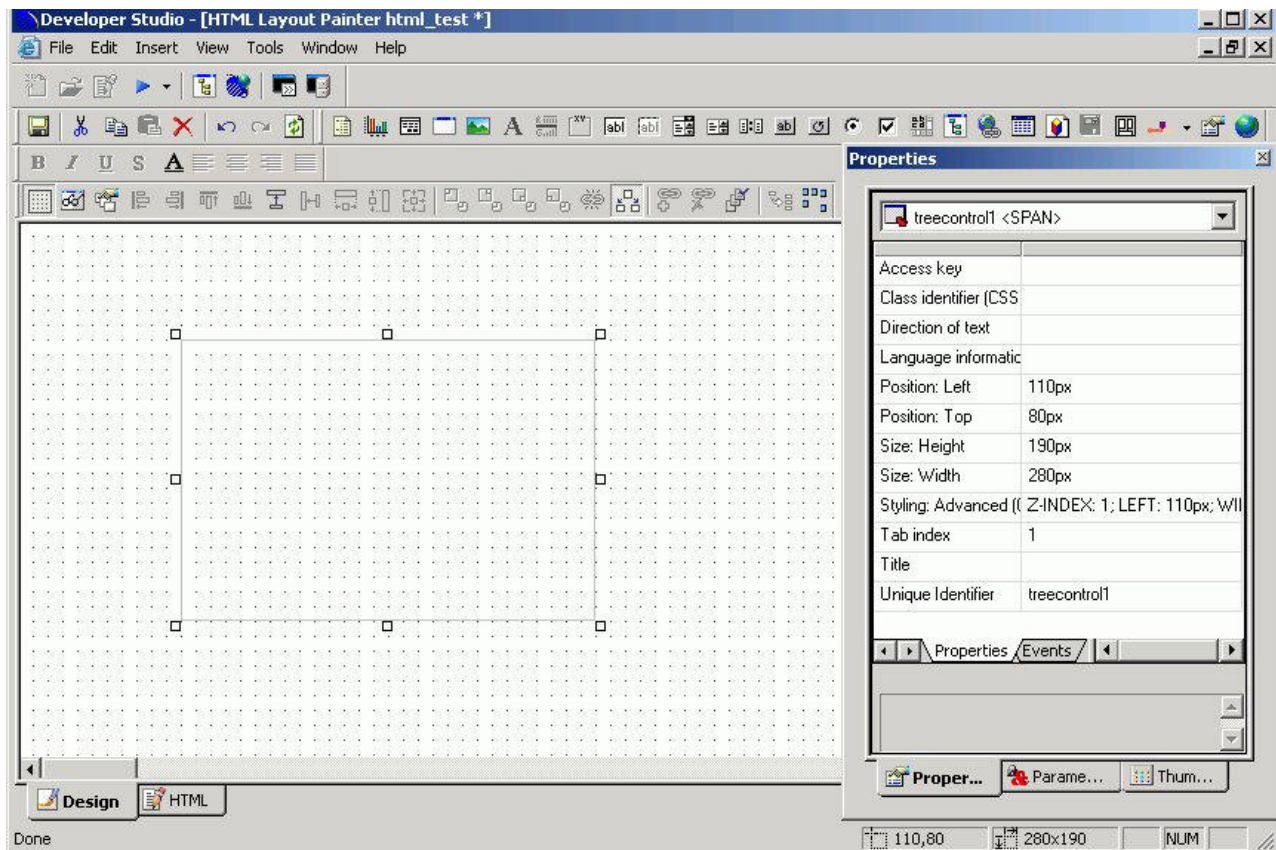
Add a Tree Control to an HTML Page Using a Data Source

1. Required: In the HTML Layout Painter, click the **Tree Control** icon from the toolbar.
2. Required: Click and drag your cursor over the canvas to select the area for the tree control.
3. Select the tree control and click the **Parameters** tab on the Properties window.
4. In the Control Values field, select **Dynamic**.
5. Select the **Data Source** option button and click the browse **(...)** button adjacent to the text box.
6. Select a multi-dimensional data source and click **OK**.
7. Expand the Dimensions folder, a Dimension, and the Hierarchies folder.
8. Select a field from the hierarchy and add it to the Value field text box and the Display field text box.

Add a Tree Control to an HTML Page Using an Existing Procedure

You can select an existing procedure to add to the tree control in an HTML report. When you select a procedure, it should use fields from the parent/child hierarchy and be set up as follows:

```
TABLE FILE file
SUM FST.dispfield
BY ParentUniqueField
BY UniqueField
BY datafield
ON TABLE PCHOLD FORMAT XML
END
```

where:

file

> Is the name of the data source.

dispfield

> Is the field whose values display in the tree control.

ParentUniqueField

> Is the field that represents the parent for the parent/child hierarchy
> (PROPERTY = PARENT_OF).

UniqueField

> Is the field that represents the unique IDs for the hierarchy members
> (PROPERTY=UID).

datafield

> Is the field whose values are passed as the parameter value.

After the procedure is set up, follow these steps:

1. In the HTML Layout Painter, click the **Tree Control** icon from the toolbar.
2. Click and drag your cursor over the canvas to select the area for the tree control.
3. Select the tree control and then click the **Parameters** tab on the Properties window.
4. In the Control Values field, select **Dynamic**.
5. Select the **Procedure** option button and click the browse **(...)** button adjacent to the text box.
6. Select a procedure and click **Open**.

Properties and Events for the Tree Control

The tree control has the following properties:

Class identifier, Direction of text, break style, Language information, Title, Unique Identifier

The tree control is associated with the following events:

onclick, ondblclick, onkeydown, onkeypress, onkeyup, onmousedown, onmousemove, onmouseout, onmouseover, onmouseup

Using a Slider Control

You may add a slider control for numeric range values in a report or graph. This enables you to use a slider bar to select from a range of values.

Add a Slider Control

1. Required: Add a slider control by doing one of the following.

    - Click the **Slider** button from the Components toolbar.
    - From the Insert menu, select **Slider**.

    The cursor changes to a crosshair.

2. Required: Click and drag the crosshair to create a slider control, and adjust it to the size you want.

    A slider control is created in the layout and assigned the name slider*(n)*, where *n* is a number.

    Note: The Slider control is determined by the default slider control type selected from the Settings Dialog Box. For details about changing the slider bar, see How to Change the Default Slider Bar.

3. To associate an existing parameter with the slider control, click the Parameters tab of the Properties window, find the row with the name of the slider control (slider*(n)*), click the Parameter field in that row, click the down arrow to the right of Unassigned at the edge of the Parameter field, and select the parameter you want to associate with the slider control.

    Note: You may also add a slider control from the New Parameters dialog box that appears when creating a report or graph with numeric range values. Slider is automatically selected as the control type if a numeric range parameter is detected. For more information about the New Parameters dialog box, see Setting Form Controls for New Parameters.

4. You may modify the static minimum and maximum values for the range by using the Parameters tab of the Properties window.
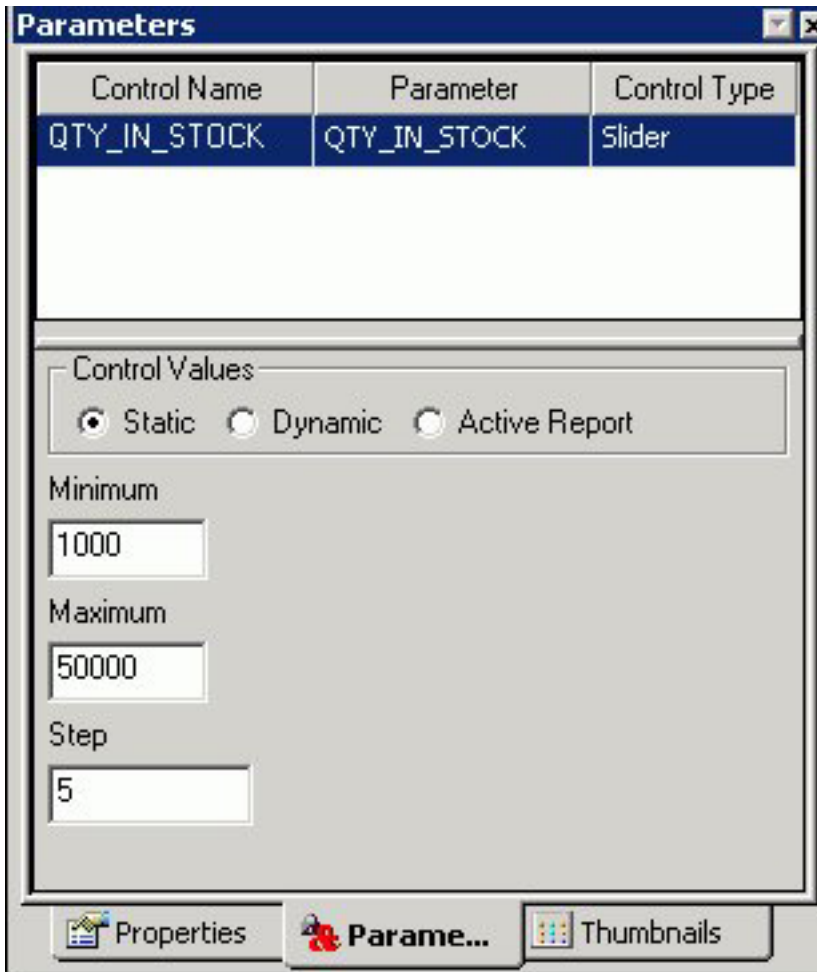
Note: Step indicates how the numbers increment on the slider bar.

5. In addition, you may change the range of values from static to dynamic by using the Parameters tab of the Properties window. Select a data file to retrieve values from.

Note: Click Ctrl + Shift to add a value to the Display field.

6. Optionally, change the slider control's properties by adjusting the properties displayed in the Properties tab of the Properties window. For details, see Slider Control Properties in the Properties Window.

7. Run the HTML page and use the slider bar to slide the parameter values up or down. You may also use the end arrows to increase or decrease the numbers in the range. The output is generated based on the selected number from the slider bar.
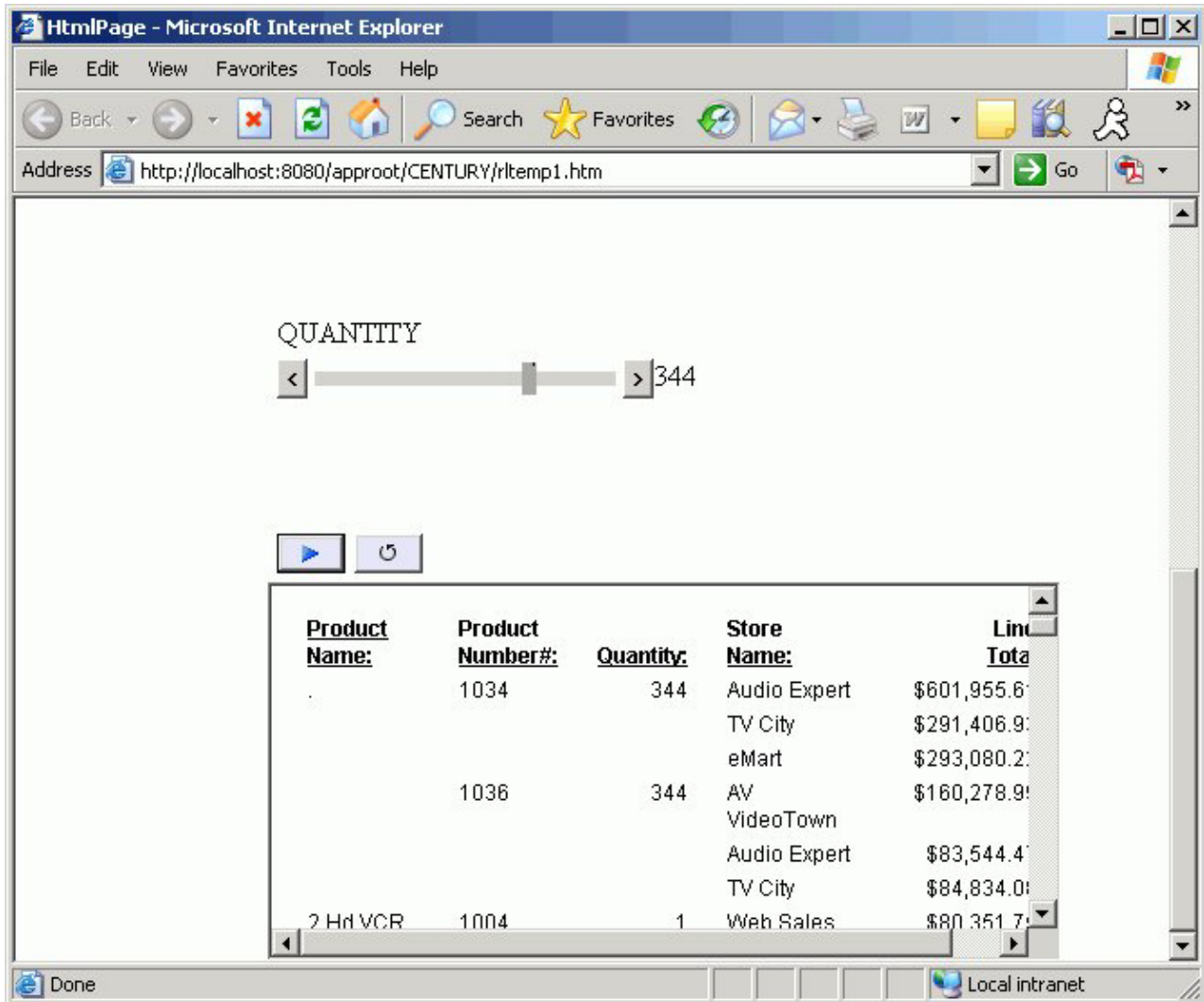
Change the Default Slider Bar

The default slider type is determined from the Settings dialog box.

1. To change the default slider type, select **Settings** from the Tool menu. The Settings dialog box opens.
2. Click the **Form's Settings** button to open the Form settings dialog box.
3. Use the drop-down list to change the default slider control type. You may select from:
   - Horizontal or Vertical Slider Simple bar. The slider bar has no end arrows, just a bar with the slider.
   - Horizontal or Vertical Slider with Color Bar and Arrows. Arrows appear at each end of the slider bar.
   - Horizontal or Vertical Slider with Color Bar, Arrows, and Edit. Arrows and an edit box appear at the end of the slider bar, showing the current value.

Note: Once a slider control is inserted into the HTML Layout, changing these options will not affect the existing sliders, the option is only applied to new slider controls. To change the existing slider bar type, delete the slider object and insert a new slider control.

Slider Control Properties in the Properties Window

When a slider control is selected, the Properties tab in the Properties window contains properties of your slider control.
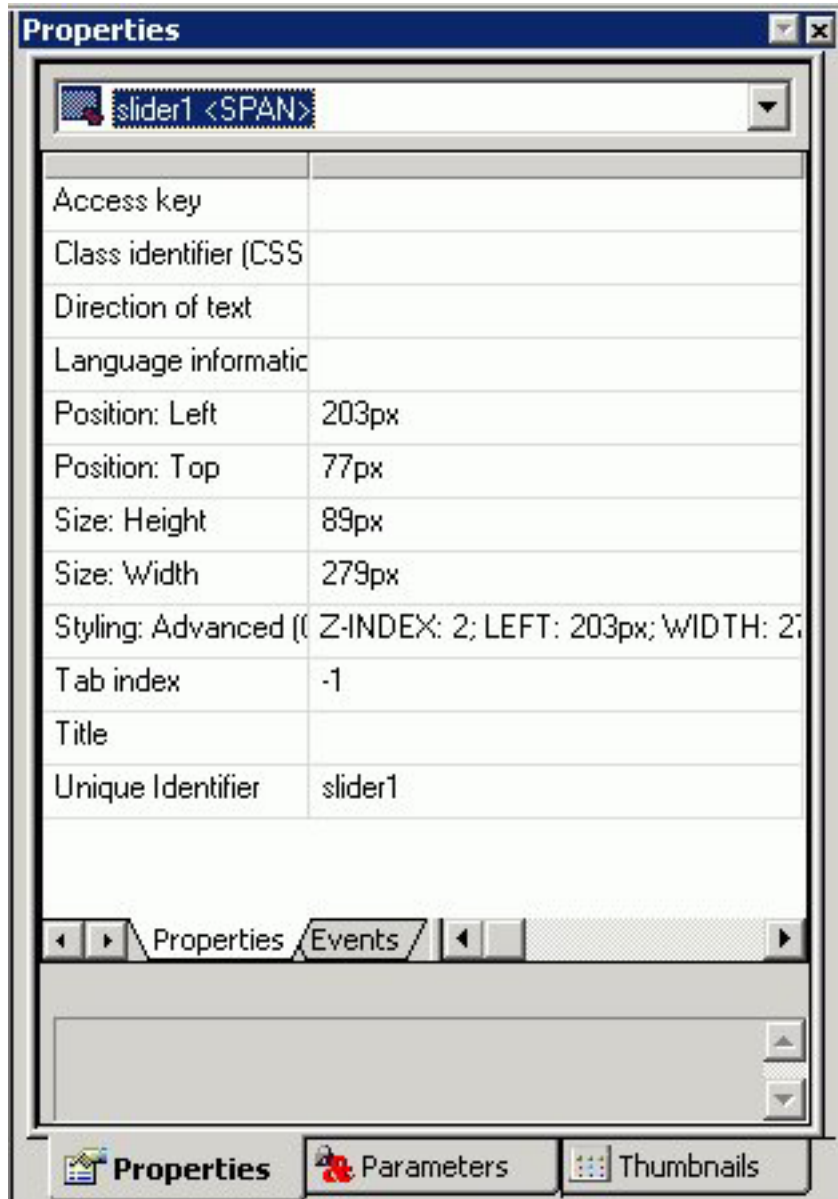
Click a property to display a description of the selected property at the bottom of the Properties window.

Adding a Dynamic Calendar

Date parameters can utilize a built-in calendar control that enables you to select the desired date or range of dates in a pop-up dynamic calendar. A procedure that is added to or referenced in the HTML Layout Painter and contains date parameters will have a Calendar control type available in the Properties tab of the Properties window.

When the Calendar control type is selected, a text box with a calendar icon will display in the Design View of the Layout. The text box is the only control available for the calendar, and the icon will always display to the right of the text box. The icon cannot be positioned independently from the text box.

Note that when programmatically returning a date to the calendar, the date must be in a FOCUS date format that specifies the complete date from the list of supported data types in Calendar Properties.

Set Calendar Properties

1. Right-click the calendar icon, and select **Calendar Properties**.

   The following menu appears.

This menu enables the developer to set the range of dates available to the user at run time. Available dates will be represented as an active hyperlink (blue and underlined) and unavailable dates will be static (black without underlines).

2. Select an option for setting the range of dates:

   - **Static.** This option will set a static date range in which the developer will select a start date and an end date using a pop-up calendar or by clicking the month, day, or year from the controls. This setting is the default.

     By default, the current date is selected and circled in red. As you scroll through the calendar with the left/right arrows, the currently selected day will remain highlighted for each month. Clicking a date will add that date to

the control. Dates can be selected by scrolling left to right, entering the month, day, and year as text, or by selecting the month, day, and year from the drop-down list and spin boxes.

- **Relative.** This option allows the developer to set a specific number of days, months, and years relative to the current date. The current date (at run time) will always be the reference or starting point and the calendar will show a number of days, months, and years relative to the current date. The range could be all in the past (for example, five years prior to the current date) or all in the future (for example, five years in the future).

- **Dynamic.** This option allows the developer to point to a procedure that returns a range of dates. Clicking the browse **(...)** button allows the developer to choose a preexisting procedure located in the current APP (data server) or Domain (MR). The procedure must return two date values on the same data line in XML format. The date values must be returned in a format that returns two digits for the month and day, and four digits for the year, for example MM/DD/YY.

Calendar Properties in the Properties Window

When a calendar is selected in the HTML Layout Painter, the Properties tab in the Properties window contains properties of that calendar.
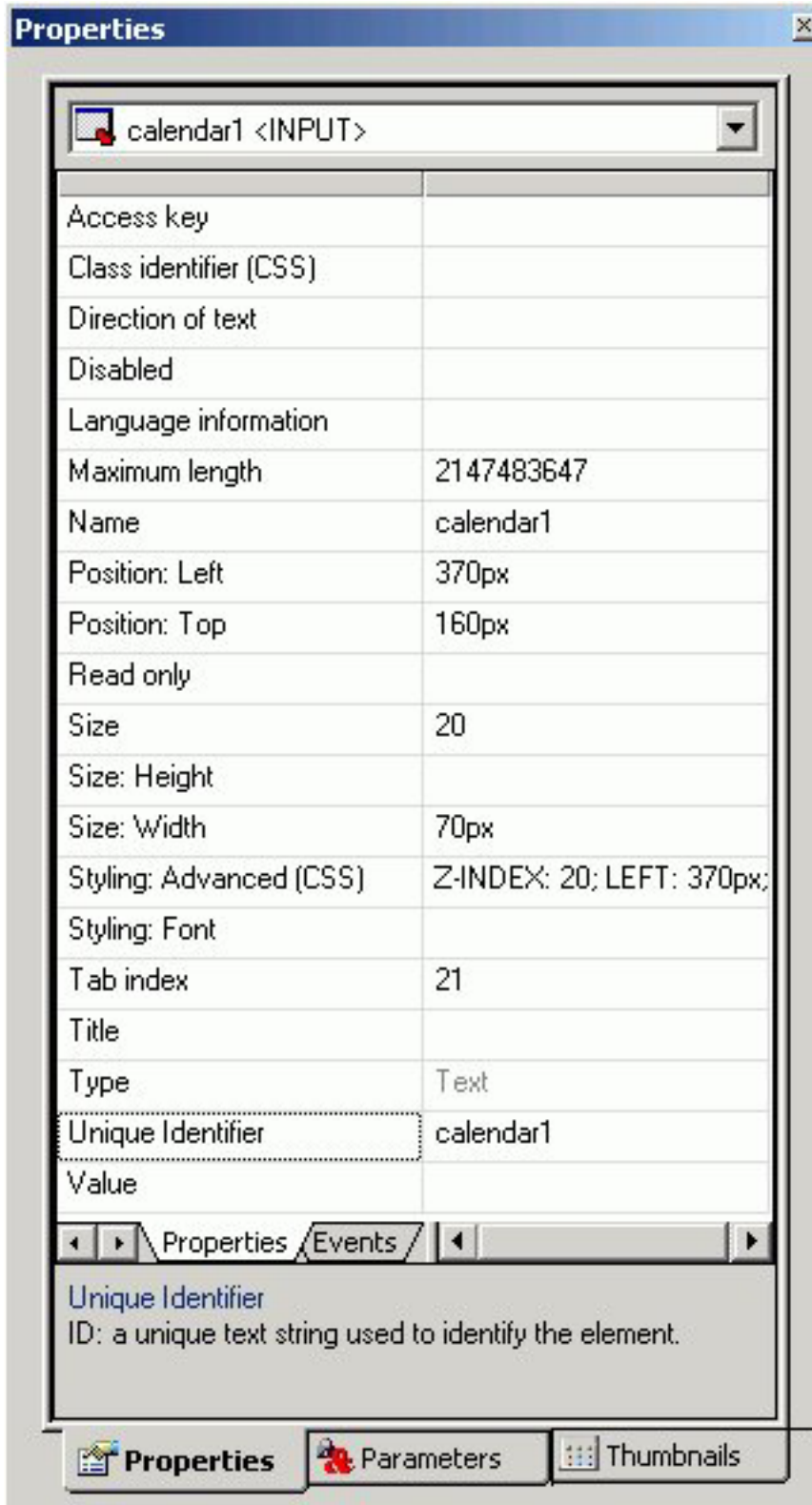
Click a property to display a description at the bottom of the Properties window.

Working With the Properties Window

The Properties window is a dockable window that consists of several tabs and components.

- A Properties tab consists of the following components:
  - Contains a hierarchical drop-down list of objects that are currently in the layout. For the selected object, there are additional subtabs on the Properties tab.

  - A Properties subtab that lists attributes for the selected object. The attributes appear on the left. Click in the right column to set the properties for the attribute.
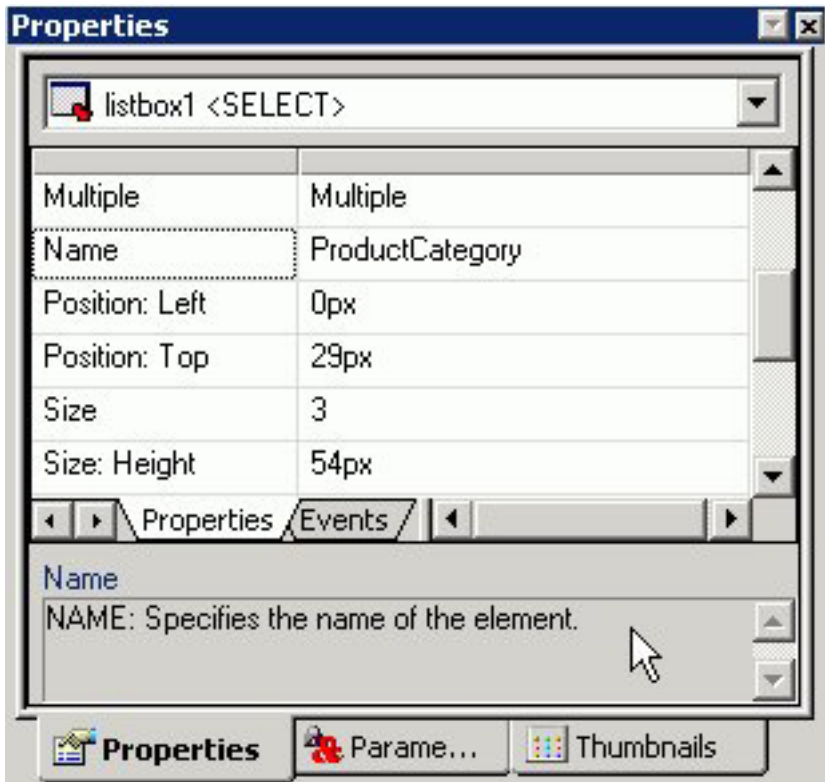
– An Events subtab lists all events that can be assigned an action for an object.

  Note: To code an event for an object, select an event and click the ellipse button in the Events tab. The HTML Layout adds the event to the HTML code and switches from Design to HTML view. In the HTML view, you are prompted to define the event. Add the appropriate JavaScript code.

  The following image is an example of the HTML code that appears when the Events tab is selected in the HTML Layout Painter.
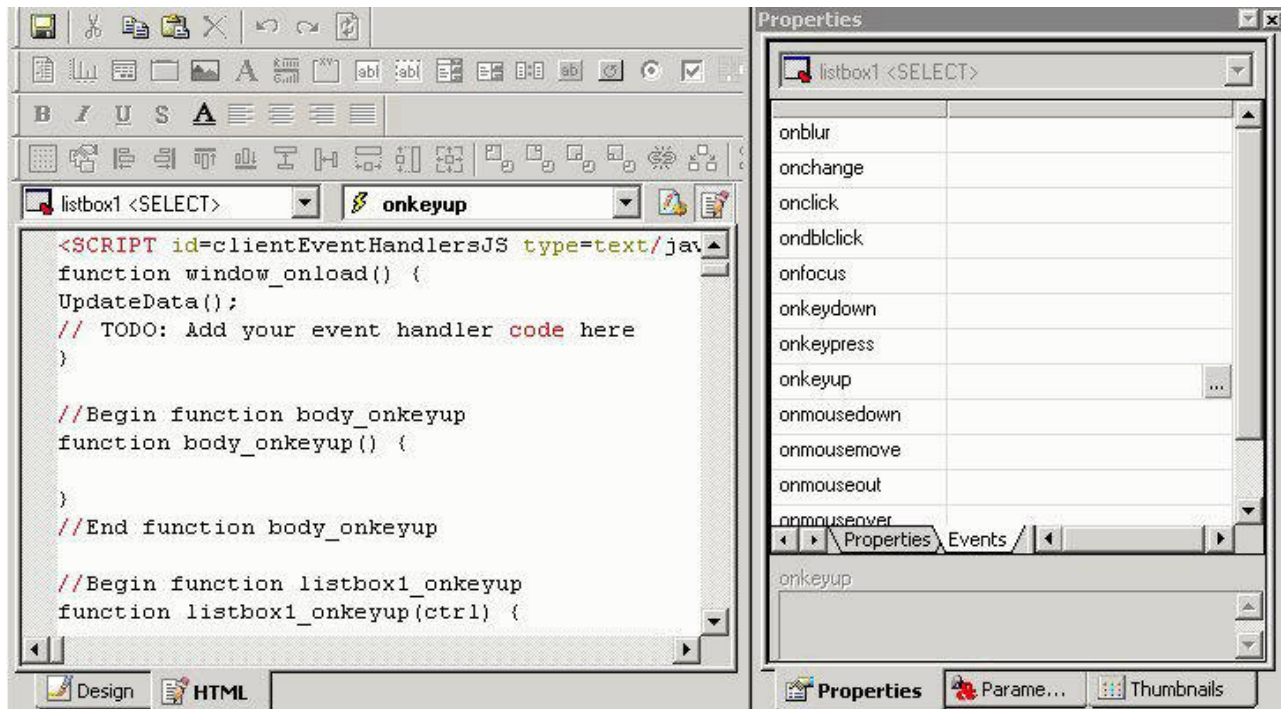
- A Parameters tab enables you to assign parameters to controls. For details on the properties than can be set for an object, see the topic on that object.
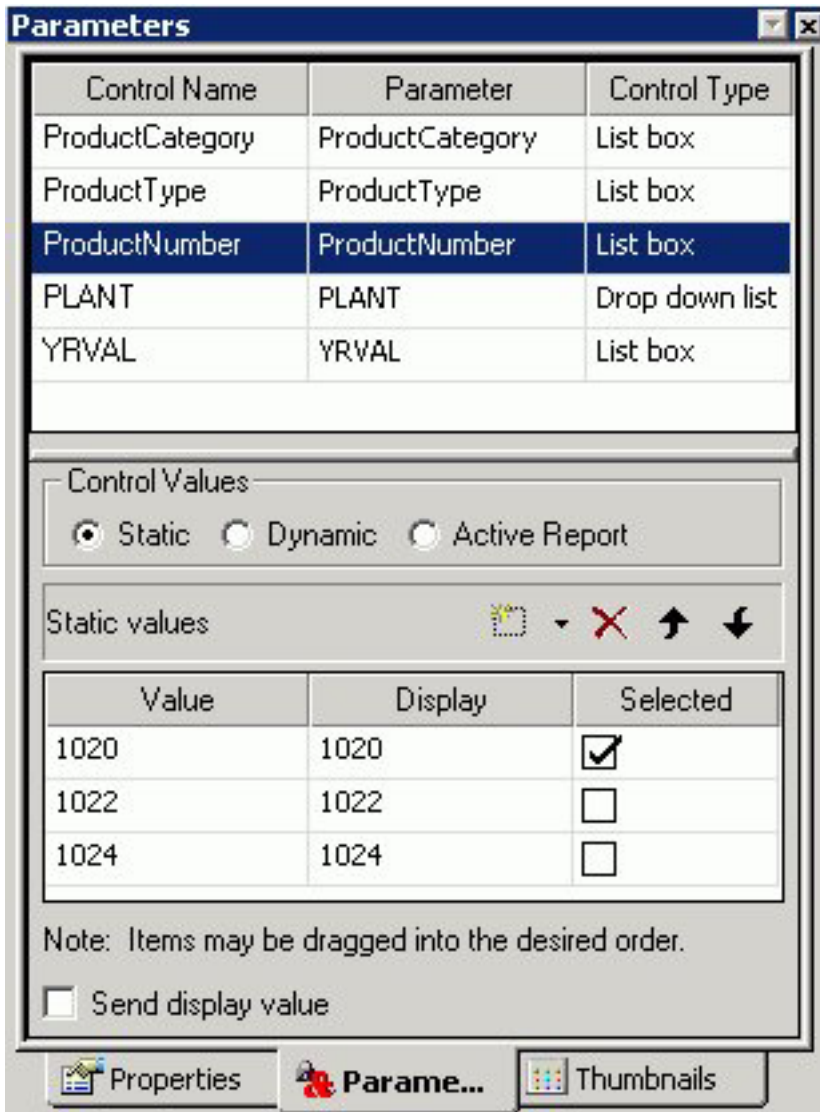
- A Thumbnails tab enables you to view thumbnails of each page in the layout. Thumbnails work similar to thumbnails in Adobe PDF, and enable you to scroll to different pages. You may also refresh the Thumbnails tab, and enlarge or reduce the thumbnails.
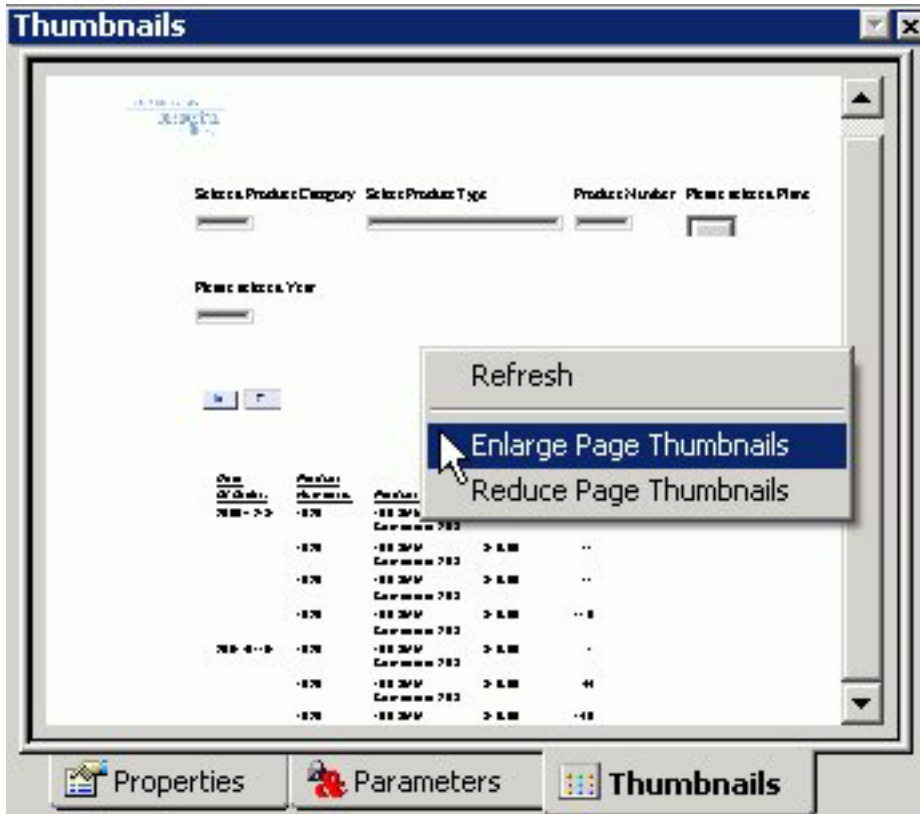
Move the Properties Window

To **undock** the Properties window, double-click the **Properties** title bar, then move the window to the desired location in the layout.

To **dock** the Properties window, double-click the **Properties** title bar.

Setting Form Controls for New Parameters

When a report contains one or more new amper variable parameters created in the Report Assistant, the New Parameters dialog box appears when you save the report. You can set the Control Type for each parameter prior to returning to the HTML Layout Painter. This eliminates the need to select each parameter individually to edit the associated Control Type setting using the Parameters tab of the Properties window.

The following image shows the New Parameters dialog box. For each parameter, you will find Name and Type fields, a Create control check box, and a drop-down menu to set the Control Type to either text box, calendar, list box, drop-down list, radio button, check box, hidden, text area, or tree control. The Type field is set to Unresolved unless the amper variable parameter is created with the Dialogue Manager using the △DEFAULT command (which sets the Type field to Default).
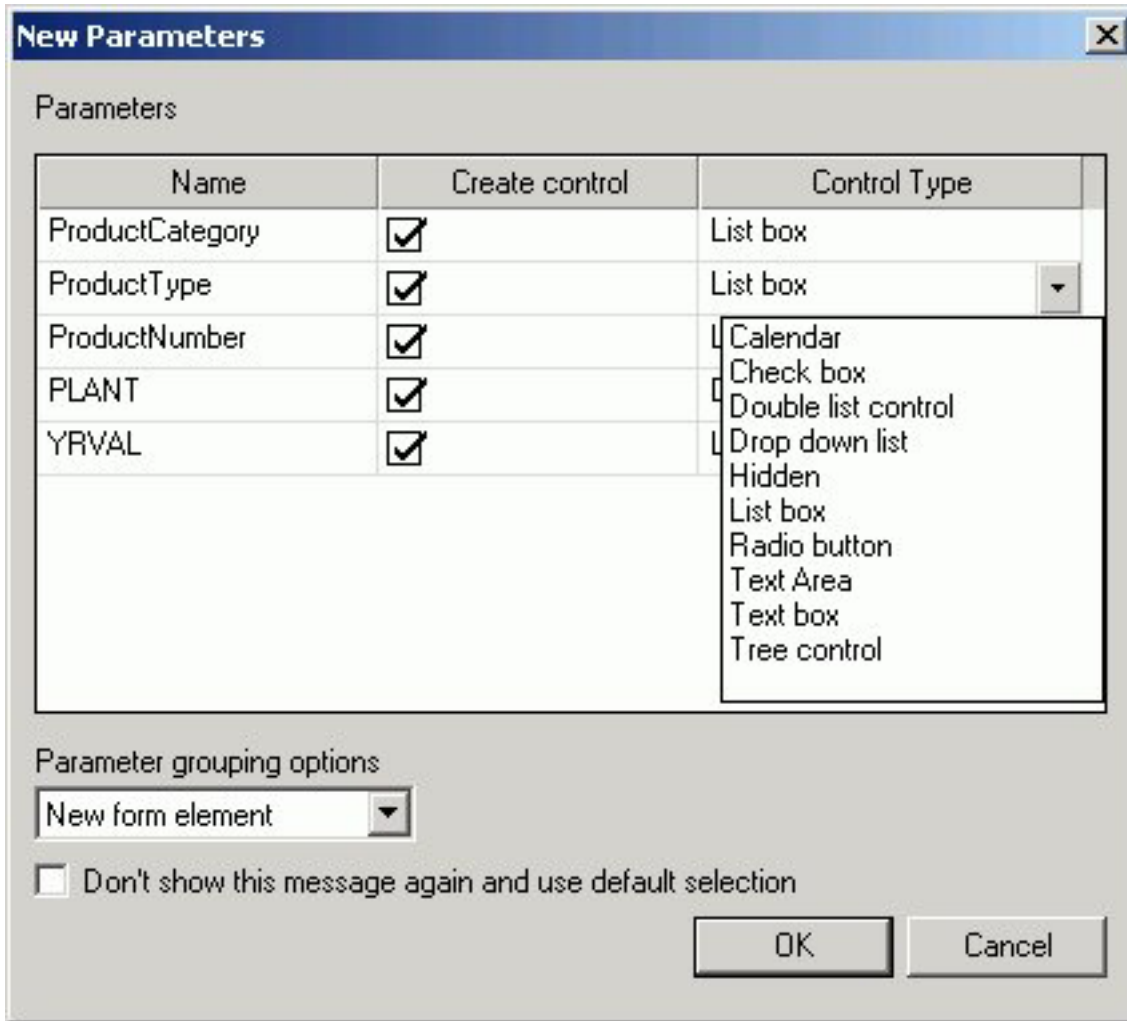
The Parameter grouping options menu in the New Parameters dialog box provides options for the placement of the form controls associated with new amper variable parameters created in the Report Assistant. Select **Do not create a form** to insert the form controls for each of the new parameters in separate locations on the HTML page. Select **New form element** to insert the form controls for all of the new parameters inside one form element you can position anywhere on the HTML page. This form element also contains submit (run) and reset buttons.

Working With the Parameters Tab in the Properties Window

The Parameters tab is a component of the Properties window. The Parameters tab consists of the following components:

- The top pane contains information on the controls in the layout.
- The bottom pane contains information on parameter values.

For more detailed information, see Associating Form Controls With Parameters.

Associating Form Controls With Parameters

When a parameter is created in the Report Assistant, a text box is automatically added to the layout and associated with the parameter. However, you can change the form control and form control type associated with the parameter.

Change the Type of Form Control Associated With a Parameter

1. In the Parameters tab, select the parameter name listed in the Parameter column.

2. Select the form control you want to use from the drop-down list in the Control Type column to the right, and your change is reflected in the report layout.

Unassign or Reassign a Parameter to an Existing Form Control

1. In the Parameters tab, select the desired form control in the Control Name column.

2. To the right of the desired form control, click the Parameter field, then select the **down arrow** that appears at the right edge of the Parameter field.

   All existing parameters are shown in the drop-down list, along with Unassigned.

3. Select an existing data field to assign the parameter to, or select **Unassigned** to remove the association to the control.

   Note: Unassigned parameters may result in no data returned for the request. An unassigned parameter must have a default value assigned in the underlying DB2 Web Query code.

Creating a Static List of Parameters

When creating a list of static parameters, you can select from the following options:

- Add new value.
- Add ignore value.
- Add everything value.
- Use values from procedure.
- Use values from external file.

When the options are added to the Value list, the display text can be customized, but the value cannot be changed.

Parameters Tab in the Properties Window (Static Values)

The following image is the Parameters tab in the Properties window with Static control values.
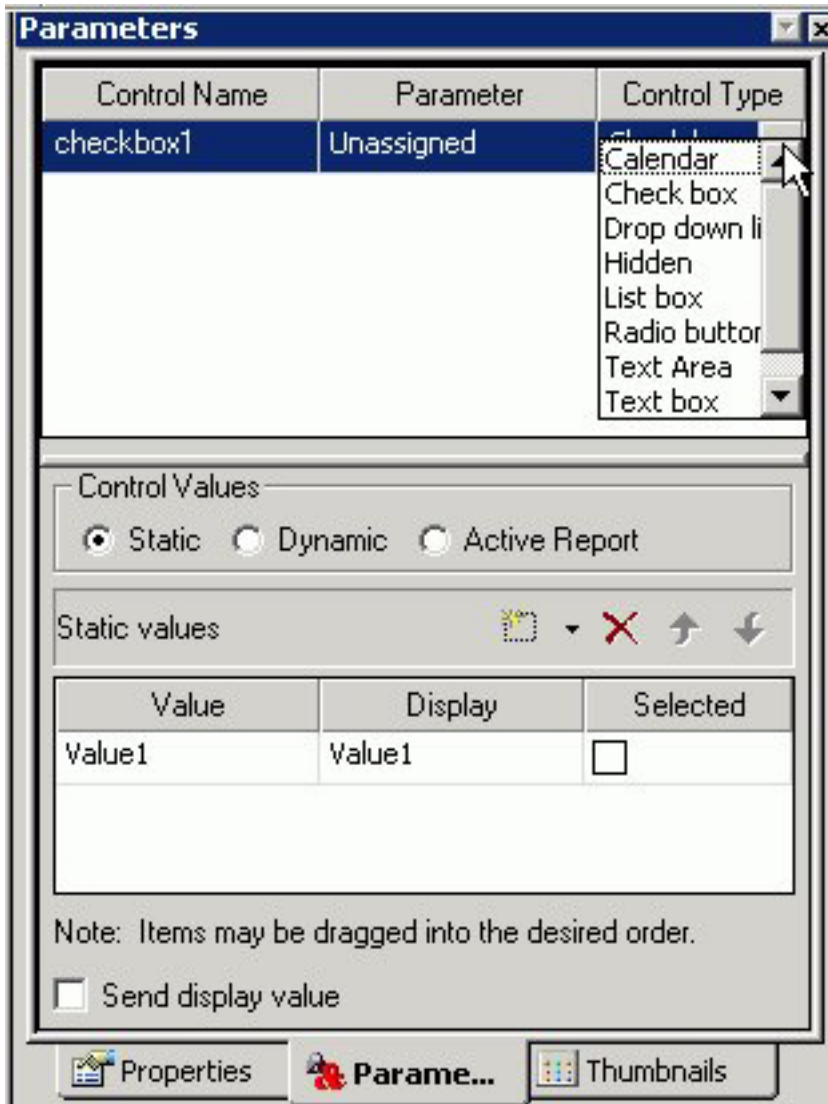
The Parameters tab contains the following fields and options when Static is selected in the Control Values section:

Control Name

> Contains the name of the form control associated with each parameter.

Parameter

> Contains the name of each parameter in the current procedure.

> **Unassigned** - Specifies the parameter is not associated with a form control.

Control Type

> Contains the type of form control associated with each parameter.

> The options are: **Calendar, Check box, Drop down list, Hidden, List box, Radio button, Text Area, Text box, Tree Control**

Control Values

> Determines whether parameter values are obtained from a static, dynamic, or active report control list.

Static - Uses a static list of parameter values you supply. A list of static values can also be created in the Report Assistant. For an example of importing static values that were created in Report Painter, see Adding Static Field Values From Report Painter.

Static values

Is a list of supplied values for a static list.

Value - Is the value to be passed to the selected parameter.

Display - Is the text that represents the parameter value in the form control the user views. **Note:** Click Ctrl + Shift to add a value to the Display field.

Selected - Is the value to act as the default value. If the form control is multi-select, more than one value can be selected.

New - Creates a new parameter value.

Delete - Deletes a supplied parameter value from the list.

Move Up - Moves the selected value up in the list.

Move Down - Moves the selected value down in the list.

The XML output is:

Add a New Static Value

To manually add new static values, do the following in the Parameters tab:

1. In the Control Values section, select **Static**.
2. Click the **New** icon, and enter parameter value information in the Static values section:
   a. In the Value column, enter the value to be passed to the selected parameter.
   b. In the Display column, enter the text that represents the parameter value in the form control the user views.
   c. In the Selected column, check the box for the value you want to be selected by default. If the form control is multi-select, more than one value can be selected.
3. Repeat Step 2 until the list contains all of the values you want to include.
4. Optionally, click the **Delete** button to eliminate any values, and the up and down arrows to rearrange the order of the values.

Adding Static Field Values From Report Painter

Instead of adding new static values, you may also import static value parameters that you created from the Variable Editor. Parameter values can be data values to limit your data or fieldnames to select the fields in your report. Both type of static lists can be created in Report Painter. This example details how to supply field values to a report through the HTML Layout Painter and Report Painter.

Note: For the purposes of this example, use the CAR Master File (car.mas) that is supplied with Developer Studio.

1. From the HTML Layout Painter, select **New Report** from the Insert menu.

   The cursor changes into a crosshair.
2. Click and drag the crosshair to create a report object and adjust it to the size you want.

3. Right-click the report object and select **Open report**.

   The Open dialog box appears.

4. Open the **car.mas** file and click **Open**.

   The Report Painter opens.

5. From Report Painter Object Inspector, right-click on Report Variables and select **New Report Variable Field**.

   The Variable Editor opens.

   - Enter **SORTVAR** in the Name and Prompt input fields, and keep the Variable Type as Single Select.
   - Select **Static List** from the Accept List section.
   - Double-click **COUNTRY** and **BODYTYPE** from the Fields from database Data Context area.

     The fields are added to the Static Accept List.
   - Click **OK** to save the variable.

6. Create a second variable:

   - Right-click on Report Variables and select **New Report Variable Field**.

     The Variable Editor opens.
   - Enter **MEASURES** in the Name and Prompt input fields.
   - Select **Multiselect AND** from the Variable Type drop-down list.
   - Select **Static List** from the Accept List section.
   - Double-click **CAR**, **DEALER_COST**, **RETAIL_COST**, and **SALES** from the Fields from database Data Context area.

     The fields are added to the Static Accept List.
   - Click **OK** to save the variable.

7. Double-click **&SORTVAR** and **&MEASURES** from the Report Variables in the Object Inspector.

   The variables are added to the report.

8. Click **&SORTVAR** and select **By** from the Columns toolbar.

   Note: If the parameter is to be used as a By field in a report, it must be the single select variable type.

9. Save and close the report.

   You are returned to the HTML Layout Painter and since you are adding a report with parameters, the New Parameters dialog box appears.

10. From the New Parameters dialog box:

    - Select **Double list control** from the Control Type drop-down list for MEASURES.
    - Select **Do not create a form** from the Parameter grouping options drop-down list.
    - Click **OK** to close the New Parameters dialog box.

    The report and associated parameters are added as static field values. You may rearrange or lengthen the controls in the layout.

11. Save and run the layout.

    Select the static field values (By field and the associated measures) and run the report.

Add an Ignore Value

The add ignore value option sends FOC_NONE to the server at run time and is intended for use with complex applications.

1. From the HTML Layout Painter, use form controls to supply parameter values for a report.

   Note: A report with parameters requires that you to select values (at run time) in order to generate the output.

2. Required: Click the **Parameters** tab in the Properties window.

   The properties of a form control, as well as the parameters associated with each form control, can be controlled with the Properties tab and Parameters tab of the Properties window.

3. Select **Add ignore value** from the Static values drop-down list.

4. Run the HTML page and select the **Ignore All** value to ignore the parameter values.

Note: Sending FOC_NONE to a compound WHERE statement will result in the entire WHERE criteria being ignored. Therefore, when working with multiple parameters that may be chained together, each parameter should be created in an individual WHERE statement as follows:

```
WHERE ( REGION EQ &REGION.(OR(Midwest,Northeast)).Region:. );
WHERE ( ST EQ &STATE.(OR(CA,CT)).State:. );
WHERE ( CITY EQ &CITY.(OR(Atlanta,Boston)).City:. );
```

Add an Everything Value

The add everything value option uses JavaScript to send every value present in the parameter list to the server at run time.

1. From the HTML Layout Painter, use form controls to supply parameter values for a report.

2. Click the **Parameters** tab in the Properties window.

3. Select **Add everything value** from the Static values drop-down list.

   Note: The Add everything value option is only available with a Multiselect OR variable type.

4. Run the HTML page and click the **Select All** parameter value to view all the parameter values.

Use Values From a Procedure

This is the default option which populates the static list with values predefined in the procedure.

1. From the HTML Layout Painter, use form controls to supply parameter values for a report.

2. Click the **Parameters** tab in the Properties window.

3. Select **Use values from procedure** from the Static values drop-down list.

4. Run the HTML page and select the parameter values from the procedure.

Import Values From an External File

This option enables you to use a local external file to provide a query limit for the selected data.

1. From the HTML Layout Painter, use form controls to supply parameter values for a report.

2. Required: Click the **Parameters** tab in the Properties window.
3. Highlight the static parameter that you want to import external values for.
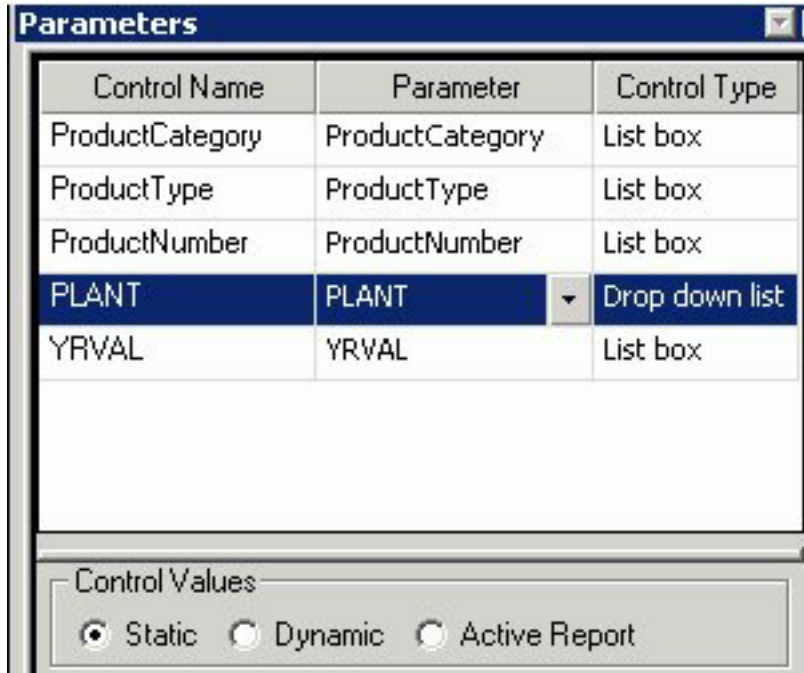
4. Required: Select **Use values from external file** from the Static values drop-down list.
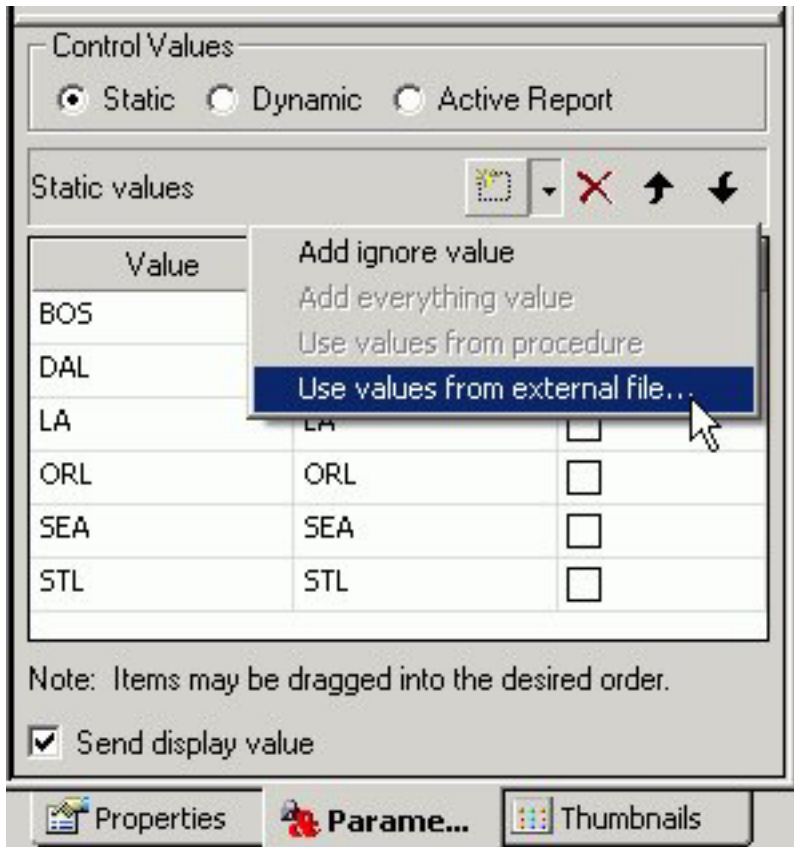
The Open dialog box appears.

5. Select a text file from your local machine and click **Open**.
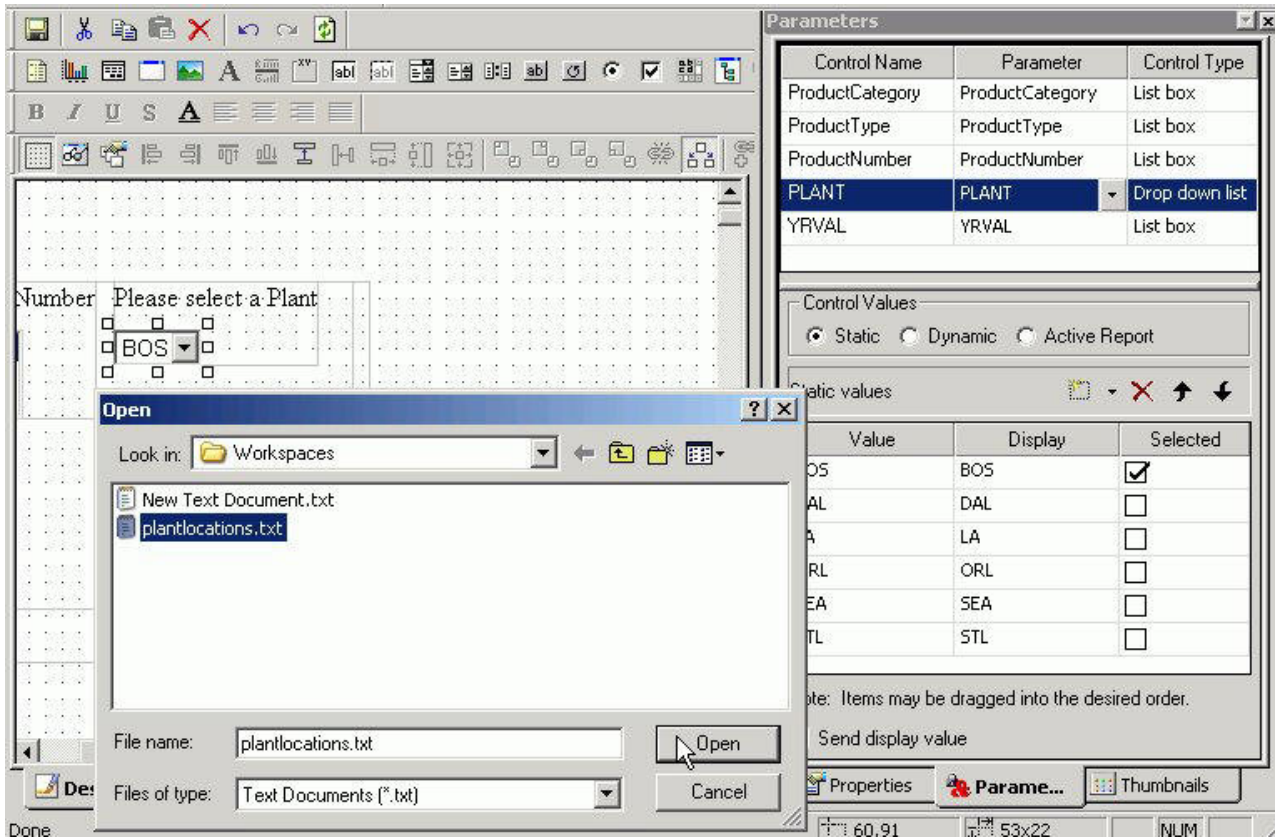
Note: The external file can be a file with single values on each line, or two values per line, comma delimited.

For example, in the following text file, **BOS** is the return value and **Boston** is the display value.
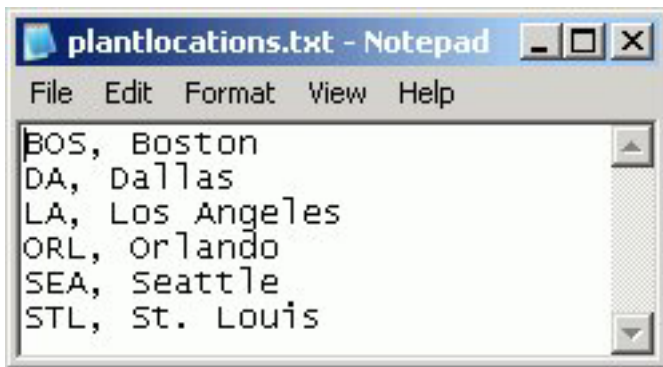
The imported values are loaded into the Static values area of the Parameters tab.
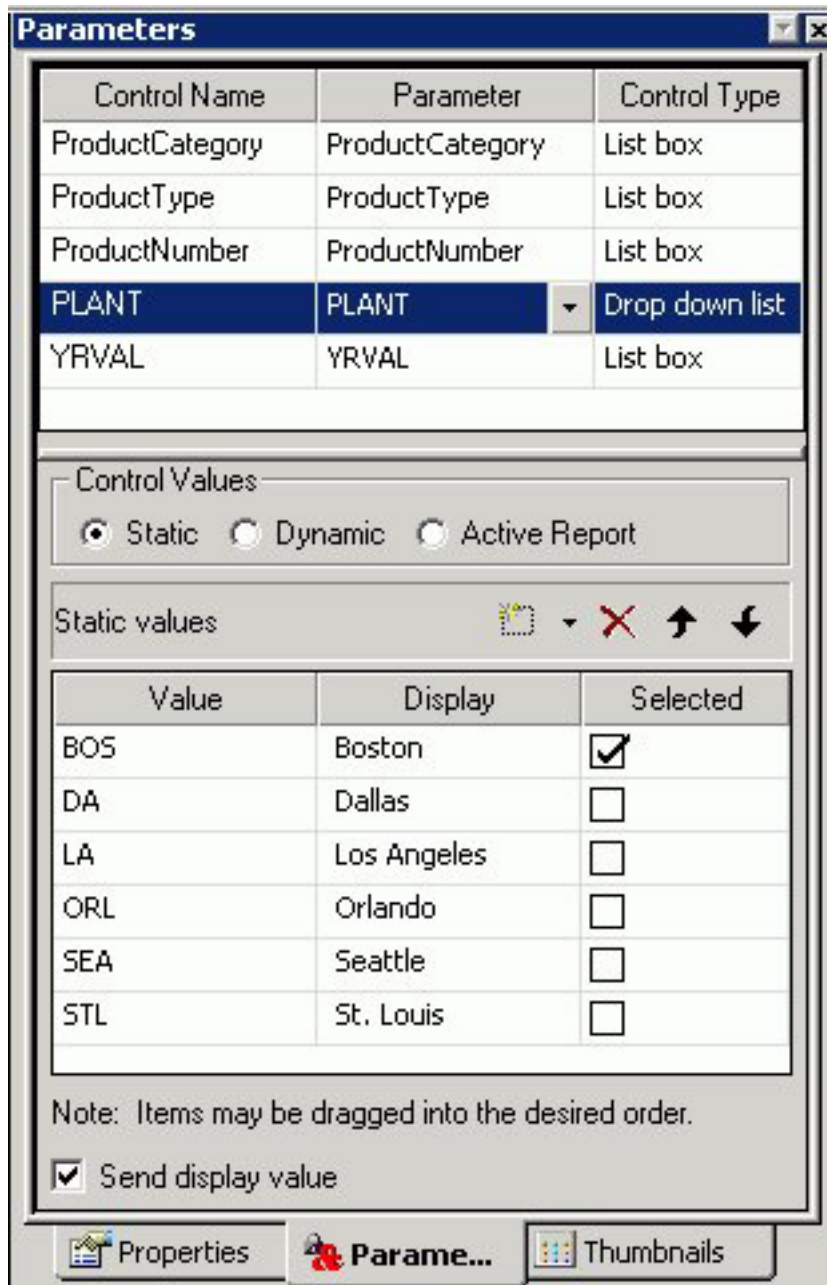
Note: If there is only one value, the value will populate both the return and the display values.

6. Run the HTML page to see the imported values for the selected parameter.

Creating a Dynamic List of Parameters

Dynamic values are available by default if a data field used in the procedure is associated with the selected form control. The Parameter field at the top of the Parameters tab displays this data field (or displays Unassigned if there is no associated parameter).

Parameters Tab in Properties Window (Dynamic Values)

The following image is the Parameters tab in the Properties window with Dynamic control values.
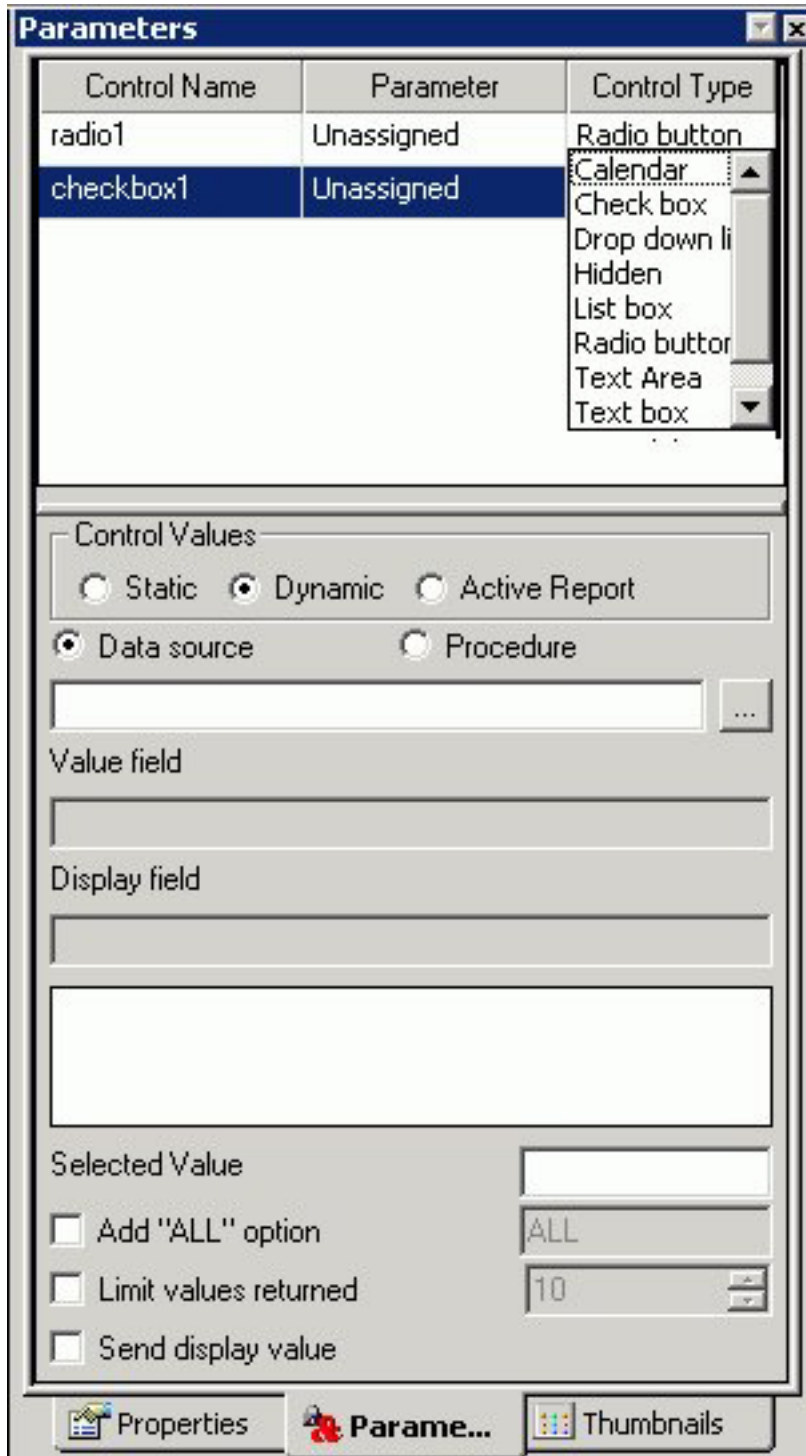
The Parameters tab contains the following fields and options when Dynamic is selected in the Control Values section:

Control Name

Contains the name of the form control associated with each parameter.

Parameter

Contains the name of each parameter in the current procedure.

Unassigned - Specifies the parameter is not associated with a form control.

Control Type

Contains the type of form control associated with each parameter.

The options are: **Calendar, Check box, Drop down list, Hidden, List box, Radio button, Text Area, Text box, Tree Control**

Control values

Determines whether parameter values are obtained from a static, dynamic, or active report control list.

Dynamic - Uses a list of values retrieved from a selected data source when the request is executed. This is the default if you use an Accept clause in a Master File to create an amper variable parameter within a procedure. For more information, see Select a Dynamic List of Values.

Data Source

Is the data source from which the values will be retrieved.

Procedure

Is the procedure that will be called.

Value field

Is the data source field from which the values will be retrieved.

Display field

Is the text that represents the parameter value in the form control the user views.

Note: Click Ctrl + Shift to add a value to the Display field.

Selected Value

Enter the value(s) to be selected as the default value whenever the procedure is run. For more information, see Use Selected Values as the Default Value.

Add ″ALL″ Option

Adds the option to select ALL data source values to the control. Alternate text can be substituted for ″ALL″ using the text field to the right. For more information, see Parameter List Options.

Limit values returned

Indicates that a specific number of field values will be retrieved from the data source. The specific number of fields is selected with the menu to the right.

Select a Dynamic List of Values

To add or select a different dynamic list of values to populate a form control, do the following in the Parameters tab of the Properties window:

1. Select the **Dynamic** option in the Control Values section.
2. To find the desired source file you want to retrieve values from, select the **Data source** option to retrieve values from a .mas file, or select the **Procedure** option to retrieve values from a .fex file. Click the browse **(...)** button, select a source file from the Get source file dialog box that opens, and click **Open**.

For the Data source option only, select the data field that has the desired values from the tree structure displayed at the bottom of the Parameters tab. The data field you select will be populated in the Value field.

3. Optionally, type a new name in the Display field for the data field populated in the Value field.

4. Optionally, select **Limit values returned**, and select or type the number of field values you want to retrieve from the data source in the box to the right of this option.

Parameter List Options

When creating a static or dynamic list of parameters, you may add an all value to the list of parameter values and show the display value in the report output.

The ALL feature allows developers to automatically add an ALL value to a list of parameter values. An ALL value does the following:

- For dynamic parameters, the ALL feature sends a value of FOC_NONE to the reporting server alerting the server to bypass or ignore the parameter altogether. (Ignoring the parameter would return all values in the data source.)

- With static parameters, the ALL value typically uses JavaScript to return all of the values displayed in the list. This prevents you from having to select every value in the list manually. When using the ALL feature with static parameters, you can select from the following options:

  - Add ignore value. This option sends FOC_NONE to the server at run time, alerting the server to bypass or ignore the parameter altogether. It is intended for complex applications.

  - Add everything value. This option uses JavaScript to send every value present in the parameter list to the server at run time. The Add everything value option is only available with a Multiselect OR variable type.

When using Dynamic or Active Report control values, you may enter the value(s) to be selected as the default value whenever the procedure is run. For more information, see Use Selected Values as the Default Value.

Send the Display Value for Static and Dynamic Controls

From the HTML Layout Painter, you may show the display value, rather than the actual data, for headings and footings in the report output.

1. From the HTML Layout Painter, import a report that contains a parameter.

   When importing a report with parameters, the New Parameters dialog box appears prompting you to create the control type.
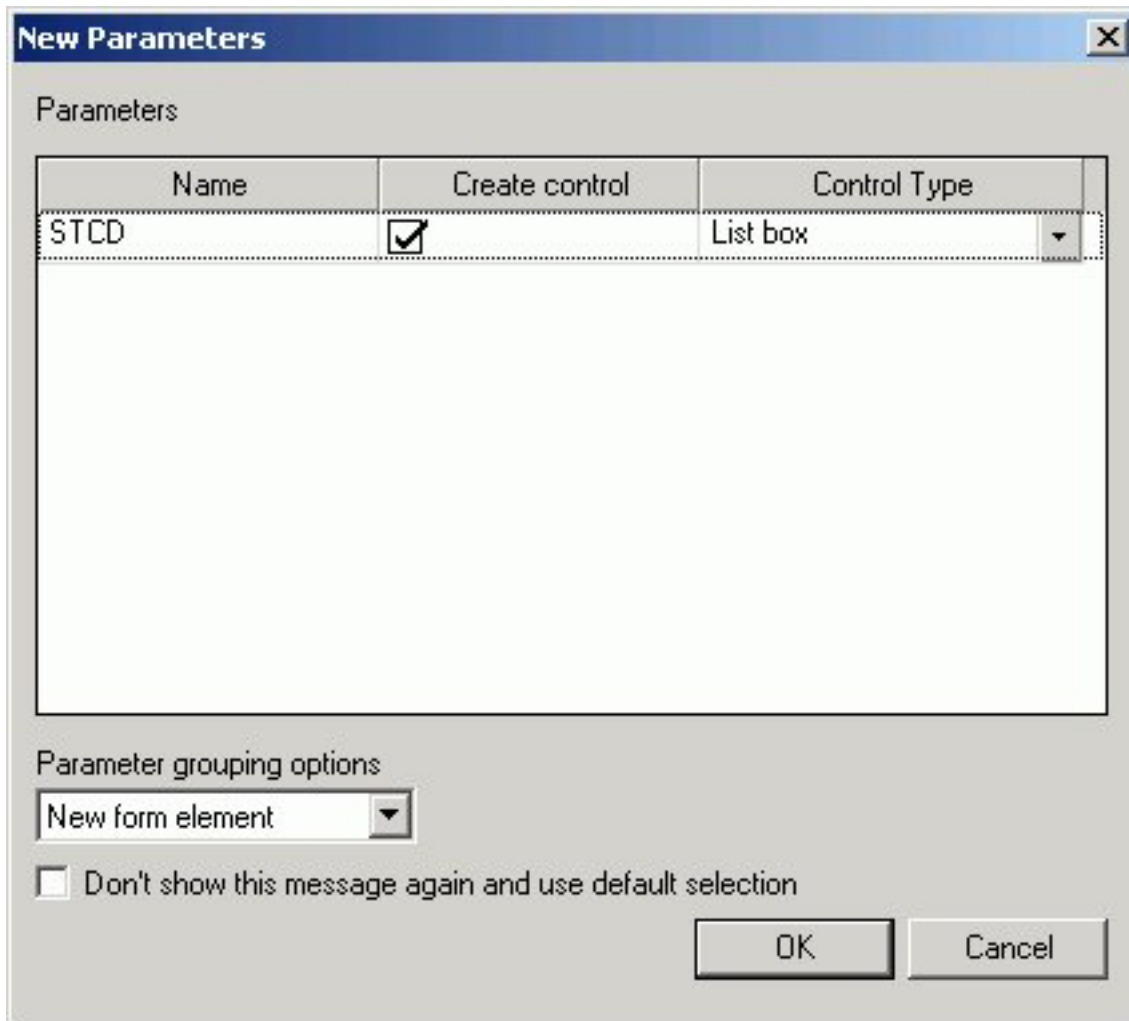
The report and control type is added to the HTML Layout Painter.
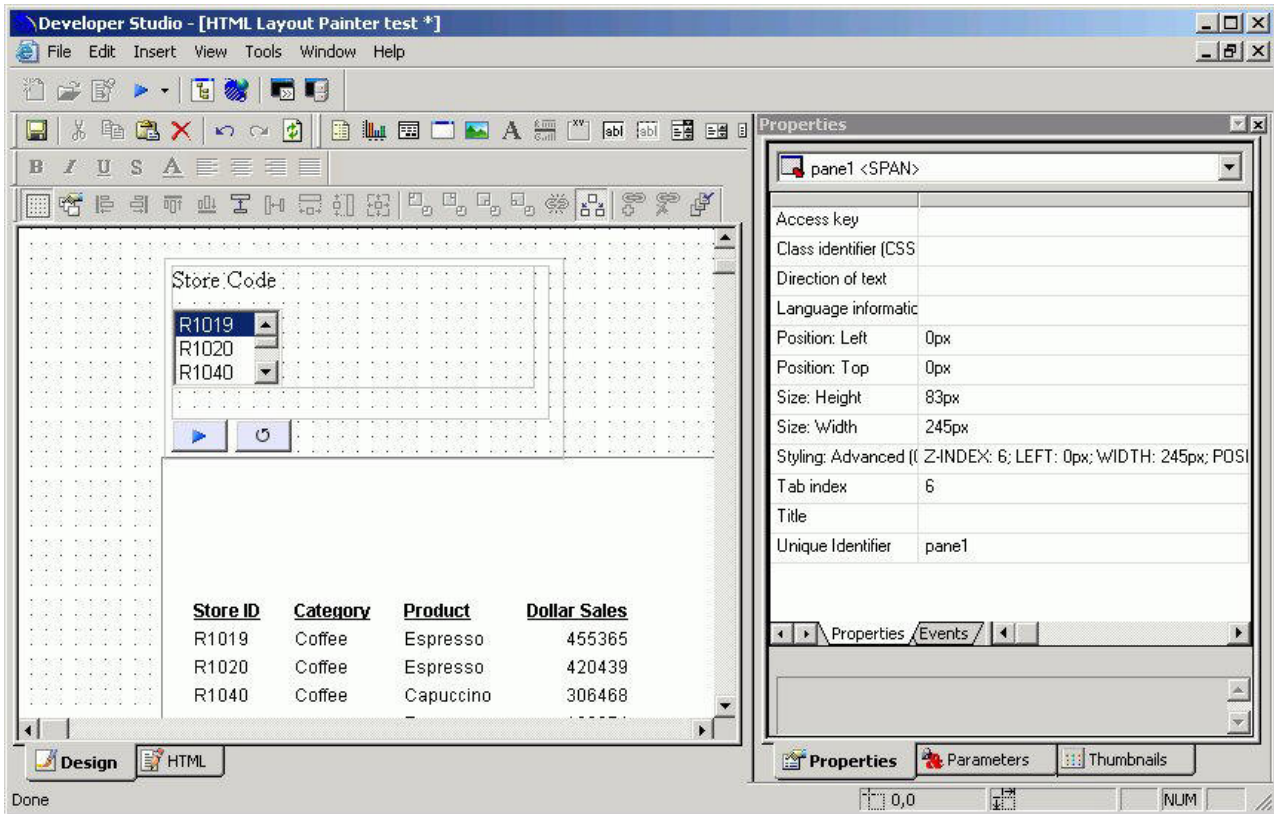
2. To show the display value for the parameter selection, use the Parameters tab of the Properties window and choose **Static** or **Dynamic** as the type of control value.

3. For a Static control type, use the Parameters tab to manually change the display value for the Static control.
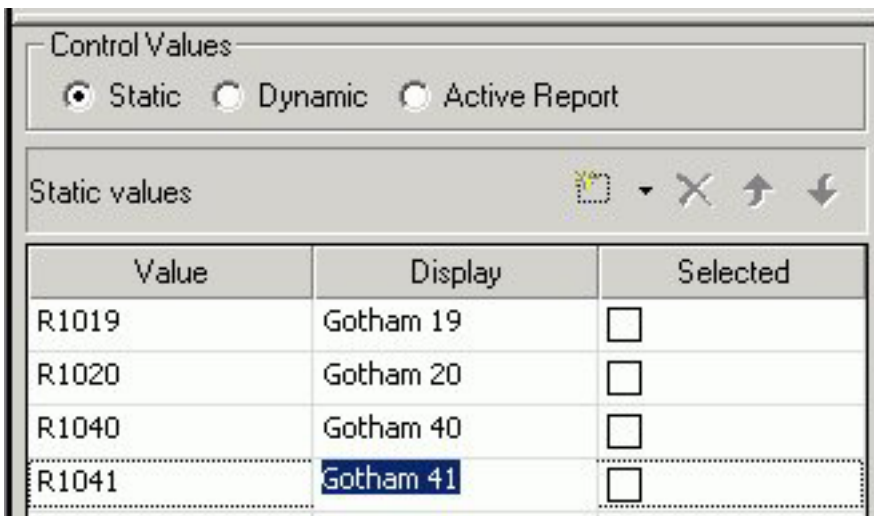
> **Note:** You may also update the display values from the Variable Editor in the Report Assistant.

4. Select the **Send display value** check box from the Parameters tab.

   The display values are shown in the control type of the HTML Layout.
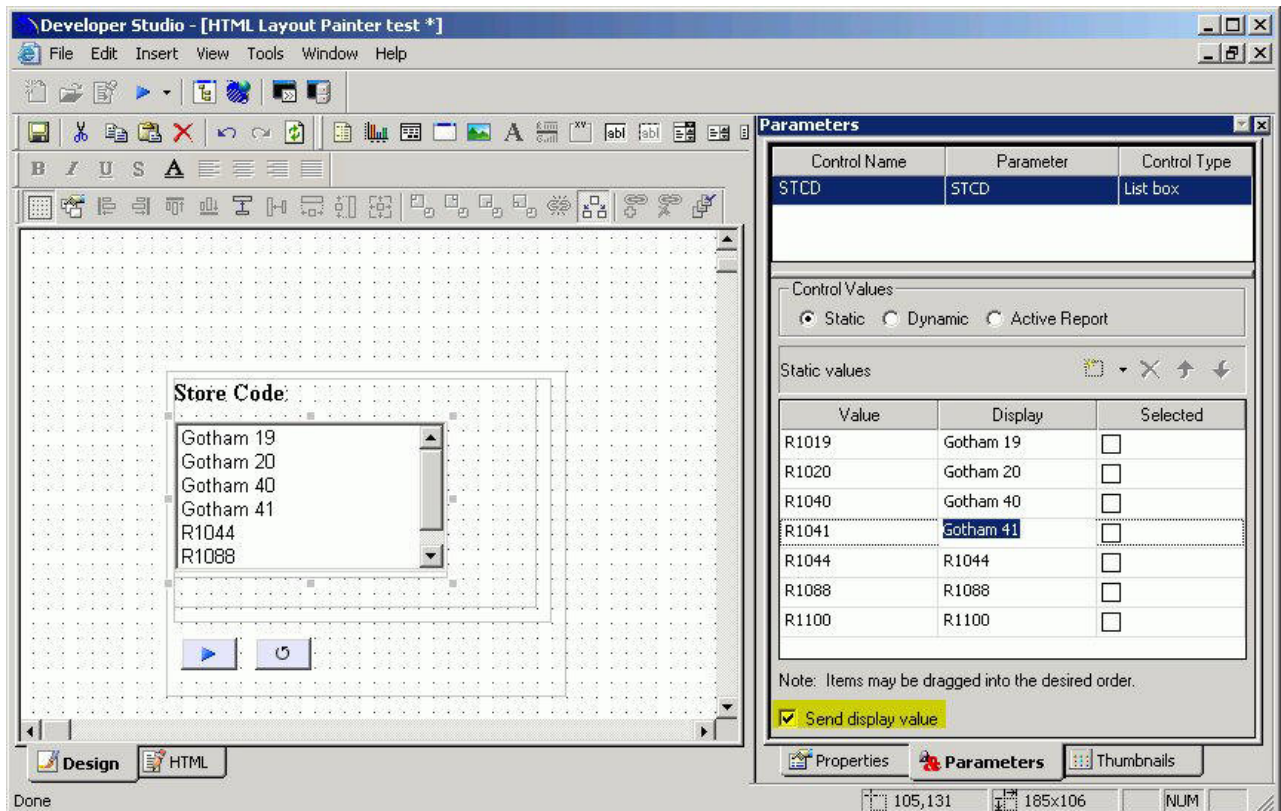
5. For a Dynamic control type, use the Parameters tab to set both the Value Field and the Display Field.

   Note: Setting the display value for Dynamic controls is only applicable if you are using a Data source that contains both a code and description field.

   - Double-click a field from the Fields list to add it as the Value Field.
   - Select a field from the Fields list, click Ctrl + Shift to add it as the Display Field.
   - Ensure that **Send display value** is checked.

   The following image is an example of the Parameters tab with Send display value selected.
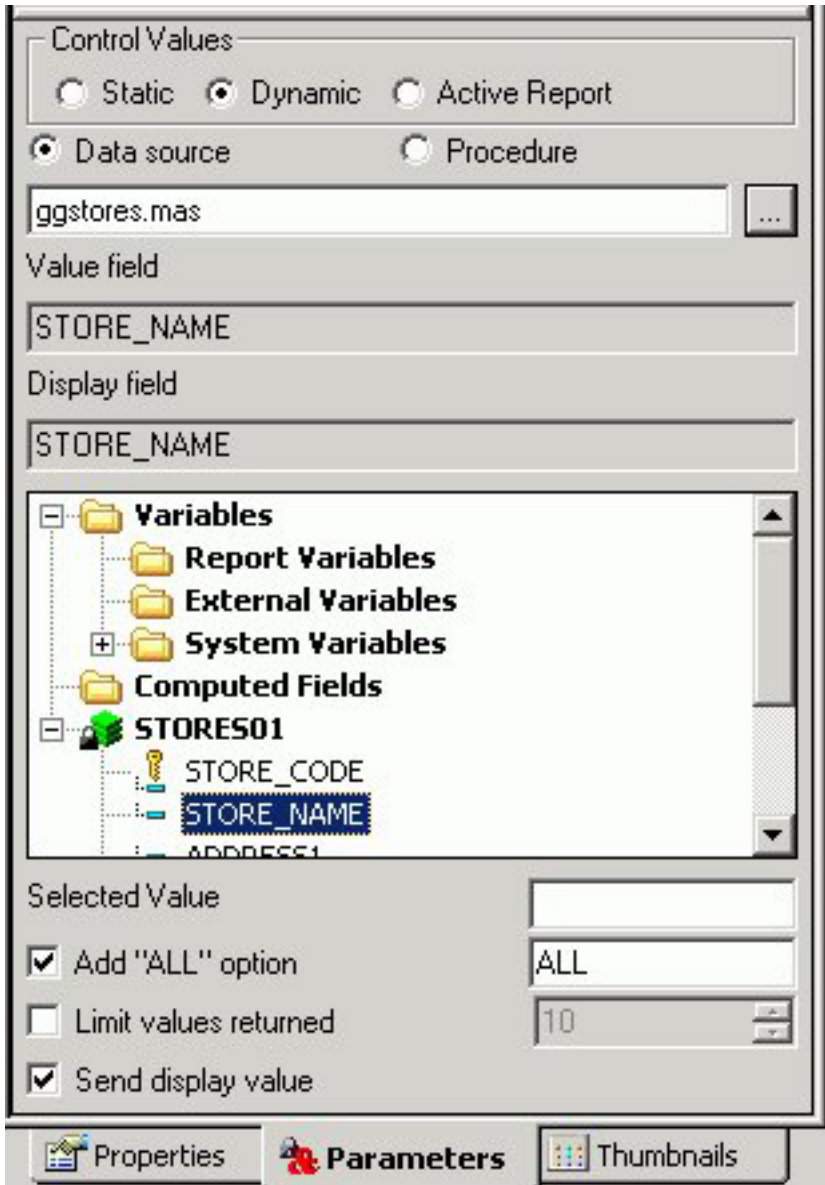
The display values are shown in the control type of the HTML Layout.
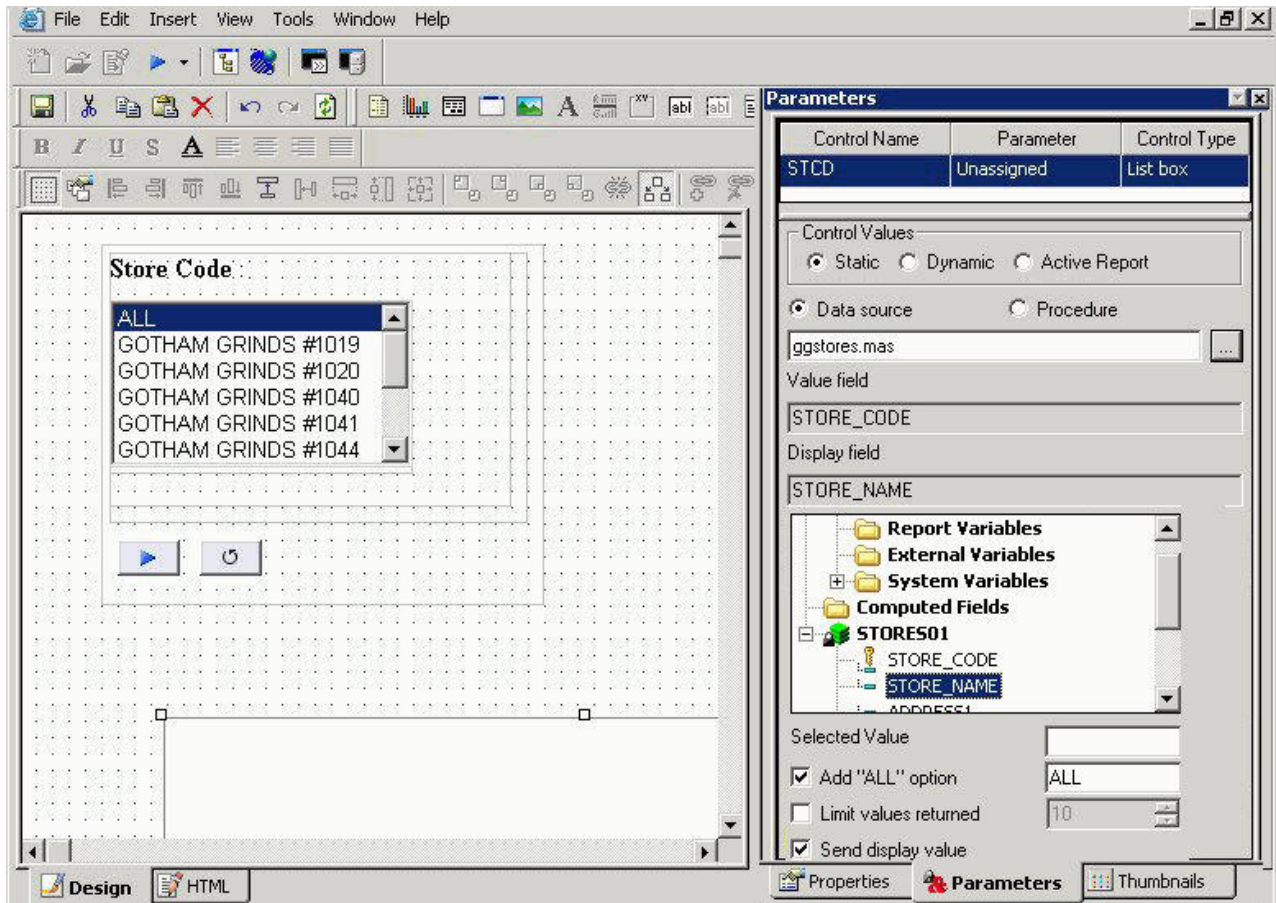
6. Add a heading to the report to show the display value. Double-click the report from the HTML layout to open and edit the report.

   The report opens in the ReportAssistant.

7. Manually add &Variable_TEXT to the report heading or footing.

8. Save and close the report to return the HTML Layout Painter.

   Note: When you create the Text variable (&Variable_TEXT), you are not prompted to set a control type for this variable when returning to the HTML Layout, as no control type is needed.

9. Run the HTML page.

10. Select the parameter for the report and run the report.

    The display value is shown in the report heading.

    For example, in the image below, Gotham 40 is selected as the Store Code and shows the display value in the report heading. The actual value, R1040, is shown in the Store ID column.
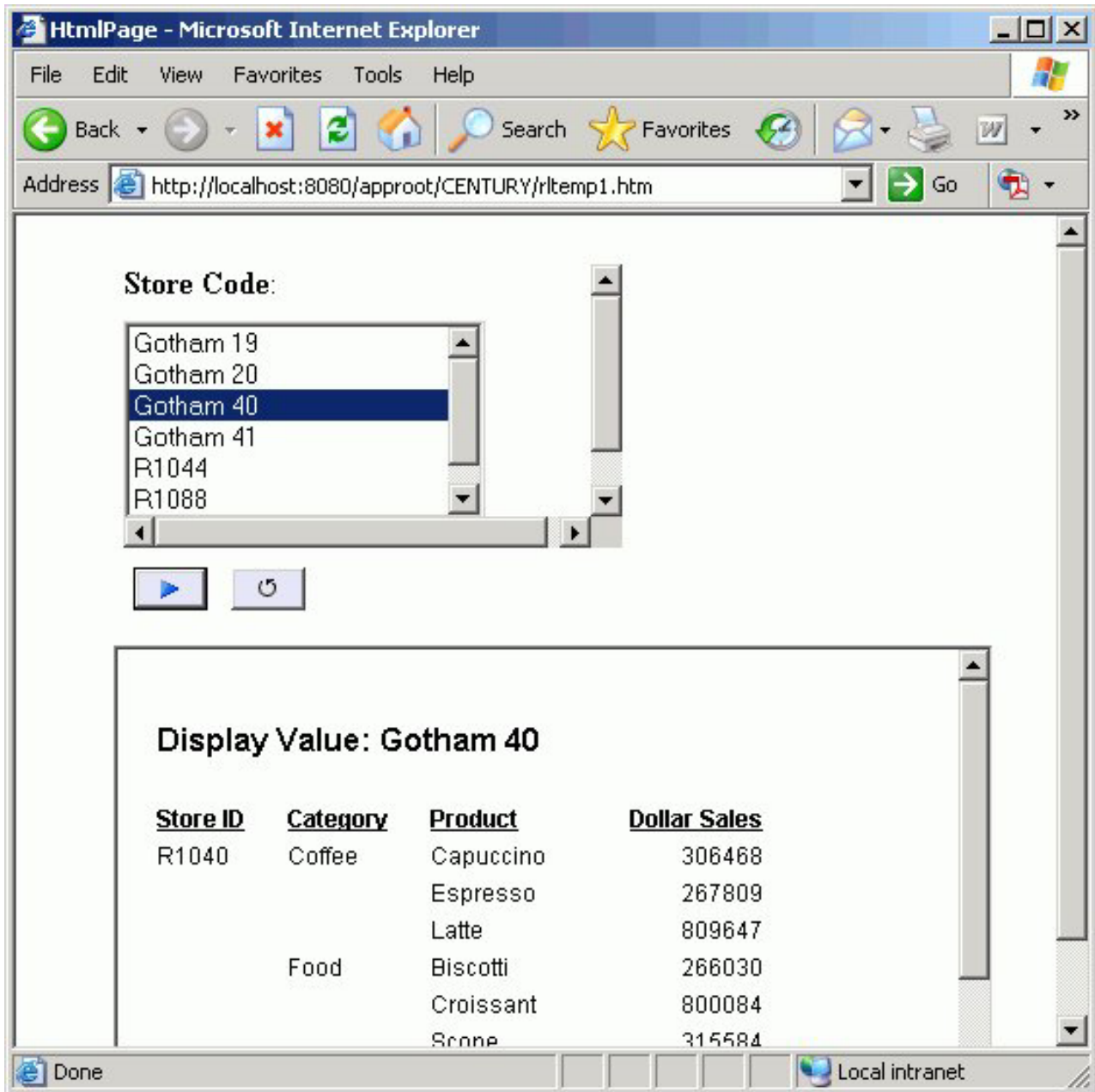
Use Selected Values as the Default Value

When using Dynamic or Active Report control values, you may enter the value(s) to be selected as the default value whenever the procedure is run.

If you import a procedure (.fex) that has a dynamic prompt value then the input box is populated with values retrieved from the data source. If the selected value is available for the report output, the value(s) are selected by default. If the selected value is not available in the report results, then the value(s) that you entered are ignored and the first value retrieved from the data source is selected.

1. Enter the exact parameter value in the **Selected Value** input field, as it appears in the report output.

Note: Parameter values are case sensitive.

2. Optionally, you may enter more than one value by using a comma between the values.

   For example: **CT, GA** are entered as the Selected Value fields for the Dynamic State parameter in the image below.
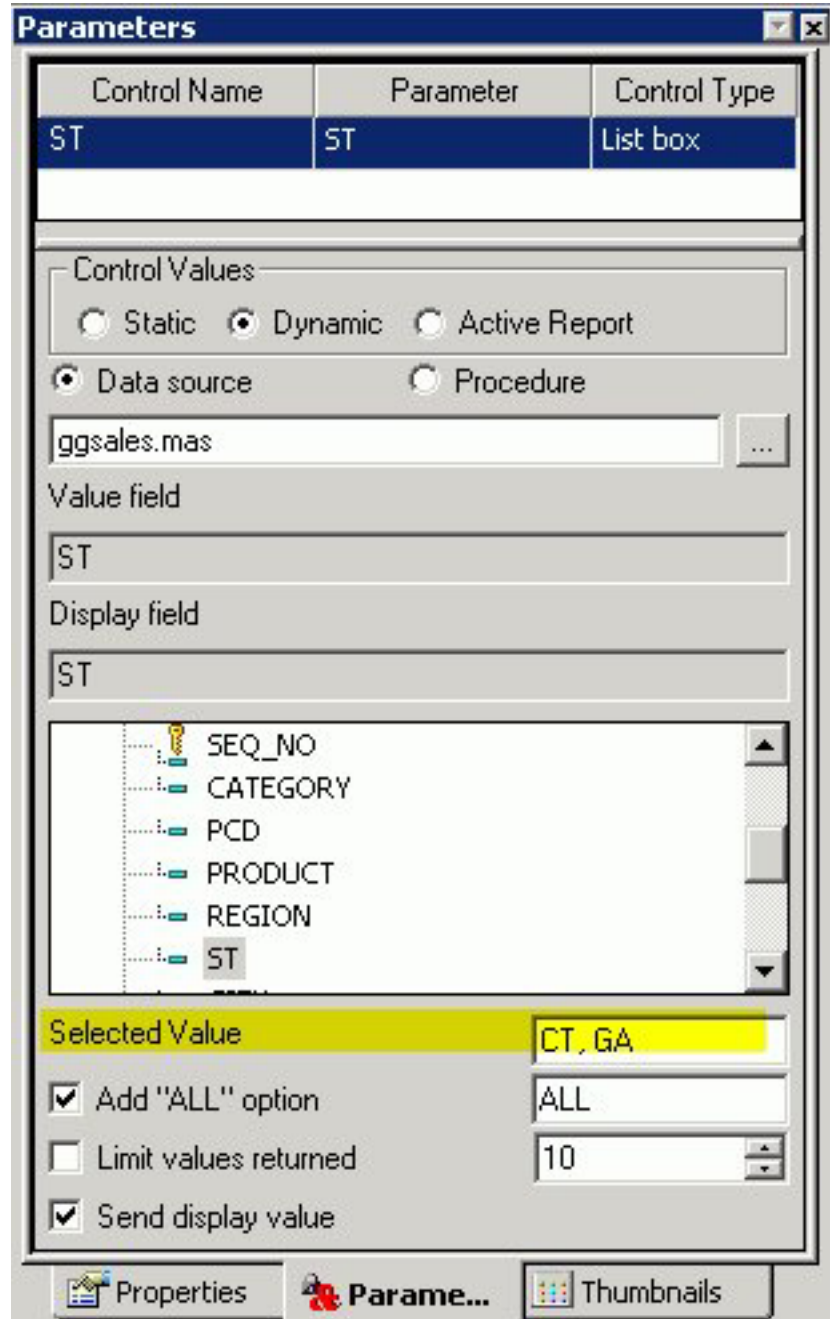
3. Run the report.

   The selected value, if available from the report results, is automatically selected (highlighted) in the parameter list.

   For example, the example below shows CT, GA as the selected values in the State parameter list in the report output.

By State
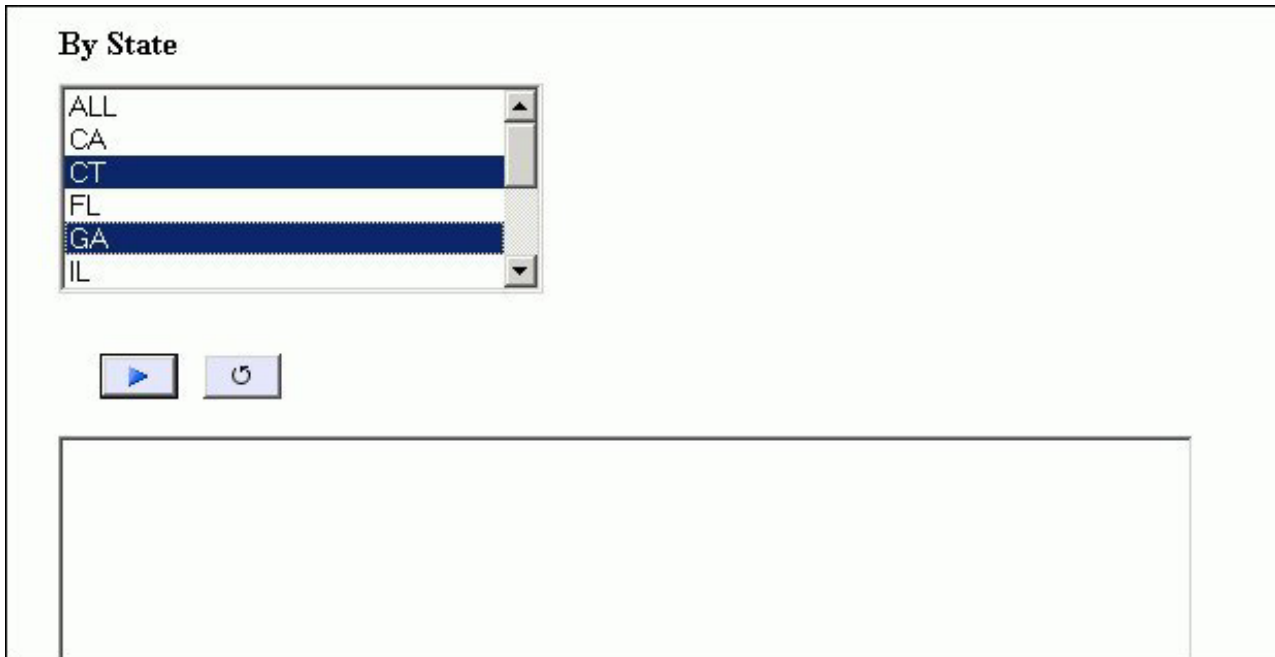
4. Click the **Run** button to run the report with the selected value parameters.

   The report output appears.

   Note: If the selected value is not available in the report results, then the value that you entered is ignored and the first value retrieved from the data source is shown.

Filtering Dynamic Parameter Values

Use the chain feature to associate two or more related parameters. When you chain parameters together, chained dynamic values are filtered as selections are made to each parameter control. For example, if you chain the PLANT parameter to the STATE parameter, only PLANT values for the currently selected STATE parameter will be available instead of all the plants in the data source. Each time a selection is made, all chained parameters will be dynamically updated. The chain feature also allows you to add, remove, and reverse the order of parameters in the chain. See other topics in this section for details.

Dynamic parameters are processed with a caching mechanism that gathers all of the necessary values prior to loading the page. This method automatically combines all of the necessary requests into a single HTTP request and maps the result sets to the appropriate controls, greatly reducing the load time involved with sending multiple requests for data.

Filter Parameter Values

1. In the Control Values section of the Parameters tab, click the **Dynamic** radio button.

2. Press the **Shift** key and then click two or more parameters to select them simultaneously and activate the chain feature buttons on the positioning toolbar.

3. Click the **Add to chain** button. The parameters you selected are labeled with numbers to indicate the order in which they will be chained when you run your report.
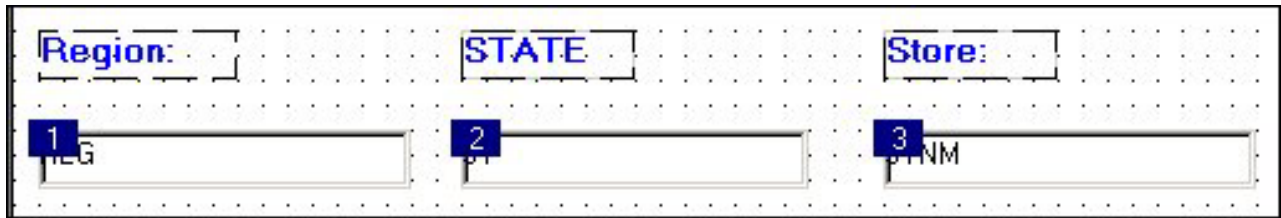
You can also reverse the order of parameters in the chain. See Reverse the Order of Chained Parameter Values for details.

Reverse the Order of Chained Parameter Values

1. Click the **Chain order** button in the positioning toolbar.
2. Click the parameter you want to display first in the chain.
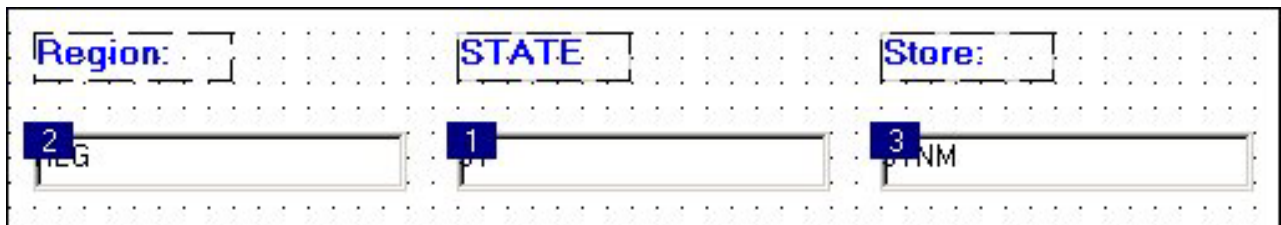
You can continue to reverse the order in the chain by clicking the numeric labels on the chained parameters. If you would like to see the numeric labels on the chained parameters, select the **Chain order** button. Double-click a parameter to designate it as the first parameter in the chain.

Remove a Parameter from the Chain

1. Click the parameter you want to remove from the chain.
2. Click the **Delete chain** button in the positioning toolbar.
3. Press the **Shift** key and click the remaining chained parameters.
4. Click the **Chain order** button in the positioning toolbar. You will no longer see a numeric label on the parameter that you removed.

Enable Cached Processing for Chained Parameters

1. Create a report with parameters in the HTML Layout Painter:
   a. From the HTML Layout Painter, select **New Report** from the Insert menu.
      The cursor changes into a crosshair.
   b. Click and drag the crosshair to create a report object and adjust it to the size you want.
   c. Right-click the report object and select **Import existing report**.
      Tip: You may create parameters directly in your report or add them later in the HTML Layout Painter.
2. To create dynamic parameters for your report in the HTML Layout Painter, right-click the report object and select **Add a filter** from the context menu.
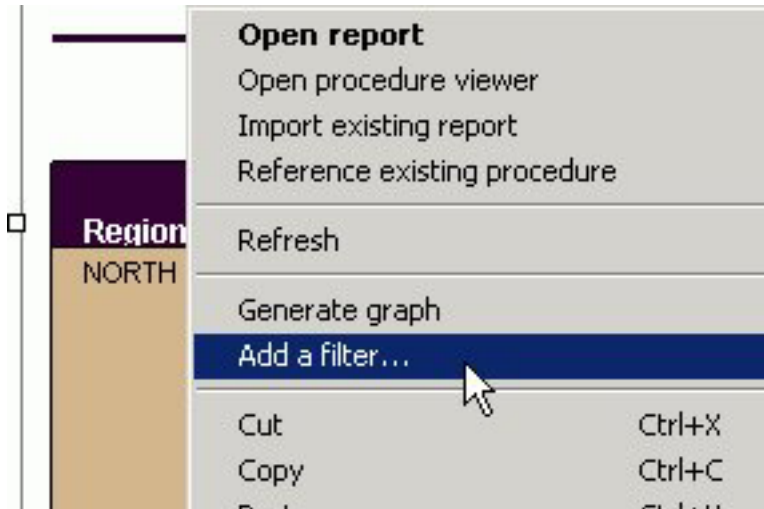
The Filter options dialog box opens.

3. Select the field to be used for the parameter, the Multiselect option, and click **OK**.
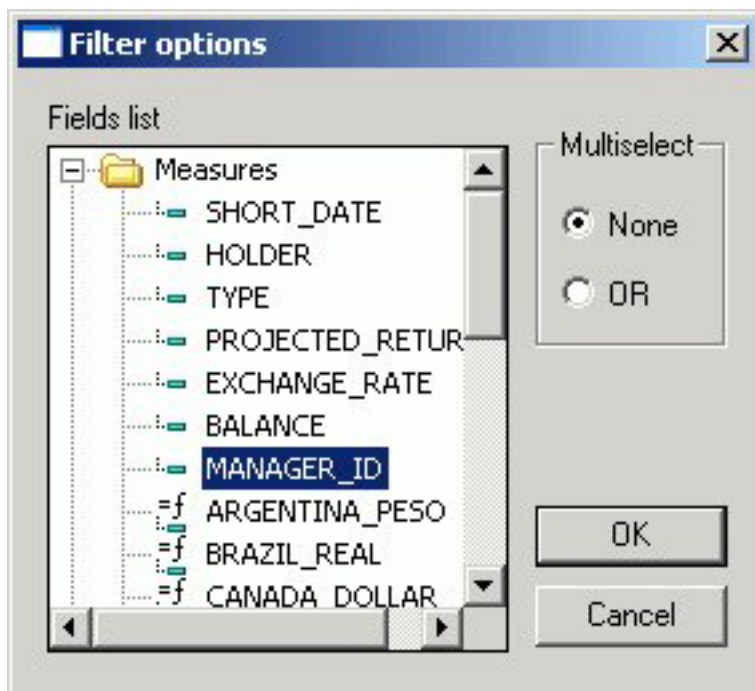
The New Parameters dialog box appears.
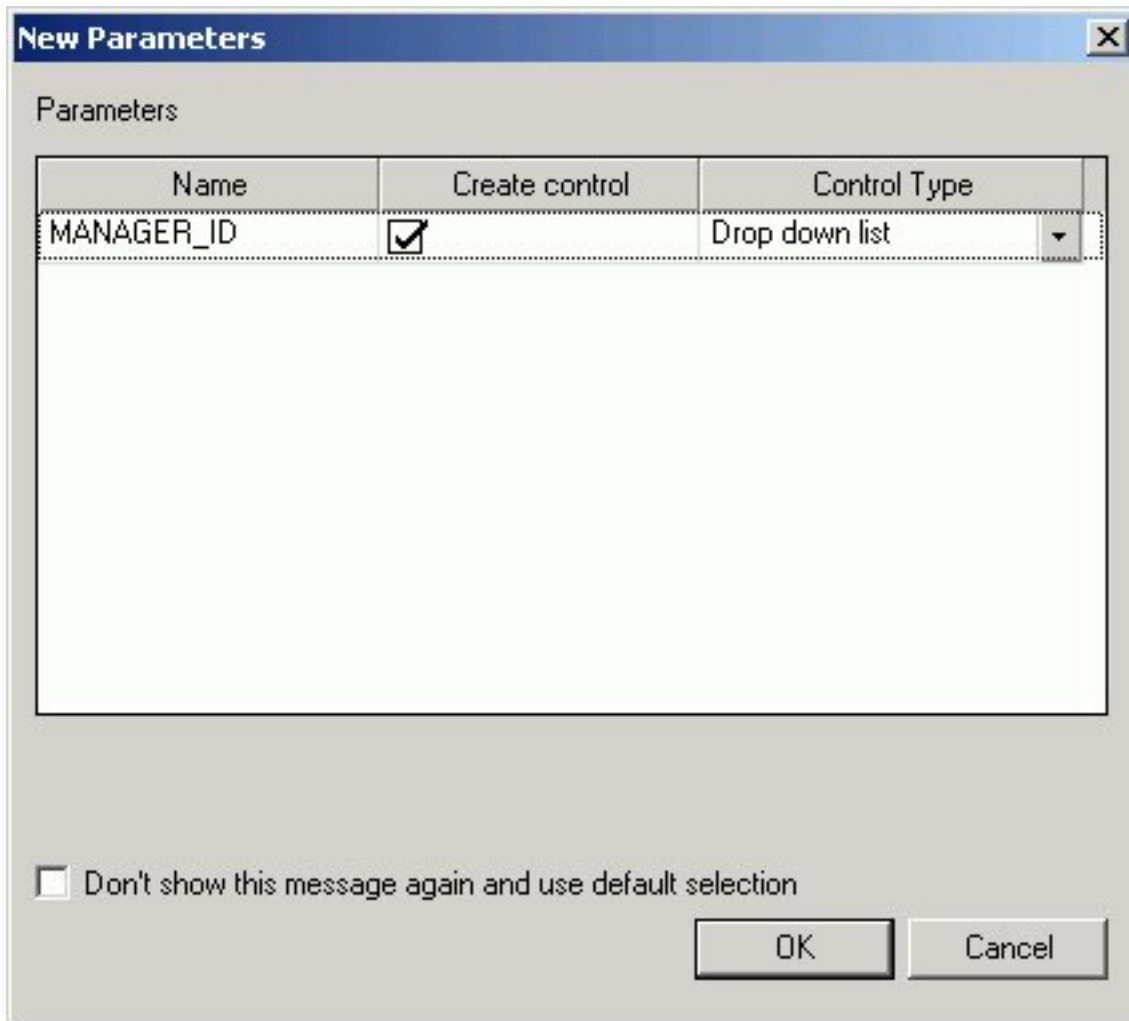
You may select a control type for the parameter from this dialog box, or adjust them later using the Parameters tab.

Note: If the New Parameters dialog box does not appear, ensure that **Show New Parameters dialog** is selected from the Settings dialog box.

4. Click **OK** to close the New Parameters dialog box.

5. The filter appears above the report object.

Repeat this procedure for each additional parameter for the report.

6. Select two or more controls from the parameters and click **Add to chain** from the Positioning toolbar.
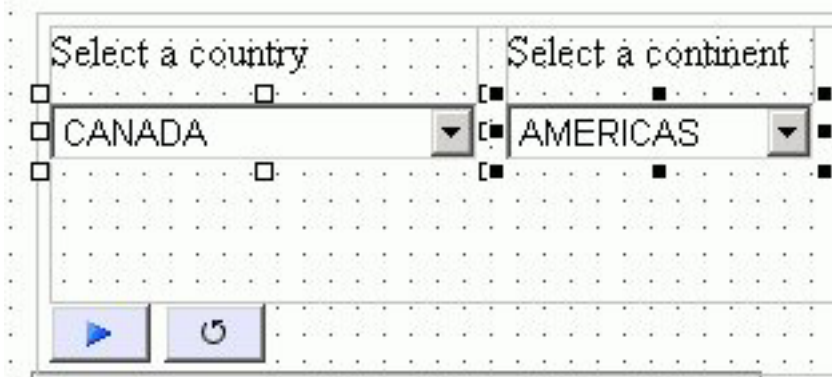
The parameters are chained together.

7. To view the chained parameters in the Design View, select **Chain order** from the Positioning toolbar.

   The parameters are numbered in the logical order that they were chained.
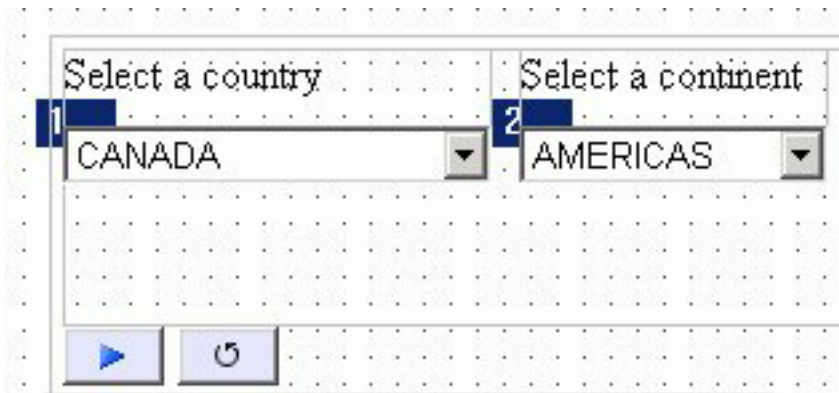
8. Select a numbered parameter, right-click and select **Cache run time data**.

When executing the document, data for the chained parameter is cached to improve performance.

Note: Cache run time data is the default option.

Applying Chaining Logic Options in the HTML Layout Painter

Previously, parameters could only be chained together with an Equal logic scenario, mainly suited for hierarchical data filtering. For example, only seeing cities within the selected state.

Additional chaining logic scenarios include such options as Not Equal to, Greater than, Less than, and so on. This is useful when reusing existing code to populate parameters that may be used multiple times in the same layout, and in different locations of the hierarchy of the chain.

For example, a form offers two lists of dates so that you can select a FROM / TO date range. By chaining these parameters together and applying the Greater than chaining logic, this ensures that when a date is selected for the FROM parameter, only dates that follow the FROM date display in the TO date control, eliminating the possibility of selecting an invalid date range.

Apply Chaining Logic to a Parameter

1. Create a report with parameters in the HTML Layout Painter.

2. Select two or more controls from the parameters and click **Add to chain** 🔗 from the Positioning toolbar.
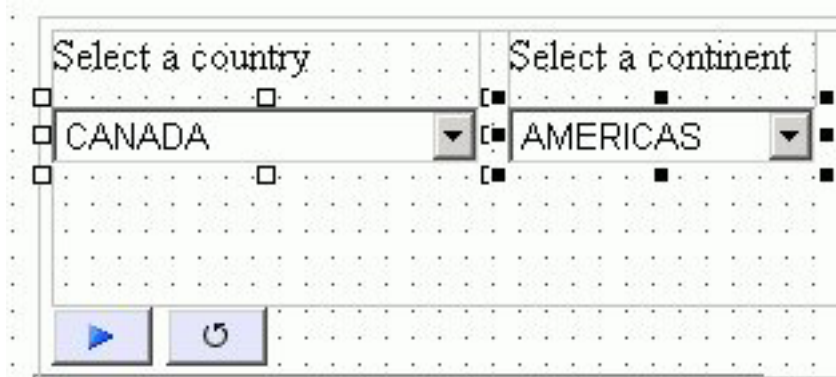
The parameters are chained together.

3. Select a parameter, right-click, and select **Chaining logic** from the context menu.
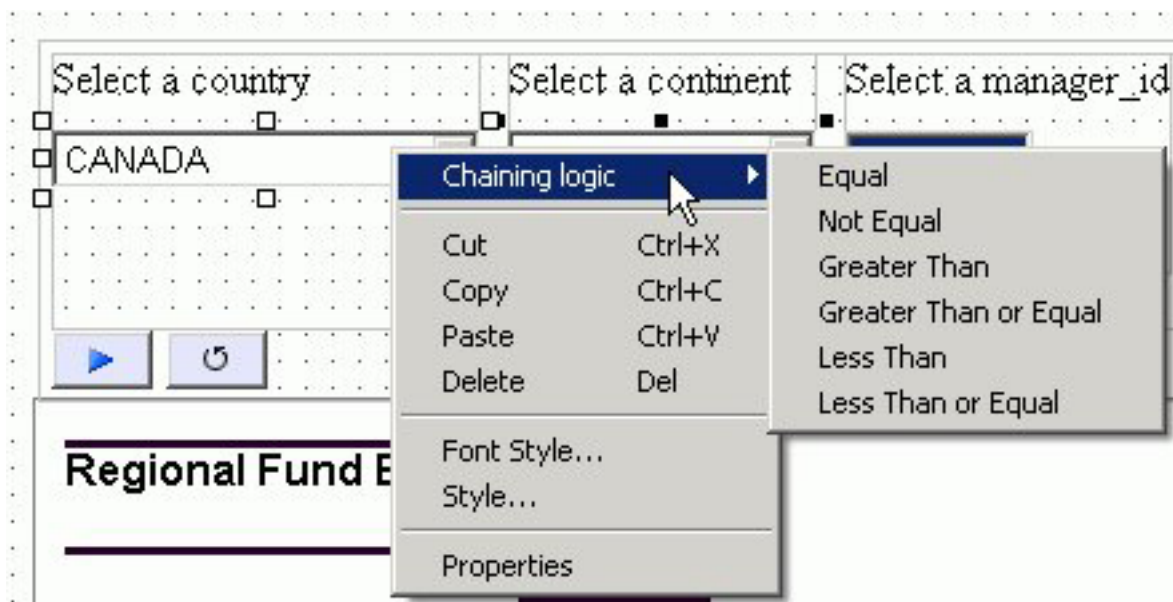
4. Select a chaining logic option from the submenu.

   Note: Equal is the default chaining logic option.

   The chaining logic is applied to the parameter selected.

Customizing Dynamic Parameters

You have the option to allow customized retrieval of dynamic parameter values. By using a custom procedure, you can use the FOCUS language to utilize temporary HOLD files, filtering, etc. This allows the HTML Layout Painter to better integrate with a multitude of large and/or proprietary data sources that may require unique retrieval methods. The procedure must meet the following criteria:

- The procedure must return a name and value pair for each parameter value that will populate the list. The first value is the submission value which is passed to the form control when you click the Submit button. The second value is the display value which is what you will see in the control. These two values can be the same or different.

- The procedure must return the two data values on a single data line in XML format (PCHOLD FORMAT XML).

Note: Different parameters in the same procedure can use both this option and the data source method.

For chained parameters, there are two options based on whether caching is enabled:

- By default, caching of parameter values in not enabled. Each parameter control must be mapped to its own procedure and the developer is responsible for writing filters or WHERE criteria to properly filter the hierarchy of controls.

- If caching is enabled for the chained parameters, a single procedure can be used to populate the controls of the chained variables. In this case, the HTML Layout Painter writes the WHERE criteria.

Hidden Parameters

You can select a HIDDEN control type in the HTML Layout Painter. This allows parameter name value pairs to be passed in the form control without the user seeing them. This option can be chosen in the Parameters tab in the Control Type column. When this control type is selected, the current control assigned to the parameter will be removed. The value of the parameter can be entered in the Value column of the Parameters tab.

Automatically Populating Fields With Parameter Values

When the name of a dynamic parameter matches a corresponding field name in a data source, the HTML Layout Painter automatically populates the field name for the parameter.

The data source is populated by a default based on the first data source specified by a TABLE FILE or GRAPH FILE command. The data source field is populated for the Value and Display field controls in the Parameters tab. This generates a "run ready" Layout report as long as the parameter names match the field names.

Supplying Parameter Values to External Reports

You can supply parameters to external reports the same way you apply them to reports created with the Report Assistant. The report layout can also contain multiple reports associated with a common set of parameters. Launch mechanisms such as a push button, hyperlink, or image can be associated to external reports that contain different sets of parameters. You can also update and target parameters for an external report to a frame or window directly within your report layout.

Before you can supply parameter values to an external report, you must create a hyperlink to the report and associate a launch mechanism to it. After you associate a launch mechanism, such as a push button, hyperlink, or image, to the external report, the report layout is populated with the parameters.

Supply Parameter Values to External Reports With a Push Button

In this procedure you will create a push button to launch an external report and target the output to a window.

1. From the Insert menu, select **Push Button**.

    The cursor changes into a crosshair.
2. Click and drag the crosshair to create a push button and adjust it to the size you want.
3. Right-click the button and select **Create hyperlink**.

    The Hyperlink Properties dialog box opens.
4. Specify the text you want to display as the hyperlink in the Display Text input field.
5. To set the action of the hyperlink to execute an external report, select the **New** icon, then select **External procedure** from the Action drop-down list.
6. Enter the external report's procedure name in the Source field, or click the browse **(...)** button to browse to the procedure.
7. Optionally, direct the output to a specific location by selecting **Window** or **Frame** in the Target Type field.
8. Specify the Target/Template Name for the target frame or window.
9. Click **OK**.

    The parameters for the report are automatically added to objects in the layout and displayed in the Parameters tab.
10. From the File menu, click **Run**.
11. Click the push button to view the external report.

    The report opens in a separate window as you specified in the Hyperlink Properties dialog box for that push button.
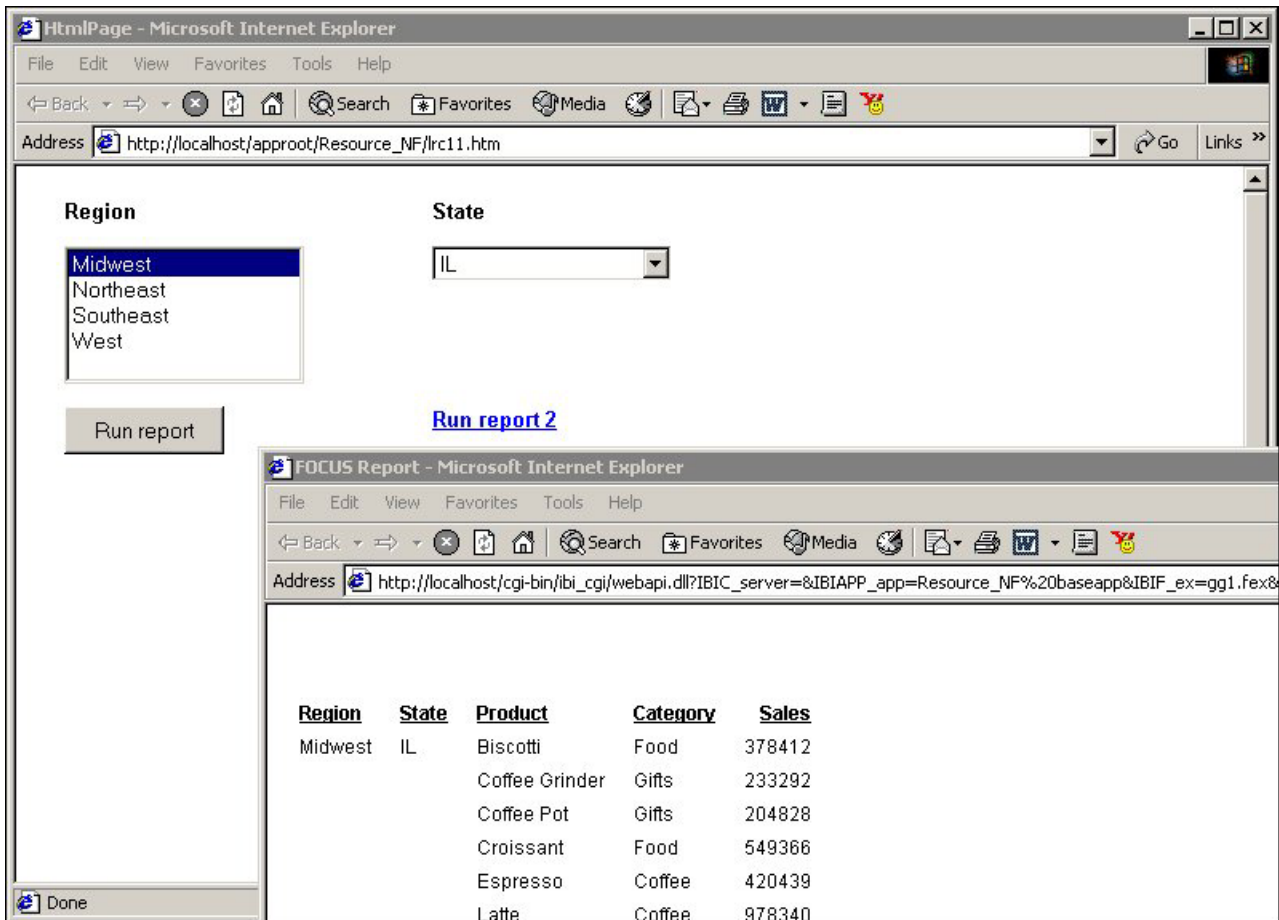
Supply Parameter Values to an External Report With an Image

In this procedure you will insert an image that will launch an external report when you click it. You will target the output to a window.

1. From the Insert menu, select **Image**.

   The cursor changes into a crosshair.

2. Click and drag the crosshair to position the image.

   The Get source file dialog box opens.

3. Navigate to the image using the Look in menu, select the image, and click **Open**.

4. Adjust the image to the desired size and location.

5. Right-click the image and select **Create hyperlink**.

   The Hyperlink Properties dialog box opens.

6. To set the action of the hyperlink to execute an external report, select the **New** icon, then select **External procedure** from the Action drop-down list.

7. Enter the external report's procedure name in the Source field, or click the browse **(...)** button to browse to the procedure.

8. Direct the output to a window by selecting **Window** in the Target Type field.

9. Specify the Target/Template Name for the target frame or window.

10. Click **OK**.

The parameters for the report are automatically added to objects in the layout and displayed in the Parameters tab.

11. From the File menu, click **Run**.

12. Click the image to view the external report.

The report opens in the window you specified in the Hyperlink Properties dialog box for that hyperlink.
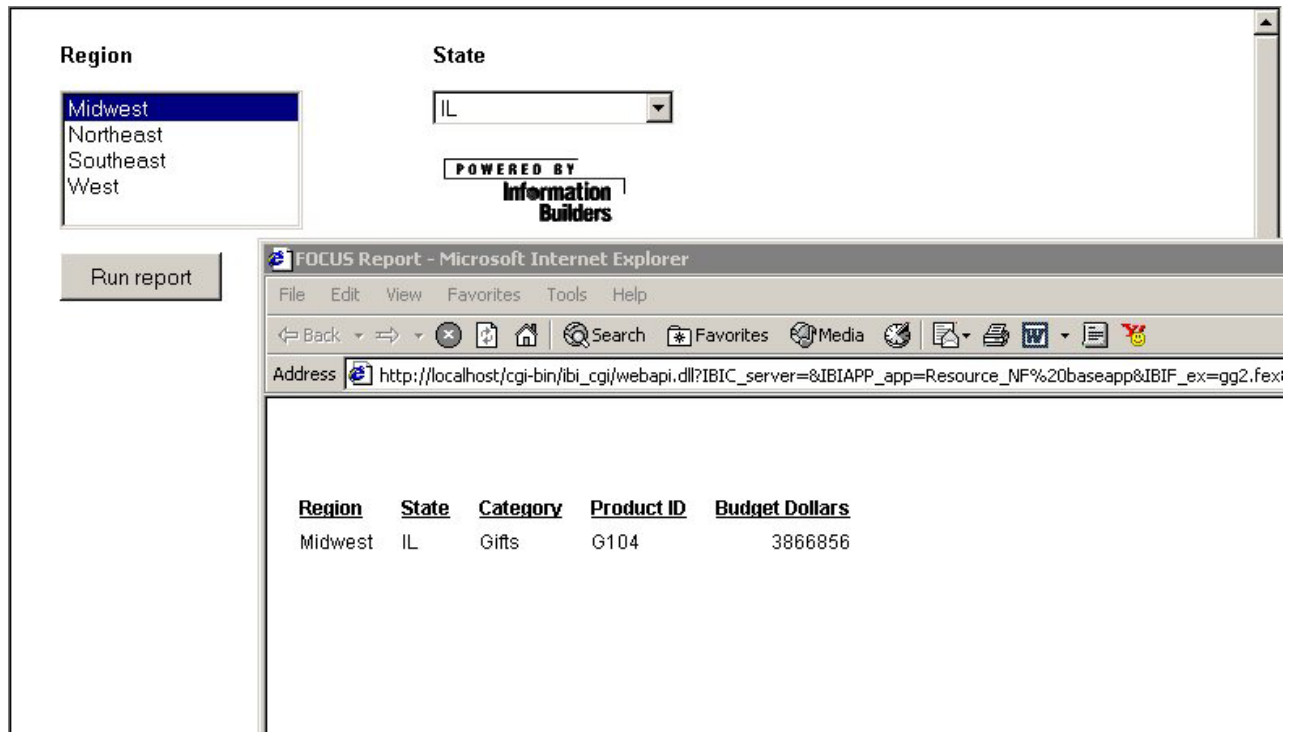
Supply Parameter Values to an External Report With a Hyperlink

In this procedure, you will create a frame and target report output from an external report to it. You will also create a hyperlink to launch the external report.

1. From the Insert menu, select **Frame**. Click and drag the crosshair to create a frame and adjust it to the size you want. Enter a name for the frame by double-clicking the **Name** field in the Properties tab of the Properties window.

2. From the Insert menu, select **Hyperlink**. Click and drag the crosshair to create a hyperlink and adjust it to the size you want.

The Hyperlink Properties dialog box opens.

3. Specify the text you want to display as the hyperlink in the Display Text input field.

4. To set the action of the hyperlink to execute an external report, select the **New** icon, then select **External procedure** from the Action drop-down list.

5. Enter the external report's procedure name in the Source field, or click the browse **(...)** button to browse to the procedure.

6. Direct the output to the frame you inserted by selecting **Frame** in the Target Type field.

7. Select the name of the frame you created from the Target/Template Name drop-down list.

8. Click **OK**.

   The parameters for the report are automatically added to objects in the layout and displayed in the Parameters tab.

9. From the File menu, click **Run**.

10. Click the hyperlink to view the external report.

    The report opens in the frame you specified in the Hyperlink Properties dialog box for that hyperlink.

Supplying Parameter Values to Multiple External Reports

In this example, you will add two reports to a layout. You will associate a push button to one report and target the output to a window. You will associate a hyperlink to the second report and target the output to a frame.

1. From the Insert menu, select **Push Button**.

   The cursor changes into a crosshair.

2. Click and drag the crosshair to create a push button and adjust it to the size you want.

   A push button is created in the layout and assigned the name button*n*, where *n* is a number.

3. Right-click the button, and select **Create hyperlink**.

   The Hyperlink Properties dialog box opens.

4. Type **Run report** in the Display Text input field.

5. Set the action of the hyperlink to execute a report by clicking the **New** icon and selecting **External procedure** from the Action drop-down list.

6. Enter the external report's procedure name in the Source field, or click the browse **(...)** button to browse to the procedure.

7. In the Target Type field, select **Window** from the drop-down list.

8. In the Target/Template Name field, select **New window** frame from the drop-down list.

9. Click **OK**.

    The parameters for the first report are automatically added to objects in the layout and displayed in the Parameters tab.

10. From the Insert menu, select **Frame**. Click and drag the crosshair to create a frame and adjust it to the size you want.

11. Double-click the Name field in the Properties tab of the Properties window and enter **frame1**.

12. From the Insert menu, select **Hyperlink**. Click and drag the crosshair to create a hyperlink and adjust it to the size you want.

    The Hyperlink Properties dialog box opens.

13. Type **Run report 2** in the Display Text input field.

14. Set the action of the hyperlink to execute a report by clicking the **New** icon and selecting **External procedure** from the Action drop-down list.

15. Enter the external report's procedure name in the Source field, or click the browse **(...)** button to browse to the procedure.

16. In the Target Type field, select **Frame** from the drop-down list.

17. In the Target/Template Name field, select the name of the frame you created **(frame1)**.

18. Click **OK**.

19. From the File menu, select **Run**.

20. Click the push button to view the first external report.

    The report opens in a separate window as you specified in the Hyperlink Properties dialog box for that push button.
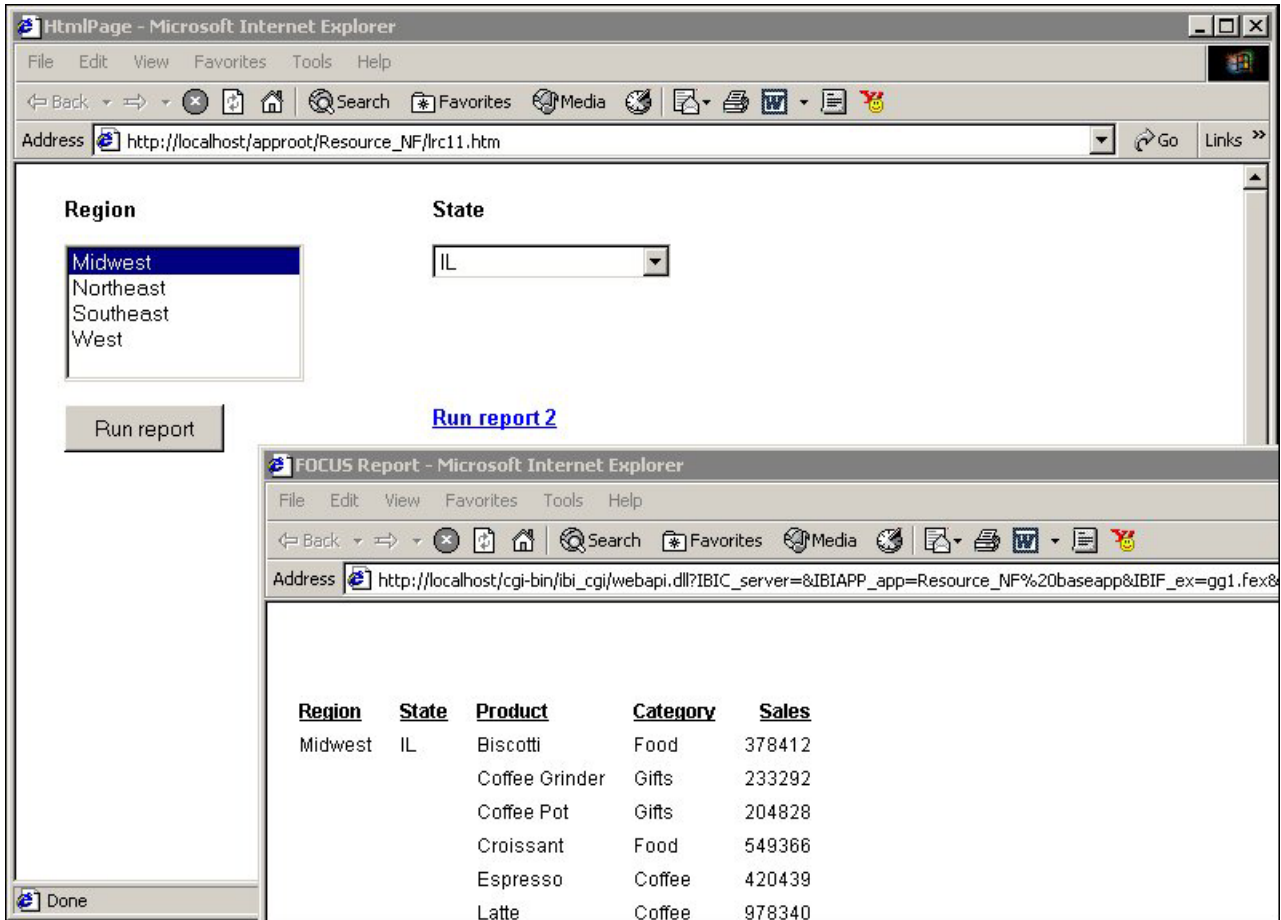
21. Click the hyperlink to view the second external report. The report opens in the frame you specified in the Hyperlink Properties dialog box for that hyperlink.

Styling Your Layout

When you create an HTML page in the HTML Layout Painter, the objects in the layout will display in your browser using the browser's default styles. You can customize the appearance of your HTML page by adding a theme or template. A template can be any HTML file containing styling information and other elements.

Adding a CSS or Script to the Layout

Cascading Style Sheets (CSS) or scripts can be added to your layout to determine the look of the Web page. This is a good way to quickly apply corporate styling to your layout or to assign global styling to a layout. It is also an easy method of assigning default styling to all elements in the entire layout with one action. Styling included in a CSS or script will display only in Preview mode.

Add a CSS or Script to the Layout

1. From the Insert menu, select **CSS/Scripts**.

   The Insert Web Files dialog box opens.

2. Click the **New** icon and navigate to the directory in which your CSS or script resides, select the file, and click **Open**.

3. Add additional files if desired. Click **OK** to add the specified files.

Setting HTML Page Properties

You can set properties for the HTML page you create in the HTML Layout Painter in the Properties tab of the Properties window.

Properties set for the HTML page will be inherited by most objects added to the layout. Once a style has been changed for an object in the layout, it cannot be styled with a template or theme. It is recommended that global styling of the HTML page be set before properties are set for individual objects.

HTML Page Properties in the Properties Window

When the background of a report is selected, the Properties tab in the Properties window contains options that control the body of the HTML page.
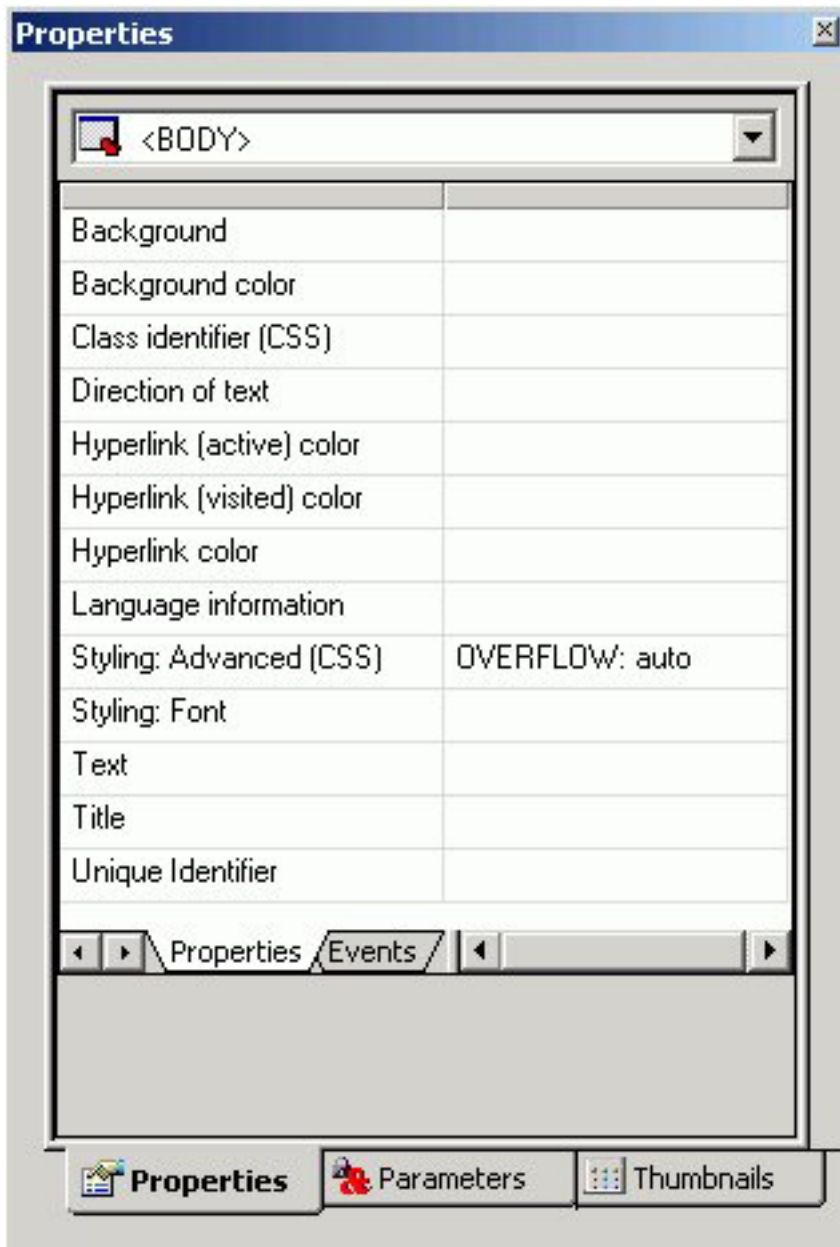
Click a property to display a description of the selected property at the bottom of the Properties window.

Specifying Browser Defaults with Style Composer

You can use the Style Composer to control default settings for font, background properties, position mode, flow control, margins, list styles, and visual effects.

Note: On the following pages, the images of the Style Composer windows show many blank fields. You may actually see the words "Not Set" populated in these blank fields when viewing these Style Composer windows in Developer Workbench.

Access the Style Composer

To access the Style Composer, complete the following steps.

1. Right-click the background of an HTML page in the HTML Layout Painter.
2. From the pop-up menu, select **Style**.

   The Style Composer window opens.
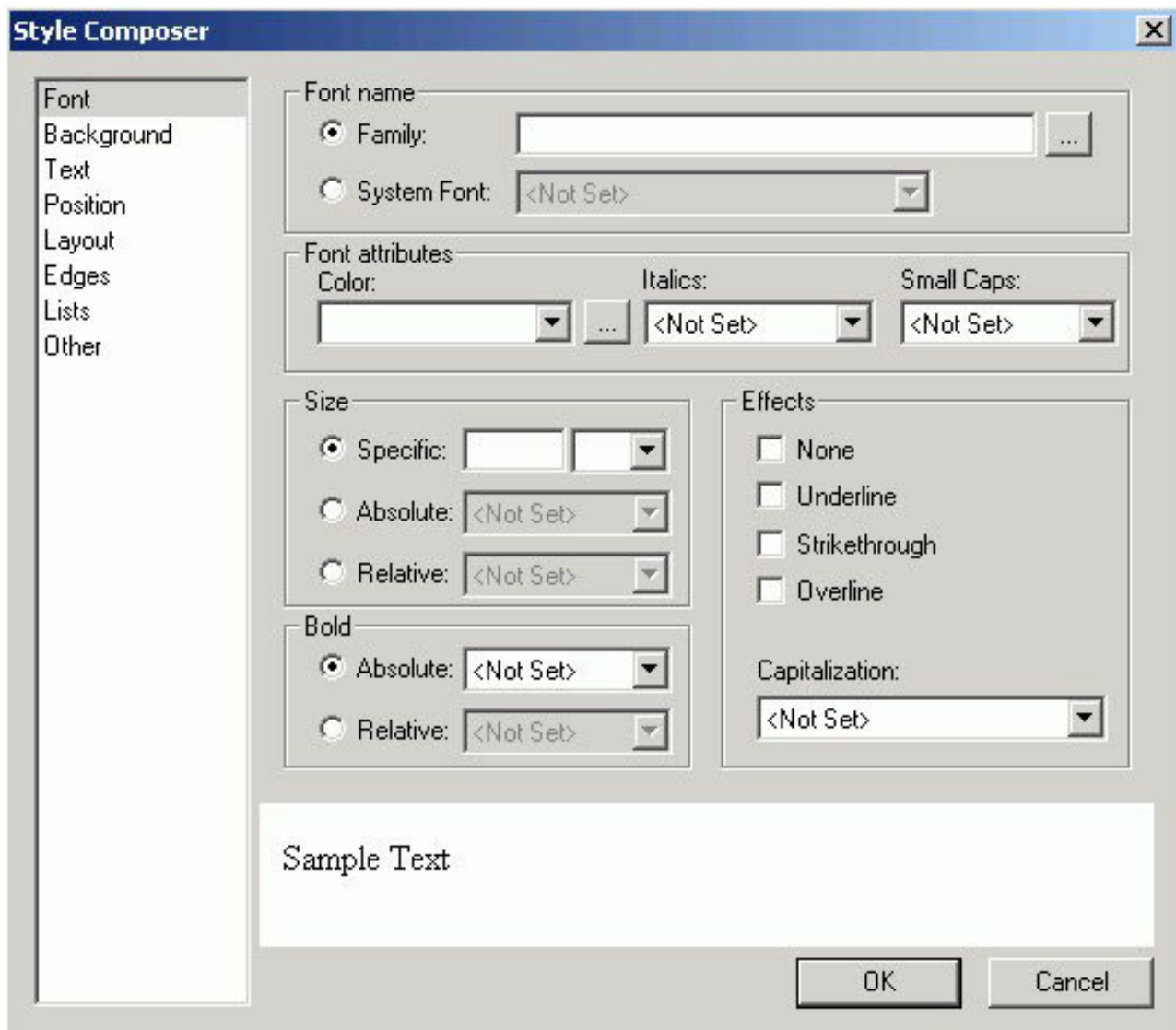
Specifying Font Styles Using the Style Composer

To specify the font styles that will be used in the browser for your HTML report, make your selections in the Font window of the Style Composer.

The Font window of the Style Composer is comprised of the following elements:

Font name

> Determines the font displayed in a browser.

> You can specify: **Family** (launches the Font Picker dialog box) or **System Font**.

Font attributes

> Determines the attributes of the font displayed in a browser.

> The options include: **Color, Italics, Small Caps**.

Size

> Determines the size of the font displayed in a browser.

> The options include: **Specific, Absolute, Relative**.

Bold

> Determines whether the font is displayed as bold in a browser.

> The options include: **Absolute, Relative**.

Effects

> Determines whether the font effects displayed in a browser.

> The options include: **None, Underline, Strikethrough, Overline, Capitalization**.

Specifying Background Properties Using the Style Composer

To specify the background styles that will be used in the browser for your HTML report, make your selections in the Background window of the Style Composer.
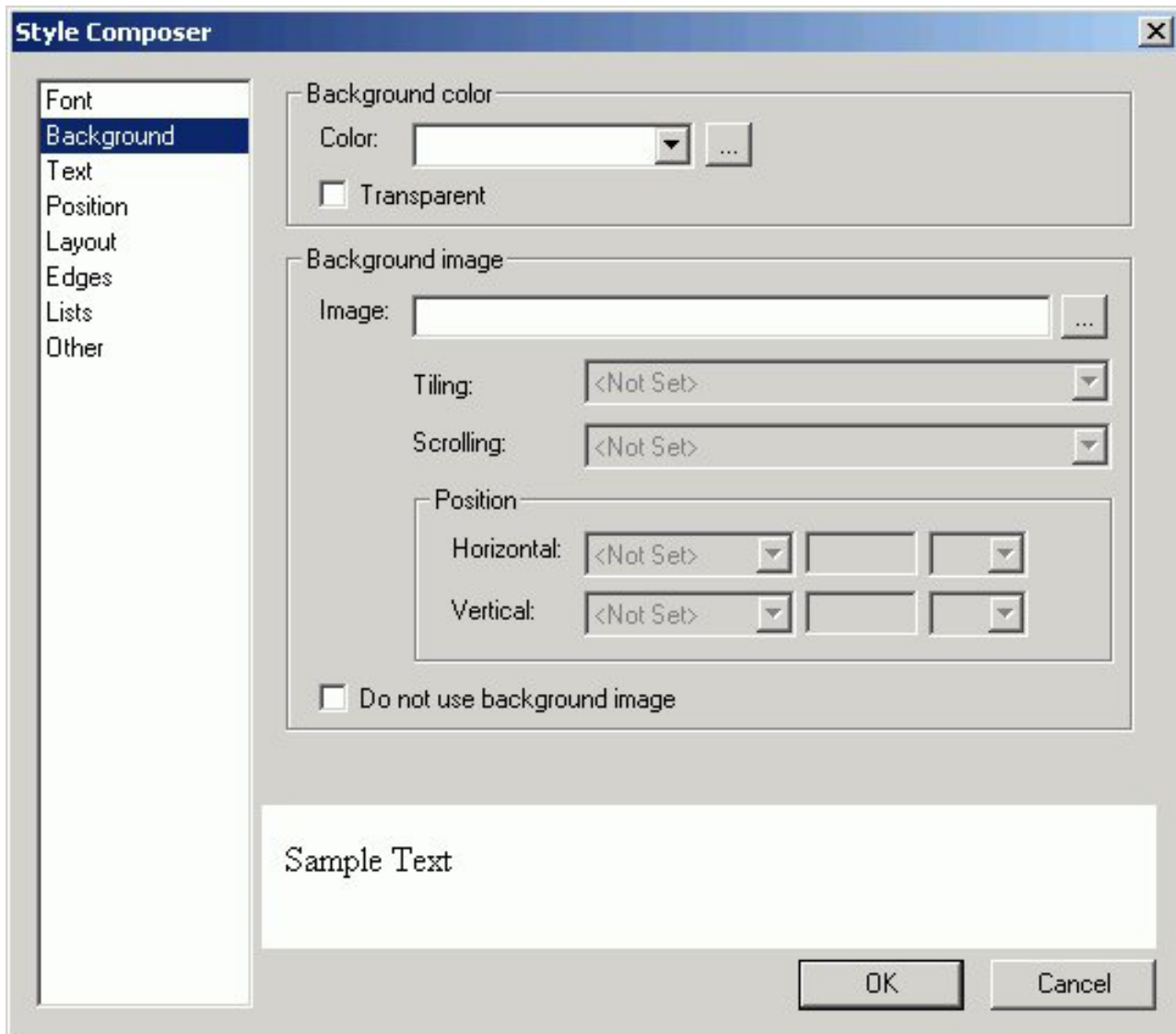
The Background window of the Style Composer is comprised of the following elements:

Background color

> Determines the font displayed in a browser.

> You can specify: **Color, Transparent**.

Background image

> Determines the properties of the background image displayed in a browser.

> The options include: **Image, Tiling, Scrolling, Position** (Horizontal and Vertical), **Do not use background image**.

Specifying Text Styles Using the Style Composer

To specify the text styles that will be used in the browser for your HTML report, make your selections in the Text window of the Style Composer.
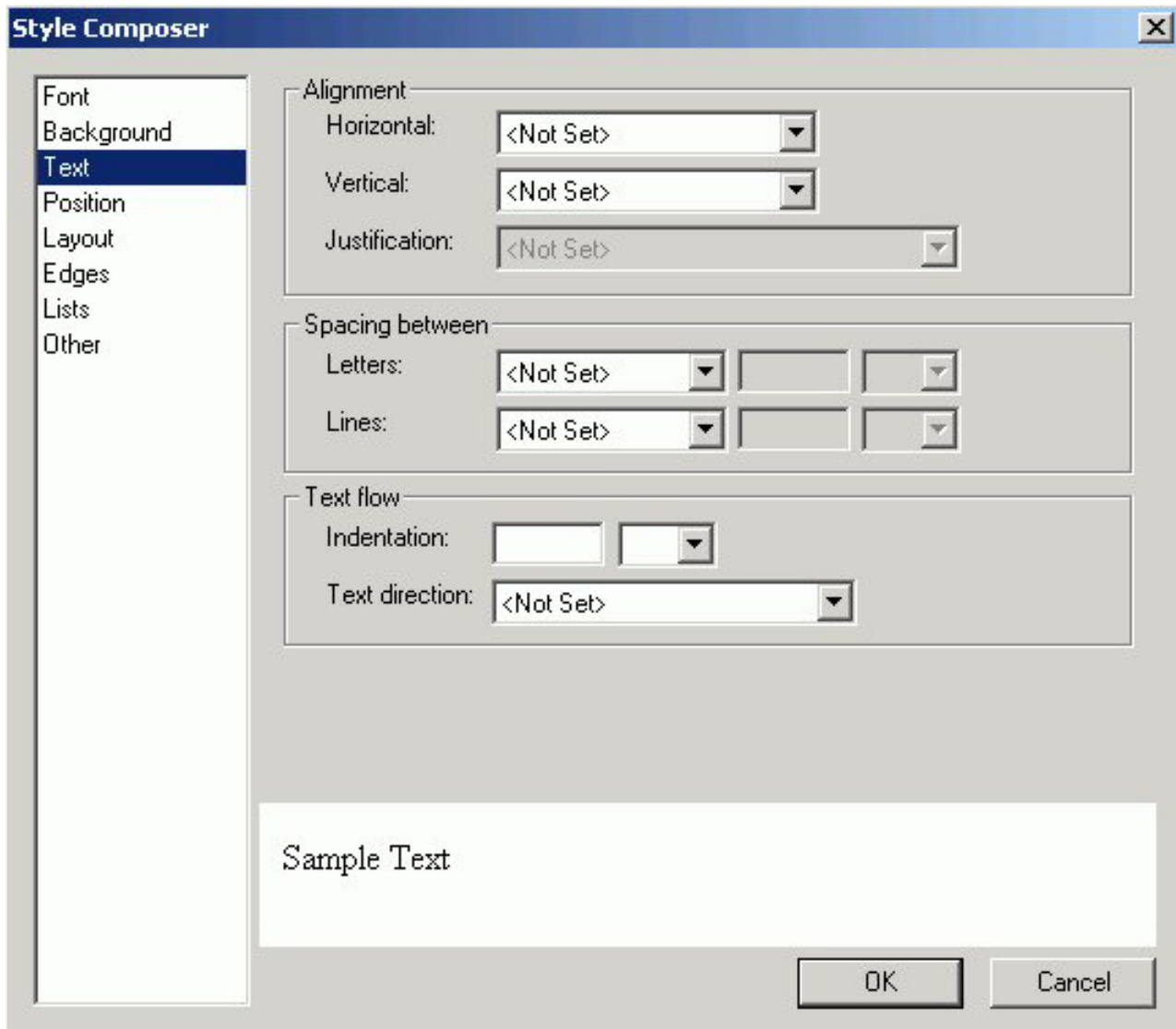
The Text window of the Style Composer is comprised of the following elements:

Alignment

> Determines the alignment of a text.
>
> You can specify: **Horizontal, Vertical, Justification**.

Spacing Between

> Determines the spacing.
>
> You can specify spacing between the following text elements: **Letters, Lines**.

Text Flow

> Determines the flow of the text.
>
> You can specify: **Indentation, Text Direction**.

Specifying Position Mode Using the Style Composer

To specify the position mode that will be used in the browser for your HTML report, make your selections in the Position window of the Style Composer.
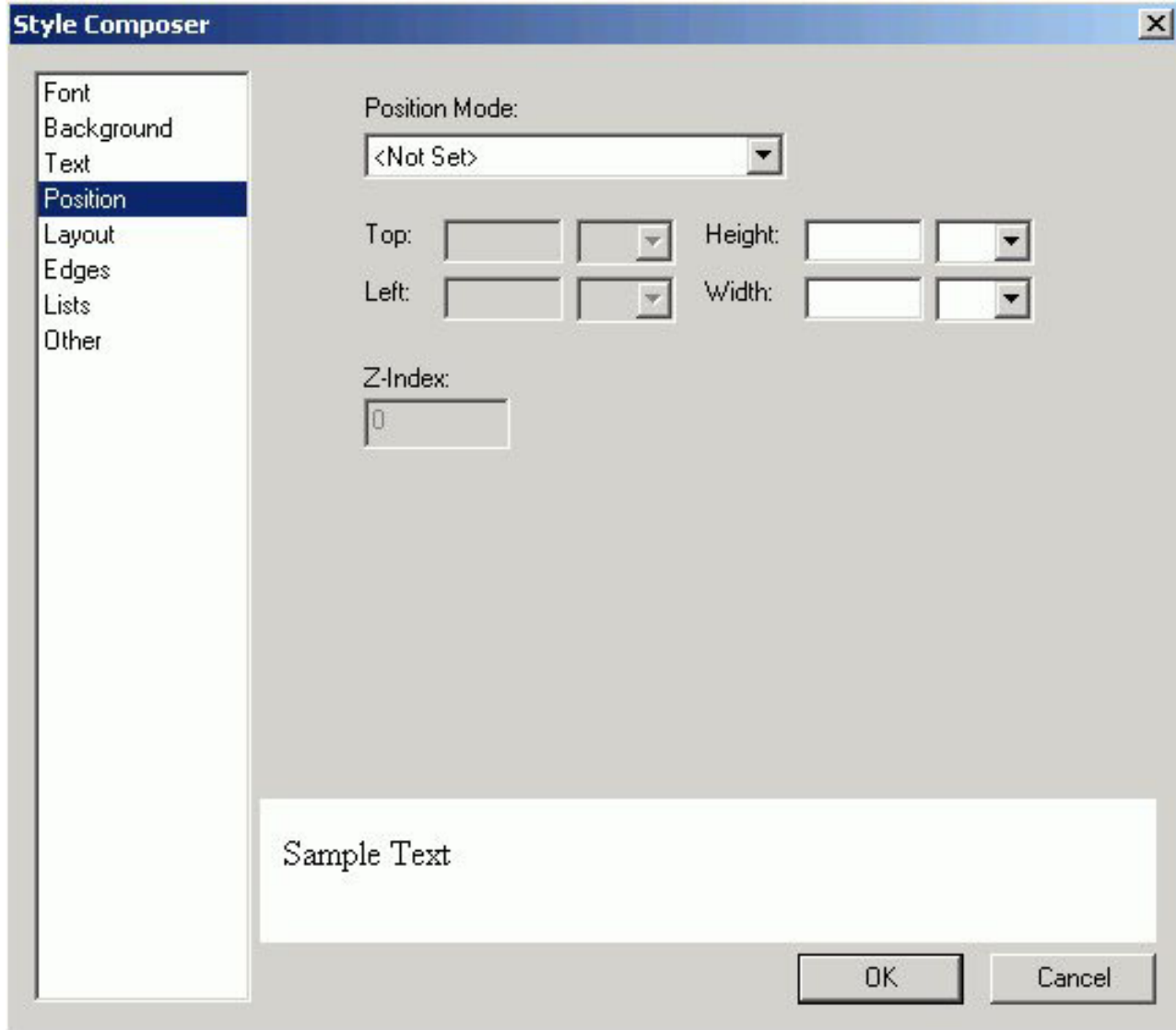
The Position window of the Style Composer is comprised of the following elements:

Position Mode

> From which you can specify: **Position in normal flow, Offset from normal flow, Absolutely position**.

Height/Width

> When Absolutely Position is selected, you can specify position indicators in the measurements.
>
> You may specify: **Top, Left, Z-Index**.

Note: Z-Index is optional. It sets or retrieves the stacking order for absolute or relatively positioned objects.

Specifying Layout Styles Using the Style Composer

To specify the layout styles that will be used in the browser for your HTML report, make your selections in the Layout window of the Style Composer.
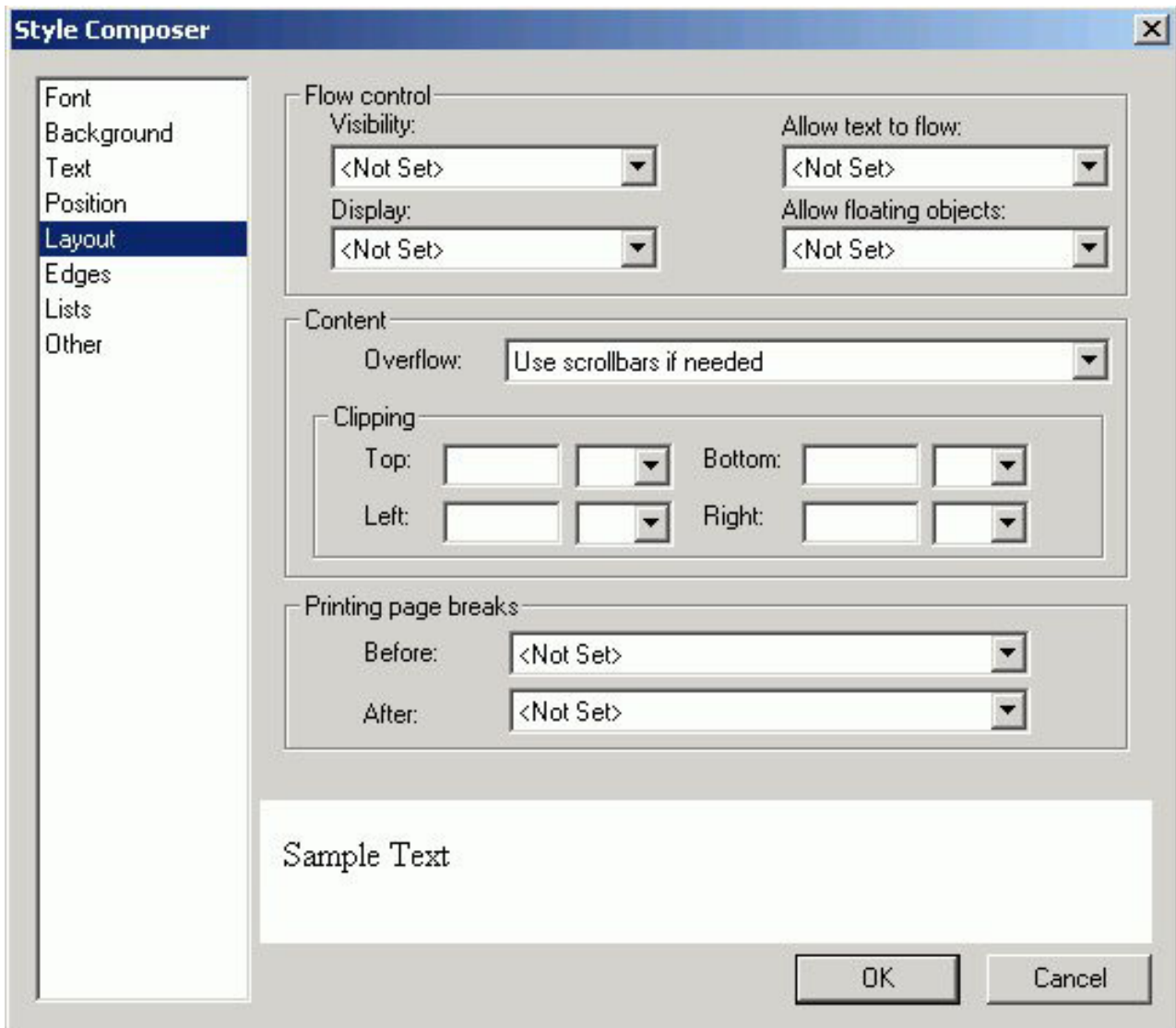
The Layout window of the Style Composer is comprised of the following elements:

Flow Control

From which you can specify: **Visibility, Allow Text to Flow, Display, Allow Floating Objects**.

Content

From which you can specify: **Overflow**.

Clipping

From which you can specify whether or not to clip the layout from the following positions: **Top, Bottom, Left, Right**.

Printing page breaks

From which you can specify: **Before, After**.

Specifying Margin Styles Using the Style Composer

To specify the margin styles that will be used in the browser for your HTML report, make your selections in the Margins window of the Style Composer, which is shown in the following image.
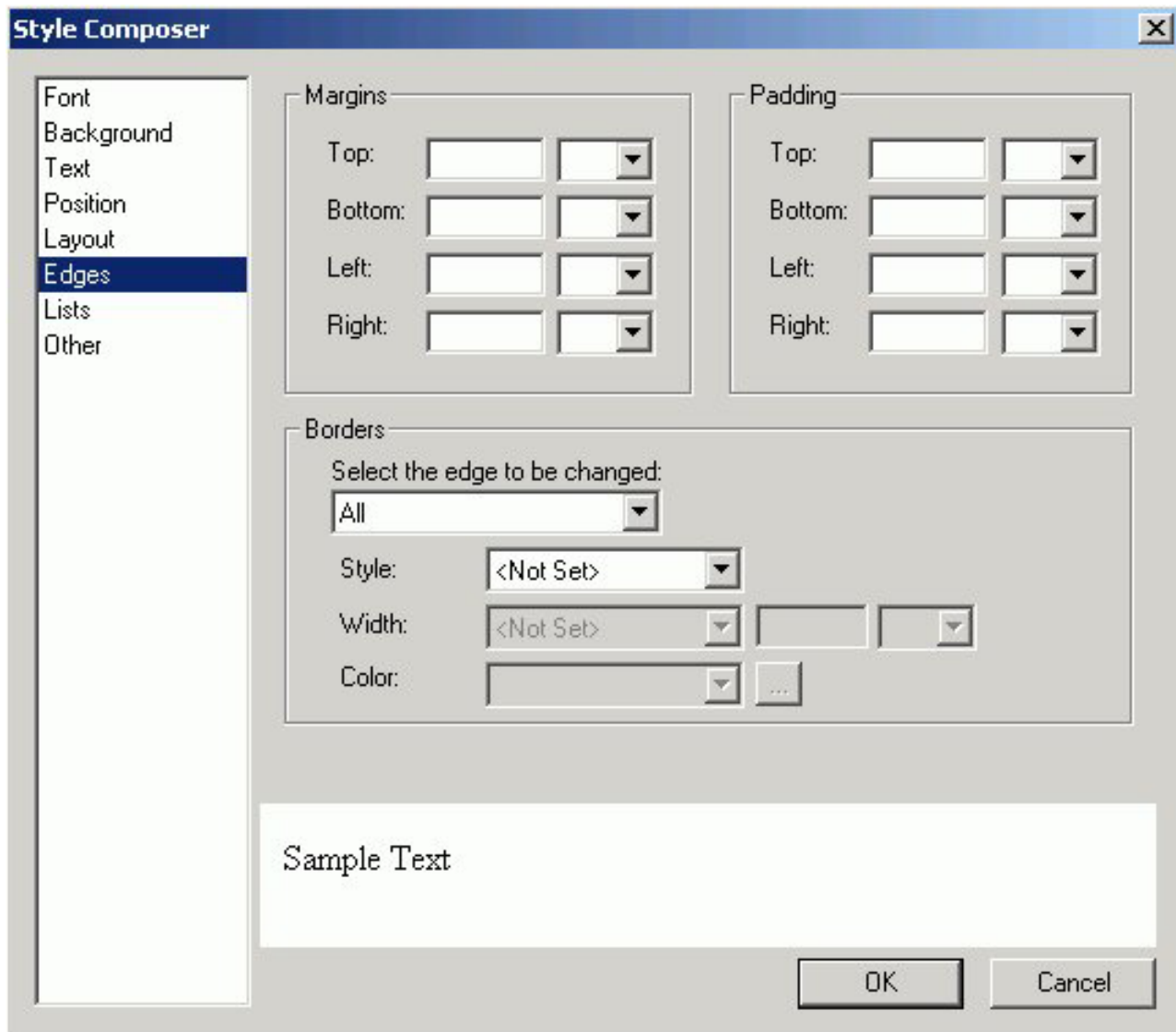
The Edges window of the Style Composer is comprised of the following elements:

Margins

From which you can specify: **Top, Bottom Left, Right** margins.

Padding

From which you can specify: **Top, Bottom, Left, Right** padding.

Borders

From which you can specify: **Select the edge to be changed, Style, Width, Color**.

Note: Borders are displayed as a single solid line in the Style Composer preview window, regardless of whether you select a single or double line border. Borders are accurately displayed in the Style Composer Design View and in the HTML output of an application.

Specifying List Styles Using the Style Composer

To specify the list styles that will be used in the browser for your HTML report, make your selections in the Lists window of the Style Composer.
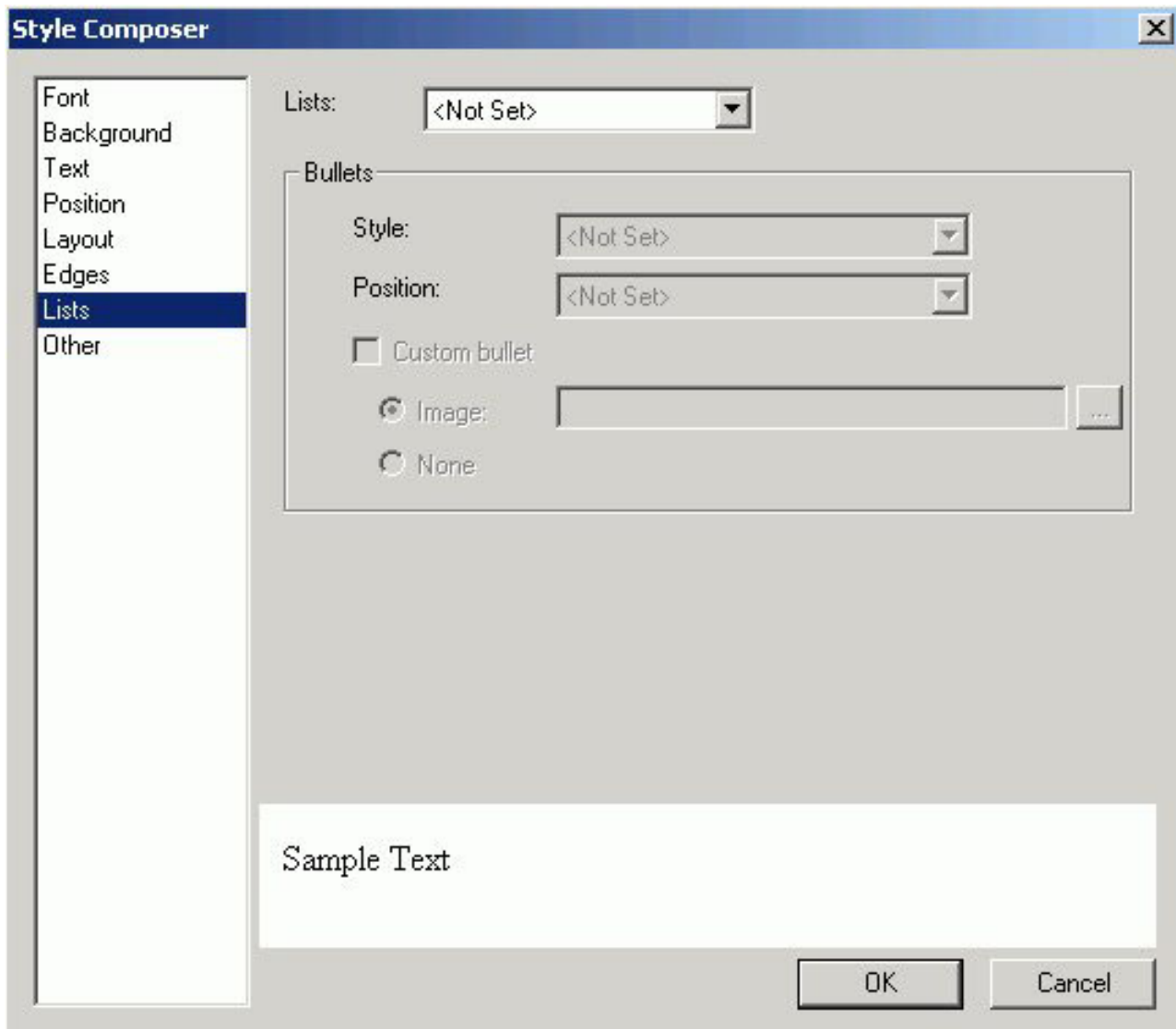
The Lists window of the Style Composer is comprised of the following elements:

Lists

From which you can specify: **Bulleted, Unbulleted**.

Bullets

From which you can specify: **Style, Position, Custom bullet**.

Specifying Interface Effects Using the Style Composer

To specify the interface styles and visual effects that will be used in the browser for your HTML report, make your selections in the Other window of the Style Composer.
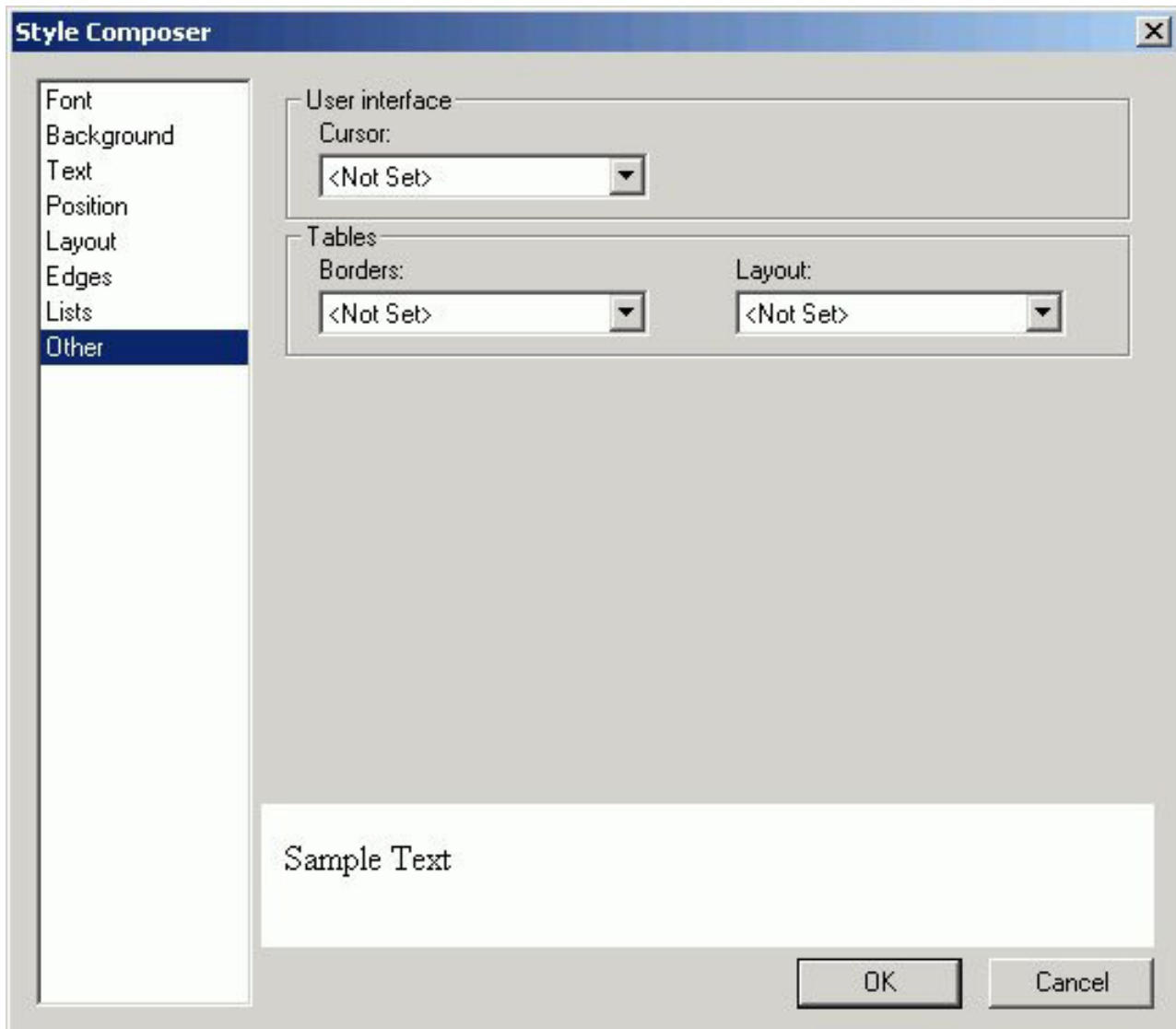
The Other window of the Style Composer is comprised of the following elements:

User Interface

From which you can specify: **Cursor**.

Tables

From which you can specify: **Borders, Layout**.

Laying Out Objects With the HTML Layout Painter

You can change the size and position of objects in your layout in the following ways:

- Set relationships between objects. This is done by setting relationships between objects and a controlling, or dominant, object. The controlling object is the last object selected. For details, see Setting Relationships Between Objects.
- Click and drag with your mouse.

Setting Relationships Between Objects

You can set relationships between underlying objects and a controlling, or dominant, object. The controlling object is the last object or placeholder selected. You can use these relationships to set properties of the selected objects according to the controlling object. Any object on the HTML page layout can be the controlling object.

You can set the following types of relationships:

- Relationships that maintain a distance between objects. This is useful since a report or graph may take up more or less room when your procedure is run than is accounted for in the layout. When setting relationships between objects in this way, you should select the corners of the objects that are closest to each other. This will ensure that the reports do not overlap if either report takes up more room than anticipated.
- Relationships that affect the positioning of objects. This allows you to ensure that objects remain aligned regardless of a change in size or position of one of the objects and that objects remain the same size even when an object's size is changed.

Set Object Size

1. Highlight multiple objects with your mouse by holding the **Shift** key to select items contiguously, or the **Control** key to select items non-contiguously. The controlling object is the last object selected.

   The size buttons on the positioning toolbar are active.

2. Click the button in the positioning toolbar that corresponds to what you want:

   Make same width sets the width of the highlighted objects to the width of the controlling object.

   Make same height sets the height of the highlighted objects to the current height of the controlling object.

   Make same size sets the height and width of the highlighted objects to the height and width of the controlling object.

All selected objects change in size according to the size of the controlling object. If an image becomes distorted when you resize it, you can restore its original size. Right-click the image, select **Image Properties**, and then **Restore Size**.

Set the Distance Between Objects

1. Highlight multiple elements with your mouse holding the **Shift** key to select items contiguously, or the **Control** key to select items non-contiguously. The controlling object is the last object selected.

2. Click the button in the positioning toolbar that corresponds to the relationship you want. The selected objects' position will be set according to the controlling object, which is the last object selected. The options are:

Relate top left maintains the distance between the controlling object's top left corner and the highlighted objects' top left corners.

Relate top right maintains the distance between the controlling object's top right corner and the highlighted objects' top left corners.

Relate bottom right maintains the distance between the controlling object's bottom right corner and the highlighted objects' top left corners.

Relate bottom left maintains the distance between the controlling object's bottom left corner and the highlighted objects' top left corners.

For details on the buttons in the positioning toolbar, see Positioning Toolbar.

Break Relationships

Click **Break** in the toolbar.

Positioning Toolbar

The positioning toolbar contains the buttons listed in the following table.

For buttons that use relationships, the relationship is controlled by the controlling object, or dominant object, which is the last object or placeholder selected. These buttons are only highlighted when more than one object is selected.

Button	Descriptions
	Toggles the grid on and off. Use the grid to assist in lining up objects in the layout. This button is recessed when the grids are enabled.
	Toggle visibility shows hidden objects. The hidden visibility option is set in the Layout section of the Style Composer.
	Specifies Tab Order.
	Aligns the highlighted objects' left edge with the left edge of the controlling object.
	Aligns the highlighted objects' right edge with the right edge of the controlling object.
	Aligns the highlighted objects' top edge with the top edge of the controlling object.
	Aligns the highlighted objects' bottom edge with the bottom edge of the controlling object.
	Aligns objects at the horizontal center point of the canvas in Design View.
	Aligns objects at their vertical center (or middle) point of the canvas in Design View.
	Sets the width of the highlighted objects to the width of the controlling object.
	Sets the height of the highlighted objects to the current height of the controlling object.

Button	Descriptions
	Sets the height and width of the highlighted objects to the height and width of the controlling object.
	Maintains the distance between the controlling object's top left corner and the highlighted objects' top left corners.
	Maintains the distance between the controlling object's top right corner and the highlighted objects' top left corners.
	Maintains the distance between the controlling object's bottom right corner and the highlighted objects' top left corners.
	Maintains the distance between the controlling object's bottom left corner and the highlighted objects' top left corners.
	Breaks the relationship set between highlighted objects. This button is only highlighted when selected objects have relationships set.
	Toggles the display of arrows illustrating the relationships between objects. This button is only highlighted when selected objects have relationships set. This button will stay highlighted when relationships are displayed. It will not be highlighted when relationships are not displayed.
	Adds a parameter to a chain. Chained dynamic values are filtered as selections are made to each parameter control. Each time a selection is made, all chained parameters will be dynamically updated. For details about chained parameters, see Filtering Dynamic Parameter Values.
	Removes a chained parameter from a chain. For details about chained parameters, see Filtering Dynamic Parameter Values.
	Activates chained parameters for viewing and editing. For details about chained parameters, see Filtering Dynamic Parameter Values.
	Synchronizes and report/graph object to an Active Report when using Active Dashboards. For more information, see Creating Active Dashboards.
	Shows the synchronized report groups when working with Active Dashboards in the HTML Layout Painter. For more information, see Creating Active Dashboards.

Customizing the Layout

All changes to your layout be can made in the Design mode of the HTML Layout Painter or the HTML editor. The changes made to a layout in an editor will be preserved when you re-enter the HTML Layout Painter.

Creating Active Dashboards

An HTML Active Report (AHTML) is a report that is designed for offline analysis.

The HTML Layout Painter has extended the functionality of Active Reports by providing the features to build an Active Dashboard; an HTML Form with one or more Active Report procedures and controls to mimic Active Report menu options, to allow global modification of multiple Active Reports in HTML Layout created pages.

The process of creating Active Dashboards in the HTML Layout Painter consists of:

- Adding an Active Report to the HTML Layout.
- Binding, or synchronizing, other report and graph objects to the Active Report in the layout.
- Configuring Active Report controls that mimic Active Report menu options at run-time.
- Exporting the HTML Layout as a FOCUS procedure (.fex) for scheduling and distributing Active Dashboards.

Binding Objects to an Active Report

You can create multiple views of an Active Report by binding a report or graph object to an Active Report. Binding, or synchronizing, is the act of configuring an association between an Active Report and other report/graph objects in the HTML Layout.

You may synchronize report/graph objects and show the synchronized report groups in the HTML Layout. The synchronize options are available from the Positioning toolbar in the HTML Layout Painter.

Note: You may only synchronize objects to one Active Report at a time. If you try to synchronize an object to a second Active Report, the first synchronization is removed.

Synchronize Report and Graph Objects to Active Reports

1. In the HTML Layout Painter, you may add an Active Report to the layout by using any of the following methods:
   - Select **New Report** from the Insert menu.
     Double-click the report object to create the Active Report in Report Painter.
   - Select **New Report** from the Insert menu.
     Right-click the report object and select **Reference existing procedure** from the context menu to add the Active Report.
   - Select **Import Existing Procedure** from the Insert menu to add the Active Report.
2. Add a report or graph object to the layout:
   - Select **New Report** or **New Graph** to from the Insert menu.
     The cursor changes into a crosshair.
   - Click and drag the crosshair to create a report or graph object and adjust it to the size you want.
3. Required: Select the objects to be synchronized:
   - Select the report/graph object as the object to be synchronized.
   - While pressing and holding the **Ctrl** key on your keyboard, select the Active Report as the report that you want to bind to.
     The synchronize buttons on the Positioning toolbar are activated.
     In the example below, the Active Report is the binding object and the graph object is about to be synchronized to the Active Report.
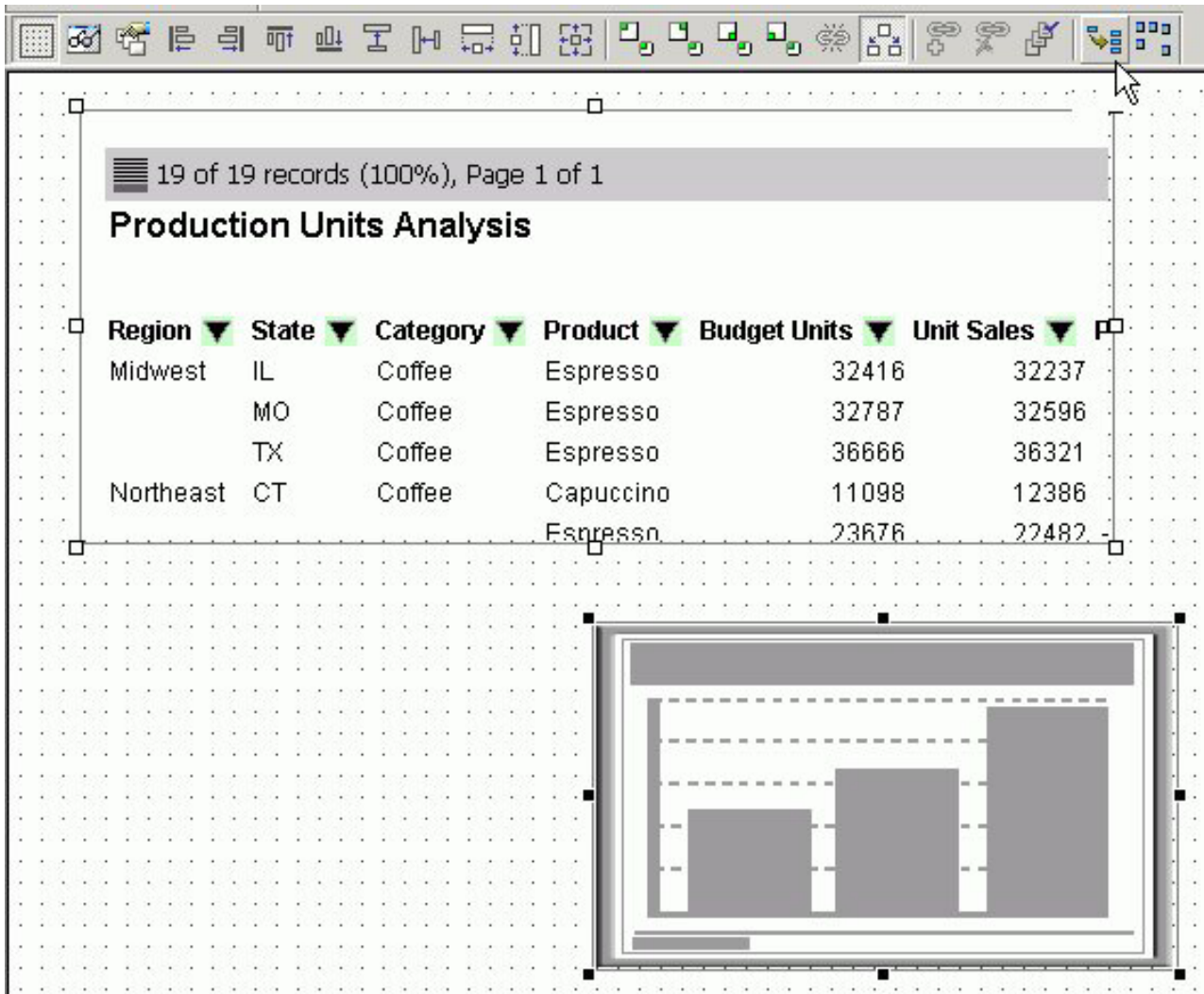
> **Note:** The binding object (Active Report) is indicated by clear boxes around the edges. The synchronized object (report/graph object) is indicated by solid black boxes around the edges.

4. Click the **Synchronize to Active Report** button on the Positioning toolbar.
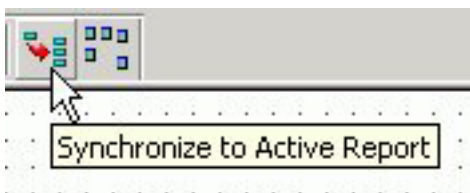
> The report or graph object is synchronized and refreshed with data from the Active Report.
>
> **Note:** There is no separate procedure associated with these Active Report objects. If you right-click on these items, there are no option to edit the procedure.

Show Synchronized Active Report Groups

Select **Show synchronized reports** from the Positioning toolbar.
The synchronized groups are shown in the layout.
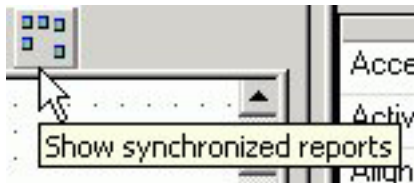
In the example below, there are two synchronized report groups. The **Production Units Analysis** graph is synchronized to the **Production Units Analysis Active Report** and the **Sales by Category** graph is synchronized to the **Sales by Category Active Report**.
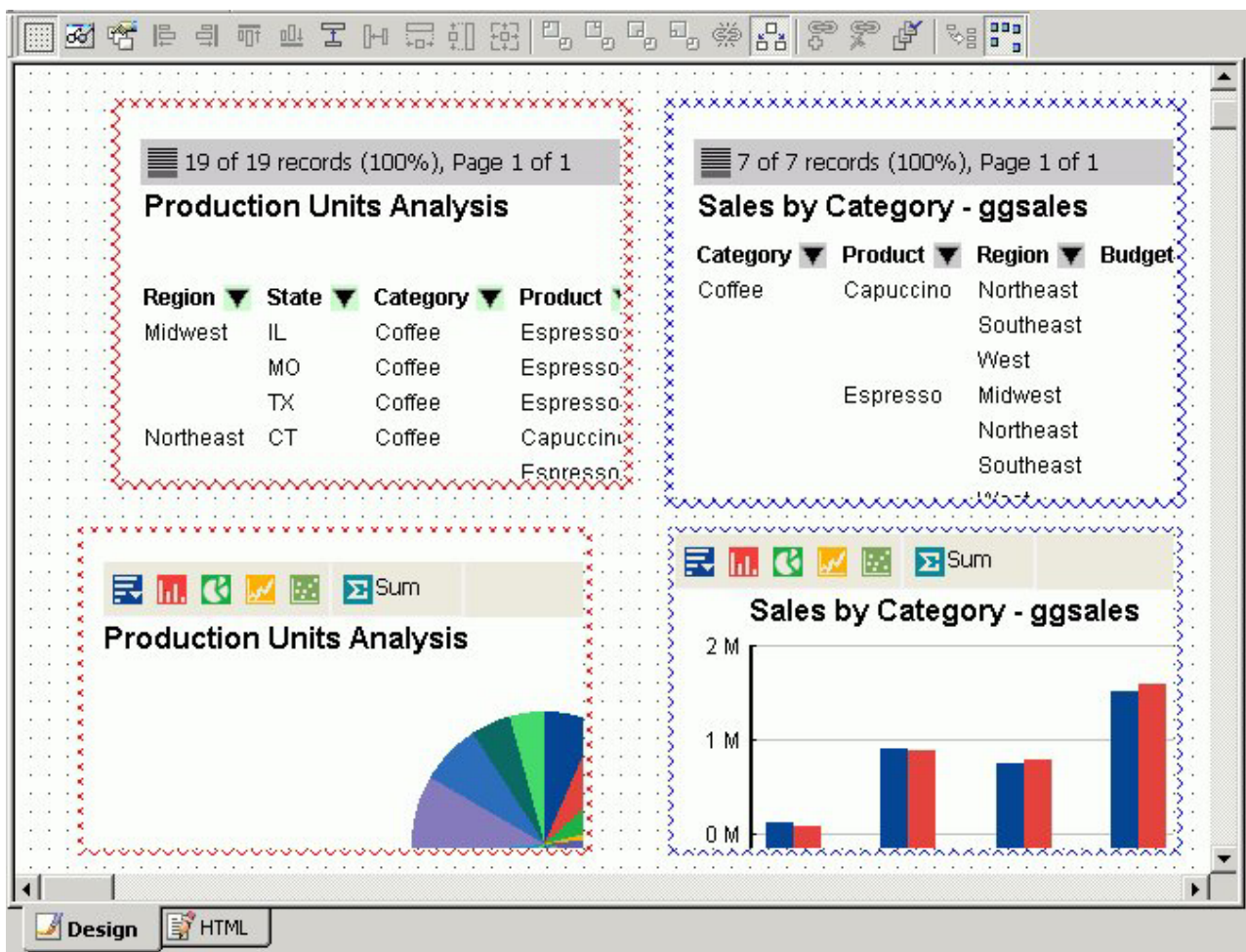
Select Properties for Synchronized Reports

You may change the Active Report object properties by using the Properties tab in the Properties window.

1. Select the synchronized Active Report object in the layout and click the **Properties** tab.

   The Active Report object properties appear.

2. Select the **Active report type** drop-down list to change the type of Active report for the object.

   The options are: **\<Not Set\>, Grid, Pivot, Bar, Line, Pie**.
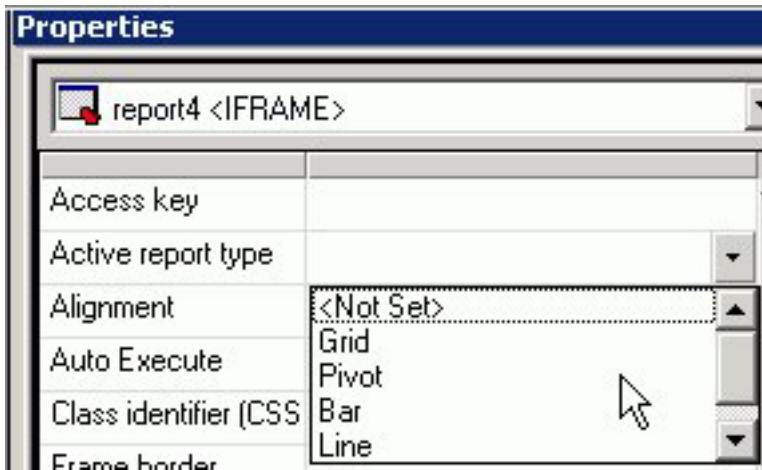
3. Optionally, If you select Bar, Line, or Pie as the Active Report type, additional X and Y axis selections are available for the synchronized report.
   * Select the **Active report X axis** drop-down list.
   * Select the **Active report Y axis** drop-down list.
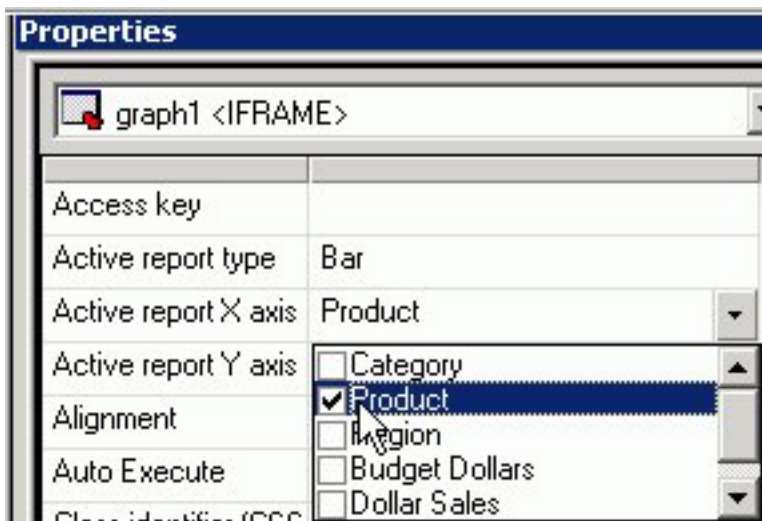
   Note: If you change the X and Y values for a graph object, the selections are not reflected in the Design View of the HTML Layout Painter. These values are applied at run-time.

Delete An Active Report Object

An Active Report object can only be populated if it is synchronized with an Active Report. To break the synchronization between a report or graph with an Active Report, delete the object.

1. Select the synchronized report or graph object in the layout.
2. Click **Delete** from Edit menu.

The Active Report object is deleted.

Configuring Active Report Controls

To add an Active Report Control, it is required that you insert a new control to the layout and configure it as an Active Report control on the Parameters tab. Selecting Active Report as the control type creates an association between the HTML Layout control and an Active Report, thereby linking actions to directly affect bound Active Reports.

Note: Active Report controls are only applicable if there are Active Reports embedded or referenced in the HTML Layout. Active Report Controls cannot be associated to any parameters in the layout. This type of control can only be associated with an Active Report in the Layout.

Add an Active Report Control to the Layout

Any HTML layout control can be configured as an Active Report control, but the following controls are the most applicable: **Check box, Drop down list, List box, Radio button**, and **Push** button.

Add a HTML Layout control that mimics an Active Report menu option:
1. Select the control type (**Drop Down List**, **List Box**, and so on), from the Insert menu.
   The cursor changes into a crosshair.
2. Click and drag the crosshair to create a the control object and adjust it to the size you want.

The HTML layout control is added as the Active Report control. You may now configure the control by using the Parameters tab.

Parameters Tab in Properties Window (Active Reports)

The following image is the Parameters tab in the Properties window with Active Report control values.
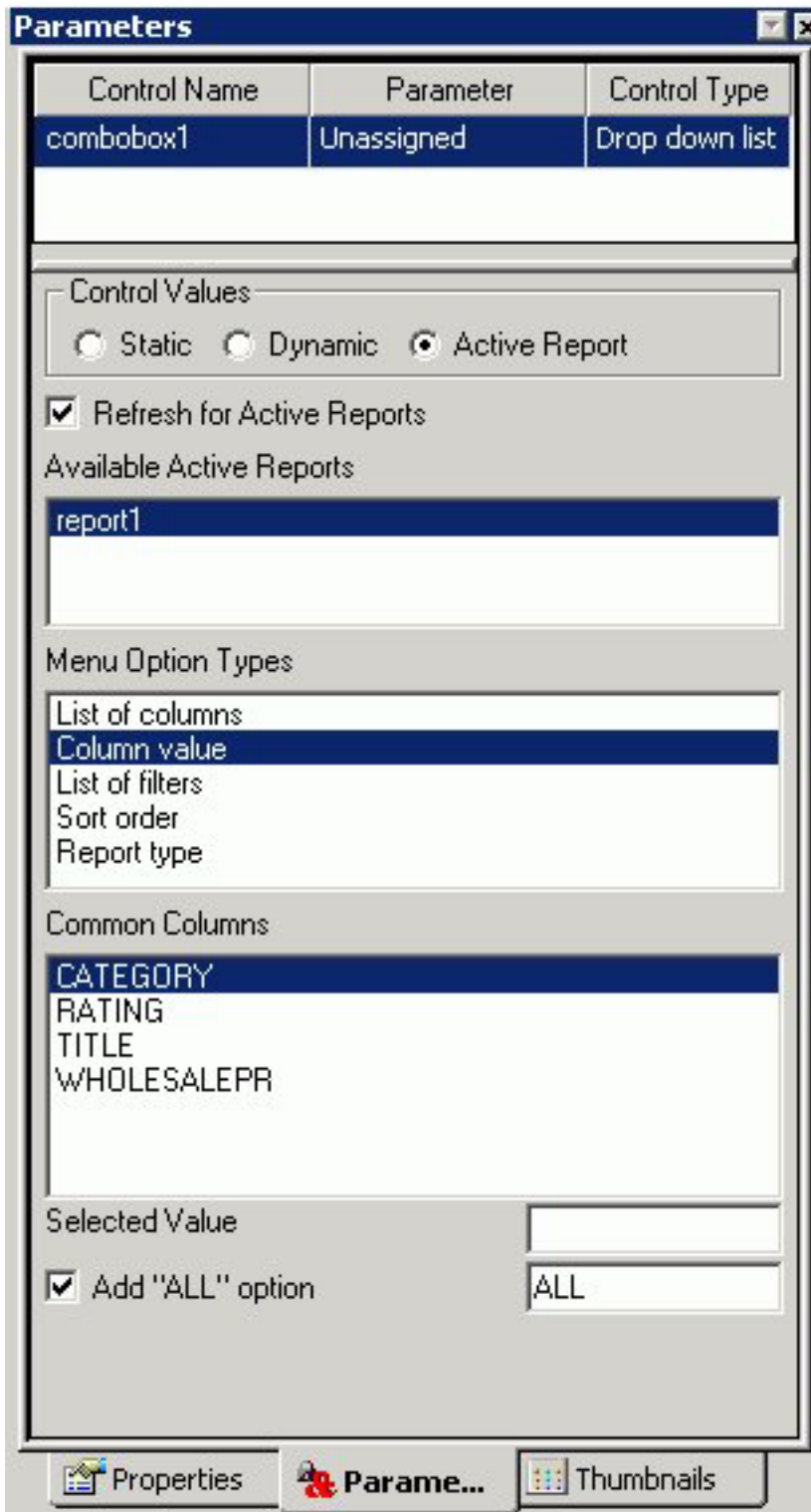
The Parameters tab contains the following fields and options when Active Report is selected in the Control Values section:

Control Name

Contains the name of the form control associated with each parameter.

Parameter

Contains the name of each parameter in the current procedure.

Unassigned - For an Active Report control, this should always be unassigned.

Control Type

Contains the type of form control associated with each parameter.

The following control types are recommended form controls when using Active Report controls: **Check box, Drop down list, List box, Radio button**.

Control values

Determines whether parameter values are obtained from a static, dynamic, or active report control list.

Active Report - Selecting the Active Report control will require you to bind the HTML Layout control to an Active Report.

Refresh for Active Reports

Enables Active Report controls to automatically modify current views of bound Active Reports when you select a new value at run-time.

Note: Refresh Active Reports is enabled by default when you select an Available Active Report from the Parameters window.

For more information about the Refresh for Active Reports option, see Refreshing Active Reports.

Available Active Reports

The Available Active Reports list binds Active Report controls to an Active Reports in the layout. Thereby, at run-time, when the Refresh Active Reports setting is triggered; the Active Reports selected in the Available Active Reports list are modified based on the current state of each Active Report control it is bound to.

Note: Available Active Reports lists all Active Reports currently embedded in the HTML Layout Painter. No Active Reports are selected by default.

Menu Option Types

The Menu Option Types determine how Active Reports are modified when the Refresh Active Reports setting is triggered. Menu Option Types configure Active Dashboard controls to sort columns, filter content and change the Active Report presentation. Multiple Active Report controls, each with different Menu Option Types settings, can be used in combination to modify the Active Dashboard.

Menu Option Types presents a list of options to designate which Active Report menu option an Active Report control inherits. To set the behavior of the current Active Report control, select only one Menu Option Types item.

Note: If no option is selected, the Active Report control will have no affect on the Active Dashboard.

List of columns

At run time, the Active Report control lists all common columns found in each bound Active Report. When the Refresh Active Reports setting is triggered, the data is sorted by the selected column.

Note: If using AS Names for a field in a report, all common columns must have the same name across all reports.

Column value

At run-time, the Active Report control lists all unique values found in a specified column, common in each bound Active Report. When the Refresh Active Reports setting is triggered, the Active Report control filters bound Active Reports based on the column value selected.

List of filters

At run-time, the Active Report control lists multiple filtering actions. This type of control does not affect bound Active Reports by itself, but only when used in conjunction with Active Report controls set to Column Value. When the Refresh Active Reports setting is triggered, the Active Report control instructs how to filter bound Active Reports based on the value selected in the Column Valued control.

The available list of filters are: **Equals, Not equal, Greater than, Greater than or equal to, Less than, Less than or equal to, Between, Contains, Contains (match case), Omits, Omits (match case)**.

Sort order

At run-time, the Active Report control lists two sorting actions; Sort Ascending and Sort Descending. When the Refresh Active Reports setting is triggered, the Active Report control sorts bound Active Reports based on the sorting action selected.

By default, the Active Report control sorts the first common column in each of the bound Active Reports.

Report type

At run-time, the Active Report control lists different Active Report presentation types; **Grid, Pie Chat, Line Chart, Bar Chart,** and **Pivot Table**. When the Refresh Active Reports setting is triggered, this Active Report control changes bound Active Reports to the presentation type selected.

Common Columns

The Common Columns list only appears when the Column value Menu Option Type is selected. Common Columns lists all common columns found in each bound Active Report. Select one field to bind to the Active Report control. At run-time, the Column value control lists all unique values across each bound Active Report's Common Columns field selected.

Note: The Common Column selection can be overridden at run-time when the Column Value control is chained to the List of columns Menu Option Type.

Selected Value

Enter the value(s) to be selected as the default value whenever the procedure is run. For more information, see Use Selected Values as the Default Value.

Note: The Selected Value option is only available for Active Report controls when the **Column Value** Menu Option Type is selected.

Add ″ALL″ Option

Adds the option to select ALL data source values to the control. Alternate text can be substituted for "ALL" using the text field to the right. For more information, see Parameter List Options.

Configure an Active Report Control

Once the Active Report control is added to the layout, you may configure the Active Dashboard control by using the Parameters tab of the Properties window.

1. Select the Active Report control in the layout and click the **Parameters** tab in the Properties window.
2. Select **Active Report** as the Control Value.

   The Active Report controls appear.
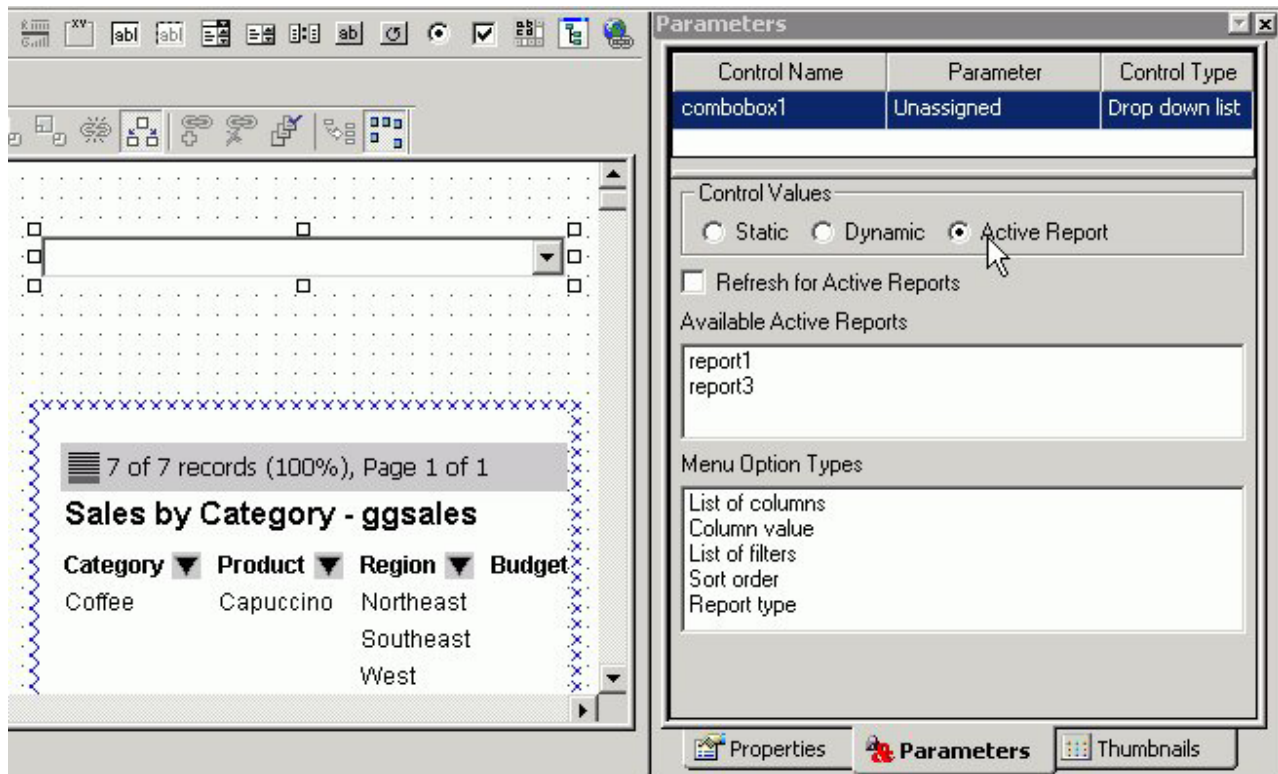
3. Bind the Active Report control to an Available Active Report in the layout:
   - Select one or more Active Reports from the list of **Available Active Reports**.
   - When an Active Report is selected, **Refresh for Active Reports** is enabled by default.

The Active Report(s) selected in the Available Active Reports list are modified based on the current state of each Active Report control it is bound to. Refresh for Active Reports refreshes current views of bound Active Reports when you select a new value at run-time.

For more information about the Refresh for Active Reports option, see Refreshing Active Reports.

4. Select the **Menu Options Types** for the Active Report control to sort, filter, list or select columns, and/or change presentations styles of the bound Active Report and the associated report and graph objects synchronized to the Active Report.
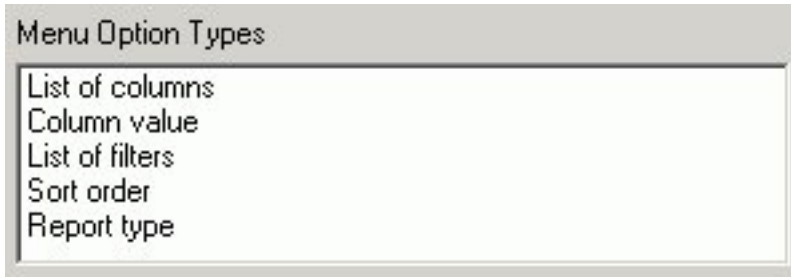
Note: If no option is selected, the Active Report control will have no affect on the Active Dashboard.

a. Select **List of columns** from Menu Option Types to list all common columns found in each bound Active Report. At run-time, the bound Active Report output is sorted by the selected column.

For example, the following Active Report control shows a list of all the columns in the bound Active Report.
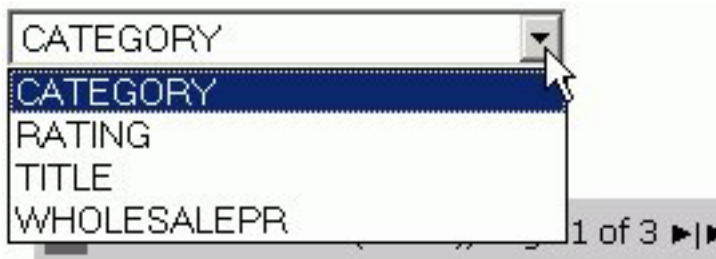
b. Select **Column value** from the Menu Option Types to list all unique values found in a specified column, common in each bound Active Report at run-time.

When Column Value is selected, the Active Report control panel dynamically presents Common Columns in each of the selected Active Reports in the Available Active Reports list.

Select one column from the **Common Columns** list.
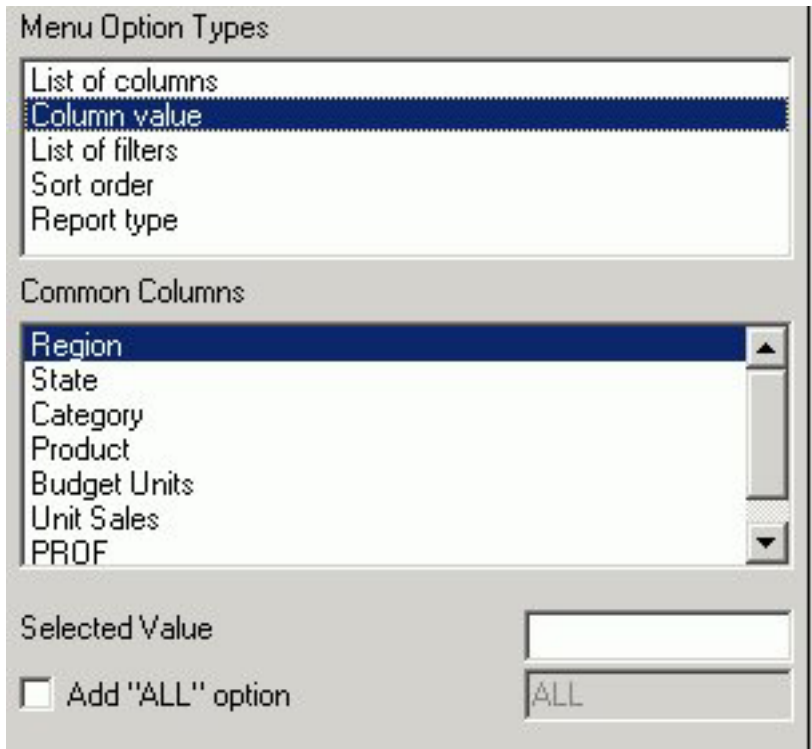
You may also use the **Selected Value** field to enter the value(s) to be selected as the default value whenever the procedure is run. The **Add ″All″ option** adds the option to select ALL data source values to the control at run-time.

For example, the following Active Report control shows **RATING** as the selected common column for the bound Active Report.
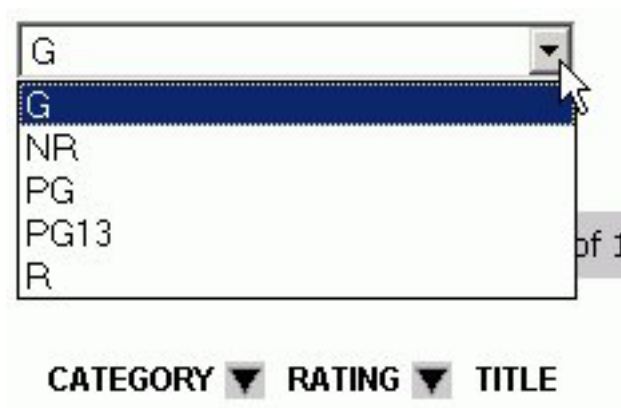
c. Select **List of filters** from the Menu Option Types to list multiple filtering actions at run-time.

Note: List of filters is used in conjunction with the Column value. Both controls should be bound to the same Active Reports. The Active Report control instructs how to filter bound Active Reports based on the value selected in the Column value control.

For example, the following Active Report control shows a list of filters in the bound Active Report.
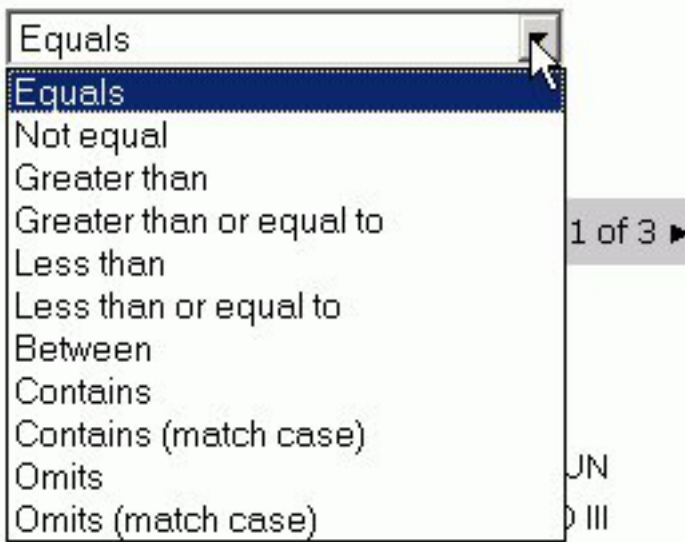
d. Select **Sort order** from the Menu Option Types to list sorting options (Ascending or Descending) at run-time. The Active Report control sorts bound Active Reports based on the sorting action selected.

Note: By default, the Active Report control sorts the first common column in each of the bound Active Reports.

For example, the following Active Report control shows a list sort order options in the bound Active Report.
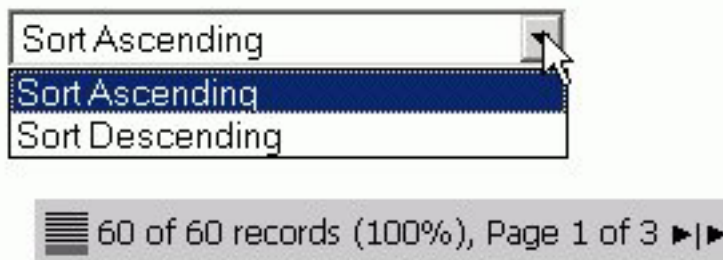
e. Select **Report types** from the Menu Option Types to list different Active Report presentation types at run-time. The Active Report control changes bound Active Reports to the presentation type selected.

For example, the following Active Report control shows report type options in the bound Active Report.
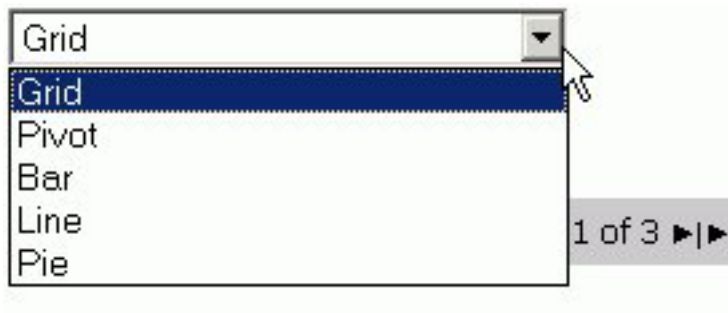
Refreshing Active Reports

The Refresh Active Reports setting enables Active Report controls to automatically modify current views of bound Active Reports when you select a new value. Each bound Active Report is modified not only by the new selection in the Active Report control with the Refresh Active Reports setting, but is based on the current state of all Active Report controls in the Active Dashboard. This action is triggered at run-time when you select a new value in an Active Report control with Refresh Active Reports set.

- Enabled (or checked) empowers the Active Report control, only after you make a new selection, to modify bound Active Reports.
- Disabled (or unchecked) prevents any bound Active Report from being modified when you select a new value in the Active Report control.

In order to update Active Reports when a value in the Active Report control changes, you must check the **Refresh Active Reports** check box on the Active Reports Control Panel.

Note: Refresh Active Reports is selected by default.

You may want to disable the Refresh option if there are multiple Active Report controls that require each control to be set before you refresh your output. If you are using multiple controls, you can associate the refresh option with a Push Button or Hyperlink, enabling you to refresh the output once all the controls are selected.

Refresh Active Reports in the Active Report Control Panel:

1. Required: From the Parameters tab of the Properties window, select **Active Report** as the Control Value.

   The Active Report controls appear.

2. Required: Select the **Refresh for Active Reports** check box.

   Note: When an Active Report is first selected, Refresh for Active Reports is enabled by default.

Refresh Active Reports With a Push Button or Hyperlink:

1. Insert a Push Button or Hyperlink to the layout:
   - Select **Push Button** from the Insert menu.
   - Select **Hyperlink** from Insert menu.

   The cursor changes into a crosshair.

2. Click and drag the crosshair to add the Push Button\Hyperlink object to the layout and adjust it to the size you want.
   - If inserting a Push Button, right-click the push button object and select **Create hyperlink** from the context menu.

The Hyperlink Properties dialog box opens.

- • If inserting a Hyperlink, the Hyperlink Properties dialog box opens.

3. Optionally, you may change the name of the Push Button\Hyperlink in the **Display Text** field.

4. From the Hyperlink Properties dialog box, select the **New** icon to add a request to execute.

5. Select **Refresh Active Reports** from the Action drop-down list.

6. Select the Source drop-down list to select which Active Reports should be refreshed.

   Note: The Source drop-down will only lists the embedded or referenced Active Reports in the layout.
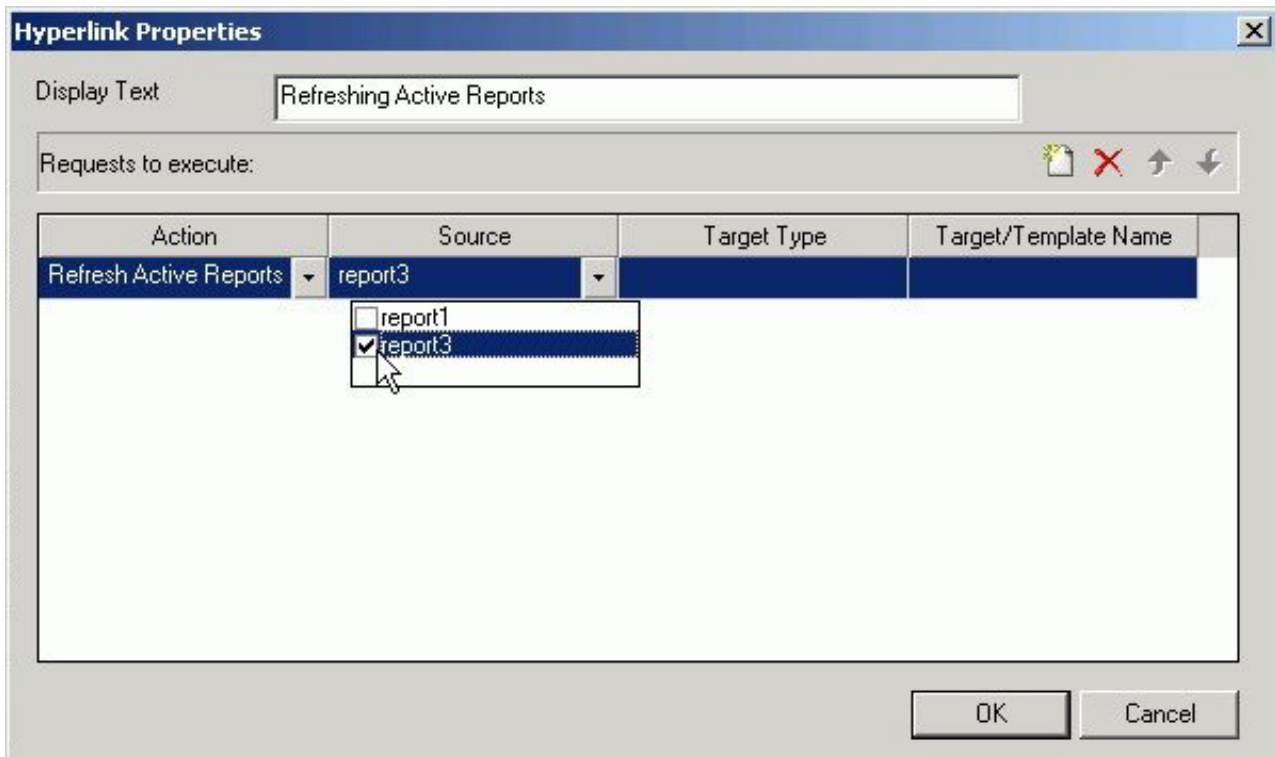
7. Click **OK** to close the Hyperlink Properties dialog box.

Exporting the Active Dashboard

A benefit of Active Reports is the ability to run these reports off-line. You may export an HTML Layout as a FOCUS procedure (.fex) to provide the ability to schedule and distribute Active Dashboards, so these too can be run off-line.

Export As Procedure converts the HTML page and embeds the code into a FOCUS procedure. The layout is no longer a Webpage or HTML file, but a FOCUS procedure with embedded HTML code. The procedure (.fex) is added to the Procedures folder (in your local projects) or in your Standard Reports group folder (Managed Reporting environment).

Once the HTML Layout is saved as a procedure, it cannot be converted back to an HTML Layout.

Note: The Export as procedure is recommended for saving Active Dashboards only, as many layout controls and report formats are not supported.

Use Export as Procedure

Once the HTML Layout is exported as a procedure, it cannot be converted back to an HTML Layout.

1. After creating an Active Dashboard in the HTML Layout Painter, select **Export as procedure** from the File menu.

   The New Procedure File dialog box opens.

2. Type a File name and click **Create**.

   Note: If the layout contains any parameter controls other than an Active Report control, the Export as procedure option is unavailable.

Conversions of Export as Procedure

Export As Procedure converts the HTML page and embeds the code into a FOCUS procedure. The layout is no longer a webpage or HTML file, but a focus procedure with embedded HTML code. This conversion requires conversion of each object on the HTML page.

The purpose of creating a procedure from an HTML Layout is to provide the ability to run these forms off-line. Because you may not be connected to an environment, the new procedure removes all referenced objects and adds them to the procedure; and layouts with controls that require multiple requests to the server will not be supported.

The following conversions occur when exporting as a procedure:

- Referenced procedures get read and written/embedded into the saved procedure.
- Report and Graph objects - Display format is converted to HTML format. For example, If the layout has embedded/referenced Excel, PowerPoint, or PDF documents, these will all be converted to HTML formatted output.

  Note: Active Reports (AHTML) format is preserved.

- Images - The reference to the image file is removed, and the image is embedded in the procedure as inline HTML code.

  Note: This code can become very long and it is recommended that only small images be used in layouts that will be exported as procedures.

Limitations of Export As Procedure

Note the following limitations when applying the Export As Procedure in the HTML Layout Painter:

- If using Static and Dynamic Parameter controls.

  The Export as Procedure menu item is disabled for any Layouts with static or dynamic parameter controls. Any scheduled report supports only one request to the server at run-time. These types of controls require multiple requests to the server, and cannot be supported off-line.

- If exporting a procedure with background images.

  Background images will not be embedded into the procedure due to size. The procedure will keep the reference. In off-line mode, if this reference cannot be resolved, no image will be displayed.

- If exporting a procedure and using CSS stylesheets.

The referenced stylesheets will not be embedded into the procedure and the procedure will keep the reference. In offline mode, if this reference cannot be resolved, no style from the CSS file will be applied.

Scheduling and Distributing Active Dashboards

If an HTML Layout has been exported as a procedure (.fex), it is now available to be scheduled with Report Broker.

For information about scheduling, see "Adding Report Broker Schedule Capability to the HTML Layout Painter" on page 597.

Controlling the HTML Layout Painter Environment

You can set properties for the HTML Layout Painter with the Settings dialog box. This dialog box allows you to set page properties such as the location of reports and graphs, as well as the display of a grid in your layouts. The changes that are made in the Settings dialog box are saved to the registry. This allows a customized environment to be created for various users.

Set Page Properties

1. Select **Settings** from the Tools menu. The Settings dialog box opens. For details, see Settings Dialog Box.
2. Make your changes, and click **OK**.

Select Form Settings From the Settings Dialog Box

1. From the HTML Layout Painter, select **Settings** from the Tools menu.
   The Settings dialog box opens.
2. Click the **Form's Settings** button.
   The Form Setting dialog box opens.
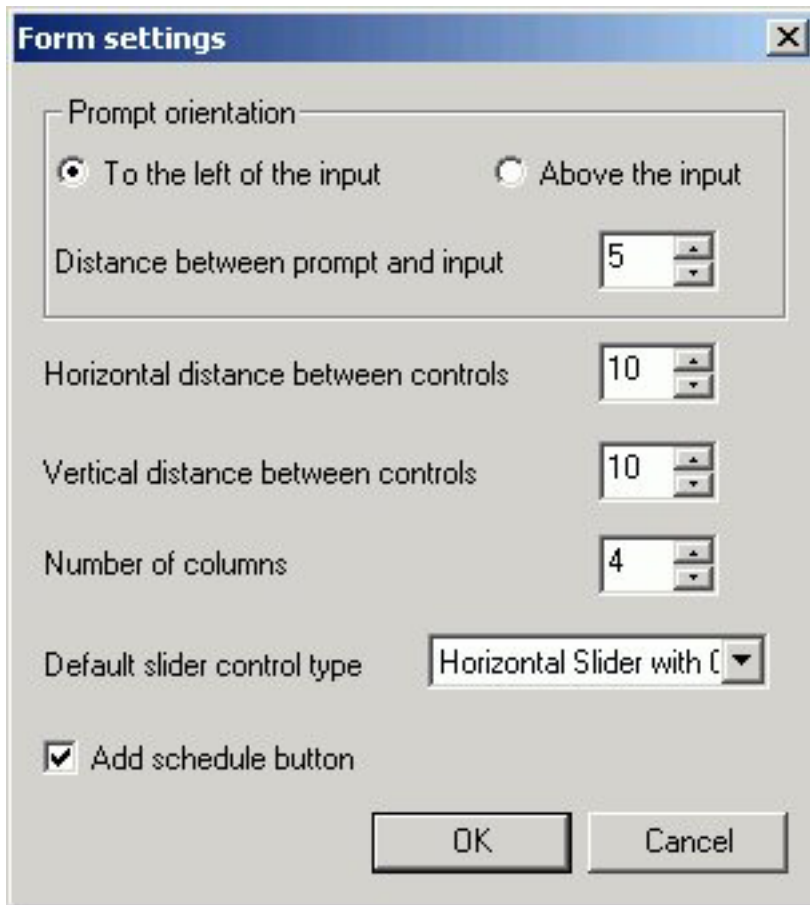
3. Select the orientation for the form object in the layout.

4. Use the spin buttons to increase or decrease the distance between the prompt, horizontal and vertical controls of the form, and the number of columns for the form.

   or

   Position the cursor in the input box and type a number.

5. Use the slider control type options to select how the slider bar appears.

   Note: Slider bars are available when using numeric range value parameters in your report or graph.

6. The Add schedule button option adds a schedule button for referenced procedures with parameters in the Managed Reporting environment. For more information, see "Adding Report Broker Schedule Capability to the HTML Layout Painter" on page 597.

7. Click **OK** to close the Form settings dialog box.

   The form settings are applied and shown in the HTML Layout Painter Design view.

8. To reset the default form settings, right-click the form object and select **AutoArrange** from the context menu.
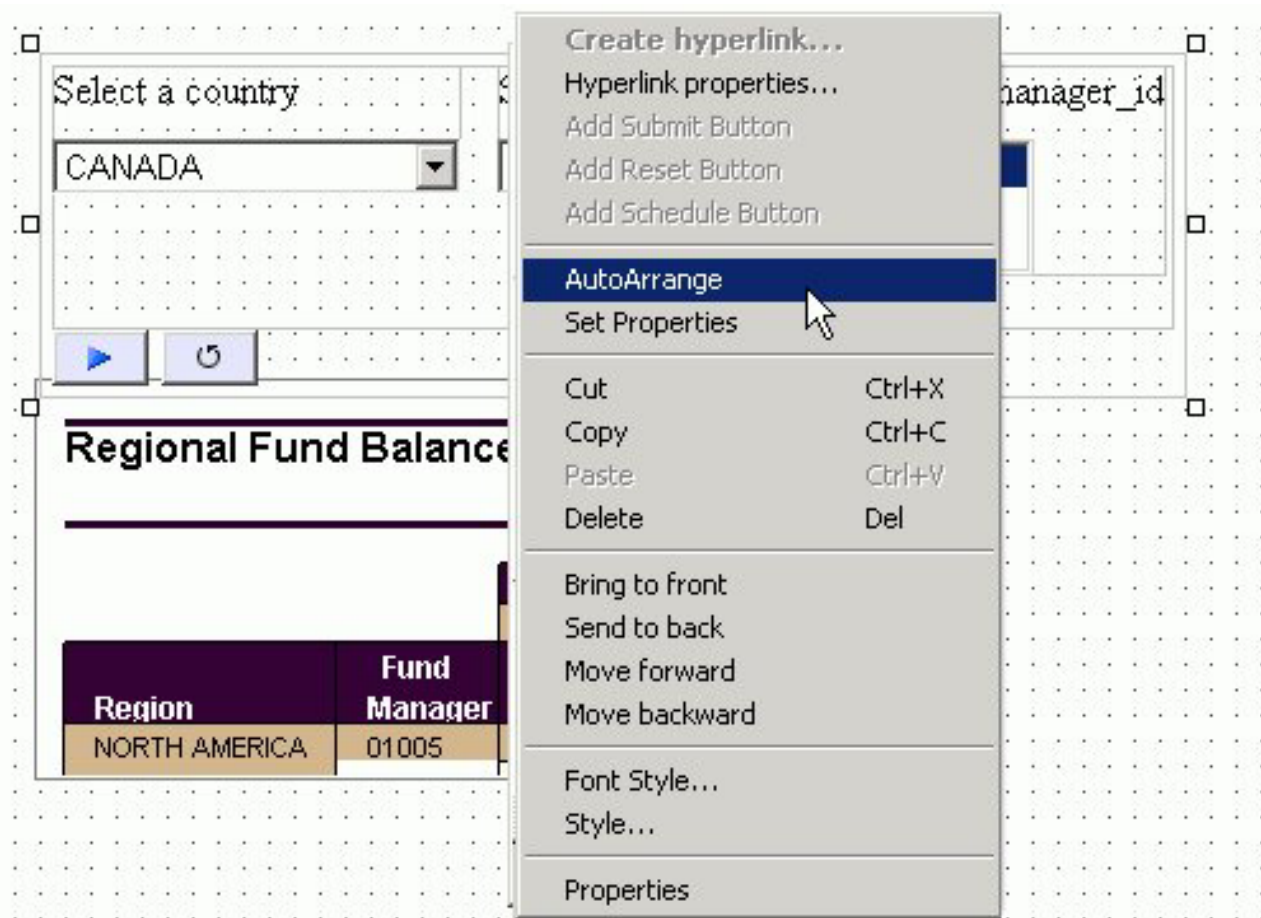
The form is auto arranged on the layout, using the default form settings.

Settings Dialog Box

The following options are available from the Settings dialog box of the HTML Layout Painter.
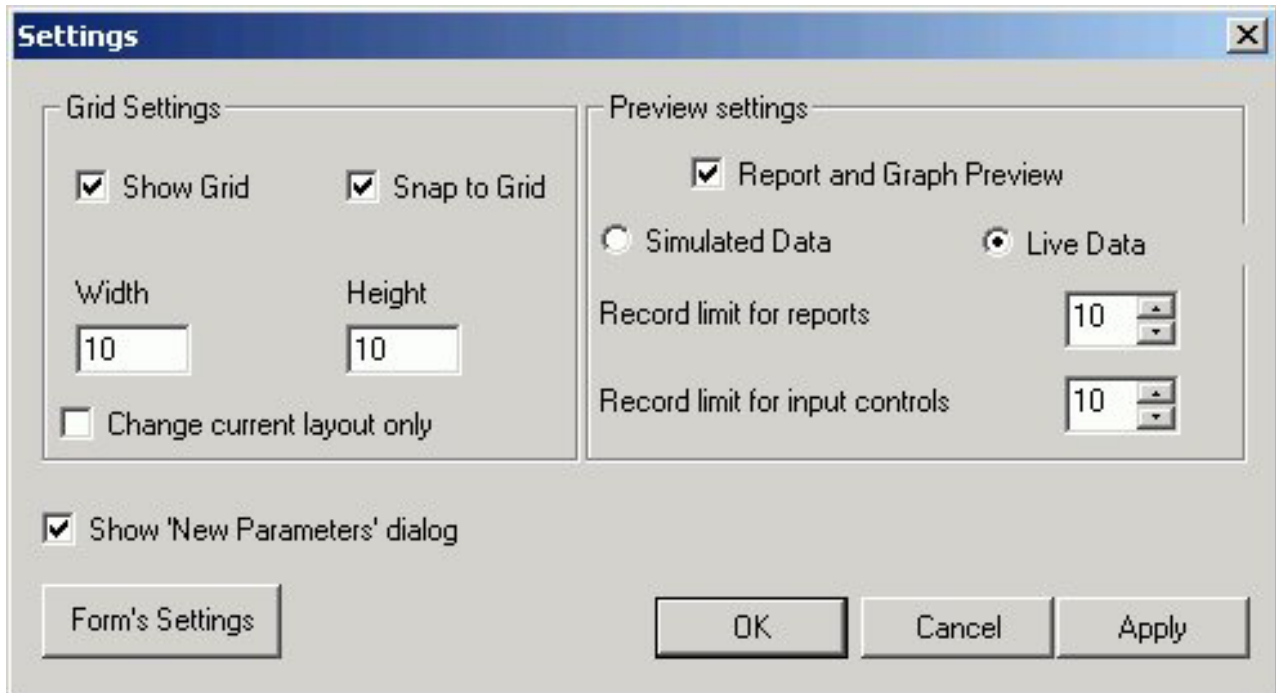
Grid Settings

Show Grid

> Displays a grid. If this is not selected, the grid is turned off for all layouts.

Snap to Grid

> Causes objects in the layout to snap to grid lines when being positioned. For detailed positioning, deselect this option.

Width

> The width of the grid in pixels.

Height

> The height of the grid in pixels.

Change current layout only

> Select this check box to apply the grid settings to the current layout.

Preview Settings

Report and Graph Preview

> Previews report and graph data in the Design View. If checked, additional options are available (Simulated Data or Live Data).

> **Note:** This option is selected by default.

> If unchecked, icons are used to represent the area in the Design View for reports and graphs. This is the fastest method of loading reports since no request are made to the server.

Simulated Data

> Selecting to preview simulated data sends a request to the reporting server that gathers formatting information from the Master file. The database is not accessed, but rather mock data is used to visually represent the report.

Note: This option is only available when previewing reports and graphs.

Live Data

Selecting to preview live data sends a request to the reporting server, and to the database, to get a snapshot of the actual data in the report.

Note: This option is only available when previewing reports and graphs.

Record limit for reports

Enables you to limit the number of records used to gather data for previewing the report in live data mode. For example, if you set 500 as the record limit for the report, then 500 rows of data are gathered for the report results.

Note: This option is only available if the Live Data option is selected.

Record limit for input controls

Sets the number or records to be shown for the input controls when gathering data for reports and graphs.

Note: This option is only available if the Live Data option is selected.

Show New Parameters Dialog

Controls whether or not the New Parameters dialog box shows when adding parameters in the HTML Layout Painter.

Note: The New Parameters dialog box is shown by default.

Form's Settings button

Options are available so that you may customize how the forms are populated and arranged in the document.

Form's Settings Dialog Box

The following options are available from the Form's Settings dialog box of the HTML Layout Painter.

Prompt orientation

Prompt refers to the descriptive text used to label the control/parameter in the output.

To the left of the input

Places the descriptive text used to label the control/parameter to the left of the input.

Above the input

Places the descriptive text used to label the control/parameter above the input.

Distance between the prompt and input

Sets the distance between the prompt and input.

Horizontal distance between controls

Sets the horizontal distance between controls.

Vertical distance between controls

Sets the vertical distance between controls.

Number of columns

Determines when the form will wrap and start laying controls out in a new row.

Default slider control type

Sets how the slider control appears. Select from the following types of slider bars:

Note: Slider controls are available when using numeric range value parameters in your report or graph.

- Horizontal or Vertical Slider Simple bar. The slider bar has no end arrows, just a bar with the slider.
- Horizontal or Vertical Slider with Color Bar and Arrows. Arrows appear at each end of the slider bar.
- Horizontal or Vertical Slider with Color Bar, Arrows, and Edit. Arrows and an edit box appear at the end of the slider bar, showing the current value.

Note: Once a slider control is inserted into the HTML Layout, changing these options will not affect the existing sliders, the option is only applied to new slider controls. To change the existing slider bar type, delete the slider object and insert a new slider control.

Add Schedule Button

Automatically adds a schedule button for referenced procedures with parameters. This option is selected by default. For more information about adding schedule capability to HTML Layout forms, see "Adding Report Broker Schedule Capability to the HTML Layout Painter."

Adding Report Broker Schedule Capability to the HTML Layout Painter

You may create a form control that adds a schedule button in the HTML Layout Painter. A schedule button enables you to schedule a report, or graph, using Report Broker. Once you have created the schedule, you can access the Report Broker HTML User Interface to edit and maintain information about the schedule.

Note: The Schedule option is only available from the HTML Layout Painter in the Managed Reporting environment if you are licensed to use Report Broker. Managed Reporting provides access to Report Broker, where you can schedule and distribute Standard Reports.

Only procedures referenced in the HTML Layout Painter are able to be scheduled.

When referencing an existing procedure with parameters in the HTML Layout Painter, the schedule button is automatically added. You may also manually create schedule buttons for your report or graph.

Add or Disable the Schedule Button for Form Controls

Form controls appear when you reference a report or graph with parameters. The HTML Layout Painter automatically adds a form control, Run button, Reset button, and Schedule button for the parameters in your layout; if your environment is licensed to use Report Broker.

The Form Settings dialog box sets when the schedule button appears with the Form controls.

Note: The Schedule button is selected by default.

To change the default behavior:
1. Select **Settings** from the Tools menu.

   The Settings dialog box opens.
2. Click the **Form Settings** button.

   The Form settings dialog box opens.
3. Deselect the **Add schedule button**.

Note: If you deselect this option, you have the ability to manually add a schedule button, for more information, see Manually Add a Schedule Button.

Add a Schedule Button to a Referenced Existing Procedure With Parameters

This procedure provides instructions on how to automatically add a schedule button to a HTML page for a referenced procedure in the layout.

Note: Access the Standard Reports folder of your Managed Reporting environment to begin this procedure.
1. From the HTML Layout Painter, select **New Report**, or **New Graph**, from the Insert menu.

   The cursor changes into a crosshair.
2. Click and drag the crosshair to create a report (or graph) object and adjust it to the size you want.
3. Right-click the object and select **Reference existing procedure** from the context menu.

   The Get source file dialog box opens.
4. Select an existing report or graph with parameters and click **Open**.

   The New Parameters dialog box appears.
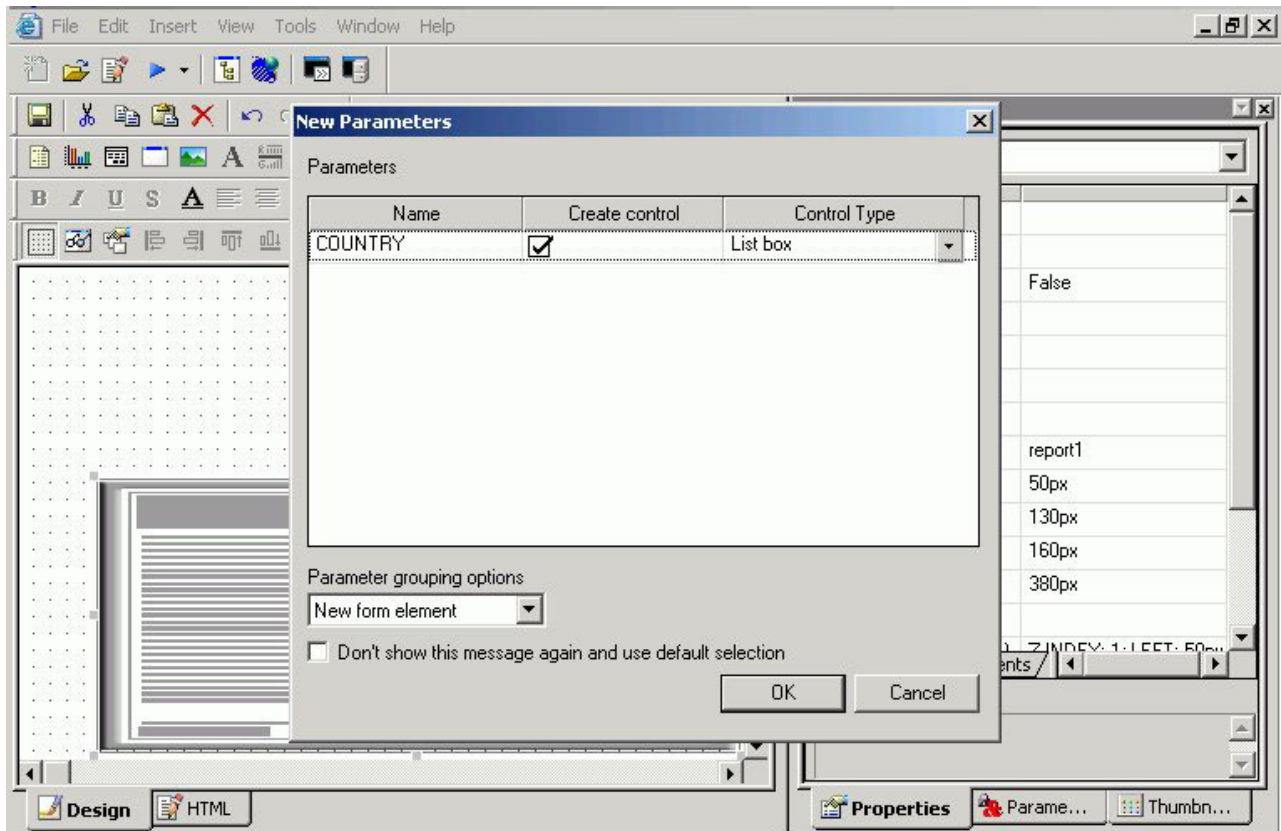
5. Click **OK** to add the new parameters and the form controls to the HTML page. The run, reset, and schedule buttons appear with the parameters on the HTML page.
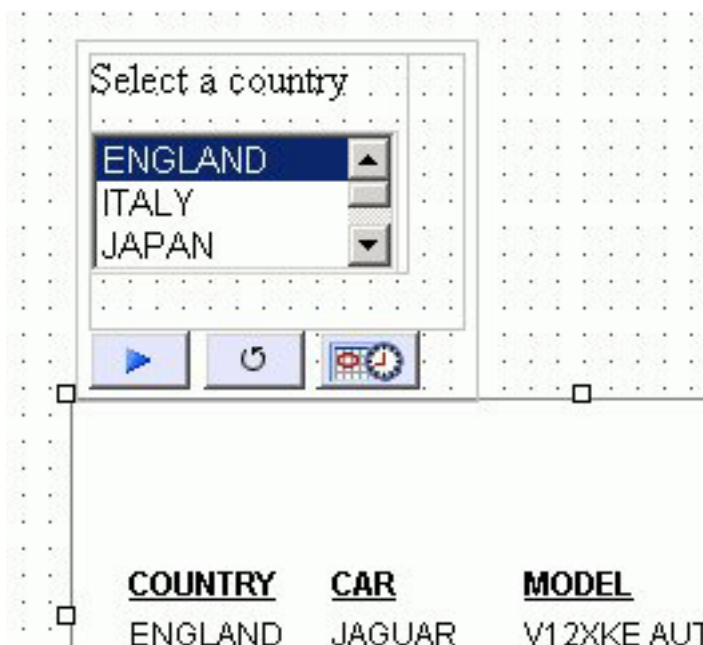
6. Optionally, select the schedule button on the HTML page to view or edit the properties. The properties appear in the Properties window of the HTML Layout Painter.
7. Double-click in the **Title** properties value field to set the title of the schedule page window.

   The default title for the schedule page window is *domain\standard reports\folder\foldername*. You may change this to the text of your choice.
8. Optionally, you may launch the Hyperlink properties for the schedule button to change the template for the schedule page.
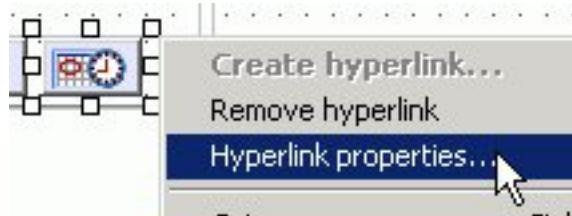
For more information about changing the template for the schedule page, see Change the Default Template for the Schedule Page.

Manually Add a Schedule Button

This procedure provides instructions on how to manually add a schedule button to a HTML page. This is useful for adding new, or additional, schedule buttons to a report or graph that has already been referenced to the HTML page, or if you are referencing a report or graph without parameters.

Note: Access the Standard Reports folder of your Managed Reporting environment to begin this procedure.

1. From the HTML Layout Painter, select a form control from the Insert menu. You may insert a push button, image, or hyperlink as a schedule button.
   a. If inserting a push button as a schedule button:
      - Select **Push Button** from the Insert menu.

        The cursor changes into a crosshair.
      - Click and drag the crosshair to add the push button object and adjust it to the size you want.
      - Right-click the push button object from the HTML page and select **Create hyperlink** from the context menu.

        The Hyperlink Properties dialog box opens.
      - You may change the name of the push button in the **Display Text** field.
   b. If inserting an image as a schedule button:
      - Select **Image** from the Insert menu.

        The cursor changes into a crosshair.
      - Click and drag the crosshair to add the image object.

        The Get source file dialog box opens.
      - Select an image and click **Open**.

        The image is added to the HTML page. You may adjust the image object to the size you want.

- Right-click the image object from the HTML page and select **Create hyperlink** from the context menu.

  The Hyperlink Properties dialog box opens.

c. If inserting a hyperlink as the schedule button:

- Select **Hyperlink** from the Insert menu.

  The cursor changes into a crosshair.

- Click and drag the crosshair to create a hyperlink object and adjust it to the size you want.

  The Hyperlink Properties dialog box opens.

- You may change the name of the hyperlink in the **Display Text** field.

2. From the Hyperlink Properties dialog box, select the **New** icon to add a request to execute.
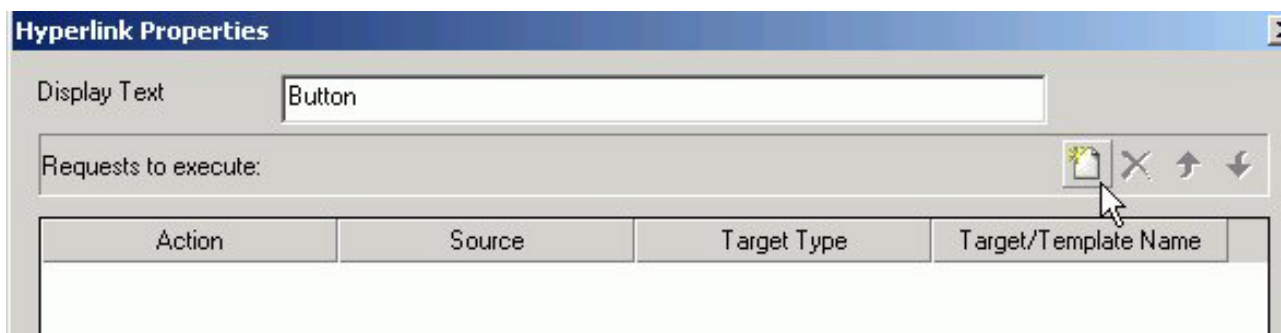
Note: The Display as Text field on the Hyperlink Properties dialog box is only available when inserting a push button or hyperlink as a schedule button.

3. Select the Action, Source, and Target/Template Name:

- Select **Schedule Report** from the Action drop-down list.
- Select the **Source** drop-down list to select the source of your Schedule Report.

  The Source drop-down will only lists the referenced procedures in the layout.

- The Target Type option is disabled and defaults to Report Broker when the Schedule Report action is selected.
- The Target/Template Name defaults to Email Library FTP when the Schedule Report action is selected.

  For more information on changing the template for the schedule page, see Change the Default Template for the Schedule Page.
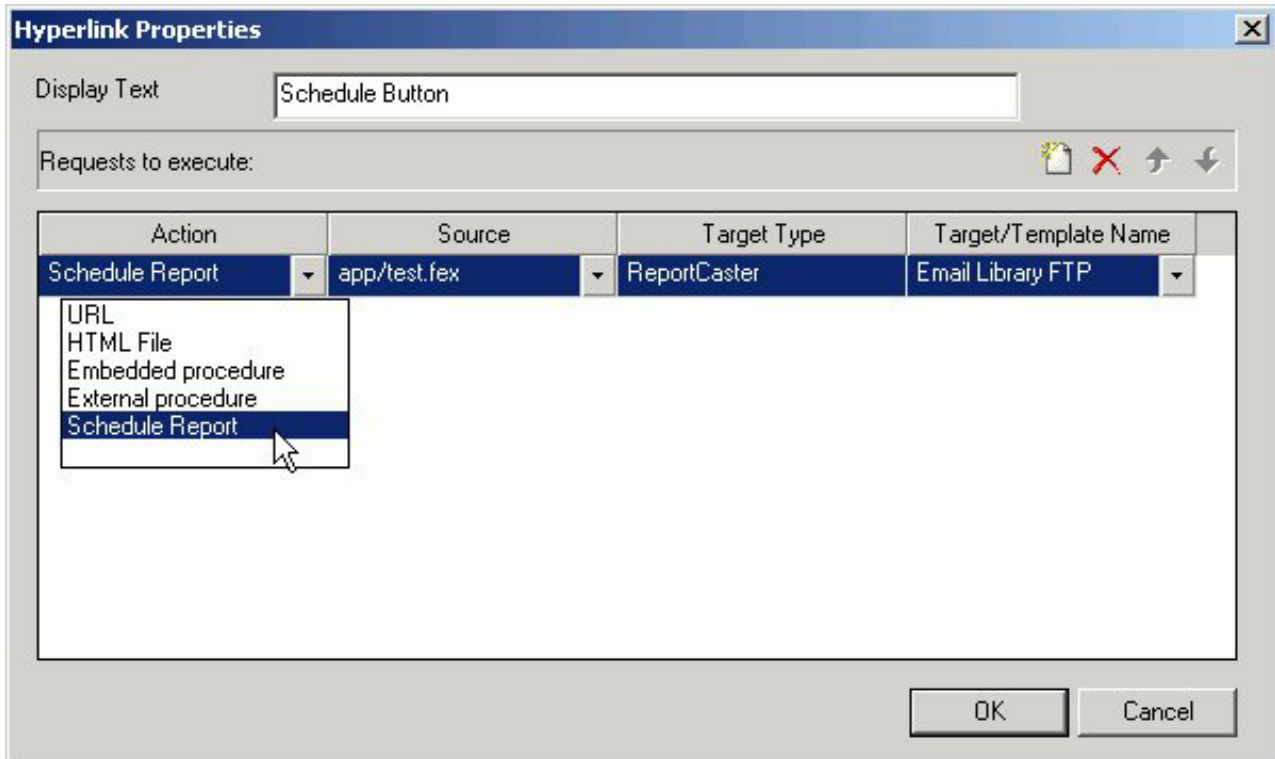
4. Click **OK** to close the Hyperlink Properties dialog box.

   The schedule button, image, or hyperlink is added to the HTML page.

5. Optionally, select the schedule object to view or edit the properties. The properties appear in the Properties window of the HTML Layout Painter.

6. Double-click in the **Title** properties value field to set the title of the schedule page window.

   The default title for the schedule page window is *domain\standard reports\folder\foldername*. You may change this to the text of your choice.

Change the Default Template for the Schedule Page

This procedure provides instructions on how to change the template for the schedule page by using the Hyperlink Properties dialog box. This is the template that appears when the schedule page is run from the report output.

1. Right-click a schedule button, image, or hyperlink from the HTML page and select **Hyperlink properties** from the context menu

   The Hyperlink Properties dialog box opens.

2. Select the Target/Template Name drop-down list to change the default template for the schedule. The options are:

   • Email Library FTP - This is the default template.

   • All

   • Email

   • Library
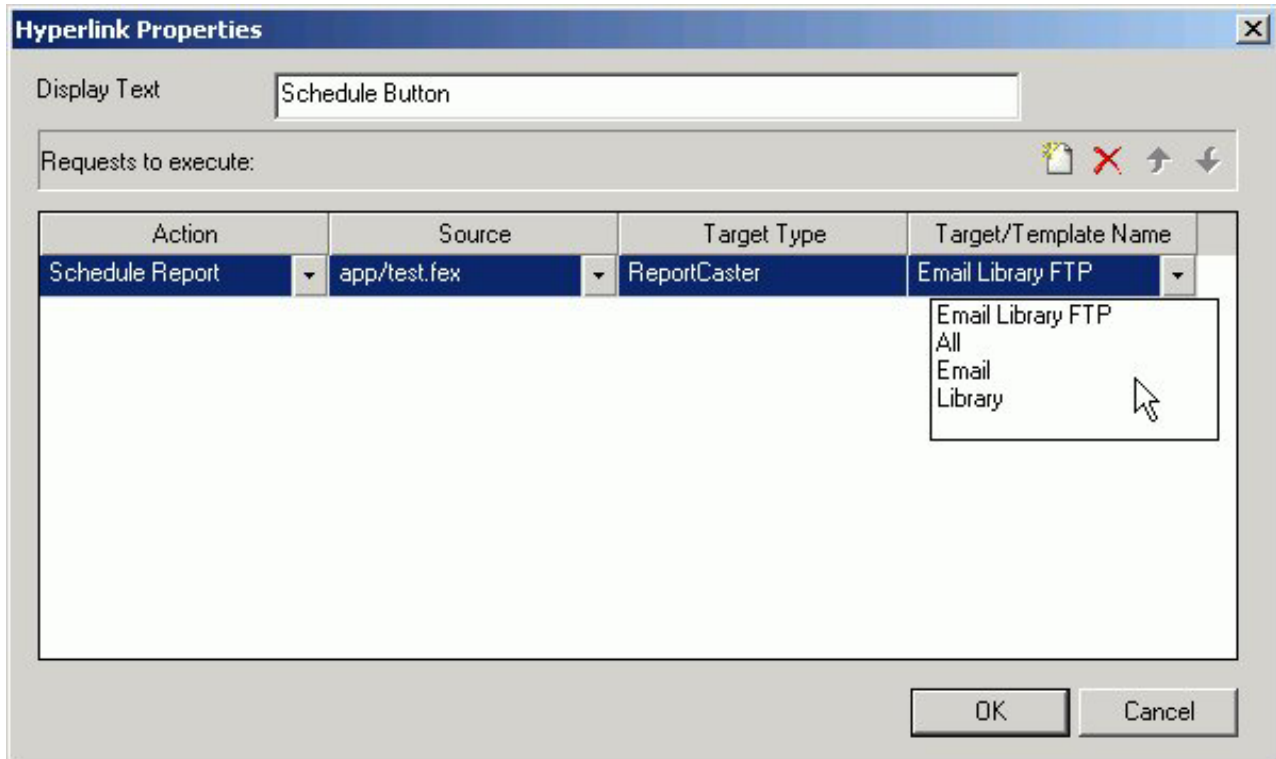
> **Note:** The template names represent the type of distribution option for the schedule in Report Broker.

3. Click **OK** to close the Hyperlink Properties dialog box.

   The selected template appears when the schedule page is run from your report output.

Report Broker Schedule Page

When you run your report, select the parameters (if applicable), and click the schedule button, the Report Broker schedule page appears.

Tip: To change the schedule template that appears at run time, use the Hyperlink Properties dialog box to change the Target/Template Name.

The report and the selected parameters are scheduled to Report Broker. From Report Broker, the values entered or selected are stored with the schedule information.

Note: You must have the Schedule user privilege to submit the schedule.

Chapter 13. Describing and Accessing Data: Overview

Introduces data source descriptions and explains how to use them.

Developer Workbench provides a set of graphical tools that you can use to describe and access many types of data sources, including:

- Relational, such as DB2.
- Multi-dimensional, such as SAP BW and Essbase.
- Procedures, such as Web Services and CICS® Transactions.
- XML, such as XML and Tamino.
- Hierarchical, such as IMS™ and FOCUS.
- Sequential, both fixed-format and free-format.
- Indexed, such as ISAM and VSAM.
- Network, such as CA-IDMS.
- Other popular data sources, such as Lotus Notes® and LDAP.

These graphical tools are designed to:

- Translate the schema of the data source into metadata that DB2 Web Query can read and report against.
- Optionally, enable you to customize and enhance the generated metadata, without having to know the subtleties of the underlying DB2 Web Query data description language.

A Note About Data Source Terminology

Different types of data sources make use of similar concepts, but refer to them differently. For example, the smallest meaningful element of data is called a *field* by many hierarchical database management systems and indexed data access methods, but called a *column* by relational database management systems.

There are other cases in which a common concept is identified by a number of different terms. For simplicity, we have standardized on a single set of terms. For example, we usually refer to the smallest meaningful element of data as a *field*, regardless of the type of data source. However, when required for clarity, we use the term specific to a given data source. Each time we introduce a new standard term, we define it and compare it to equivalent terms used with different types of data sources.

How Applications Interpret Data

When your application accesses a data source, it needs to know how to interpret the data that it finds. To accomplish this it reads a synonym, which is a term for the generated *metadata* associated with the particular data source.

Your application needs to know about:

- The overall structure of the data. For example, is the data relational, hierarchical, multi-dimensional, sequential? Depending upon the structure, how is it arranged or indexed?

- The specific data elements. For example, what fields are stored in the data source, and what is the data type of each field—character, date, integer, or some other type?

The synonym provides an alias for the data source that tells the server how tables are described and where to find them.

The primary component of a synonym is a Master File. The Master File describes the structure of a data source and its fields. For example, it includes information such as field names and data types.

For some types of data sources, an Access File supplements the Master File. An Access File includes additional information that completes the description of the data source. For example, it includes the full data source name and location. The nature of the information in the Access File is specific to each data source. You need one Master File—and, for some types of data sources, one Access File—to describe a data source.

How to Obtain a Synonym

You can generate synonyms using the Metadata menu option in Web Query, which enables you to explore DBMS catalogs and select the objects for which you wish to create synonyms. The tool prompts for all the information it needs to create a synonym for a particular data source and stores the generated synonym on the server.

There are some prerequisites: you must be authorized to use the data against which you plan to report and you must have configured an adapter to access that type of data. In fact, when you begin to create a synonym, Developer Workbench opens the adapter configuration window. The option to create the synonym becomes available only after the adapter is successfully configured.

Adapters are available for many data sources. Every adapter is specifically designed for the data source that it accesses, and, as a result, is able to translate between SQL or DB2 Web Query and the data management language (DML) of the data source. Adapters provide solutions to product variations, including product differences in syntax, functionality, schema, data types, catalogs, data representations, message processing, and answer set retrieval. It is the adapter that manages the synonym creation process. For related information, see How an Adapter Works.

How an Adapter Works

The adapter manages the communication between the data interface and the data source, passing data management requests to the data source and returning either answer sets or messages to the requestor. To perform these functions, the adapter:
- Translates the request to the applicable DML.
- Attaches to the targeted data source, using standard attachment calls. The adapter then passes the request to the data source.
- The data source processes the request.
- The results or error conditions are returned to the client application for further processing.

What You Can Do With a Synonym

Once you have generated a synonym, you can report against it using all of Web Query's reporting tools. In many instances, the configured adapter and the generated synonym are all you need to access your data and create reports and graphs.

However, you may wish to enhance the synonym in order to implement particular capabilities that are supported in the DB2 Web Query data description language. To do this, you can use a second data description tool, the **Synonym Editor**.

When you use the Synonym Editor, there is no need to know the data description language since the graphical tool displays all viewable and editable attributes of the synonym components. If you make changes to the generated synonym, the Synonym Editor validates your entries and displays messages if they violate the underlying syntax of the data description language.

Ways to Enhance a Synonym

Here are just a few of the attributes you might want to add to the synonym to enhance your data access and reporting capabilities. You can:

- Create a cluster Join view by linking available synonyms to create a multi-segment (multi-table) file for reporting.
- Add dimensions for OLAP analysis.
- Add virtual columns (DEFINEs) and columns for aggregated values (COMPUTEs).
- Add filters to specify data selection criteria.
- Add group definitions for data sources that support groups.
- Add meaningful titles and descriptions, including Multilanguage variations.
- Change the format of fields (for example, the size of an alpha field or the format of a date field).
- Create business views of the metadata in order to limit the fields available to any retrieval request that references the business view and to group fields together based on their roles in an application.
- Define parent/child hierarchies for cube data sources such as SAP BW and Essbase.

The Synonym Editor is fully described in Using the Synonym Editor.

How an Application Uses a Synonym

Synonyms are stored apart from the associated data source. Your application uses a synonym to interpret the data source in the following way:

1. It identifies, locates, and reads the Master File for the data source named in a request.
2. It locates and reads the Access File for the data source named in the request, if that type of data source requires an Access File.
3. It locates and reads the data source(s).

The data source contents are interpreted based on the information in the synonym.

Chapter 14. Using the Synonym Editor

Describes the Synonym Editor and how you may use it to view and edit synonyms.

This chapter describes the Synonym Editor and how you may use it to view and edit synonyms.

Note: The Synonym Editor is accessible only through the DB2 Web Query Developer Workbench.

Accessing the Synonym Editor

The following image is an example of how to access the Synonym Editor from DeveloperWorkbench.
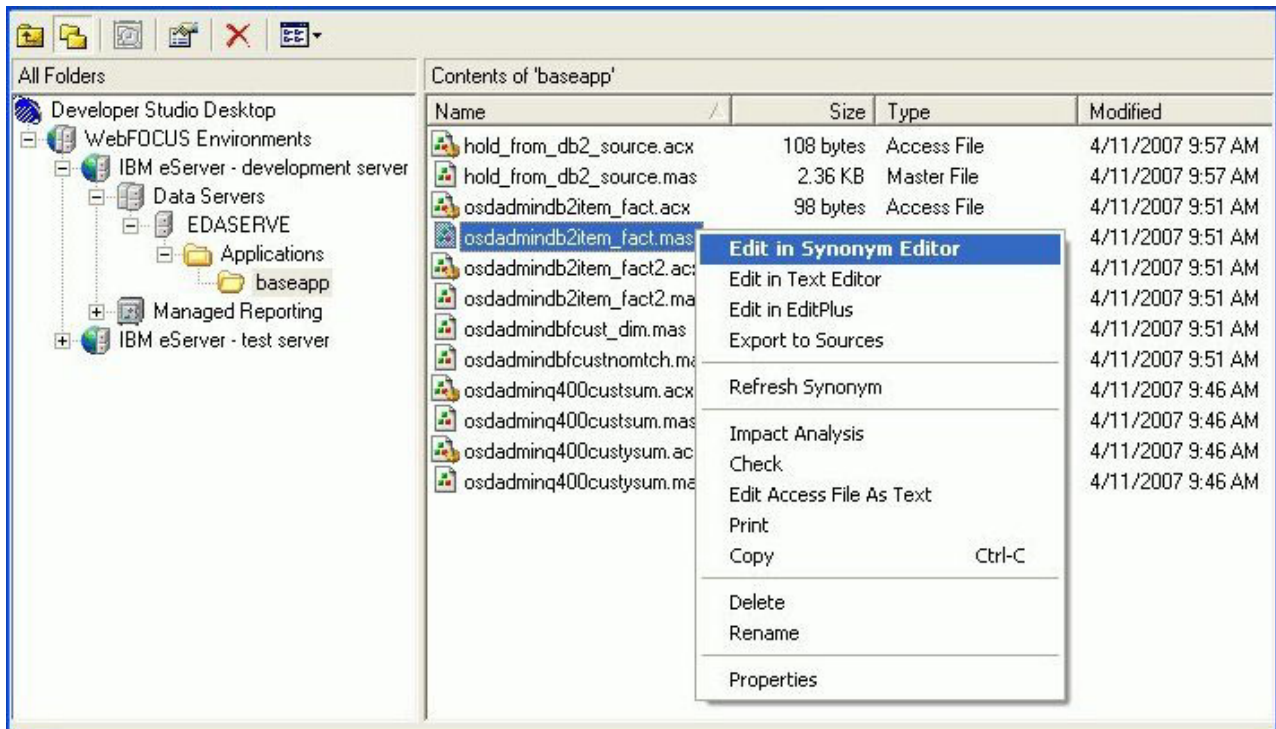
A synonym consists of a set of attributes that describe the data source. It provides the metadata for a data source, which enables an adapter to access and interpret the corresponding data.

Synonym Editor Layout

The Synonym Editor occupies the Developer StudioWorkbench workspace and displays Tree View, Modeling View, Synonym Text View, and Access File Text View tabs.

Synonym Editor - Tree View Tab

The Tree View tab shows a hierarchy of segments and columns on the left, with the attributes and values of the selected item on the right.

Note: The attributes available depend on the type of synonym.

The image below is an example of an SQL data source with a key column selected.
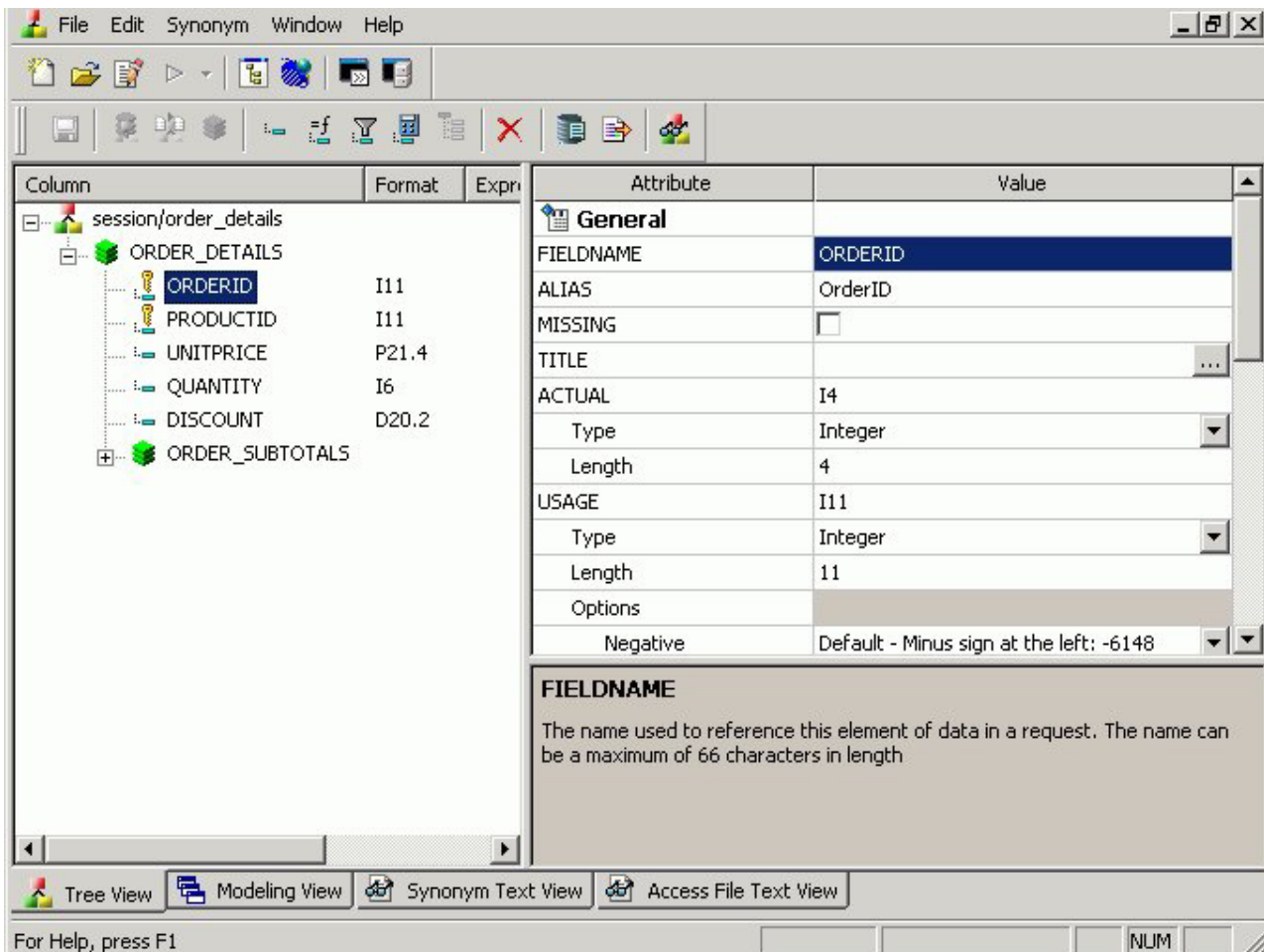
Note: Information about the attribute that has focus is displayed at the bottom of the attribute list. In this case, an explanation of the FIELDNAME attribute appears.

The following objects may appear on the Tree View tab.

Object	Function
	Synonym. General icon used to indicate synonyms, visible on top of the Tree View tab. It provides information about the file name used and the application it resides in.

Object	Function
	Segment.
	Indicates a segment in a synonym. The root or parent segment appears first in the tree.
	Key Column.
	Indicates a key column.
	Column.
	Indicates a general column.
	Virtual Column (Define).
	Indicates a virtual or defined column.
	For more information about virtual columns, see Adding Virtual Columns (DEFINE) in a Synonym.
	Index Column.
	Indicates the native DBMS has an index for quick retrieval of values for this field.
	Filter.
	Indicates a Master File filter.
	Compute field.
	Indicates a computed field.

Synonym Editor - Segment Pop-up Menu

When you right-click a segment in the Synonym Editor, the following pop-up menu appears in the Tree View tab.

The image below is an example of an SQL data source with the root segment selected.

The following options are available:

Add
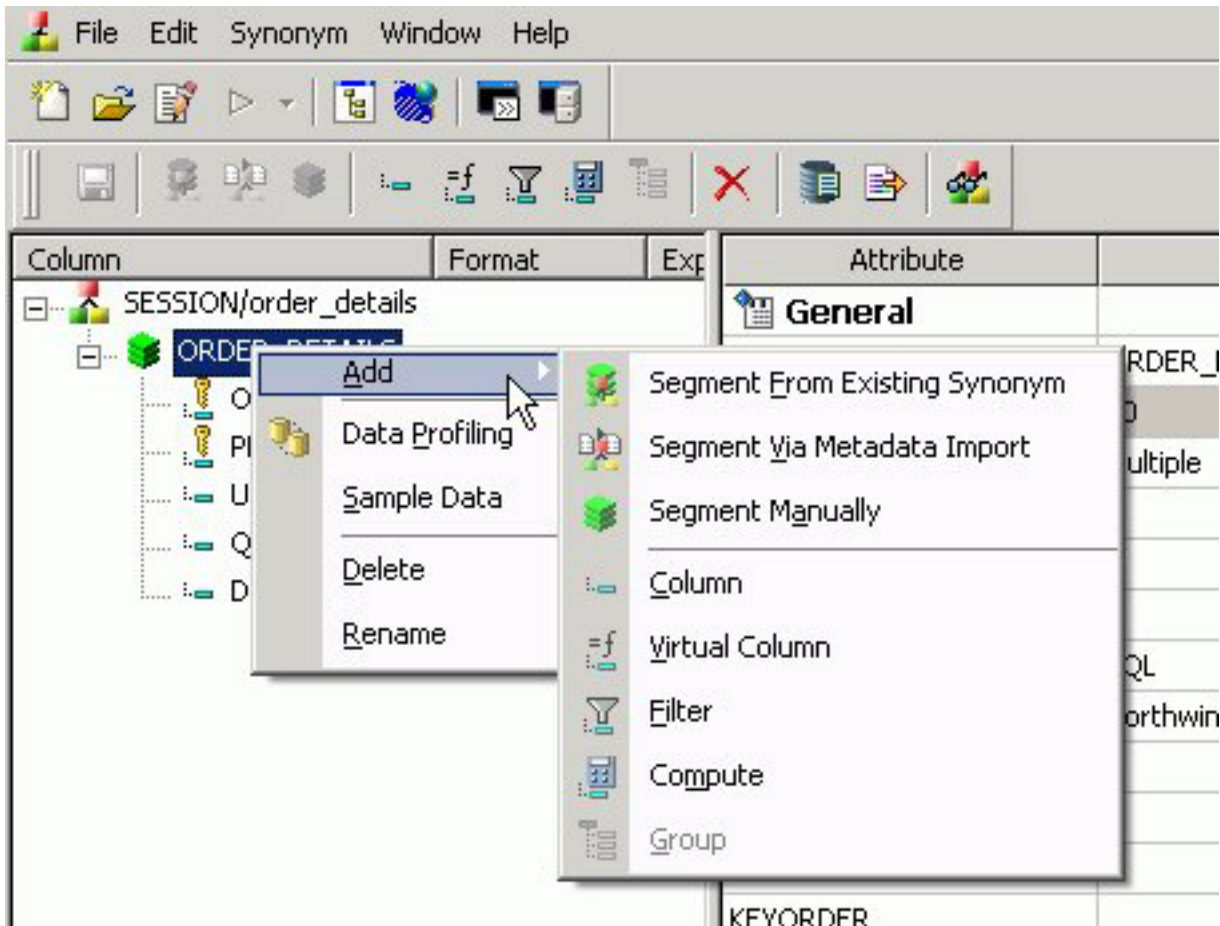
> Enables you to add one of the following:

> **Segment From Existing Synonym**

>> Adds an existing synonym to the current synonym.

> **Segment Via Metadata Import**

>> Enables you to create and add a new synonym to the current synonym through the Create Synonym tool.

> **Segment Manually**

>> Adds a synonym that must by coded manually.

> **Column**

>> Adds a column to the segment.

> **Virtual Column**

>> Adds a virtual column to the segment.

> **Filter**

>> Adds a filter to the segment.

> **Compute**

>> Adds a calculated value to the file.

Group

Adds a group column to the segment.

Data Profiling

Provides the characteristics of the data for a segment.
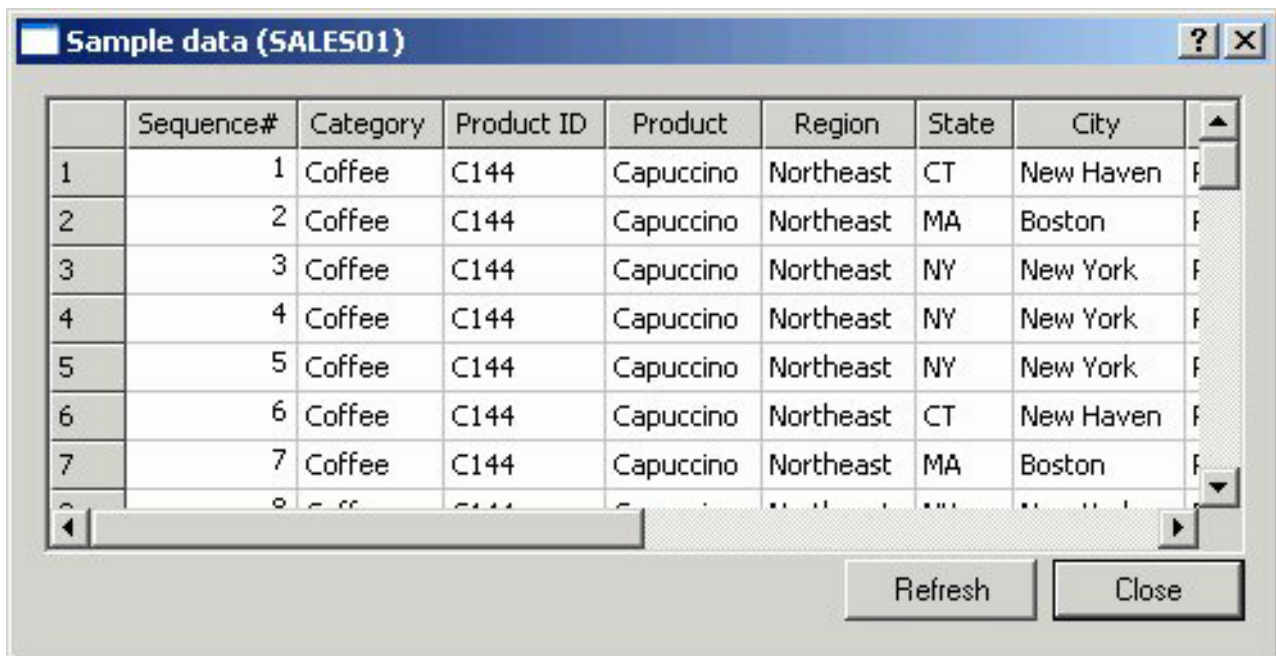
Data Profiling is available from the right-click context menu for all columns in the Master File hierarchy tree.

Sample Data

Displays sample data in the workspace.

Sample data is available from the Synonym Editor toolbar, the right-click context menu for all columns in the Master File hierarchy tree, and from the Modeling View.

The following image is an example of the sample data that appears for a segment:

Note: Sample Data is a great way to test the synonym for field data and to view the type of records returned. This can assist when performing Joins, testing connectivity to data sources, and so on.

Delete

Deletes the segment.

Rename

Enables you to rename the segment.

Synonym Editor - Column/Field Pop-up Menu

When you right-click a column in the Synonym Editor, the following pop-up menu appears in the Tree View tab.

The image below is an example of an SQL data source with a column selected.

The following options are available:

Add

> Enables you to add one of the following:

> **Column**

> > Adds a column to the synonym.

> **Virtual Column**

> > Adds a virtual column to the synonym.

> **Filter**

> > Adds a filter to the segment.

> **Compute**

> > Adds a calculated value to the file.

> **Group**

> > Adds a group column to the synonym.

Impact Analysis

> Displays an Impact Analysis report for the particular column in the workspace. An Impact Analysis report identifies the procedures that access a Master File or field within a Master File. For detailed information on Impact Analysis, see Chapter 15, "Analyzing Metadata and Procedures," on page 673.

Data Profiling

Provides the characteristics of the data for a column. Data Profiling is available from the right-click context menu for all columns in the Master File hierarchy tree.

Sample Data

Displays sample data in the workspace.

Sample data is available from the Synonym Editor toolbar, the right-click context menu for all columns in the Master File hierarchy tree, and from the Modeling View.

The following image is an example of the sample data that appears for a column.
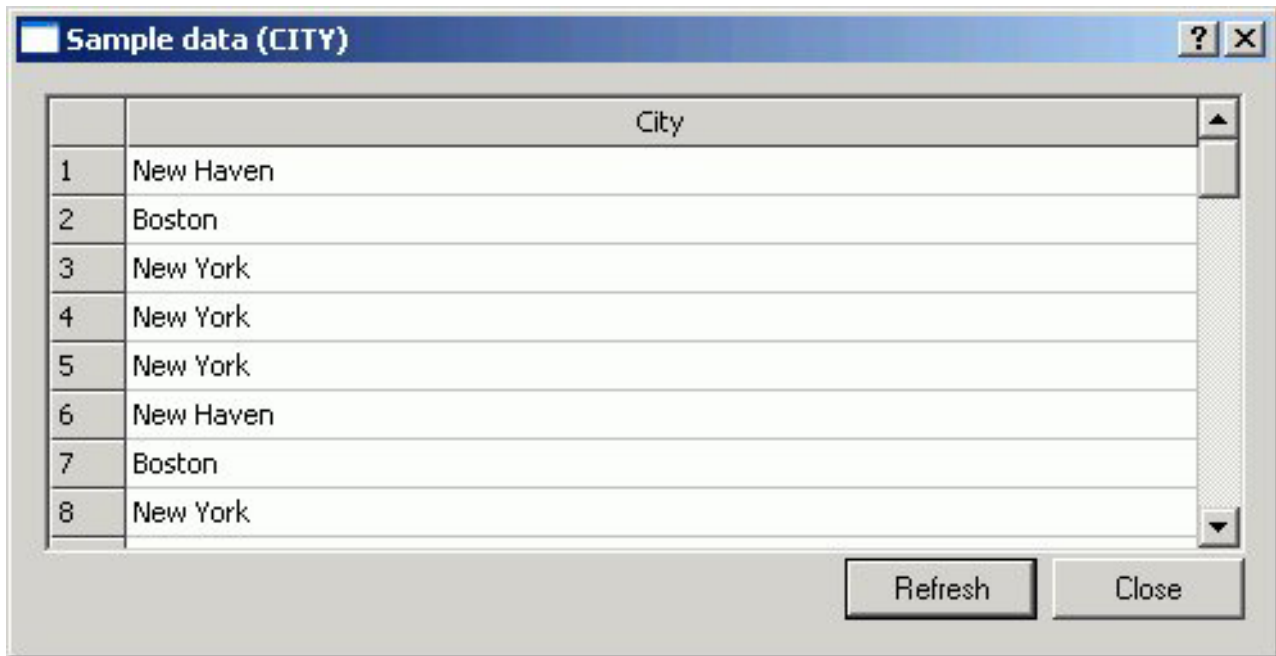
Note: Sample Data is a great way to test the synonym for field data and to view the type of records returned. This can assist when performing Joins, testing connectivity to data sources, and so on.

Decompose Date

Decomposes date fields into virtual columns representing Year, Month, Day, and Quarter fields.

Note: Decompose date is visible for date fields.

Delete

Deletes the column.

Rename

Allows you to rename the column.

Synonym Editor - Modeling View Tab

The Modeling View tab shows a graphical representation of the synonym. Use the Modeling View to define dimensions for OLAP analysis, view join properties, create cluster joins, and add or edit segments.

The image below is an example of an SQL data source in the Modeling View.
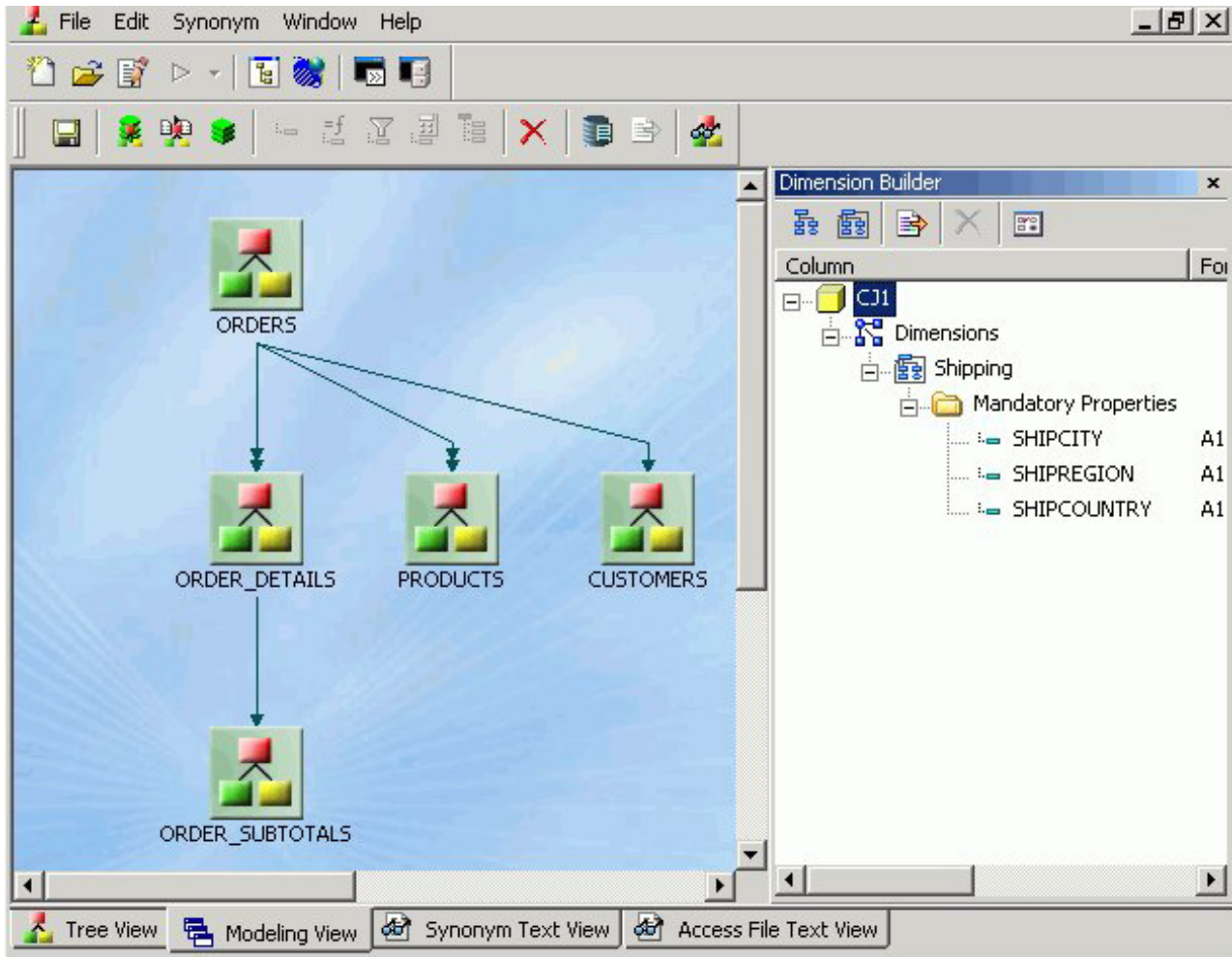
For more information about the Modeling View, see Enhancing Synonyms Using the Modeling View.

Synonym Editor - Synonym Text View Tab

The Synonym Text View tab shows the description of the underlying table.

You can print the Master File code by selecting **Print** from the File menu.

Synonym Editor - Access File Text View Tab

The Access File Text View tab shows the description of a synonym's Access File, which is used to access the database.

Note: You can print the Master File code by selecting **Print** from the File menu.

The image below is an example of an SQL data source in the Access File Text View tab.
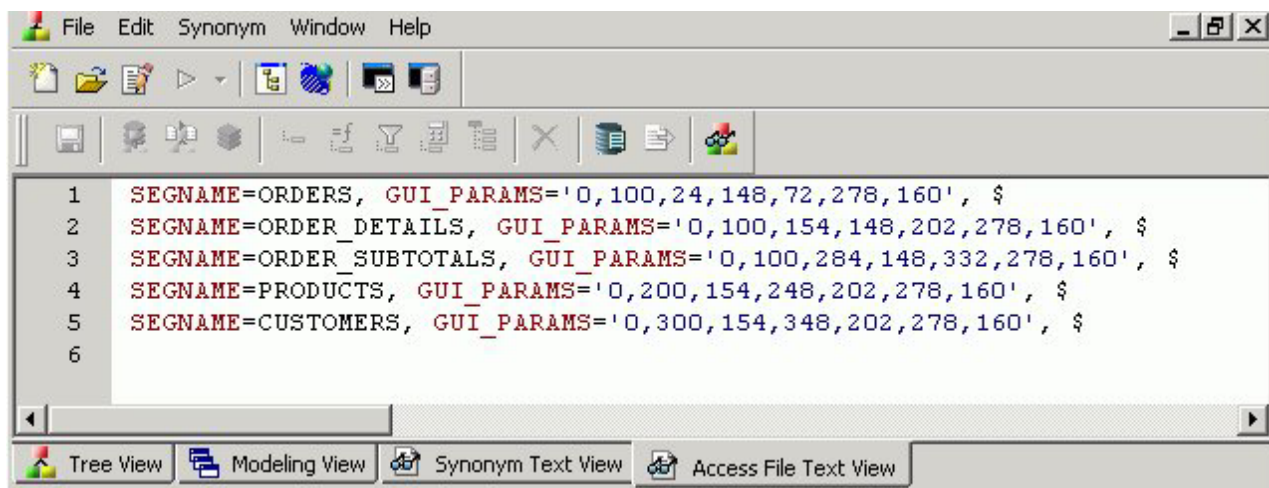
Note: The text views are read-only. You cannot edit the underlying description from these tabs.

Synonym Editor Toolbar

The Synonym Editor toolbar contains buttons that provide quick access to commonly performed functions. The behavior of the button is determined by the selected object. Therefore, certain toolbar buttons may be inactive.

| Button	Definition
	Saves the edits made to the synonym.
	Enables you to add segments from an existing synonym by using the Select Synonym dialog box.
	Enables you to add segments via metadata import by using the Create Synonym tool. This tool creates a synonym and includes it as a segment in the synonym from which the tool was launched.
	Enables you to add segments manually, assigning values to segment attribute fields in the Synonym Editor. **Note:** You would only use this approach if you are coding a Master File from scratch, as you would for a FOCUS data source.
	Adds a column field.
	Adds a virtual column (DEFINE) field. For more information about virtual columns, see Adding Virtual Columns (DEFINE) in a Synonym.
	Adds a Master File filter. For more information about filters, see Creating Filters in a Synonym.

Button	Definition
	Adds a COMPUTE field. For more information about computed fields, see Adding Computed Fields (COMPUTE) in a Synonym.
	Adds a group. For more information about groups, see Adding Group Fields in a Synonym.
	Deletes the selected item.
	Enables you to view and refresh sample data for the object.
	Enables you to create a business view and create a custom Master File that can use selected fields from the original synonym. In addition, you may customize field names, titles, and descriptions. For more information about business views, see Creating Business Views in Developer Studio.

Viewing and Editing Synonym Attributes

The Synonym Editor enables you to view and edit a synonym's attributes.

View and Edit Synonym Attributes

To view and edit synonym attributes:

1. From the Data Servers area, open a synonym by double-clicking a Master File from the Master Files folder.

   The Synonym Editor opens to the Tree View tab, which shows a hierarchy of segments and columns on the left, with the attributes and values of the selected item on the right.

   Note: The attributes available depend on the type of synonym.

   The image below is an example of an SQL data source with a key column selected.
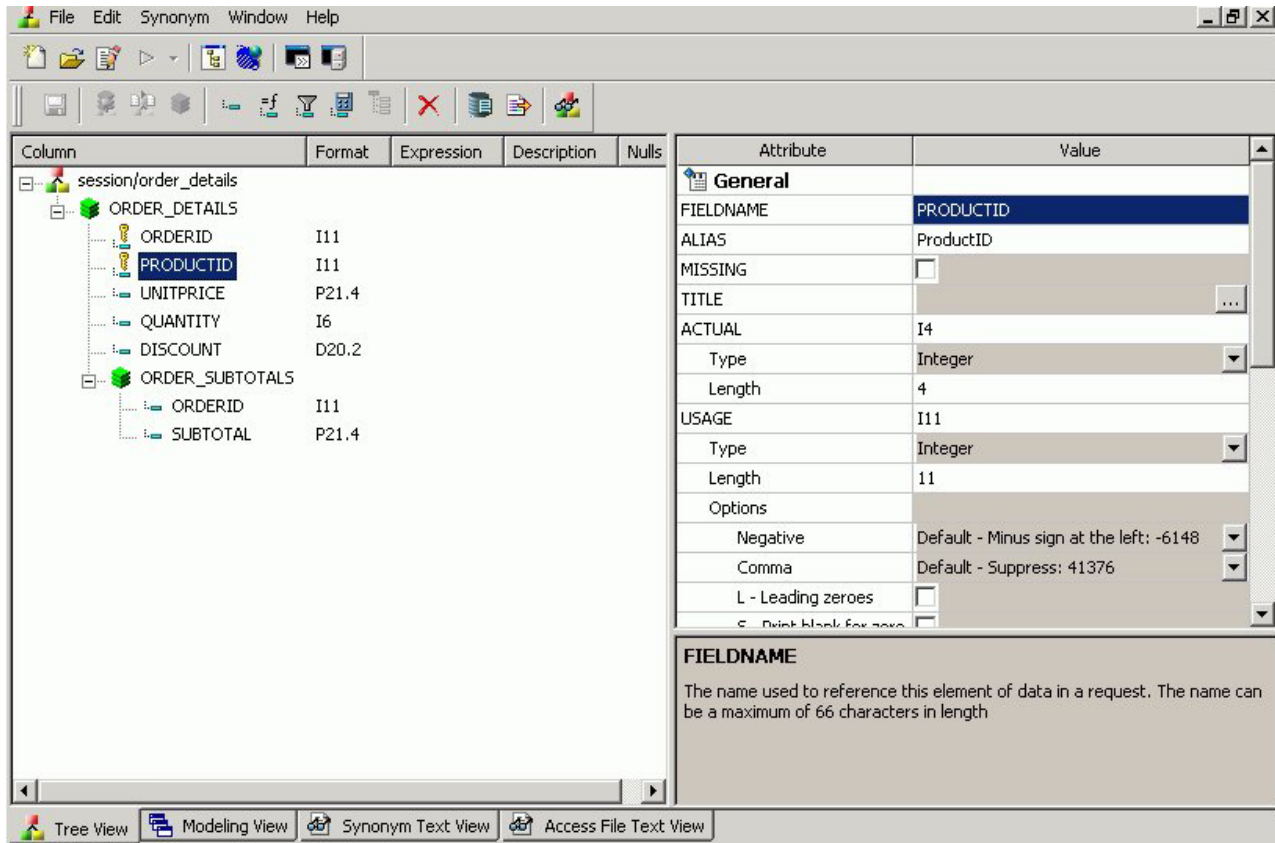
Note: The Format, Expression, Description, and Nulls values are viewable in the hierarchy with the columns. To edit these values, use the corresponding attribute fields on the right-hand side of the Synonym Editor.

2. You can change the attribute values by typing in new values or by using the drop-down menus and check boxes.

   Note: The Synonym Editor does not let you make any changes that would render the Master File unusable. Therefore, you cannot edit any value field that is highlighted gray. In addition, if a change does not have proper syntax/format applied, the field may appear in red text. Messages and warnings appear if trying to save a file with an error.

3. Save changes by clicking **Save** from the File menu.

4. To close the Synonym Editor, select **Close** from the File menu or click the control button in the upper right corner.

Note: If you close the Synonym Editor without saving your changes, you are prompted to do so.

File Attributes Summary

The image below is an example of an SQL data source with the synonym file name selected.
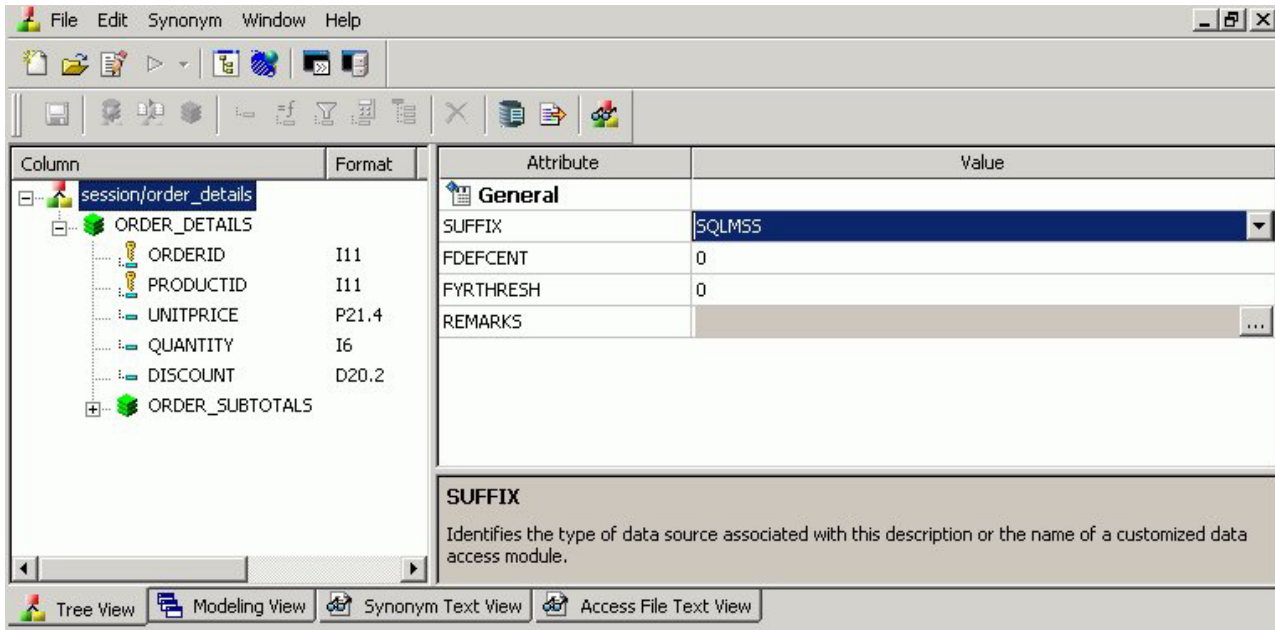
Note: Information about the attribute that has focus is displayed at the bottom of the attribute list. In this case, an explanation of the SUFFIX attribute appears.

Synonyms can have the following file attributes:

General

> **SUFFIX**
>
>> Identifies the type of synonym or data source.
>
> **FDEFCENT**
>
>> Defines the default century value, specifying a century number for handling cross-century dates.
>>
>> **Note:** Use the default setting (0) unless you wish to retrieve data from a earlier century. For example, 19xx.
>
> **FYRTHRESH**
>
>> Defines the base years—to represent the lowest year to which the century value applies (FDEFCENT).
>>
>> **Note:** Use the default setting (0) unless you wish to retrieve data from a earlier century. For example, 19xx.
>
> **REMARKS**
>
>> Enables you to include descriptive information at the file level and specify multiple language descriptions for the synonym. Remarks are displayed along with the file name during reporting.
>>
>> For more information about multilingual descriptions, see Setting Up Multilingual Titles and Descriptions.
>
> **DATASET**
>
>> Identifies the physical location of the data source to be used in the file name, including the extension and the location of the data file.

Note: The attributes available depend on the type of synonym.

Segment Attributes Summary

The image below is an example of an SQL data source with a segment selected.
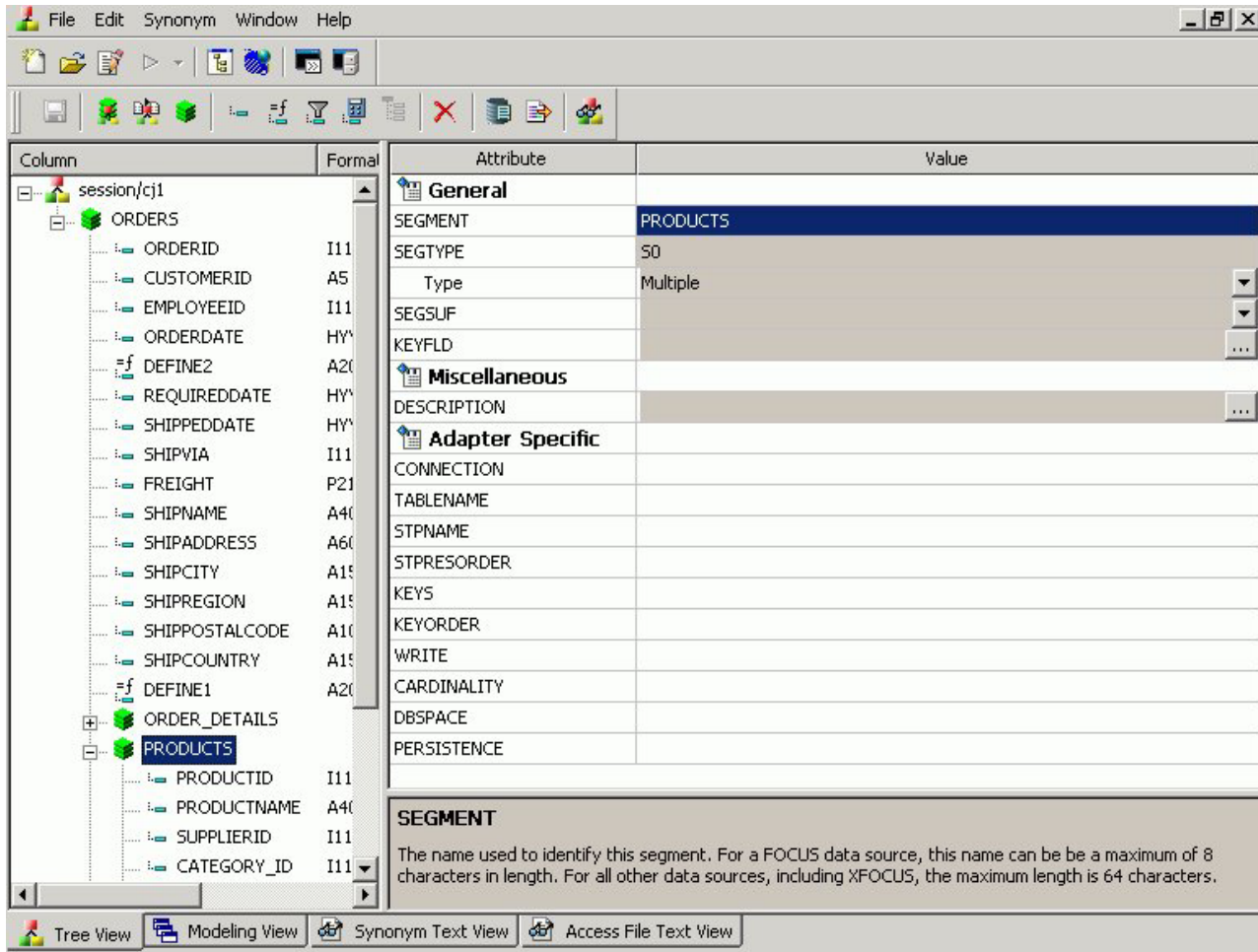
Note: Information about the attribute that has focus is displayed at the bottom of the attribute list. In this case, an explanation of the SEGMENT attribute appears.

Segments in a synonym can have the following attributes:

General

    SEGMENT

        Is the name of the segment.

    SEGTYPE

        Specifies the type or relationship that a segment has to its parent and indicates which of the segment's fields are key fields, and in what order they are sorted.

        Type - Identify the segment type and sorting options from the Type drop-down list.

        Keys - Records are sorted in a data source by key fields. Enter the number of key fields that you want to use for sorting. For example,

no two employees can have the same employee ID number, so you can use that field as the key. A segment instance can have more than one field that makes up the key; that is, two or more field values may be used to distinguish records.

SEGSUF

SEGSUF is used when part of the data source being described by the Master File is of a different data source type than that declared for the entire structure.

Note: SEGSUF is the data source type of a segment and any descendants it might have, where that type differs from the SUFFIX value.

Miscellaneous

DESCRIPTION

Contains a description or comments about the segment.

For more information about multilingual descriptions, see Setting Up Multilingual Titles and Descriptions.

CRFILENAME

Is the name of the cross-referenced data source.

CRSEGNAME

Is the name of the cross-referenced segment.

CRKEY

Identifies the common join field for the cross-referenced segment.

Note: These cross-referenced values (CRFilename, CRSegname, CRKey) are available for FOCUS data sources.

Adapter Specific

Note: Adapter Specific fields are shown if an Access File component has been generated with the synonym.

CONNECTION

Indicates the host server or data source for synonyms.

TABLENAME

Identifies the table or view. It may contain the owner ID, as well as the table name. For some synonyms, it must also contain the data source name.

KEYS

Identifies how many columns constitute the primary key.

KEYORDER

Identifies the logical sort sequence of data by the primary key.

WRITE

Specifies whether write operations are allowed against the table.

CARDINALITY

Defines the number of records that are found in the original data source when the synonym was created.

Note: The attributes available depend on the type of synonym.

Column/Field Attribute Summary

The image below is an example of an SQL data source with a key column selected.

Note: The attributes available depend on the type of synonym.
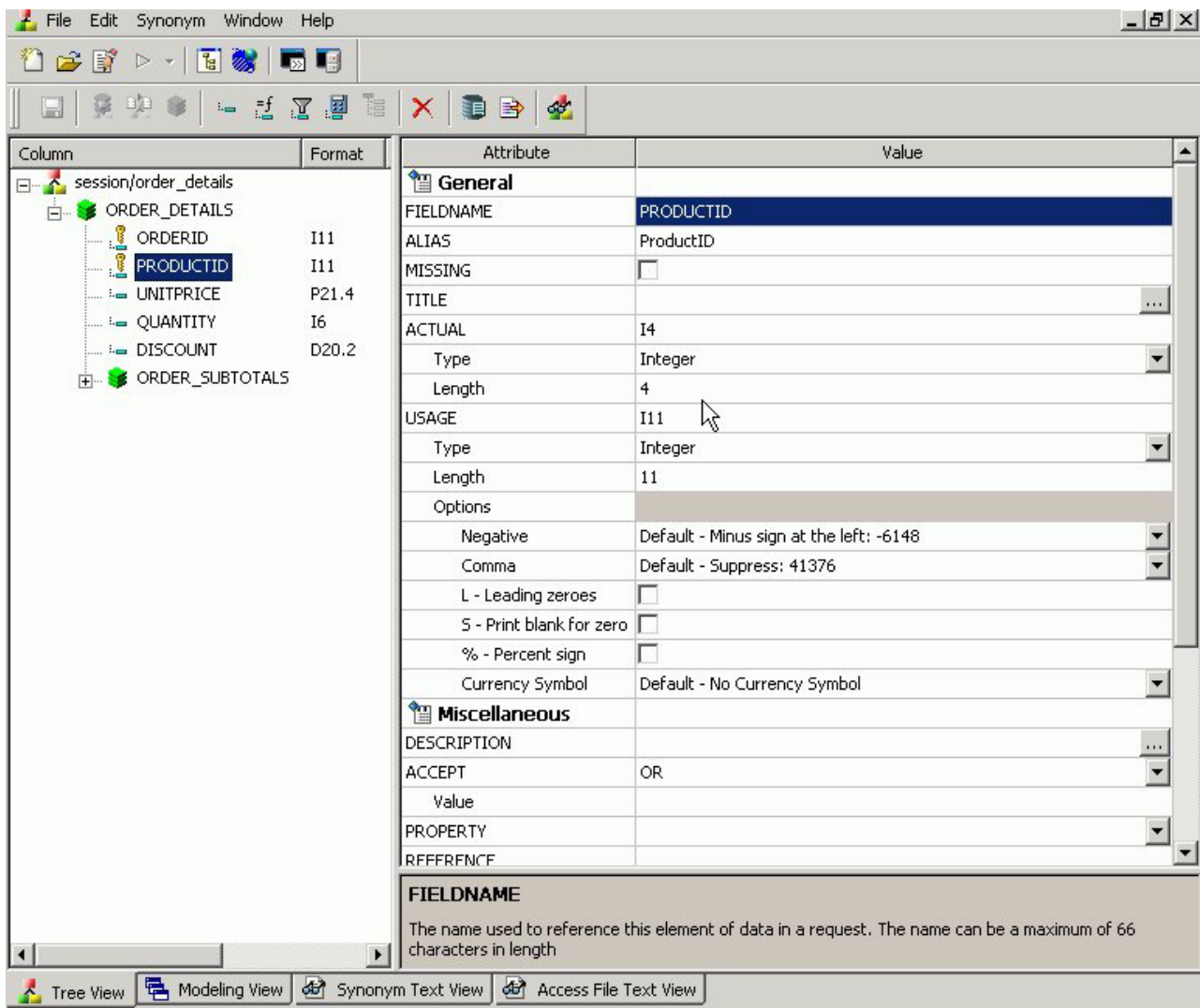
Note: Information about the attribute that has focus is displayed at the bottom of the attribute list. In this case, an explanation of the FIELDNAME attribute appears.

Columns in a synonym can have the following attributes:

General

    FIELDNAME

Is the name of the column.

ALIAS

Assigns an alternative name for a column, or the real column name for a DBMS synonym.

MISSING

Controls how null data is handled, that is, if no transaction value is supplied.

TITLE

Supplies a title to replace the column name that is normally used in reports and enables you to specify multiple language titles for the column or field.

For more information about multilingual titles, see Setting Up Multilingual Titles and Descriptions.

USAGE

Describes the data type and format for the column for usage or display.

Note: Additional attributes, DEFCENT and YRTHRESH, are available if the Usage field is set to Date, Time, or DateTime (Timestamp) format. Use these attributes to enter the century and year threshold values for the column/field.

Miscellaneous

DESCRIPTION

Contains a description or comments about the column or field.

For more information about multilingual descriptions, see Setting Up Multilingual Titles and Descriptions.

ACCEPT

Specifies criteria for validating data.

OR enables you to specify an acceptable value.

FROM-TO enables you to specify a range of acceptable value fields.

FIND enables you to supply file and field names to instruct DB2 Web Query where to search for a data source and for a list of acceptable values. You supply the field name of the data field for which the validation criteria are being assigned, the file name of the target FOCUS data source where the field can be found, and the field name of the target data field that contains the validation criteria.

Note: Find is only available for FOCUS data sources and does not apply to OLAP-enabled Master Files. Note also that, in the Maintain environment, Find is not supported when developing a Master File.

WITHIN

Contains the name of a field to be included in a dimension.

These WITHIN statements are added to the synonym through the Dimension Builder to OLAP enable FOCUS files and relational tables. This enables you to perform OLAP analysis using the OLAP Control Panel, or to use it with the FML Painter.

FIELDTYPE

Identifies an indexed column.

Note: FIELDTYPE=R indicates a read-only column, which will not be updated by DM flows. This setting is useful for columns are that automatically assigned a value by the RDBMS.

ACCESS_PROPERTY

Specifies access options for the column's data.

INTERNAL defines a column that does not appear in sample data or in the list of available columns. Restricts the field from showing in any of the Field Lists in the reporting tools.

NEED_VALUE defines a column that requires a value to access the data.

Select By defines a column by value, range, or multivalues.

AUTHRESP defines a column that describes the result of an authentication operation. Correct response values must be provided in the ACCEPT attribute (using the OR predicate if more than one value is acceptable).

AUTHTOKEN defines a column that contains a response token to be passed as an input value to the operation to be executed.

HELPMESSAGE

Appends a help message to a column.

Note: The attributes available depend on the type of synonym.

Setting Up Multilingual Titles and Descriptions

You can open a synonym in the Synonym Editor and provide text for the title, caption, and description in multiple languages. These descriptions appear in the specified language in reports generated against the synonym.

The Multilingual Titles dialog box is available from the Remarks, Title, and Description attribute value fields in the Synonym Editor.

Note: The attributes available depend on the type of synonym.

Set Up Multilingual Titles and Descriptions

1. From the Data Servers area, double-click the Master File or select **Edit in Synonym Editor** from the File menu.

   The Master File opens to the Tree View tab in the Synonym Editor.

2. To add multilingual text:

   - For **Title**, click a column from the Master File hierarchy of columns on the left.
   - For **Remarks**, click the root level of the Master File (application/filename) on the left.

- For **Descriptions**, click a column, segment, or custom field from the Master File hierarchy of fields on the left.

The corresponding attributes and values appear on the right.

3. Click the browse **(...)** at the end of the value field for either Remarks, Title, or Description.

The Multilingual dialog box opens.

4. From the Specify Titles/Descriptions for different languages drop down list, choose the language in which you want the remarks (descriptions) and/or titles to be displayed.
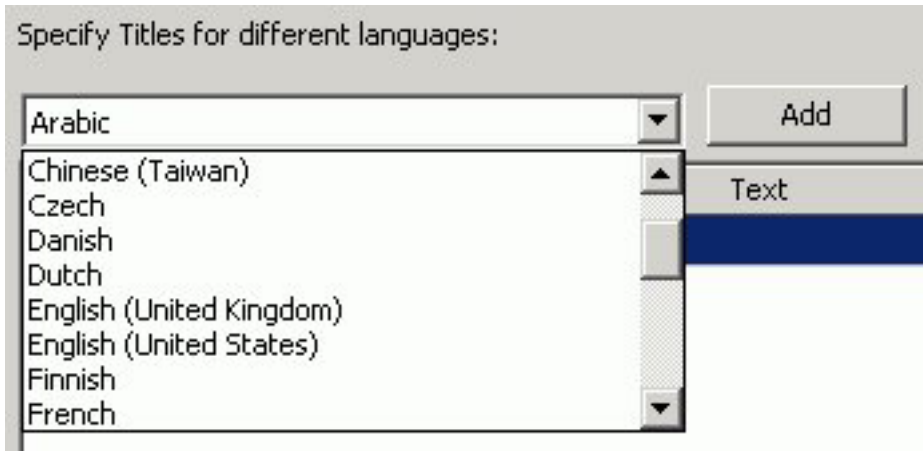
5. Click **Add**.

The selected language is added below the default language (which is determined by your code page selection).
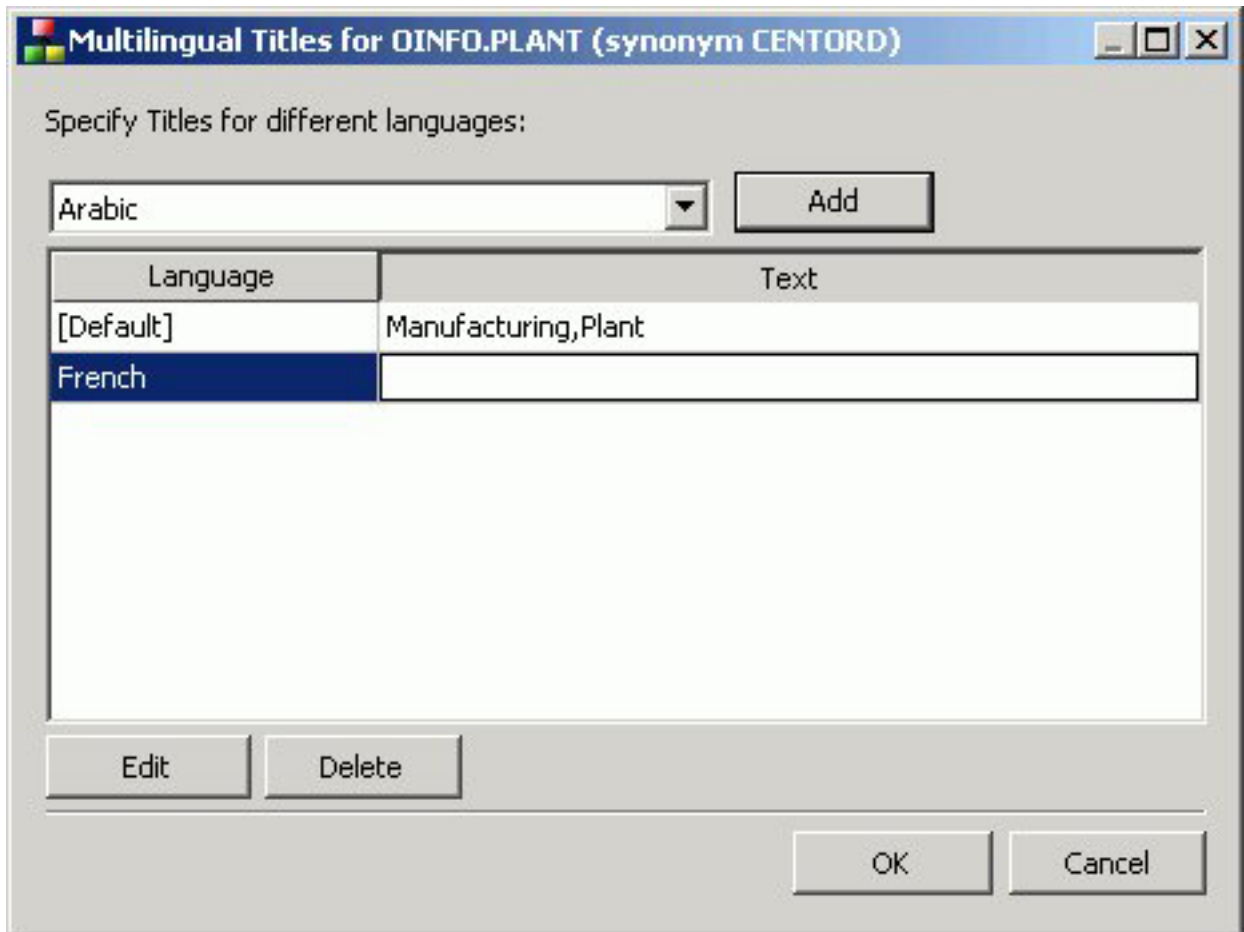
6. Type in a description/title in the Text field.
7. Required: You may add, edit or delete additional titles/descriptions:
   To add an additional language:
   a. Required: Select another language from the drop-down list.
   b. Required: Click **Add**.
   c. Required: Type in a title/description for the field.
   To edit an existing specified title/description:
   a. Select the title/description and click **Edit**.
      You may also double-click the title.
   b. Manually type a title/description name.
   To delete a specified title:
   a. Select the title/description and language to be deleted.
   b. Click **Delete**.
8. Required: Click **OK** to close the Multilingual Titles dialog box.
9. Required: Click **Save** from the File menu to save the synonym.
10. To close the synonym, select **Close** from the File menu or click the control button in the upper right corner.

Enhancing Synonyms Using the Modeling View

The Synonym Editor Modeling View tab provides a visual presentation for the synonym for which positioning is preserved and stored in the Access file. Use the Modeling View to define dimensions for OLAP analysis, view join properties, create cluster joins, and add or edit segments. For more information about using the Dimension Builder in the Modeling View, see Defining Dimensions for OLAP Analysis.

Note: The Modeling View is not available for Cube data sources.

Enhance Synonyms Using the Modeling View

Use the Modeling View to enhance a synonym by adding a segment.

1. From the Data Servers area, open a synonym by double-clicking a Master File from the Master Files folder.

   The Synonym Editor opens.

2. Click the **Modeling View** tab.

   The Synonym Editor Modeling View tab opens in the workspace.

3. Right-click a segment in the workspace.

   The following pop-up menu appears.
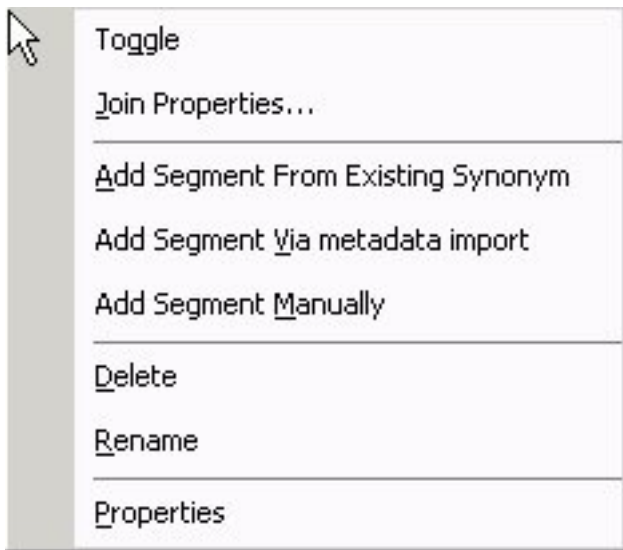
4. Add segment through one of the methods listed:
   - Add Segment From existing synonym adds an existing synonym as a segment to the current synonym.
   - Adding Segment Via metadata import enables you to add segments by using the Create Synonym tool. This tool creates a synonym and includes it as a segment in the synonym from which the tool was launched.
   - Add Segment Manually enables you to add segments manually to the current synonym.

   Note: You would only use this approach if you are coding a Master File from scratch, as you would for a FOCUS data source.

5. Required: Optionally, you can double-click the arrowed lines between tables to open the Join Properties dialog box and determine how the tables are linked.

Tip: You may also right-click a file icon and select **Join Properties** from the context menu.

In addition, when creating a Cluster Join, there are two additional options for linking tables: Join All (default) is shown by a double arrow and Join Unique is shown as a single arrow.
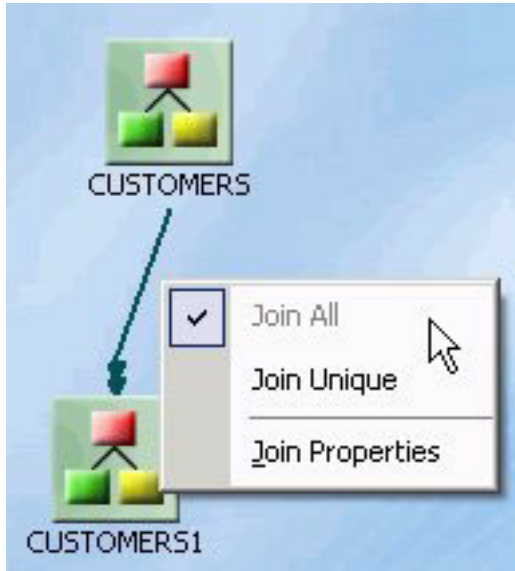
Note: Join options are not available for FOCUS files.

## Edit Synonyms Using the Modeling View

To edit synonyms using Modeling View:

1. From the Data Servers area, open a synonym by double-clicking a Master File from the Master Files folder.

   The Synonym Editor opens.

2. Click the **Modeling View** tab.

3. Right-click a segment icon.

   A pop-up menu appears, providing options for adding segments.

   Note: Options for adding segments are also available from the icons above the workspace.
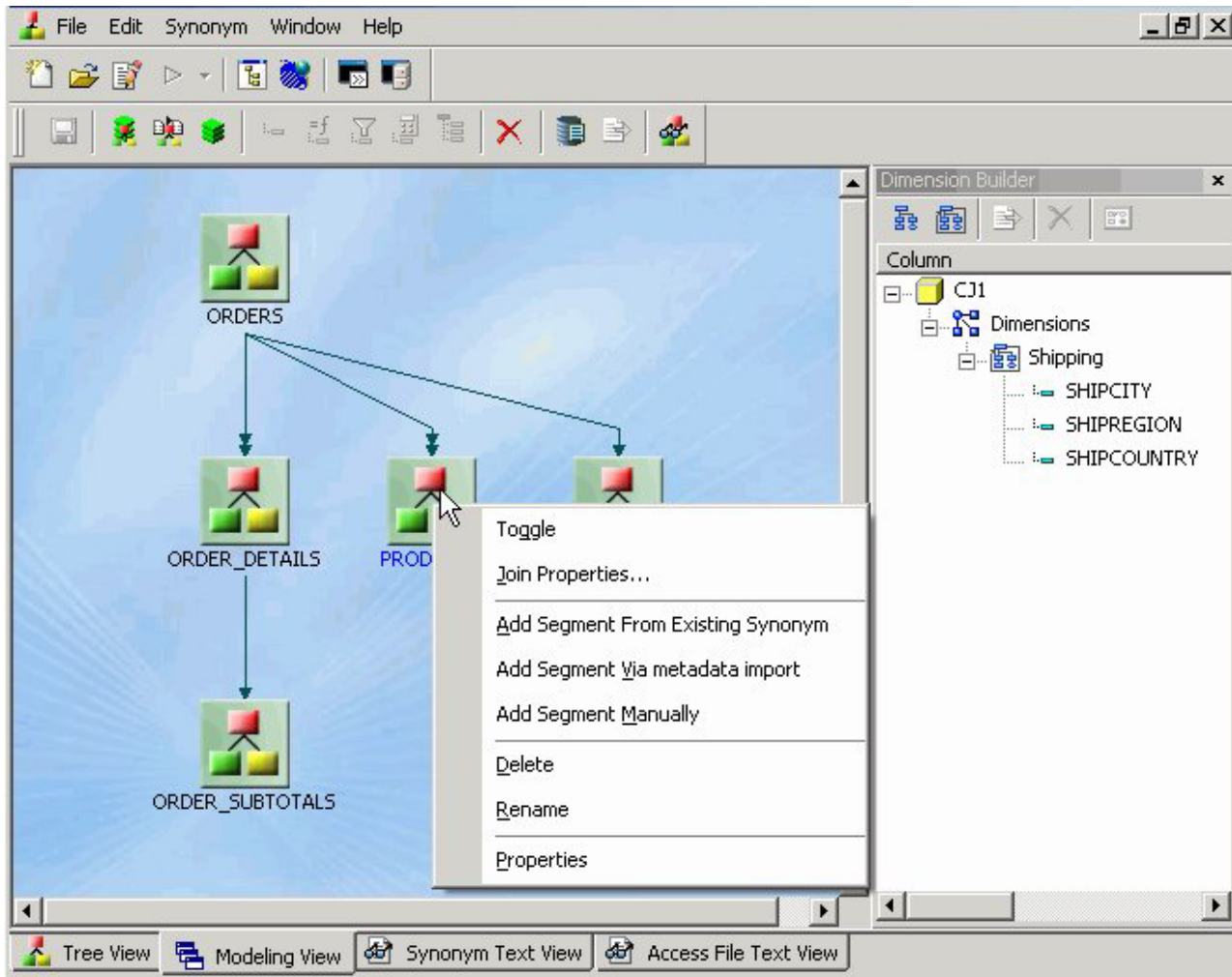
To Add a Segment From an Existing Synonym:

a. Required: Select **Add Segment From Existing Synonym** from the pop-up menu.

The Select Synonym dialog box opens.
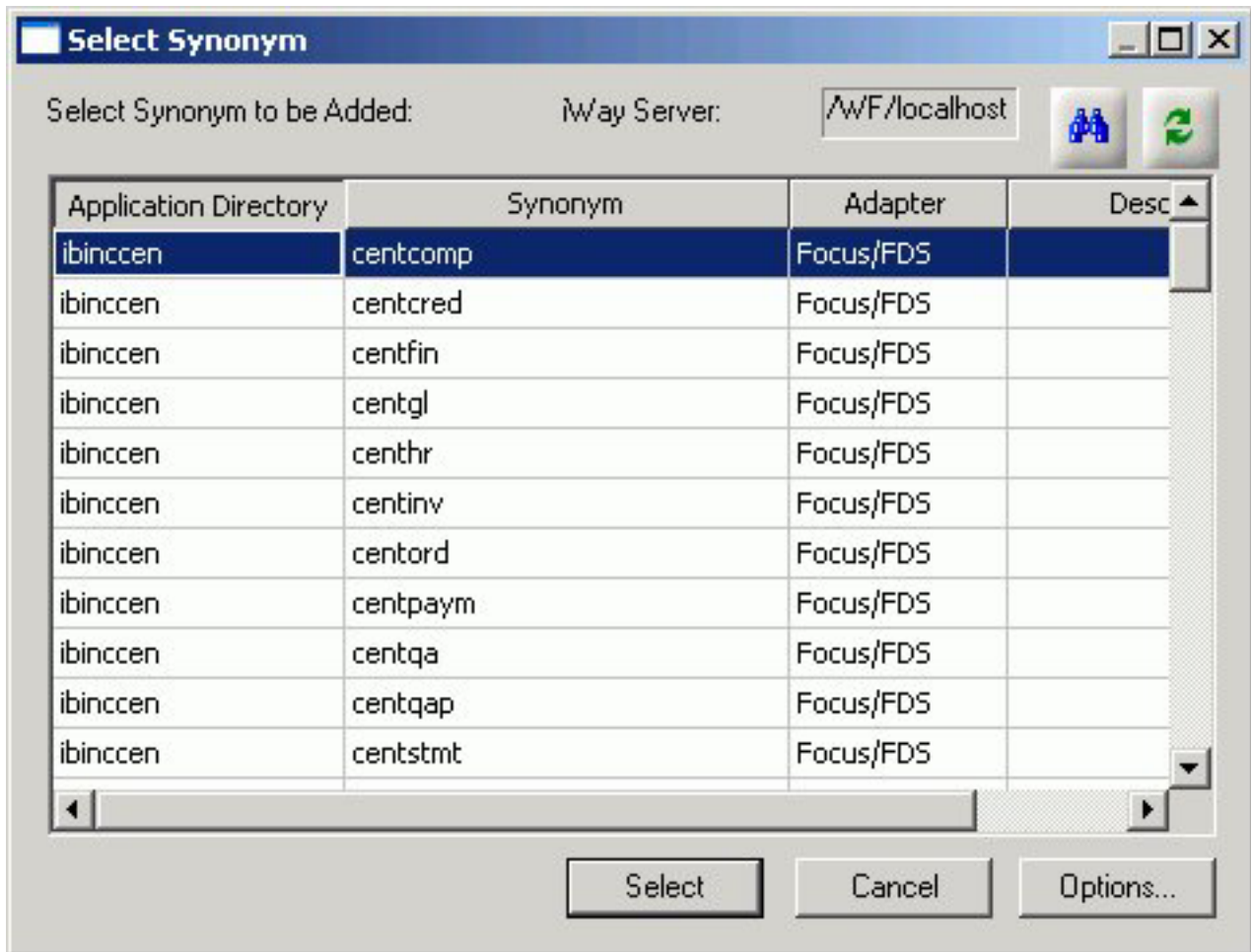
b. Click a synonym and click **Select**.

The segment is added to the synonym.

To Add a Segment Via Metadata import:

a. Select **Add Segment Via metadata import** from the pop-up menu.

This method enables you to launch the Create Synonym tool, create a synonym, and incorporate it as a new segment in the synonym from which you initiate the import.

The Create Synonym dialog box opens.

b. Create a synonym.

The segment is added to the synonym.

To Add a Segment Manually:

a. Select **Add Segment Manually**.

The segment is added to the Modeling View and a default field is created, using a default name SEG name.

b. Right-click the segment and select **Properties**.
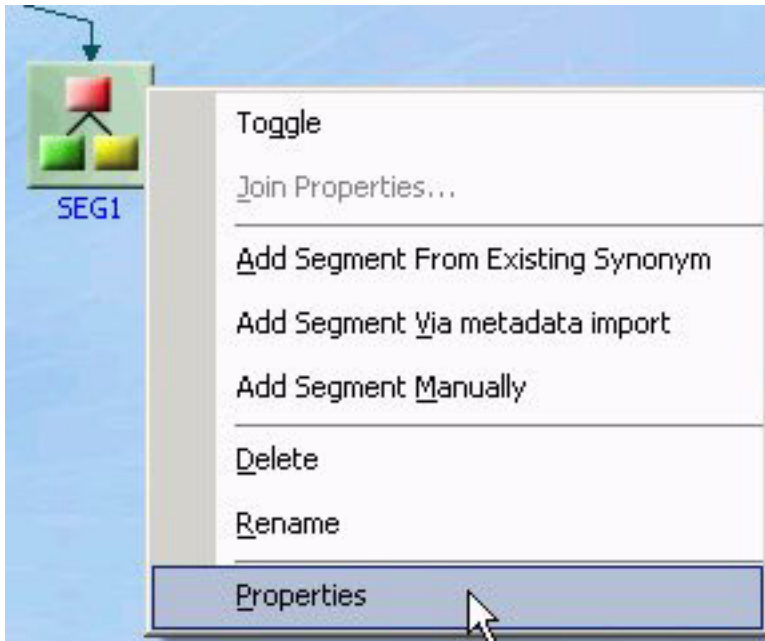
The Segment Properties dialog box opens.
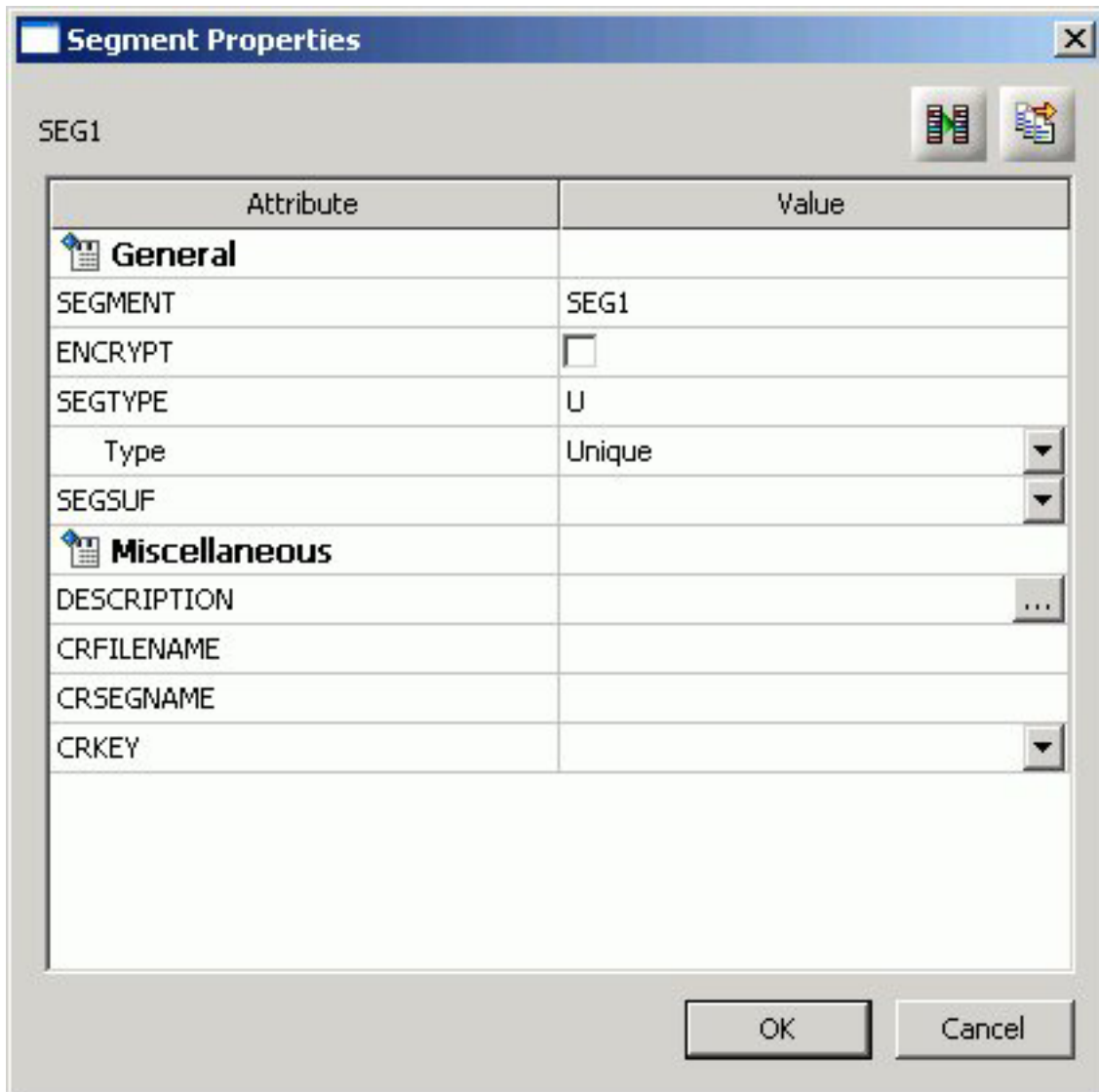
c. Required: Edit the segment's properties, if needed, and click **OK**.

d. Switch to the tree view to add or change attributes for the new segment(s).

The following image shows a synonym which has had several segments added in the Modeling View tab.
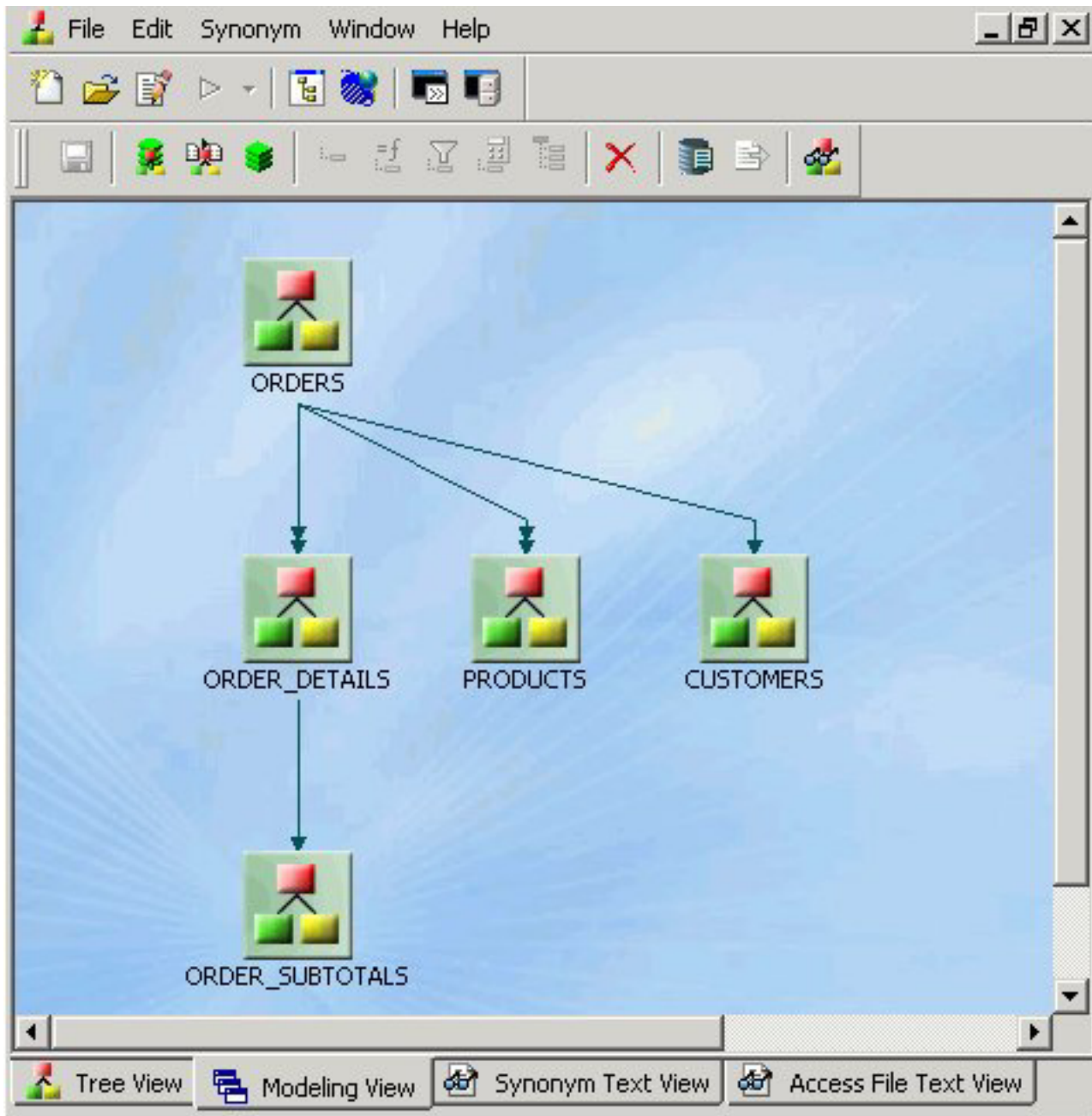
You can select the Synonym Text View or Access File Text View tab to see the resulting segment attributes.

Modeling View Pop-up Menu

When you right-click a synonym or segment in the Synonym Editor Modeling View tab, the following pop-up menu appears.

The following pop-up menu appears.
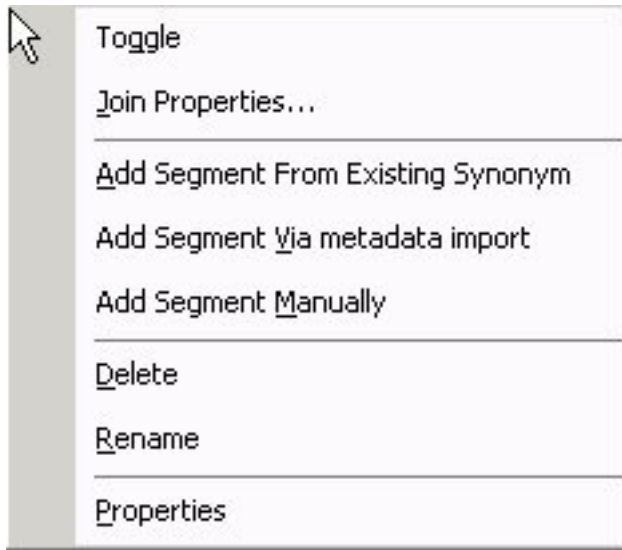
The pop-up menu has the following options:

Toggle

> Changes the view from a file icon to a table view, which enables you to see columns, sample data, and sample data parent keys.

> **Tip:** You may also double-click a file icon to open the table view. Double-click the table to close the table view, or click the X button from the toggle toolbar to close.

Join Properties

> Provides access to the Join Properties window.

> **Note:** Join options are not available for FOCUS files.

Add Segment From Existing Synonym
> Enables you to add an existing synonym as a segment to the current synonym.

Add Segment Via metadata import

> Enables you to create a new synonym through the Create Synonym tool and add it to the current synonym.

Add Segment Manually

Adds a segment to the current synonym, that must by coded manually.

Delete

Deletes the segment.

Rename

Enables you to rename the segment.

Properties

Launches the Segment Properties dialog box where you can edit the segment.

Join Properties Dialog Box

When you select Join Properties from a segment in the Synonym Editor Modeling View tab, the Join Properties dialog box appears.

The Join Properties dialog box contains Left and Right Source Columns, Join Type, and Join Condition options. Use the left and right source columns to create join maps and view sample data.
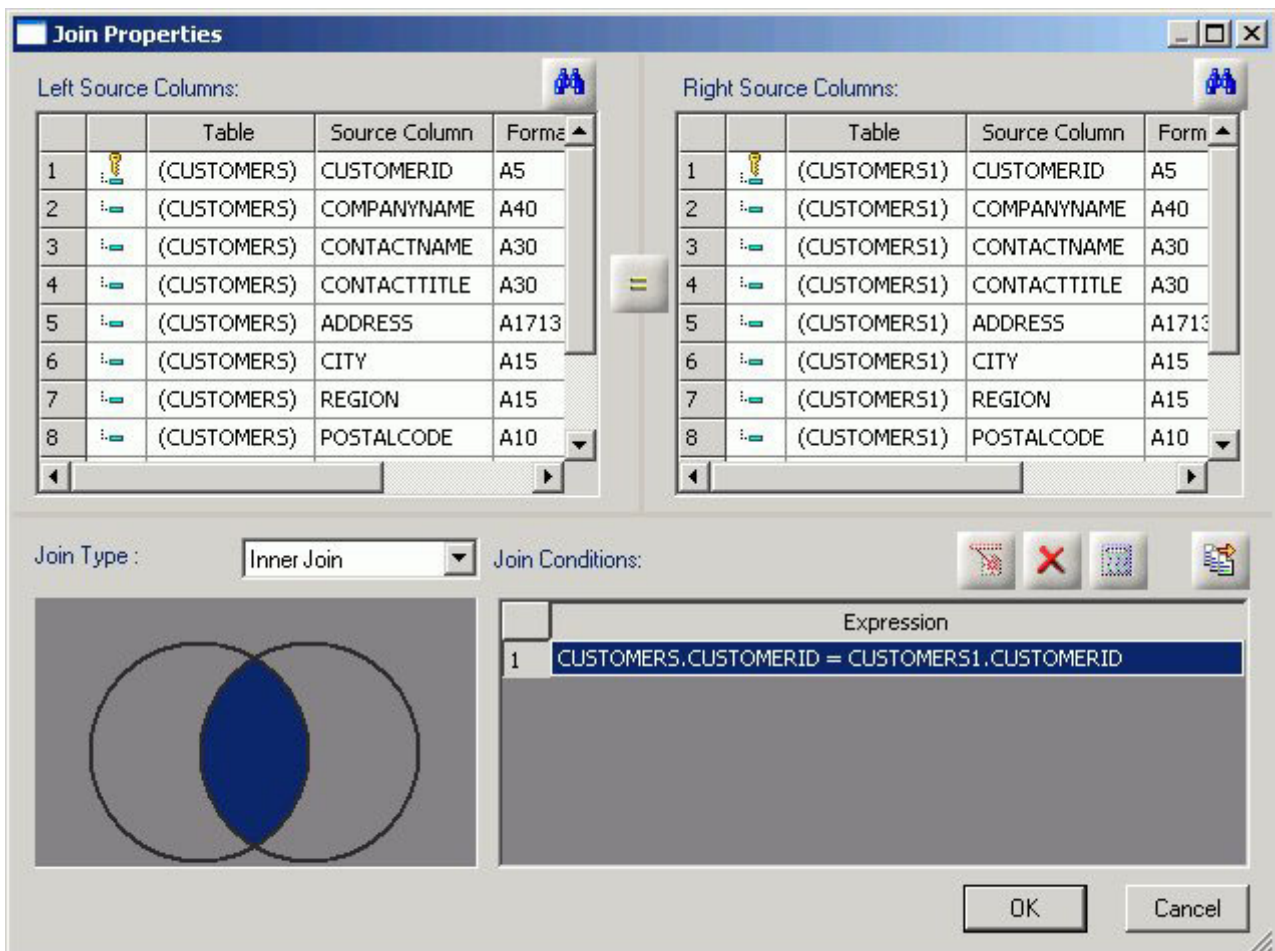
Note: Join options are not available for FOCUS files.

Creating Cluster Joins

Cluster Joins enable you to create a new file structure by linking existing synonyms of two or more relational tables using the same or mixed data sources. For example, you may join a DB2 table and an Oracle table, and so on. Use Cluster Joins to create new views in the metadata by linking together physical tables and easily report against the new view/structure. You can create cluster joins by using the Modeling View of the Synonym Editor.

The Master File description that is created combines the fields of the joined tables within a single file. The Access File from the combined file contains information about the actual location of the data sources, the Join information, and shows how the tables are linked.

The total number of tables that you can add to the tool is 64 (using 63 joins) that results in a new Master File that has a maximum of 64 segments.

Create a Cluster Join by Enhancing Existing Synonyms

Use the Modeling View to enhance an existing synonym by adding a segment.

1. From the Projects or Data Servers area, open a synonym by double-clicking a Master File from the Master Files folder.

   The Synonym Editor opens.

2. Click the **Modeling View** tab.

   The Synonym Editor Modeling View tab opens in the workspace.

3. Right-click a segment in the workspace.

   The following pop-up menu appears.
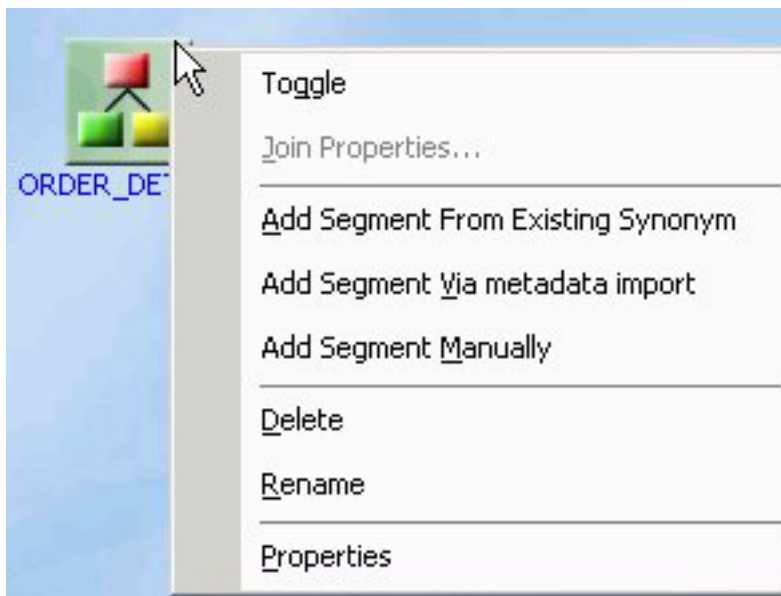
4. Add tables (segments) through one of the methods listed:

   a. Add Segment From Existing synonym.

      Select a synonym to be added and click **Select**.

      Note: Use this method if you are creating a cluster join with an existing table/synonym.

> **Tip:** Click **Save As** from the Modeling View File menu if you do want to modify the original synonym.

b. Required: Adding Segment Via metadata import enables you to add segments by using the Create Synonym tool. This tool creates a synonym and includes it as a segment in the synonym from which the tool was launched.

> **Note:** Use this method if you are creating a cluster join and need to use a synonym that does not exist. This option enables you to create the synonym and continue to create the cluster join.

c. Add Segment Manually enables you to add segments manually, assigning values to segment attribute fields in the Synonym Editor.
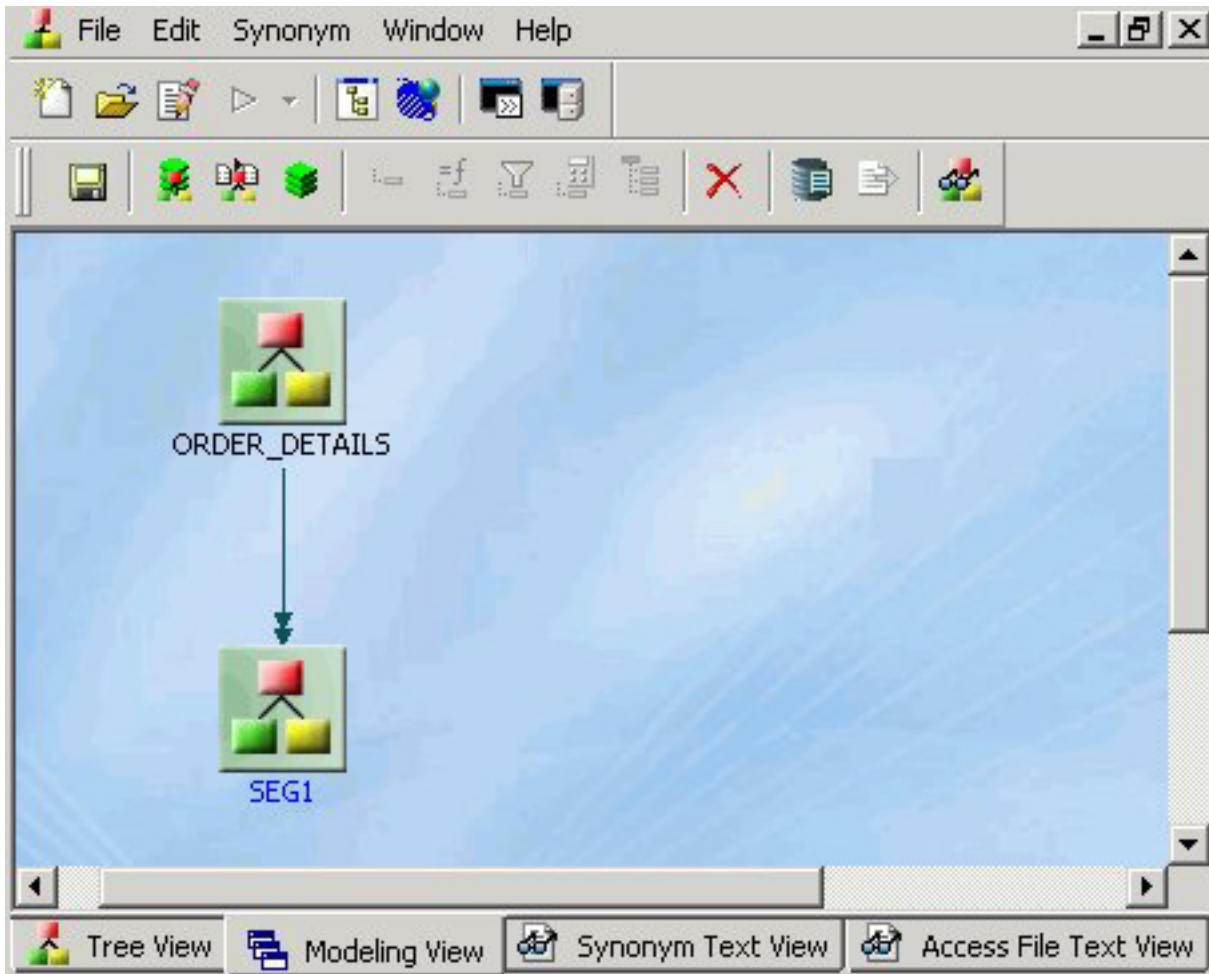
> **Note:** Use this method if you are coding a Master File from scratch, as you would for a FOCUS data source.

The segment is added in the Modeling View.

5. Right-click the arrowed lines between tables to view the join options.
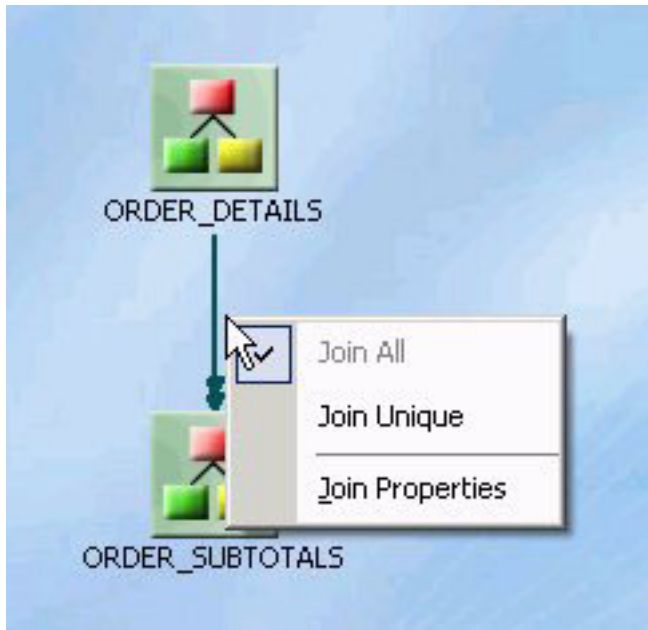
Note: These join options are not available, and do not appear, for FOCUS tables. The only way to view these options are by using Relational tables.

- Join All indicates a multiple instance (one-to-many) type of join. At run time, each host record can have many matching records in the cross-referenced file.

  Note: Join all is the default option.

- Join Unique indicates a single instance (one-to-one) type of join. At run time, each host record has, at most, one matching record in the cross-referenced file.

- Join Properties opens the Join Properties dialog box, from which you can determine how the tables are linked. You may:

  – Create Inner, Outer, or Cross Joins.

    Note: Outer joins are subdivided into Left Outer Joins, Right Outer Joins, or Full Outer Joins.

  – Combine records from the selected tables by mapping source columns.
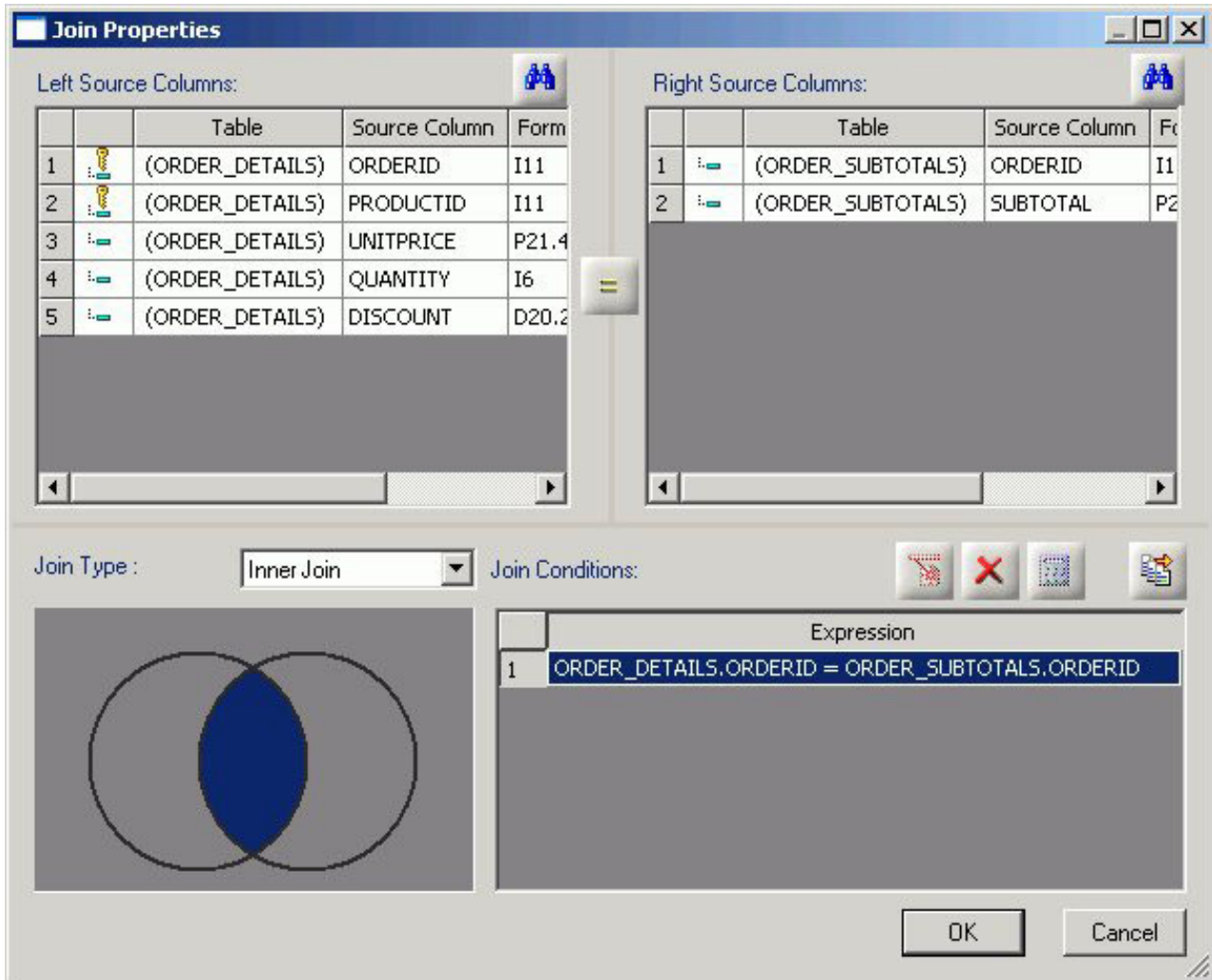
Use the Tree and Text View tabs of the Synonym Editor to view or edit the details of these tables (segments).

Tip: Click **Save As** from the Modeling View File menu if you do want to modify the original synonym.

Create a Cluster Join Using a New Synonym

Another way to create a cluster join is to start with an empty synonym:

1. Right-click a Master Files folder, select **New**, then **Synonym via Synonym Editor**. The Add Master File dialog box opens.

2. Enter a unique file name in the File name field.

3. Click **Open**.

   The Synonym Editor opens to the Modeling View tab.

4. Right-click in the workspace and select from one of the available options to start building the new view.
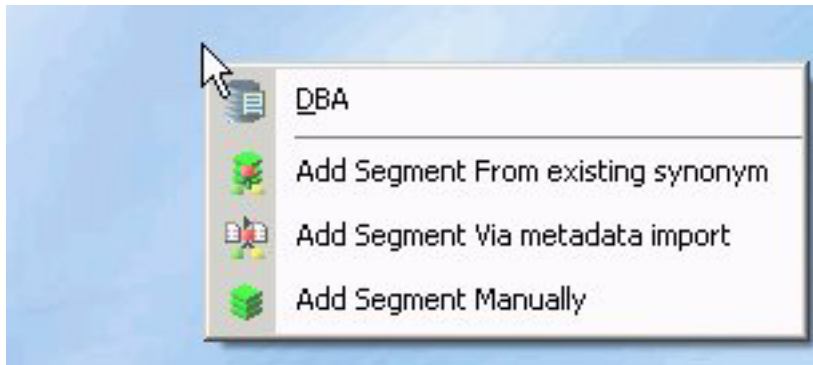
Defining Dimensions for OLAP Analysis

Synonyms can be modified to support Online Analytical Processing (OLAP). The Synonym Editor provides tools to create OLAP hierarchies and dimensions. OLAP enables you to drill down or roll up on hierarchical data, pivot fields from columns to rows (or vice versa), and slice-and-dice information by filtering or querying data sources based on specified criteria thresholds.

You OLAP-enable the Master File by using the Synonym Editor to create dimension(s) at the field level and associate fields with each dimension.

Note: OLAP is a reporting facility; it is not relevant to data maintenance projects.

Using the Dimension Builder in the Synonym Editor Modeling View

The Dimension Builder enables you to create logical views based on enterprise data (relational or legacy data sources) for multi-dimensional analysis without manually editing metadata. The Dimension Builder works with relational and FOCUS data sources.

Add a Parent/Child Hierarchy
1. From the Synonym Editor, click the **Modeling View** tab.

   The Dimension Builder appears on the right side of the window.
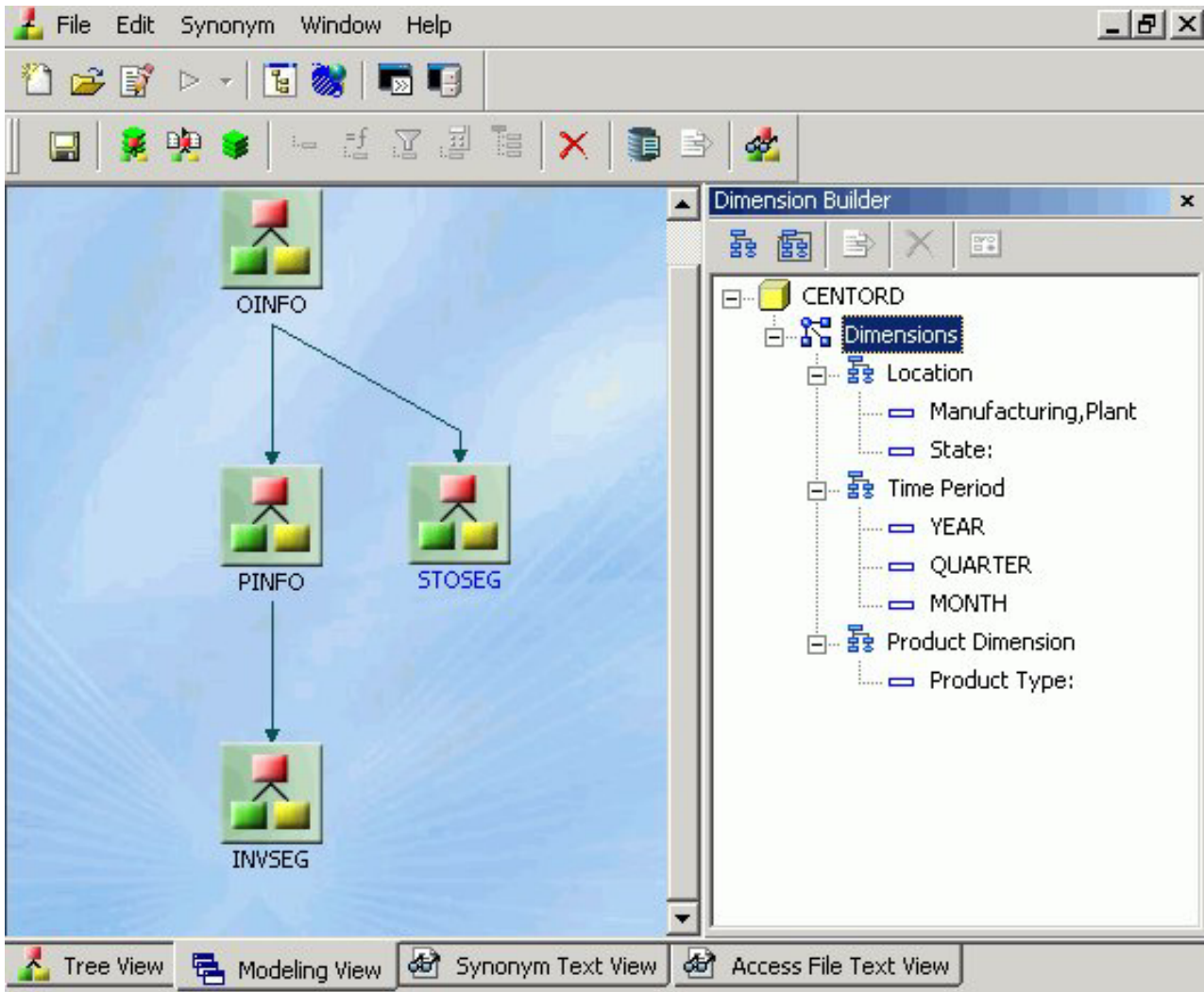
> **Note:** If the Dimension Builder is not visible, ensure that **Dimension Builder** is selected from the Synonym menu.

2. Right-click the dimension and select **Parent/Child** from the Add Hierarchy submenu.
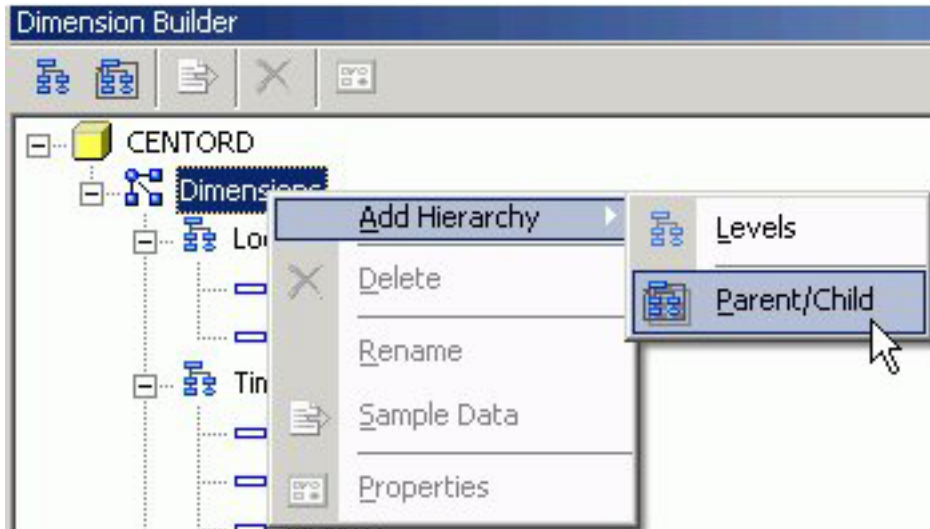
Note: You may also select this option by clicking the **Add Parent/Child Hierarchy** button from the Dimension Builder toolbar.

The Mandatory Properties for Parent/Child Hierarchy dialog box appears.

3. Select a field from the Field Tree and click **Assign** to assign a Unique ID for the hierarchy.
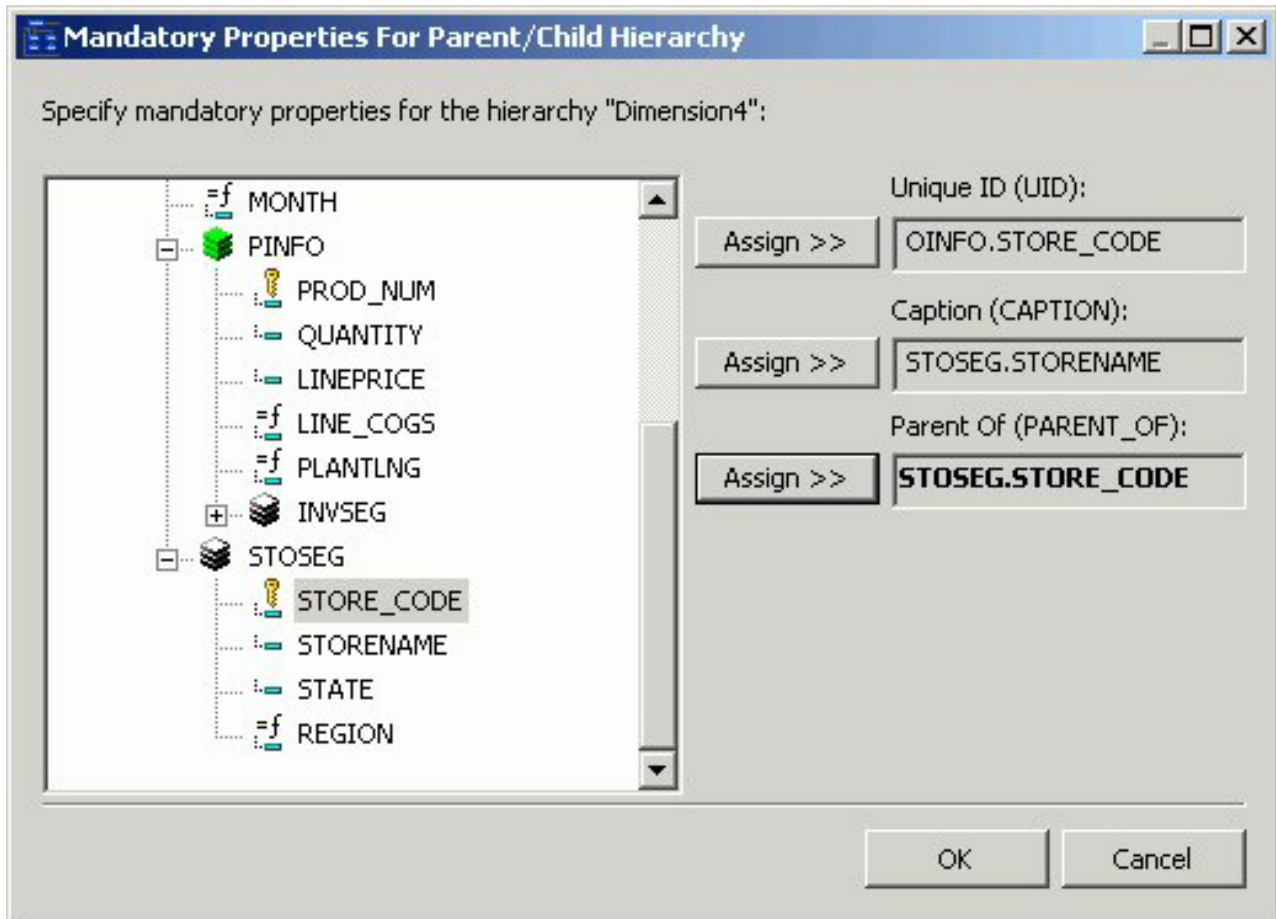
4. Repeat Step 3 for the Caption and Parent hierarchy properties.

5. Required: Click **OK** to close the Mandatory Properties for Parent/Child Hierarchy dialog box.

   The Mandatory Properties are added to the Dimension Builder.
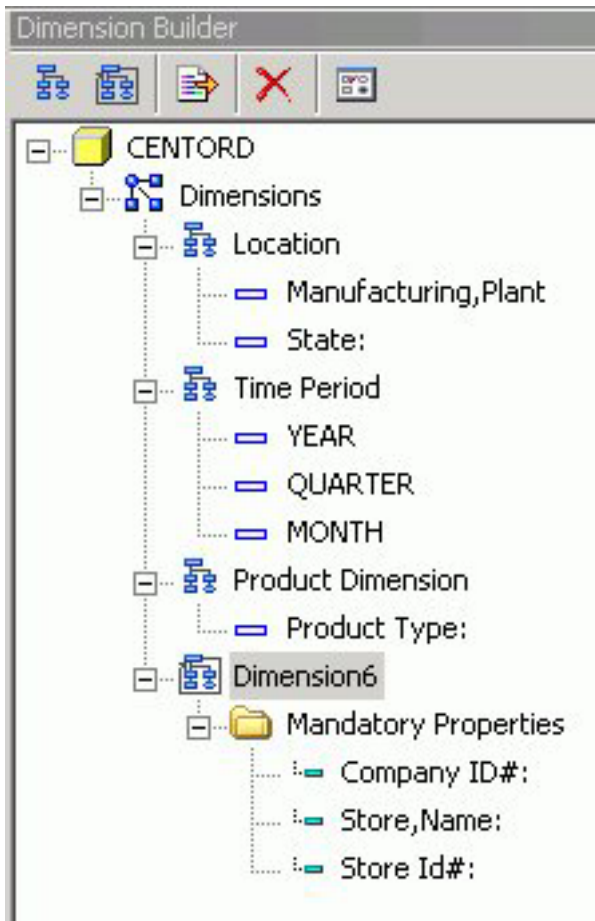
6. Use the right-click menu to rename the dimension, view properties, or view sample data for the dimension.
7. Click **Save** from the File menu to save the dimension.

Delete a Dimension

To delete a dimension, right-click a dimension and select **Delete**, or click the **Delete** button from the Dimension Builder toolbar.

Add Levels to the Hierarchy

This process enables you to edit an existing Master File, add tables (for non-FOCUS Master Files), and create and modify dimensions.

1. From the Synonym Editor, click the **Modeling View** tab.
2. Select the **Add Levels Hierarchy** button from the Dimension Builder toolbar.

   A level is added to the hierarchy. Use the right-click menu to rename the dimension or keep the default name.
3. Right-click the **file** icon in the Modeling View to toggle to the table view, which enables you to see columns, sample data, and sample data parent keys.
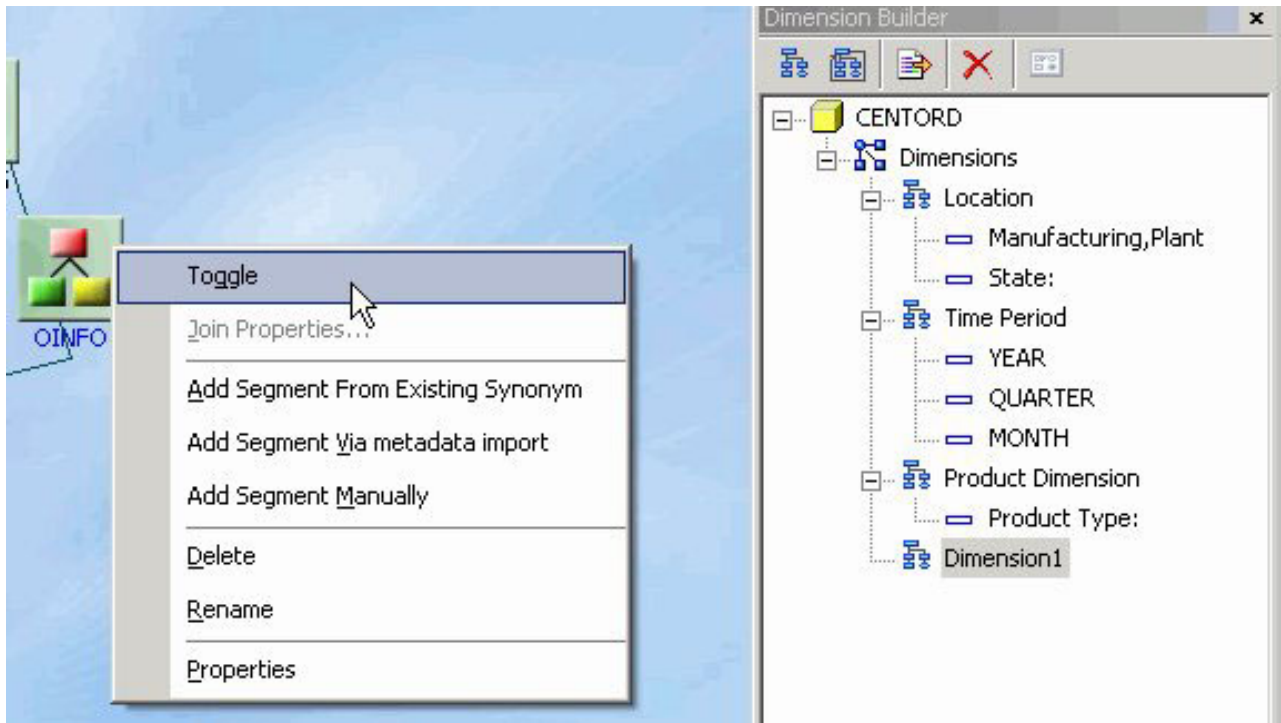
The table view appears in the Modeling View.

Tip: You may also double-click a file icon to open the table view.

4. Using the Columns tab of the table view, drag and drop selected fields to the Dimension Builder hierarchy folder.
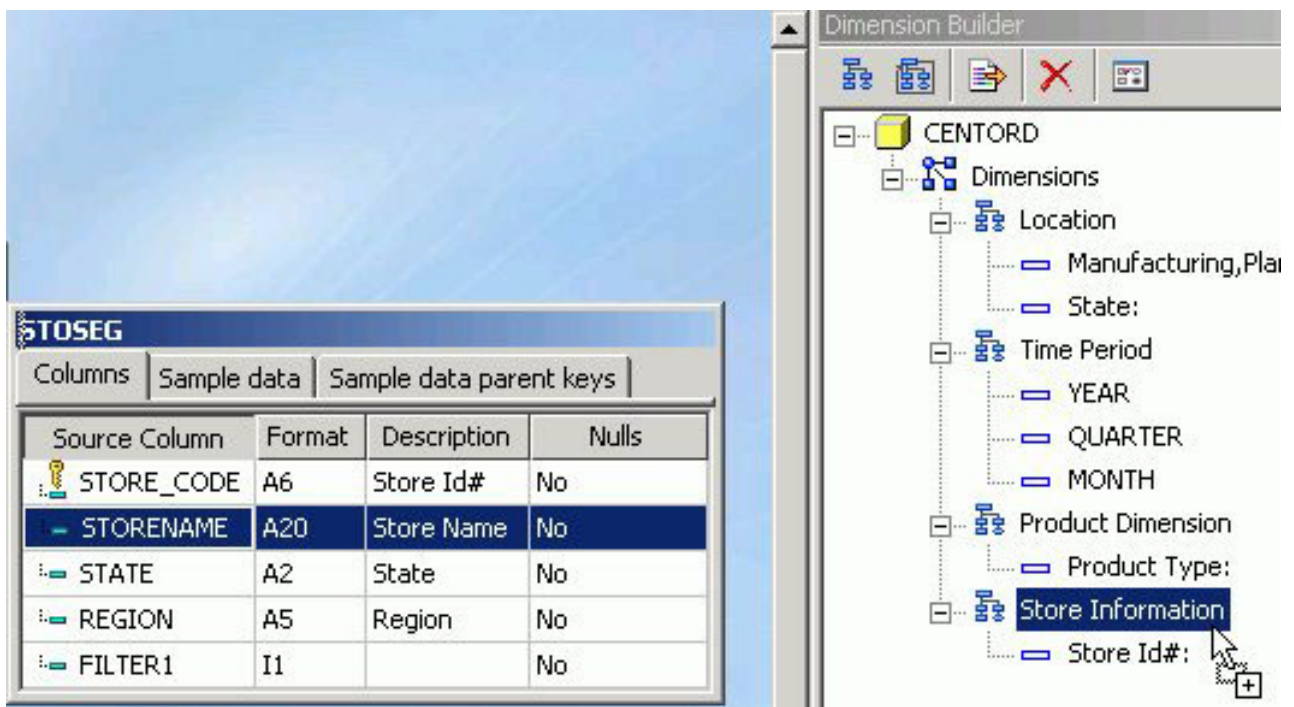
You may also drag and drop fields to and from other hierarchies using the Dimension Builder.

5. Click **Save** from the File menu to save the dimension.

   The dimension is saved and store in the Master File.

Dimension Builder Toolbar

You can access the following commands from the Dimension Builder toolbar:

Button	Definition
	Adds a level hierarchy.
	Enables you to create a parent/child hierarchy and assign mandatory properties for the hierarchy.
	Deletes the selected item.
	Enables you to view and refresh sample data for the selected field.
	Enables you to switch back to properties in the Tree View tab of the Synonym Editor.

Creating Business Views

By defining a Business View of a Master File, you can limit the fields available or create a subset of fields from the original Master File. Fields can be grouped into meaningful folders and field names, titles, and descriptions can be customized for each Business View. The Business View points to its underlying Master File, and all of the actual field information comes from the original Master File when the Business View is used in DB2 Web Query tools (such as Joins and Defines) and in reports.

Fields in a Business View are organized into folders. Each folder contains a group of fields. The fields in a folder can come from different segments in the original Master File. The Business View may contain existing fields and can include existing custom fields (DEFINEs), COMPUTES, and Filters. Custom fields are associated with a specific segment in the original Master File and are subject to the same rules as real fields. A report can reference fields from multiple folders if they all lie along a single path in the original Master File.

The Business View Master File is stored as if it were a real Master File. However, it has no SUFFIX attribute. Instead, it contains a VIEW_OF attribute that identifies it as a Business View. It can be saved to a different application from the original Master File.

Note: Impact Analysis searches Business Views in addition to FOCUS procedures. This enables you to see if changes in the original Master File will impact fields used in the Business View. For detailed information on Impact Analysis, see Chapter 15, "Analyzing Metadata and Procedures," on page 673.

You may create a Business View for an existing Master File by using the Synonym Editor.

Create a Business View Using the Synonym Editor

1. From the Data Servers area, navigate to the Master Files folder where you wish to create the Business View and highlight the Master File to be altered.

   Note: You may create a Business View anywhere where you can select a Master File.

2. Required: Double-click the Master File or select **Edit in Synonym Editor** from the File menu.

   The Master File opens to the Tree View tab in the Synonym Editor.

3. Select **Create Business View** from the Synonym menu.

   The Business View window opens to the Business View Tree tab.

   A default File name and folder appears. Additional folders may be created for grouping fields.

   Note: A Business View Master File may contain only one root folder.
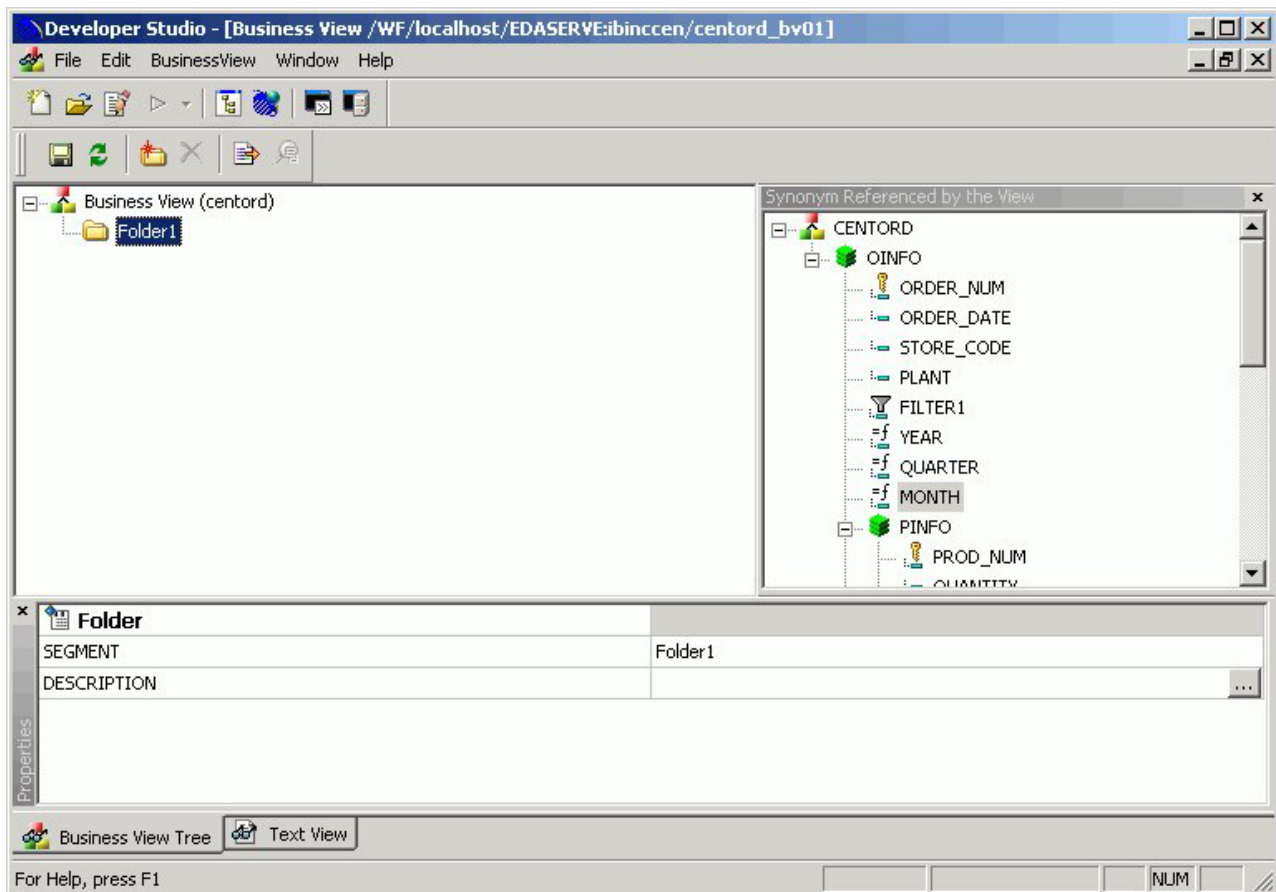
The Properties Bar on the bottom of the Business View window shows information for the selected fields and folders.

Tip: Use the Properties bar to change the title and descriptions for the field names. Items that cannot be edited are grayed out.

4. To add additional folders for the Business View, select **New Folder** from the BusinessView menu.

   Note: Multiple sub-folders can be created and folder may be empty for organizational purposes.

5. Required: Select fields from the Master File listed on the right and drag to the appropriate folder on the left. Use Shift-click and Ctrl-click to select multiple fields.

Note: If needed, fields may be duplicated and placed under multiple folders.

The selected fields appear in the Business View Tree tab.
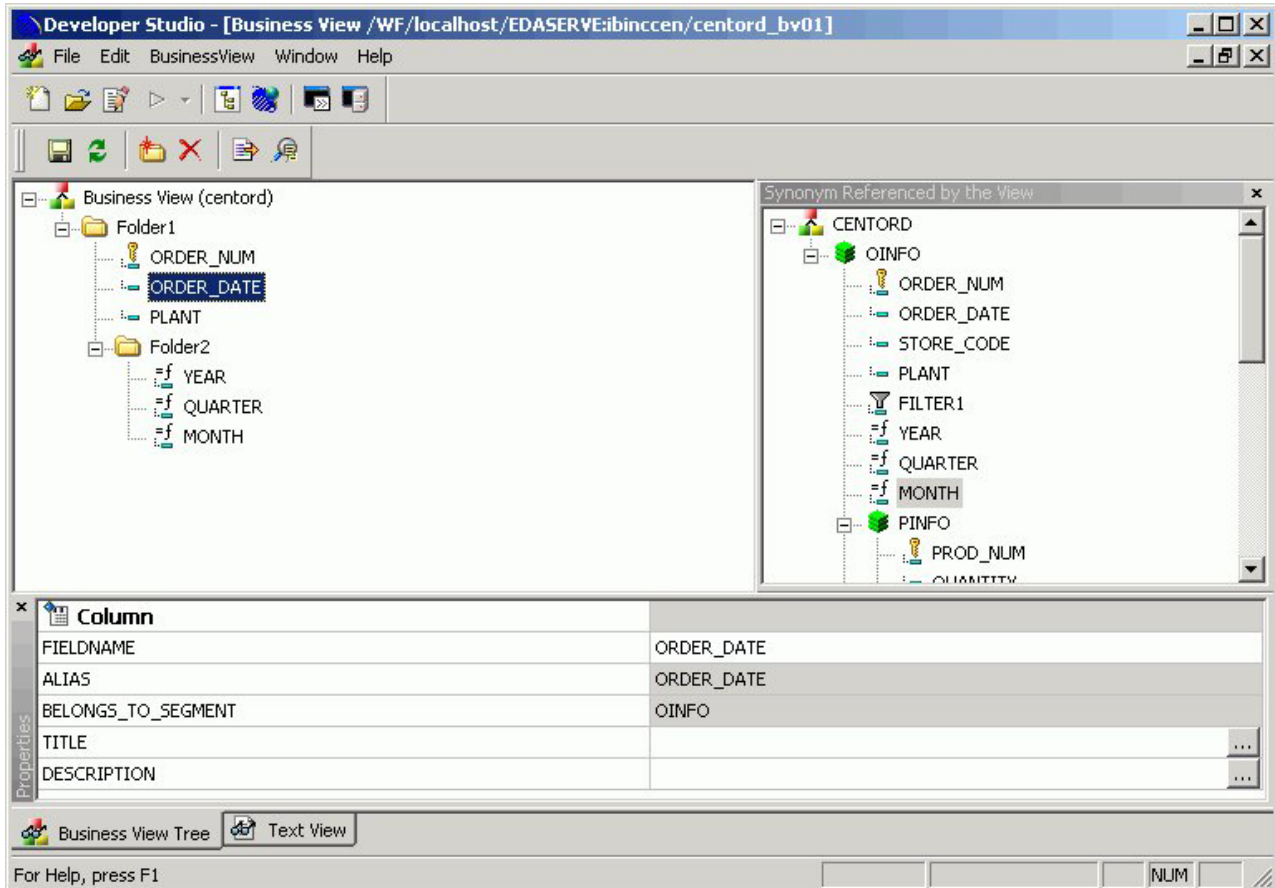
6. Select **Save As** from the File menu to save the Business View as a Business View Master file.

Note: The Business View Master File may be saved in a different application then the main files.

7. Select **Close** from the File menu to return to the Synonym Editor.

Business View Menu Options

The following menu items are available from the Business View window.
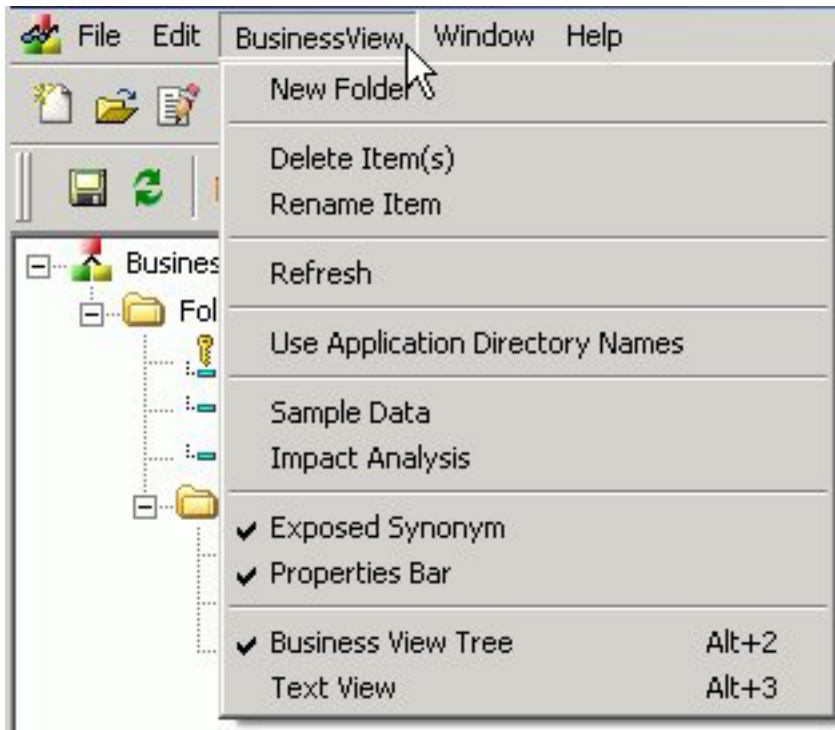
New Folder

> Adds a new folder to the Business View. You may also access this option
> from the right-click context menu and from the new folder icon, located on
> the Business View toolbar.

Delete Items

> Deletes the selected item.

Rename Item

> Enables you to rename a selected field or folder. You may also rename
> items from the Properties bar and the from the right-click context menu.

Refresh

> Refreshes the Business View. A Refresh icon is also available on the
> Business View toolbar.

Use Application Directory Names

> Specifies the application name where the original file exists. Information is
> written in the new master file. The following image is an example of the
> application directory name for the business view.
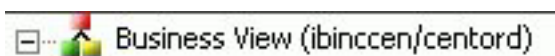
> **Note:** This option is turned off by default and is only available during the
> creation of the business view.

Sample Data

> Test the business view for field data and view the type of records returned.

Impact Analysis

Opens the Impact Analysis tool which enables you to analyze data, control search criteria, save reports, and interactively open and edit procedures based on search results. For detailed information on Impact Analysis, see Chapter 15, "Analyzing Metadata and Procedures," on page 673.

Exposed Synonym

Shows the Synonym referenced by the Business View.

Properties Bar

Shows the Properties Bar where you may edit properties for the selected fields and folders. Items that cannot be edited are grayed out.

Note: The Properties Bar may be moved or docked anywhere on the Business View window.

Business View Tree

Shows the contents of the Business View in the Business View Tree. This is the default option.

Text View

Shows the contents of the Business View in the Text View. You may also select this option by clicking the Text View tab from the bottom of the Business View window.

Note: The text view is read-only. You cannot edit the underlying description from this tab.

Usage Notes for Business Views

- The detailed information about fields, such as USAGE and ACTUAL formats or indexes remain in the original Master File.
- All information about Cluster Master Files remain in the original Master File.
- When a Master File description contains more than one field with the same name, as can occur when files are joined, the BELONGS_TO_SEGMENT attribute identifies which instance of the field name is being referenced in the Business View.
- Folders can be empty for organizational purposed. For example, Region can have empty folders called North, South, East, and West.
- You can issue an SQL SELECT command against a Business View. However, a Direct SQL Passthru request is not supported against a Business View.
- Business Views support alternate file views and fully qualified field names.
- The SEG. operator against a Business View folder displays all of the fields in that folder, not all of the fields in the real segment.
- Requests against a Business View cannot reference any fields or segments not in the Business View.
- All HOLD formats are supported against a Business View.
- All adapters for non-FOCUS data sources support retrieval requests against a Business View.
- Business Views are not supported with data source maintenance commands such as Maintain or REBUILD.
- Business Views are not available for Cube Data Sources.
- For OLAP enabled Master Files, dimensions are not visible while creating a business view. Only fields from the original Master File will be visible.

- When using Business Views, the following SET command must be specified: SET FOCTRANSFORM=ON. This SET command can be placed in a procedure or can be set in the global server profile or specified in a user profile and can be set while using the Business View or regular Master Files. For thin client tools (Power Painter, HTML tools, and so on), it needs to be in the profile as end-users do not have access to the SET tool or ability to access an editor and manually enter the required command.

  Adding this SET command in the profile is a good practice as it will avoid errors during reporting in case command is missing and users will not need to be concerned about what type of file they are using during reporting (Business View or existing master file).

Using a Business View Master File

When you use the Business View Master File in the DB2 Web Query tools and reports, the Field formats, descriptions and titles will be retrieved from the original Master File, unless they are customized and a title and description is available through the Business View.

You may access the Business View Master Files from all development areas of Developer Workbench.

The Table list shows all available Master Files including the Business Views that are available. The Remarks column shows a description from original Master File unless the Business View file contains it own remarks.

The following image is an example of a Business View Master File in the DB2 Web Query Table List, that appears when creating a report in the Data Servers area.
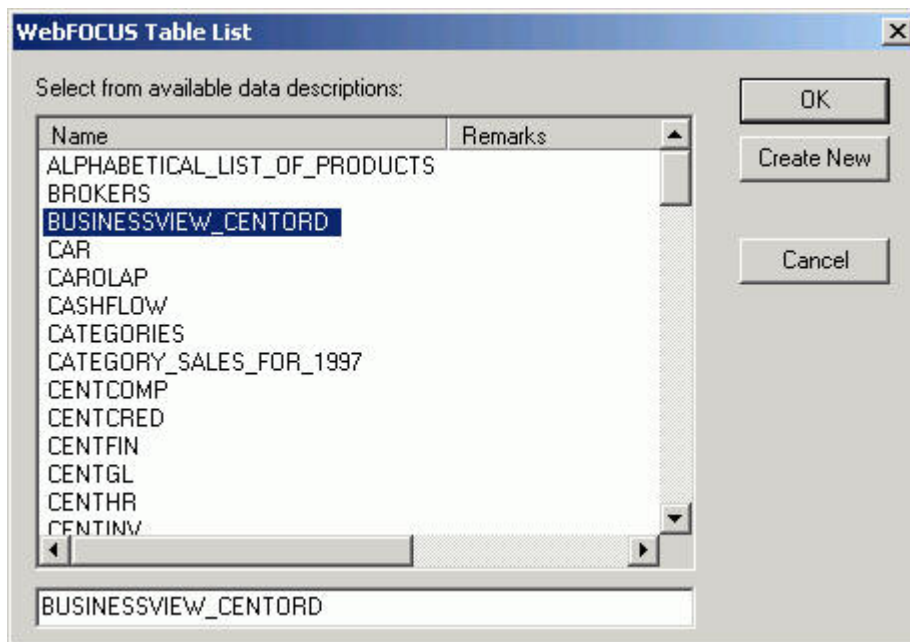
In the Report Assistant, when using a Business View Master File with the Report Assistant, only fields from the Business View are shown in the Fields list.

The following image is an example of a Business View Master File in the Report Assistant.
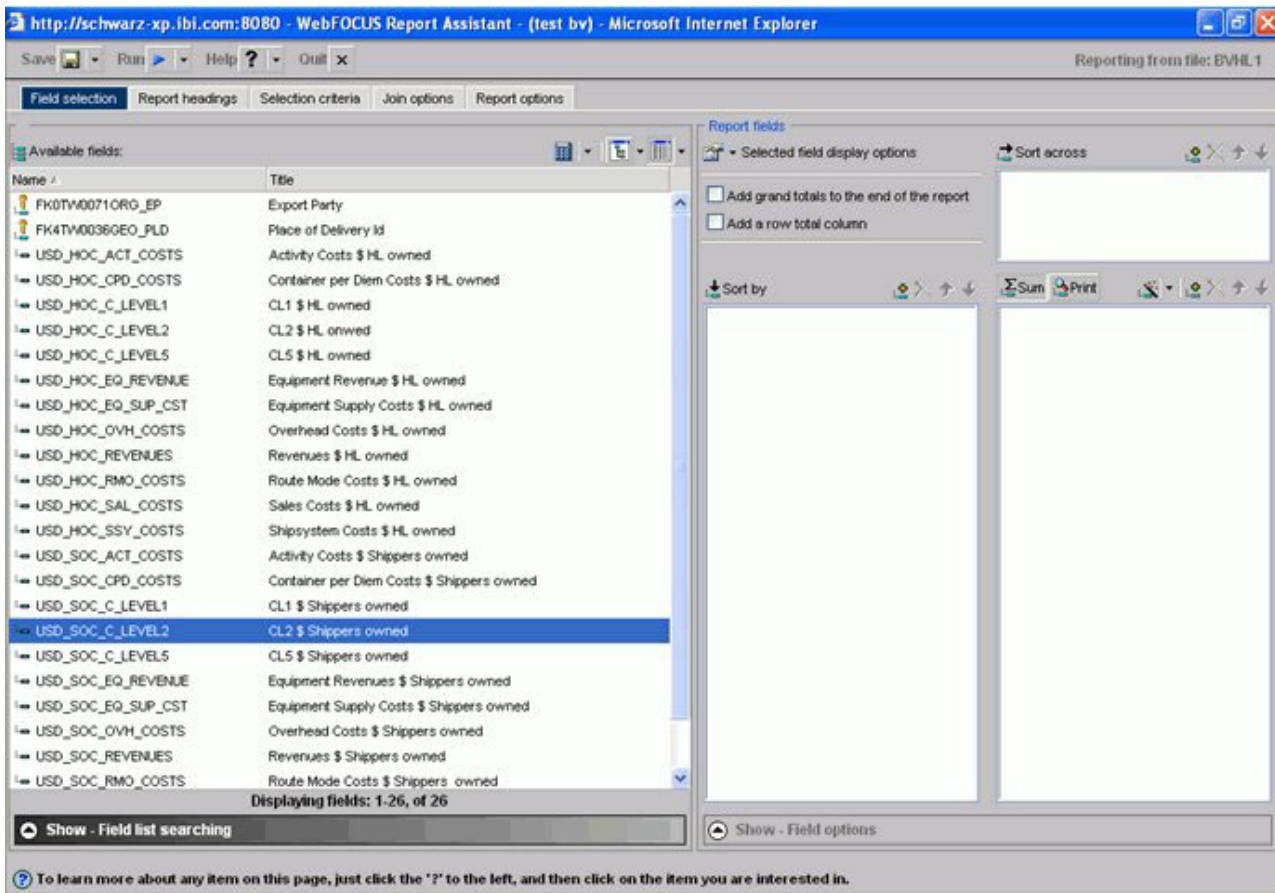
Adding Virtual Columns (DEFINE) in a Synonym

You may create a DEFINE field as a custom field in the Synonym Editor. A custom field can be used in a request as though it is a real data source field. Virtual columns (DEFINES) are available when the data source is used for reporting.

A virtual column can contain an expression, a constant, or a column name.
- If the virtual column is a complex expression, you can create the expression with the Virtual Column Calculator or just type it into the Expression field.
- If the virtual column is a simple expression, like a constant value, you can type the value in the Expression field.

Virtual columns are designated by the following icon:

=f

After creating a virtual column, you can test it by right-clicking the synonym and selecting **Sample Data**. Sample Data appears in a separate dialog box.

Create a Virtual Column in a Synonym

To create a virtual column in a synonym:

1. From the Data Servers area, open a synonym by double-clicking a Master File from the Master Files folder.

   The Synonym Editor opens.

2. Right-click a segment (or column) and click **Add**.

3. Select **Virtual Column** from the Add submenu.

   The Virtual Column Calculator opens.
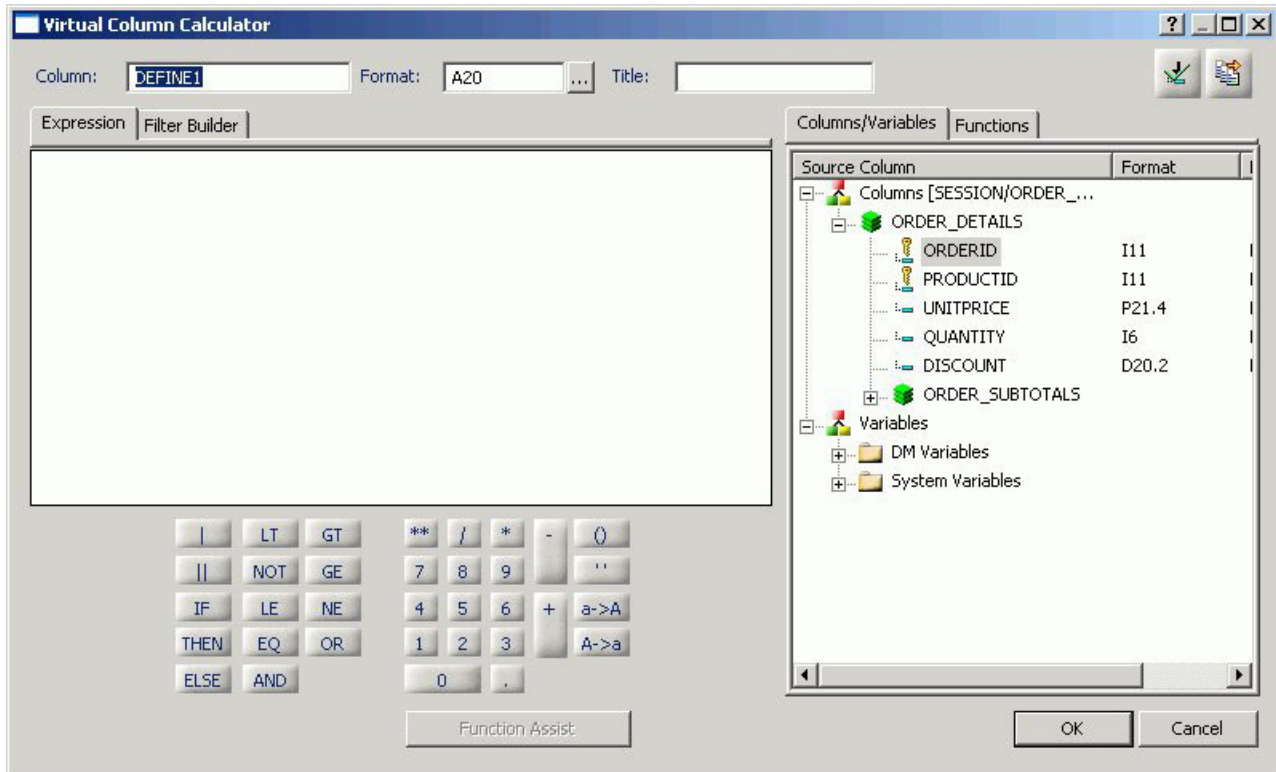
4. From the Virtual Column Calculator, type a name for the column in the Column input field, or use the default define name.

5. Required: You may enter a descriptive title for the virtual column (DEFINE) in the Title input field.

   Tip: From the Synonym Editor, click the browse **(...)** button at the right of the TITLE and DESCRIPTION value fields to specify multiple language titles.

6. Use the Expression tab and the calculator buttons to build the expression for the virtual column (DEFINE).

   or

   Use the Filter Builder tab to build the expression.

   a. From the Filter Builder tab, use the drop-down lists to select the filter Column, Relation, and Type.

      Note: Parameters are not supported.

   b. Click the browse **(...)** button at the right of the Value input field.

The Value Selection dialog box opens.

   c. Select from the available value(s) and use the arrows to add or remove values.

   d. Click **OK** to close the Value Selection dialog box and return to the Filter Builder tab.

      The expression is added to the value field.

Column: FILTER1   Format: I1   ...   Title:

Expression | Filter Builder

	Column	Relation	Type	Value
✕	REGION ▼	EQ ▼	Value ▼	'CENTRAL AMERICA' OR 'SOUTH AMERICA' ...

Columns/Variables

Source Column	Format
Columns [IBISAMP/TRADES]	
TRADES	
TRADER_ID	A5
DATE_OF_TRADE	MDYY
CONTINENT	A10
REGION	A15
COUNTRY	A15
HOLDER	A4
TYPE	A9
BUY_SELL	A1
AMOUNT	D16
Variables	
DM Variables	
System Variables	

OK   Cancel

e. To add another filter, double-click a column or variable from the Columns/Variables tab on the right side of the Virtual Column Calculator. The filter is added to the Filter Builder. Add the expression value.

f. To delete an expression from the Filter Builder, click the red X in the row of the filter that you are deleting. The filter is removed from the Filter Builder.

7. You may select the Check expression and Sample Data buttons, located on the top right of the Virtual Column Calculator, to verify that the expression is valid and to view sample data for the filter.

8. Click **OK** to close the Virtual Column Calculator and return to the Synonym Editor.

   Note: To edit the DEFINE, Title, or Expression, you may do so directly from the Synonym Editor or you may click the browse **(...)** button at the right of the EXPRESSION value field to relaunch the Virtual Column Calculator.

9. If no columns from the synonym are used in the expression or have been defined, you can use the WITH option to identify the logical home of the defined calculation. You can also use the WITH option to move the logical home for the virtual column to a lower segment than it would otherwise be assigned to (for example, to count instances in a lower segment).

   Tip: You can click and drag the DEFINE field and move it to a different segment in the Tree View tab, which also changes the segment association.

10. Specify the Missing Data options for columns that allow null data. You can allow all missing data.

11. Click **Save** from the File menu to save the synonym.

12. To close the Synonym Editor, select **Close** from the File menu or click the control button in the upper right corner.

For more information about expressions and virtual column (DEFINE) attributes, see Defining Attributes and Creating Expressions for Custom Fields.

Creating Filters in a Synonym

Filters are created in the Master File through the Synonym Editor and can be used in a Business View file or in reporting tools such as Power Painter, HTML Report Assistant, and so on. You can also use filters to perform other data checking and validation, and sort data based on the conditions that you create.

Filters are created under a specific segment and by default they have association with the selected segment. Filters can also be created without segment association.

Create Filters in a Synonym

1. Required: From the Data Servers area, double-click a synonym from the Master Files folder, or right-click the synonym and select **Edit in Synonym Editor**.

   The Synonym Editor opens.

2. Required: Right-click a segment or field and select **Filter** from the Add submenu. The Filter Calculator opens.
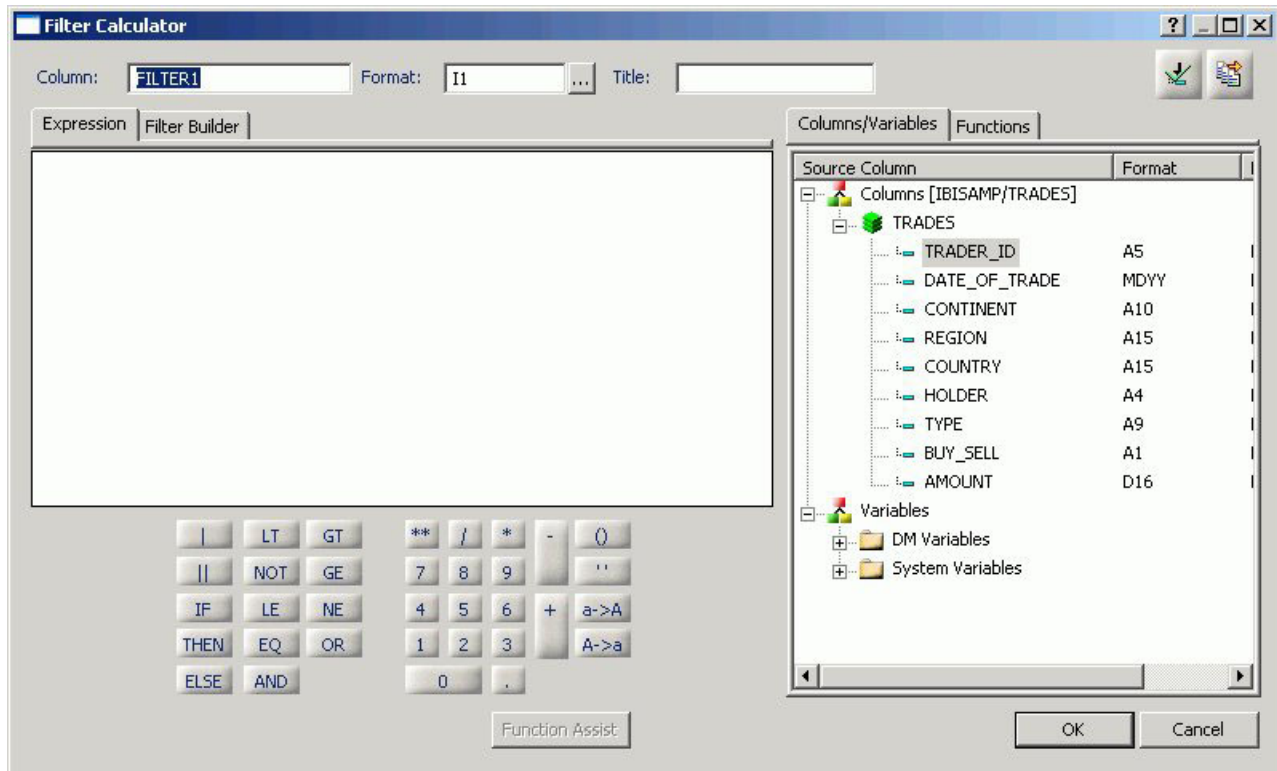
3. From the Filter Calculator, type a name for the filter in the Column input field, or use the default Filter name.

   Note: It is recommended that filters have a descriptive name to help identify the filter action during reporting.

4. The Format field shows a default value of I1.

   Note: The format field cannot be changed. Values for filters return 0 for false and 1 for true.

5. You may enter a descriptive title for the filter in the Title input field.

Tip: From the Synonym Editor, click the browse **(...)** button at the right of the
TITLE and DESCRIPTION value fields to specify multiple language titles.

6. Use the Expression tab and the calculator buttons to build the expression for
   the filter. or Use the Filter Builder tab to build the expression.

   a. From the Filter Builder tab, use the drop-down lists to select the filter
      Column, Relation, and Type.

      Note: Parameters are not supported with Master File Filters.

   b. Click the browse **(...)** button at the right of the Value input field.

      The Value Selection dialog box opens.

   c. Select from the available value(s) and use the arrows to add or remove
      values.

   d. Click **OK** to close the Value Selection dialog box and return to the Filter
      Builder tab. The expression is added to the value field.

	Column	Relation	Type	Value	
✕	REGION ▼	EQ ▼	Value ▼	'CENTRAL AMERICA' OR 'SOUTH AMERICA'	...

Column: FILTER1   Format: I1 ... Title:

Expression Filter Builder

Columns/Variables

Source Column	Format	
Columns [IBISAMP/TRADES]		
TRADES		
TRADER_ID	A5	
DATE_OF_TRADE	MDYY	
CONTINENT	A10	
REGION	A15	
COUNTRY	A15	
HOLDER	A4	
TYPE	A9	
BUY_SELL	A1	
AMOUNT	D16	
Variables		
DM Variables		
System Variables		

OK   Cancel
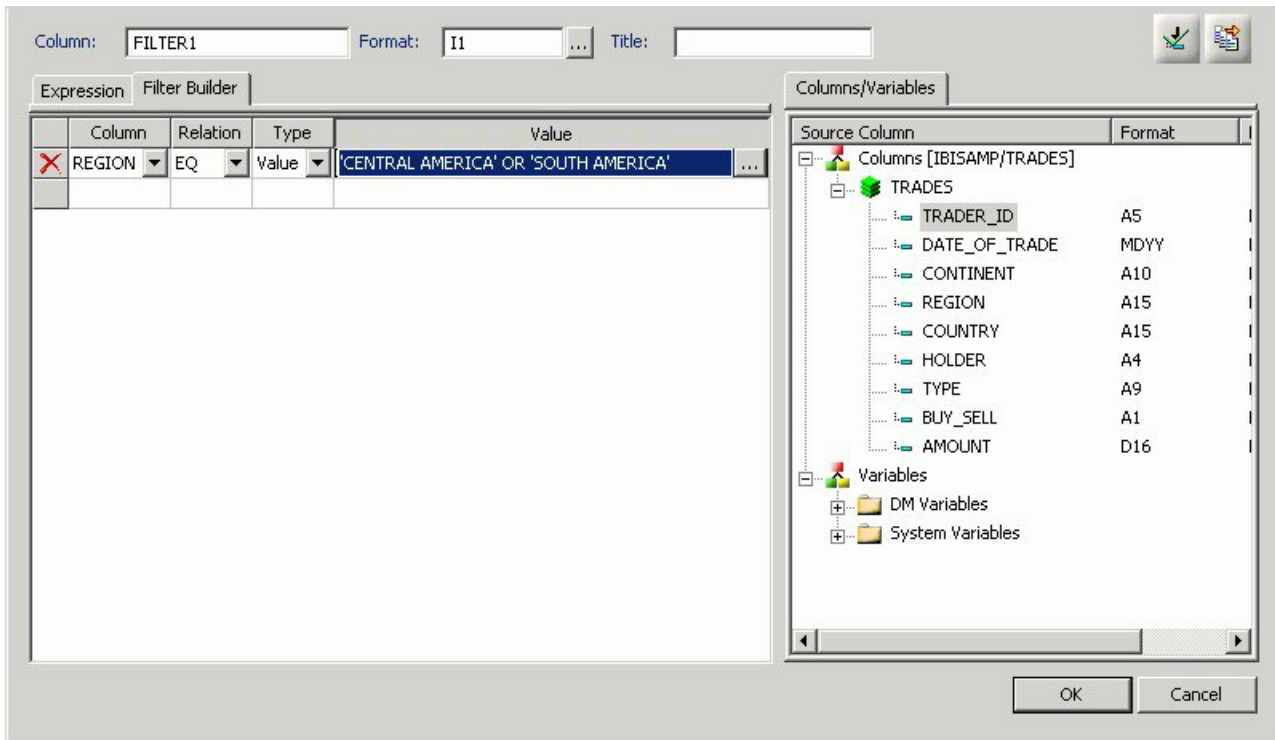
e. To add another filter, double-click a column or variable from the Columns/Variables tab on the right side of the Filter Calculator. The filter is added to the Filter Builder. Add the expression value.

f. To delete an expression from the Filter Builder, click the red X in the row of the filter that you are deleting. The filter is removed from the Filter Builder.

7. You may select the Check expression and Sample Data buttons, located on the top right of the Filter Calculator, to verify that the expression is valid and to view sample data for the filter.

8. Click **OK** to close the Filter Calculator dialog box to return to the Synonym Editor.

   Note: To edit the Filter, Title, or Expression, you may do so directly from the Synonym Editor or you may click the browse **(...)** button at the right of the EXPRESSION value field to relaunch the Filter Calculator.

9. Required: To create a Filter without segment association, use the WITH drop-down list to select a blank segment.

   Note: Filters are created under a specific segment and by default they have association with the selected segment. Filters can also be created without segment association.

> **Note:** If using a field that appears in multiple segments, the WITH segment associated should be the lowest level segment, or it should be left empty to prevent errors.
>
> In addition, If no fields from the synonym are used in the expression or have not been computed, you can use the WITH option to identify the logical home of the Filter calculation. You can also use the WITH option to move the logical home for the Filter field to a lower segment than it would otherwise be assigned to (for example, to count instances in a lower segment).

10. Click **Save** from the File menu to save the synonym. The filter is saved as part of the synonym.
11. To close the Synonym Editor, select **Close** from the File menu or click the control button in the upper right corner.

Adding Computed Fields (COMPUTE) in a Synonym

You may create a Computed field as a custom field in the Synonym Editor. The procedure for adding a custom field is similar to the procedure used to add a Defined field. The Computed field is identified as the Master File Computed Field and is differentiated from the Defined fields and the other Computed Fields.

Note: Computed fields from the Master File appear in the Windows versions of the Graph Assistant and in the Report Assistant fields list.

Create a Computed Field in the Synonym Editor

To create a computed column in a synonym:

1. Required: From the Data Servers area, open a synonym by double-clicking a Master File from the Master Files folder.

   The Synonym Editor opens.
2. Required: Right-click a segment (or column) and click **Add**.

3. Required: Select **Compute** from the Add submenu.

   The Compute Calculator opens.
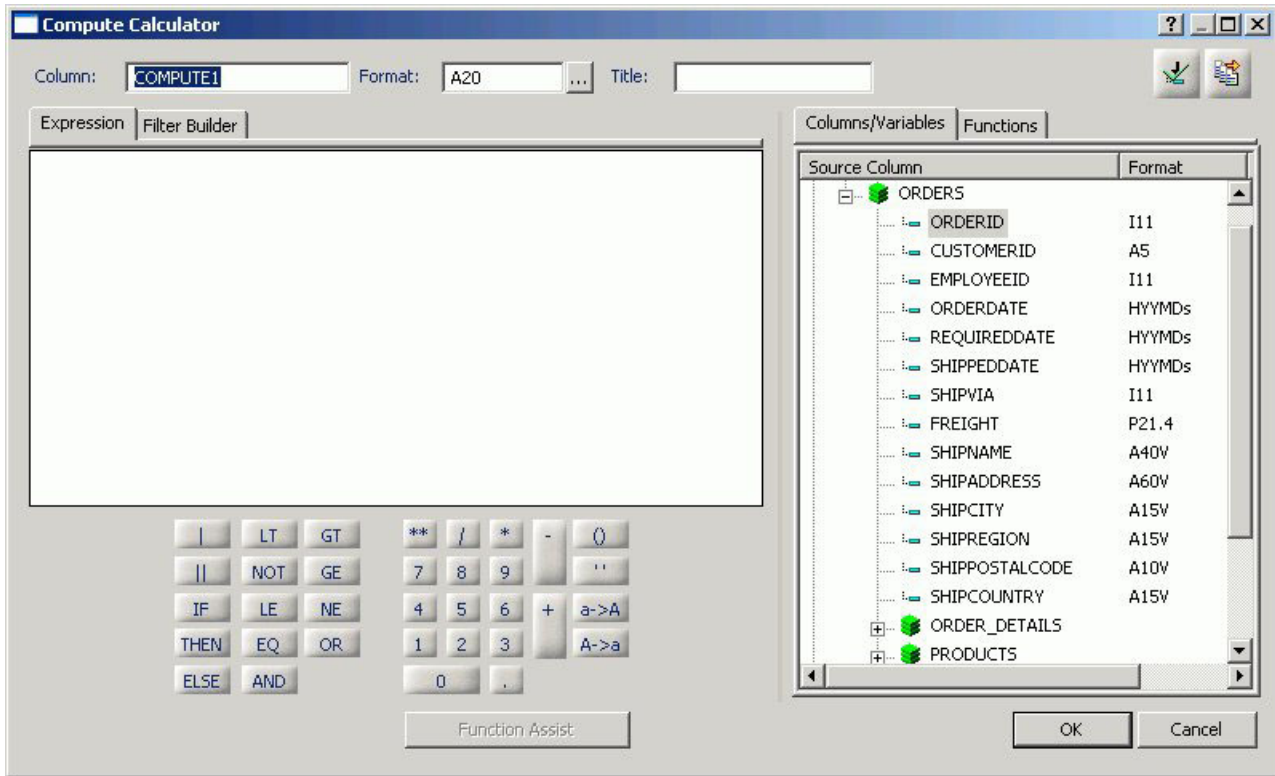
4. From the Compute Calculator, type a name for the column in the Column input field, or use the default compute name.

5. You may enter a descriptive title for the computed field (COMPUTE) in the Title input field.

   Tip: From the Synonym Editor, click the browse **(...)** button at the right of the TITLE and DESCRIPTION value fields to specify multiple language titles.

6. Use the Expression tab and the calculator buttons to build the expression for the computed field (COMPUTE). or

   Use the Filter Builder tab to build the expression.

   a. From the Filter Builder tab, use the drop-down lists to select the filter Column, Relation, and Type.

      Note: Parameters are not supported.

   b. Click the browse **(...)** button at the right of the Value input field.
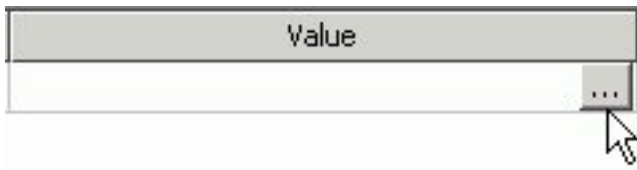
The Value Selection dialog box opens.
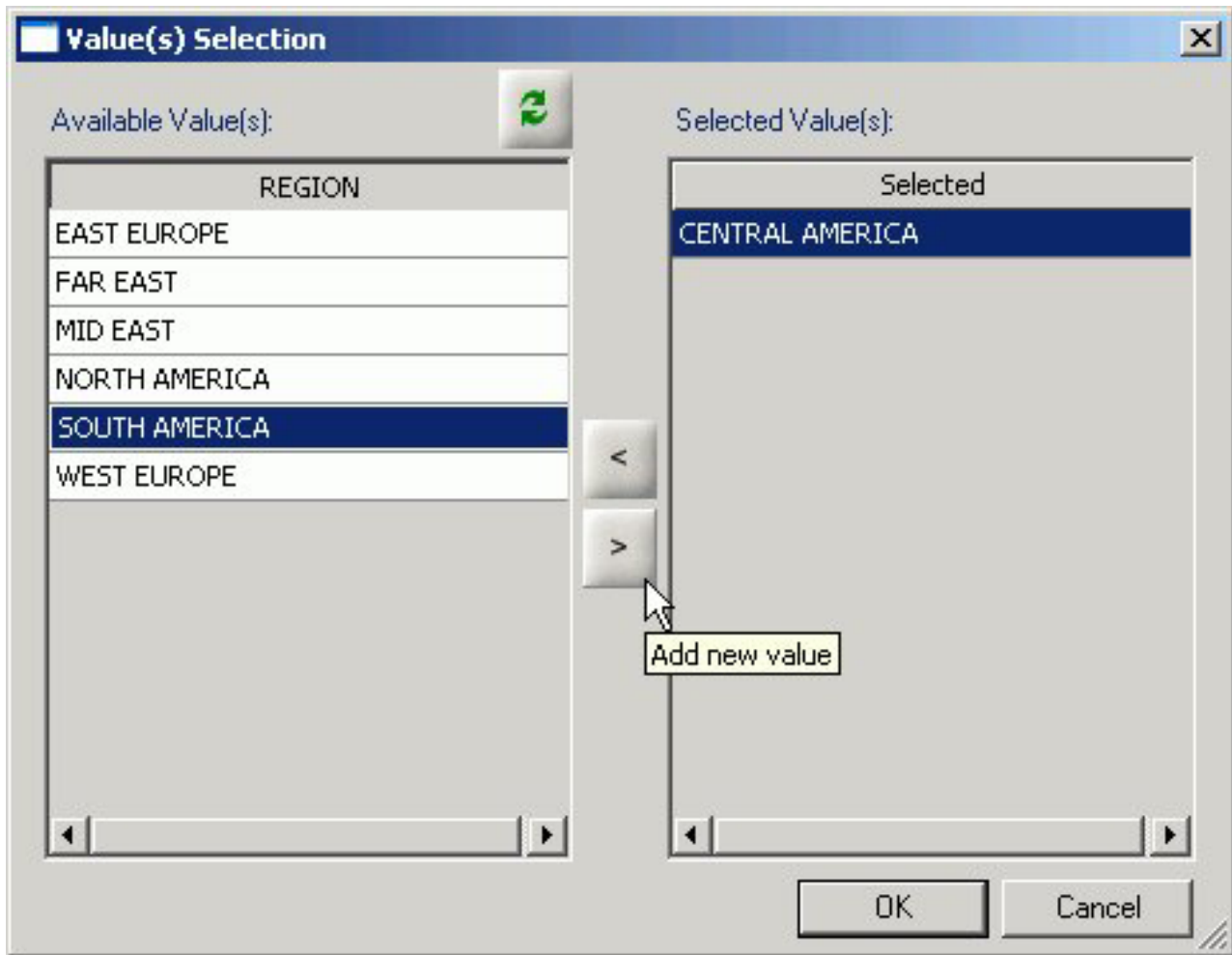
c. Select from the available value(s) and use the arrows to add or remove values.

d. Click **OK** to close the Value Selection dialog box and return to the Filter Builder tab.

The expression is added to the value field.

e. To add another filter, double-click a column or variable from the Columns/Variables tab on the right side of the Compute Calculator. The filter is added to the Filter Builder. Add the expression value.

f. To delete an expression from the Filter Builder, click the red X in the row of the filter that you are deleting. The filter is removed from the Filter Builder.

7. You may select the Check expression and Sample Data buttons, located on the top right of the Compute Calculator, to verify that the expression is valid and to view sample data for the filter.

8. Click **OK** to close the Compute Calculator and return to the Synonym Editor.

> **Note:** To edit the Compute, Title, or Expression, you may do so directly from the Synonym Editor or you may click the browse **(...)** button at the right of the EXPRESSION value field to relaunch the Compute Calculator.

9. Specify the Missing Data options for columns that allow null data. You can allow all missing data.

10. Click **Save** from the File menu to save the synonym.

11. To close the Synonym Editor, select **Close** from the File menu or click the control button in the upper right corner.

For more information about expressions and COMPUTE attributes, see Defining Attributes and Creating Expressions for Custom Fields.

Defining Attributes and Creating Expressions for Custom Fields

A custom field is a field whose value is not stored in the data source but can be calculated from the data that is there. You can create a custom field in your synonym by adding a virtual column (DEFINE), Master File filter (FILTER) and a Computed Field (COMPUTE). The fields are available whenever you access the corresponding data source in a reporting tool.

You can define attribute values and create expressions for custom fields by using the Synonym Editor.

Custom Field Attributes

The following attributes may be available for custom fields (DEFINE, FILTERS, and COMPUTE) in the Synonym Editor.

Note: The attributes available depend on the type of synonym and the type of custom field selected. The image below is an example of an SQL data source with the virtual column (DEFINE) selected.
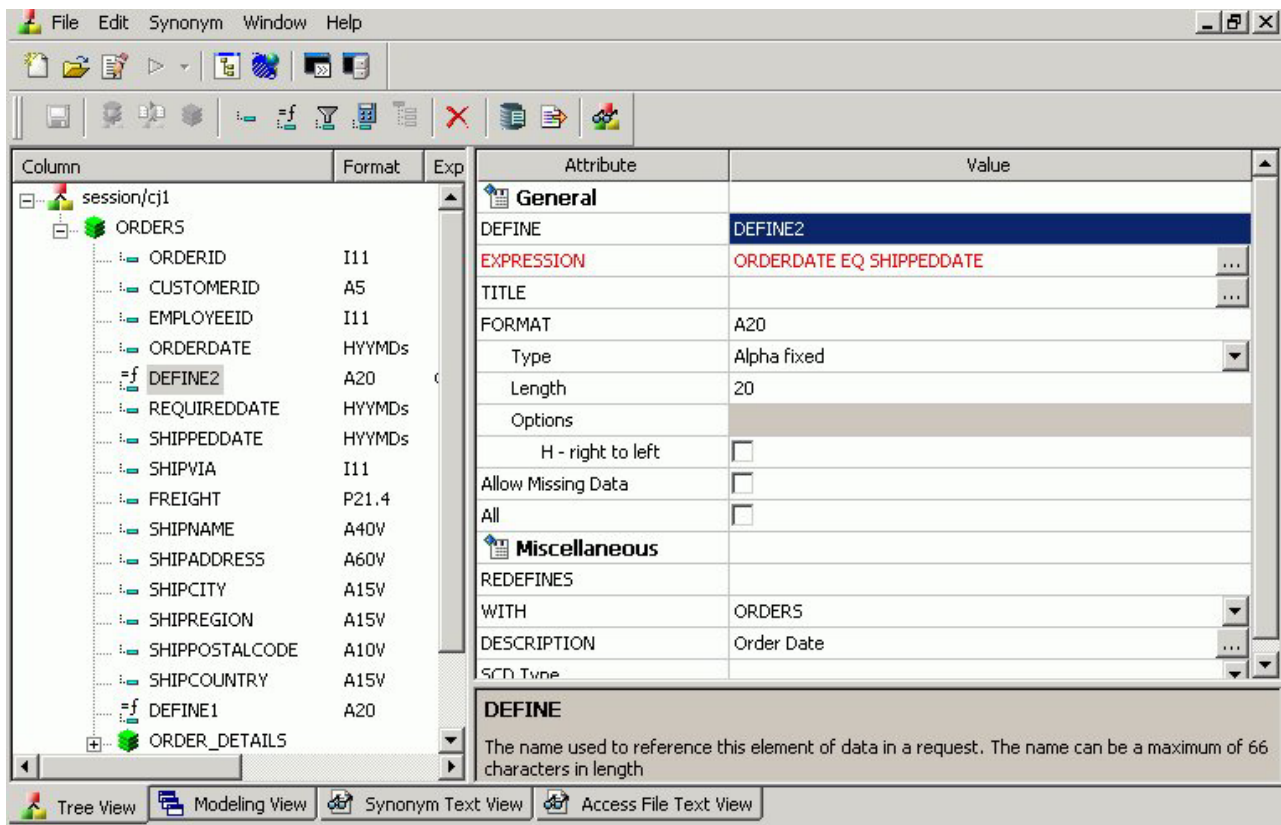
Custom fields (DEFINE, FILTER, COMPUTE) typically have the following attributes:

General

> **DEFINE**
>
> > Is the name of the virtual column.
> >
> > **Note:** This attribute only appears when a virtual column (DEFINE) is selected.
>
> **FILTER**
>
> > Is the name of the Master File filter field.
> >
> > **Note:** This attribute only appears when a virtual Filter field is selected.

COMPUTE

> Is the name of the computed field.
>
> **Note:** This attribute only appears when a virtual computed field is selected.

EXPRESSION

> Is the calculation that creates the virtual column.

TITLE

> Supplies a title to replace the column name that is normally used in reports and enables you to specify multiple language titles for the virtual column.

FORMAT

> Describes the data type and format for the virtual column.
>
> **Note:** This attribute only appears for DEFINE and COMPUTE custom fields.

Allow Missing Data

> Allows missing data if not, transaction value is supplied.
>
> **Note:** This attribute only appears for DEFINE and COMPUTE custom fields.

All

> Allows all missing data if not transaction value is supplied.
>
> **Note:** This attribute only appears for DEFINE and COMPUTE custom fields.

Miscellaneous

WITH

> If no columns from the synonym are used in the expression or have been defined, you can use the WITH option to identify the logical home of the defined calculation. You can also use the WITH option to move the logical home for the virtual column to a lower segment than it would otherwise be assigned to (for example, to count instances in a lower segment).
>
> **Note:** This attribute only appears for DEFINE and FILTER custom fields.

DESCRIPTION

> Contains a description or comments about the virtual column.

WITHIN

> Contains the name of a field to be included in a dimension.
>
> These WITHIN statements are added to the synonym through the Dimension Builder to OLAP enable FOCUS files and relational tables. This enables you to perform OLAP analysis using the OLAP Control Panel, or to use it with the FML Painter.
>
> **Note:** This attribute only appears for DEFINE and FILTER custom fields.

SCD Type

Sets slowly changing dimensions. This option is only available for existing relational targets.

A **surrogate key** is the first column in the table and has an SCD type of **blank**. Other columns with a blank SCD type have no SCD processing done to them.

Logical Key Field is the database key.

Activation Flag indicates that the row is current.

Begin Date/End Date indicates date range for the row values. A null end date indicates the row is current.

Type I (overwriting history) designates columns whose database values are overwritten with new values.

Type II (preserving history) designates columns that whose database rows are flagged as inactive or assigned an end date; new rows are inserted with the new values.

blank (non-key columns) indicates that database values are not changed.

Note: The attributes available depend on the type of synonym.

Calculators for Custom Fields

To launch the Virtual Column Calculator, Filter Calculator, or Compute Calculator, click the browse **(...)** button at the right of the EXPRESSION value field in the Synonym Editor.

The selected calculator opens., depending on the type of custom field that you are creating.
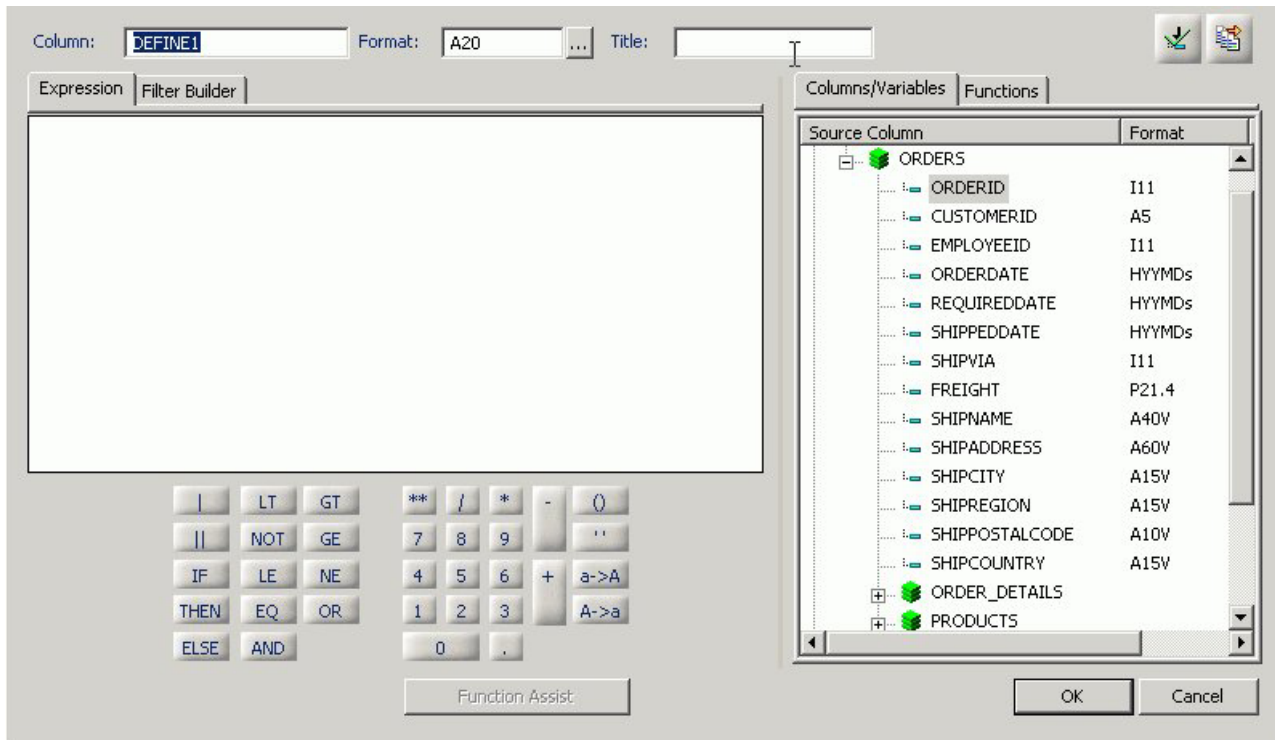
The calculator has the following fields/options:

Column

>   The name of the object being created (virtual field, filter, computed field).

Format

>   Is the fields format.

Expressions tab

>   Location for typing an expression. You can add data source fields from the Columns/Variables tab, functions from the Functions tab, and numbers and operators from the calculator as you type.

Filter Builder tab

>   Displays the filter building window from which you can add and delete columns, choose the relation, type, and select values for your filter.

Columns/Variables tab

>   Displays a hierarchical list of available source columns, and System Variables folders that you can use in creating an expression.

Functions tab

>   A function is a program that returns a value. This tab lists the built-in functions that you can use to derive the value of a temporary field.

Function Assist button

>   Enables you to specify parameters for the function through a dialog box when creating or editing a transformation.

Calculator buttons

>   Insert numbers and operators.

The following operators are available:

| (single concatenation bar)

Concatenates two values, retaining any trailing blanks after the first one. For example, if FIRST_NAME and LAST_NAME were both in A15 format, the expression

```
FULL_NAME = FIRST_NAME | LAST_NAME
```

would produce a column like the following:

```
MICHAEL      SMITHSONJANE        JONES
.
.
.
```

|| (double concatenation bar)

Concatenates two values, suppressing any trailing blanks in the first. For example, to construct the full name and insert a comma, the syntax

```
FULL_NAME = LAST_NAME || (', ' | FIRST_NAME)
```

would produce a column like the following:

```
SMITHSON, MICHAEL JONES, JANE
.
.
.
```

The concatenation in the parentheses is done first (preserving the blank space after the comma), and the result is then concatenated to LAST_NAME, suppressing the trailing blanks of LAST_NAME.

IF

Establishes a conditional test.

THEN

Specifies the action to perform if the result of a conditional test is TRUE.

ELSE

Specifies the action to perform if the result of a conditional test is FALSE.

LT

Returns the value TRUE if the value on the left is less than the value on the right.

NOT

Returns the value TRUE if the operand is false.

LE

Returns the value TRUE if the value on the left is less than or equal to the value on the right.

EQ

Returns the value TRUE if the value on the left is equal to the value on the right.

AND

Returns the value TRUE if both operands are true.

GT

Returns the value TRUE if the value on the left is greater than the value on the right.

GE

Returns the value TRUE if the value on the left is greater than or equal to the value on the right.

NE

Returns the value TRUE if the value on the left is not equal to the value on the right.

OR

Returns the value TRUE if either operand is true.

**

Raises a value to the specified power.

( )

Adds parentheses.

″

Inserts two single quotation marks. Enter alphanumeric test values between these.

a->A

Converts selected text to uppercase.

A->a

Converts selected text to lowercase.

Check expression button

Verifies the validity of the expression.

Sample data button

Produces sample data for the expression.

Adding Group Fields in a Synonym

For data sources that support groups, you can assign a unique name to multiple fields to create a group field. A group field is created by two or more alphanumeric fields, physically next to each other. A group field provides an efficient means for grouping similar or logically connected fields that will be accessed as a single unit, but do not warrant a separate segment.

Note: In the Maintain environment group fields are supported in a Master File as long as they are not group keys, that is groups that are actual fields, as supported by VSAM. If you create a group field in the Maintain environment, the group will not be visible, only the fields that make up the group will be visible.

Add a Group Field to a Segment

1. Required: From the Data Servers area, open a synonym by double-clicking a Master File from the Master Files folder.

The Synonym Editor opens.

2. Required: Right-click a segment (or column) and click **Add**.

3. Required: Select **Group** 🔳 from the Add submenu.

   Note: The groups option may not be available for some data sources.

   A group is added to the synonym and its attributes and values appear on the right.
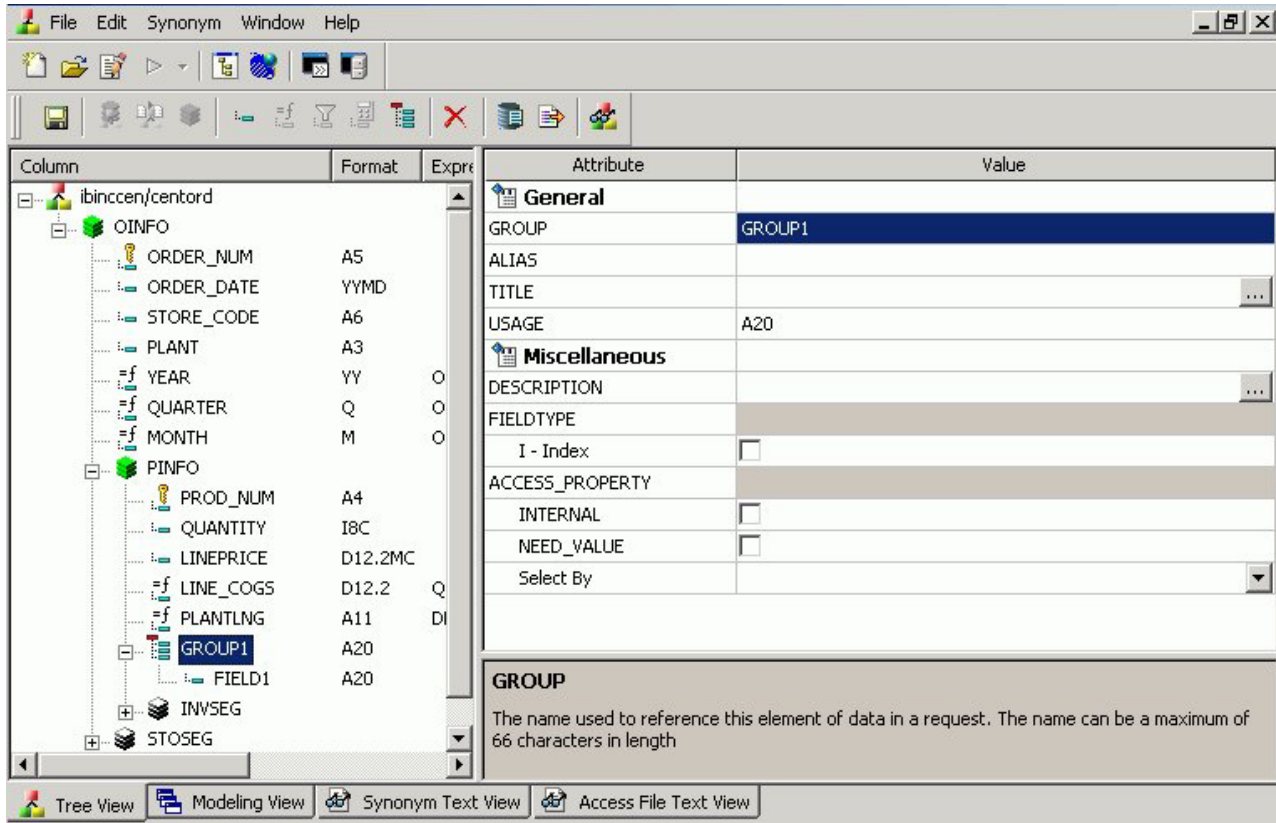
4. Type a name for the group in the **GROUP** field.

5. Select the USAGE **Type** value for the group. Specify the length, the decimal places (if applicable) and any display options for the group.

6. Click the **I - Index** box if you want the group to be indexed.

7. Select the ACCESS_PROPERTY values to specify access options for the group's data.

8. Optionally, you can specify the TITLE and DESCRIPTION display options.

9. Click **Save** from the File menu to save the synonym.

10. To close the Synonym Editor, select **Close** from the File menu or click the control button in the upper right corner.

Add a New Field to a Group Field

1. Required: Right-click the group field to which you want to add a new field and select **Add**.

2. Required: Select **Column** 🔳 from the Add submenu.

   A field is added to the group and its attributes and values appear on the right.
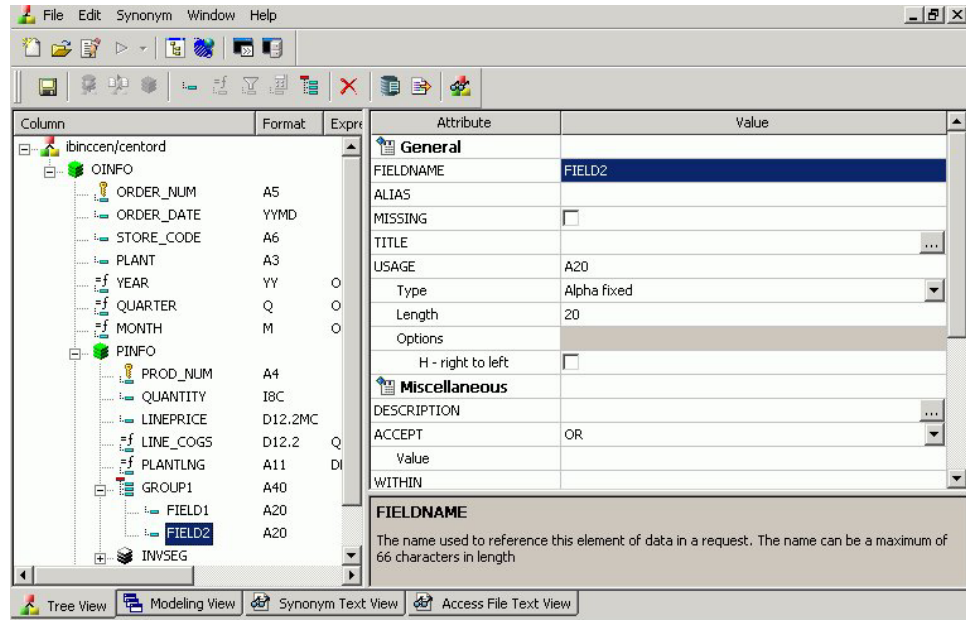
3. Supply the required information for the group field. For more information about field attributes, see Viewing and Editing Synonym Attributes.

Add an Existing Field to a Group Field

1. Click the field you want to add to the group field.

2. While holding the left mouse button down, drag the field and drop it on the group field name.

   The field is added to the group field.

Delete a Group Field From a Segment

1. Required: Right-click the group and select **Delete**.

   A confirmation appears stating that all fields \ columns within the group will be deleted.
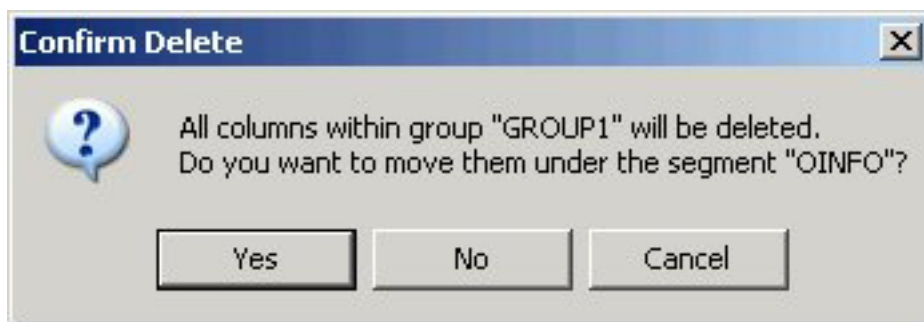
2. Required: Click **Yes** to delete the group and move the fields (within the group) under the root segment of the synonym.

   Click **No** to delete the group and all of the fields within the group.

   Click **Cancel** to close the Confirmation Delete dialog box and return to the Synonym Editor.

Delete a Field From a Group Field

Right-click the field, then select **Delete**.

Group Field Attributes
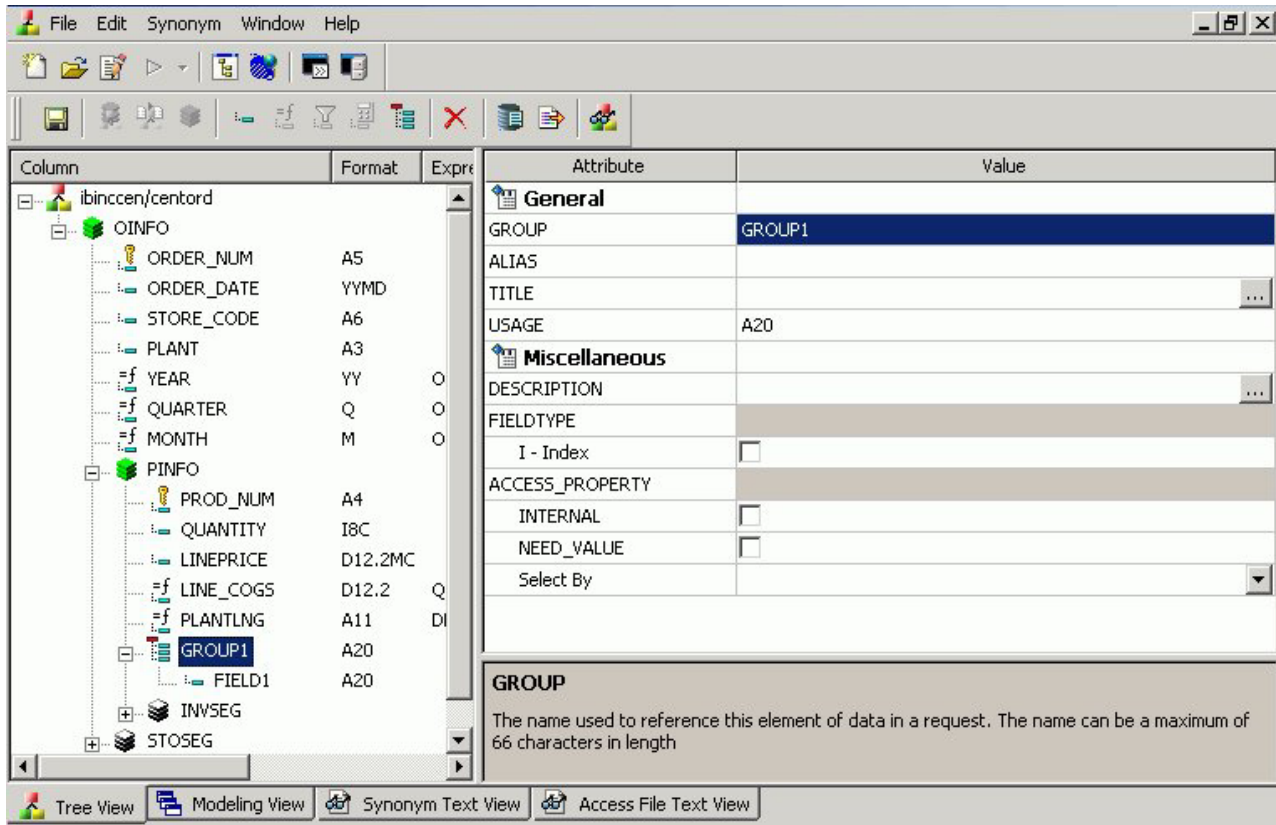
Group fields in a synonym can have the following attributes:

General

> **GROUP**
>
> > Is the name of the group.
>
> **ALIAS**
>
> > Assigns an alternative name for a group.
> >
> > If you create a report, the group name appears as a column heading unless you have specified an alternate title for the group. Aliases cannot be used as column titles.
>
> **TITLE**
>
> > Supplies a title to replace the group name that is normally used in reports and enables you to specify multiple language titles for the group.
>
> **USAGE**
>
> > Contains the format for the group field. Since the group field is made by concatenating together several other fields, the Synonym

Editor determines what this format needs to be. For example, if the group field has two alphanumeric fields in it, each 20 characters long (A20), then the group field must be alphanumeric and 40 characters long (A40). The group field is always alphanumeric, regardless of what the fields that make it up are.

Miscellaneous

DESCRIPTION

Contains a description or comments about the group. The description displays in Field lists and on the status bar.

Field descriptions also appear as bubble help in OLAP-enabled reports. If you do not include a description, bubble help shows the field name (column title).

FIELDTYPE

Identifies an indexed group. You can index the values of a field to enhance data retrieval performance. To do so, select the Index check box when you add a field and before you add the data. An index is an internally stored and maintained table of data values and locations that enhance the performance of data retrieval. A Master File can have several associated indexes, but the combined total of indices and segments cannot exceed 64.

Note: FIELDTYPE=R indicates a read-only column. This setting is useful for columns are that automatically assigned a value by the RDBMS.

Tip: You can turn on the index after adding data to a field, however, you will have to use the Rebuild Index option to create the index.

ACCESS_PROPERTY

Specifies access options for the column's data.

INTERNAL defines a column that does not appear in sample data or in the list of available columns. Restricts the field from showing in any of the Field Lists in the reporting tools.

NEED_VALUE defines a column that requires a value to access the data. Indicates that a selection is needed in the Report Request (WHERE condition).

Select By defines a column by value, range, or multivalues.
- If Value is checked, only one value should be defined for selection in the Report request.
- If Range is checked, a range selection should be defined in the Report request.
- If Multivalues is checked, multiple values are allowed for selection in the Report request.

Note: The attributes available depend on the type of synonym.

Chapter 15. Analyzing Metadata and Procedures

Describes how to analyze procedures using Impact Analysis, and how to view Data Profiling for the columns in a synonym.

Describes how to analyze procedures using Impact Analysis, and how to view Data Profiling for the columns in a synonym.

Note: Impact Analysis is accessible only through the DB2 Web Query Developer Workbench.

Analyzing Procedures With the Impact Analysis Tool

You can use the Impact Analysis tool to generate a report that identifies the procedures that access a specific Master File or field within a Master File. This tool helps you analyze the potential impact of modifying or deleting Master Files or fields. The Impact Analysis tool enables you to analyze data, control search criteria, save reports, and interactively open and edit procedures based on search results.

Impact Analysis searches Business Views in addition to FOCUS procedures. This enables you to see if changes in the original Master File will impact fields used in the Business View. For more information about Business Views, see Creating Business Views in Developer Studio.

The Impact Analysis tool is accessible from Developer Workbench or from within the tools in the Synonym Editor and Business Views. When Impact Analysis is launched from the Synonym Editor or Business Views, it searches files based on the application path of the reporting server. From the interface, it enables you to select applications/domains to be searched.

View Impact Analysis Results from the Synonym Editor

When Impact Analysis is launched from within the Synonym Editor, it searches files based on the application path of the reporting server.

1. From the Projects or Data Servers area, double-click the Master File or select **Edit in Synonym Editor** from the File menu.

   The Master File opens to the Tree View tab in the Synonym Editor.

2. Select the Synonym File name, or a column field, right-click and select **Impact Analysis** from the context menu.

   The following image shows the context menu used to launch the Impact Analysis tool.

Add ▶
Impact Analysis
Data Profiling
Sample Data
─────────
Delete
Rename
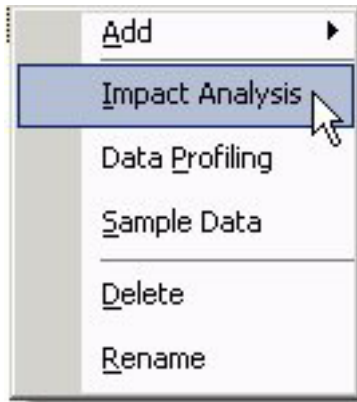
> **Note:** Impact Analysis is also available from the Business View tab.
>
> The Impact Analysis results are displayed in a report spreadsheet.

3. You may use the results toolbar to view server messages, print the report, copy data as text, and export the report.

	Found In	Found In App	Found In Type	Usage	Line	Used In	Identifier
1	dblgg	ibisamp	FEX	MODIFY	4517		
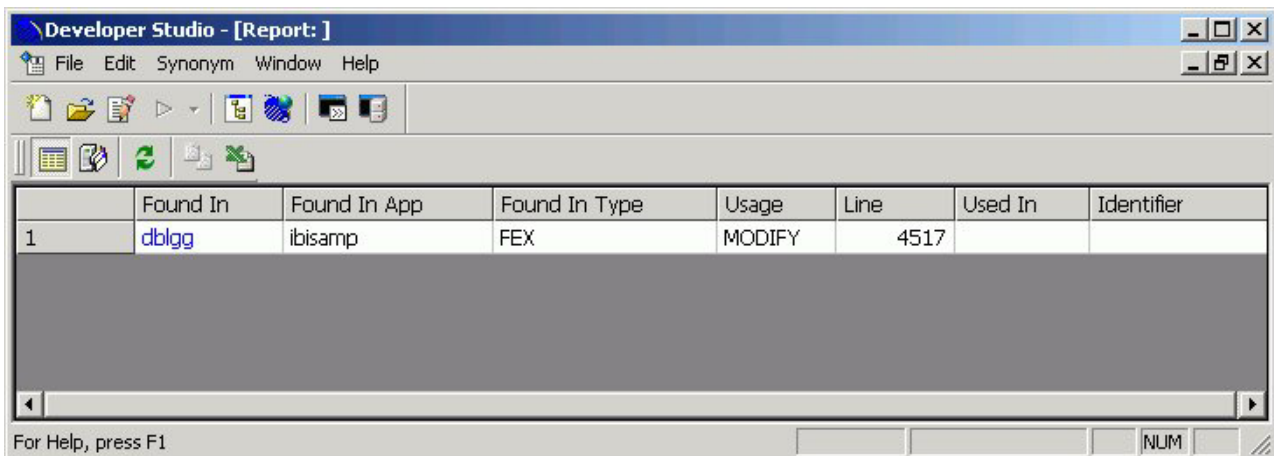
4. Click **Save** from the File menu to save the Impact Analysis results.
5. Click **Close** from the File menu to close the Impact Analysis results window and return to the Synonym Editor.

Use the Impact Analysis Tool From the Interface

1. From the Projects or Data Servers area, highlight a Master File and select **Impact Analysis** from the File menu.

   Note: You may also select a **Impact Analysis** from the right-click context menu of a Master File. This interface enables you to select applications or domains to search.

   The Impact Analysis tool opens with the New Report tab displaying the selected (Master) File Name and default Search Paths.

2. To search for all procedures that access a specific Master File or field, perform one of the following:
   - **For the Master File you selected when opening the tool,** it will be searched by default. (Go to the next numbered step.)

- **For a different Master File,** click the ellipsis button to the right of the File Name search field and select a different Master File in the Open dialog box that opens.
- **For a single field within a Master File,** after you select the desired Master File, click the ellipsis button to the right of the Field Name search field and double-click a field name in the Master File pop-up box.

3. Optionally, to search for procedures in directory paths not listed by default in the Search Paths pane, add more search paths by clicking the folder icon above the Search Paths area and selecting one or more folders in the Browse for Folder dialog box that opens.

   Note: You can also delete search paths by highlighting a search path and clicking the **Delete** icon above the Search Paths area.

4. Click **Analyze** to display a report in the Impact Analysis Results pane.

   The following image shows the New Report tab of the Impact Analysis dialog box populated with File Name search criteria, multiple Search Paths, and a report displayed in the Impact Analysis Result pane.
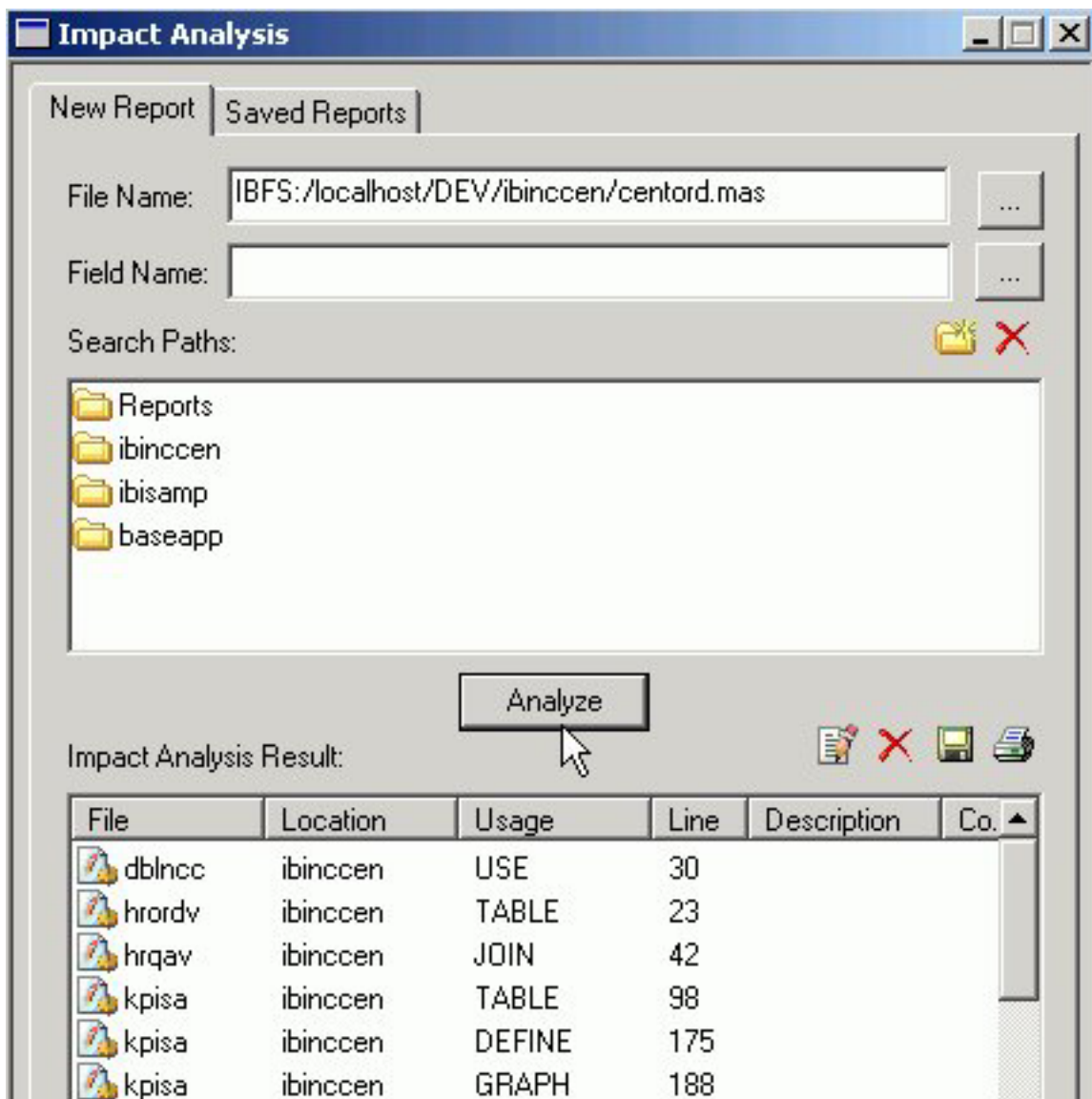
You have options to edit procedures, print the report, delete an item in the report, and export a report.

Exported reports are XML formatted, have an .IAR extension, and are saved in the following default directory (unless you specify a different location):

drive:\ibi\DevStudio*releasenumber*\bin

5. Click the **Saved Reports** tab to access all previously created reports.

All Impact Analysis reports are automatically saved in the following XML formatted file (unless you manually delete a report):

drive:\ibi\DevStudio*releasenumber*\bin\IARepository.dita

Information is appended to this file as new analysis reports are performed. You have options to view reports, import previously exported reports, and delete reports.

Viewing Data Profiling Characteristics

Data Profiling provides data characteristics for the columns in a synonym. You can display the characteristics for all the columns in a synonym or segment, or for an individual column.

Note: Data Profiling is not available if your adapter is not configured correctly.

For alphanumeric columns, Data Profiling provides the segment, format, count of distinct values, total count, patterns count, maximum, minimum, and average length, minimum and maximum values, and number of nulls. Patterns count shows the number of patterns found in each alphanumeric column.

For numeric columns, Data Profiling provides the segment, format, count of distinct values, total count, maximum, minimum, and average value, and number of nulls.

Data Profiling for an individual column provides access to Statistics, Patterns, Values, and Outliers reports.

Data Profiling a Synonym or Segment

Data Profiling provides information on all the columns in a synonym or segment. You can also drill down to the Values or Patterns reports for an individual column from a synonym or segment's Data Profiling report.

View Data Profiling for a Synonym or Segment

To view the data profiling information for a synonym or segment's columns:

1. From the Projects or Data Servers area, open a synonym by double-clicking a Master File from the Master Files folder.

The Synonym Editor opens to the Tree View tab.

2. Right-click the synonym or segment name and, select **Data Profiling**.

The Data Profiling information displays in the workspace.
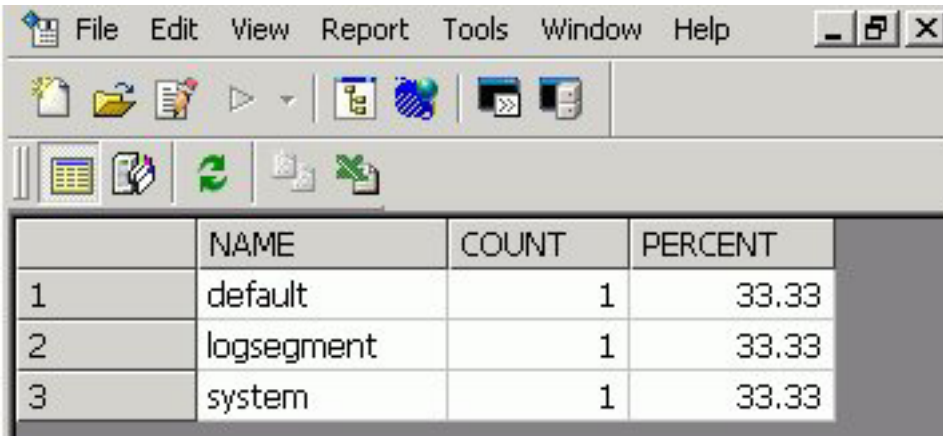
You may use the data profiling results toolbar to view server messages, print the report, copy data as text, and export the report.
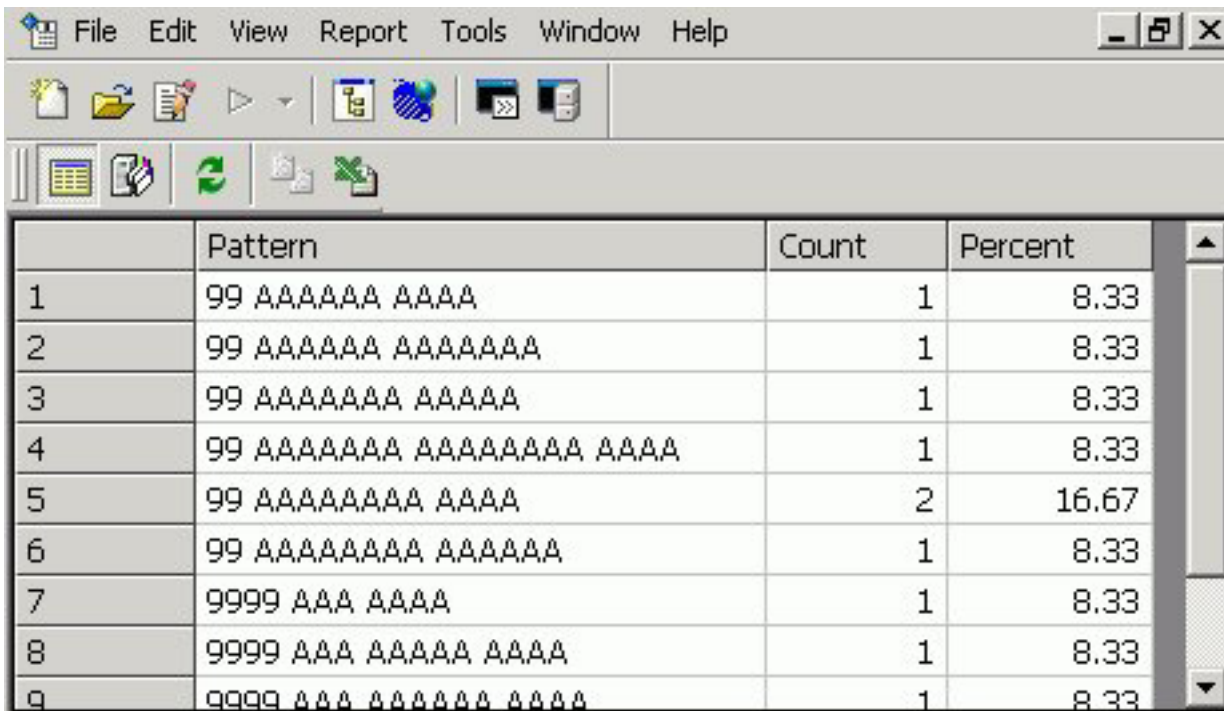
3. Optionally, you can click a column name or patterns count (for alphanumeric columns) to drill down to the Values or Patterns reports, respectively.

This is a partial Values report produced by clicking a column name.

	NAME	COUNT	PERCENT
1	default	1	33.33
2	logsegment	1	33.33
3	system	1	33.33

This is a Patterns report produced by clicking a Patterns Count value.

The image below is an example of an address column.

	Pattern	Count	Percent
1	99 AAAAAA AAAA	1	8.33
2	99 AAAAAA AAAAAAA	1	8.33
3	99 AAAAAAA AAAAA	1	8.33
4	99 AAAAAAA AAAAAAAA AAAA	1	8.33
5	99 AAAAAAAA AAAA	2	16.67
6	99 AAAAAAAA AAAAAA	1	8.33
7	9999 AAA AAAA	1	8.33
8	9999 AAA AAAAA AAAA	1	8.33
9	9999 AAA AAAAAA AAAA	1	8.33

For pattern analysis, a ″9″ represents a digit, an ″A″ represents any upper case letter, and an ″a″ represent any lower case letter. All printable special characters are represented by themselves, and unprintable characters are represented by an ″X″.

Data Profiling a Single Column

Data Profiling for an individual column provides access to four reports:

- **Statistics** shows the same information as a Data Profile report for a synonym or segment.

  For alphanumeric columns, the Statistics report provides the segment, format, count of distinct values, total count, patterns count, maximum, minimum, and average length, minimum and maximum values, and number of nulls.

For numeric columns, the Statistics report provides the segment, format, count of distinct values, total count, maximum, minimum, and average value, and number of nulls.

- **Patterns**, which is only available for alphanumeric columns, shows patterns of letters, digits, and special characters, as well as counts and their percent.
- **Values** shows unique values and their percents.
- **Outliers** shows the ten highest and lowest distinct values and their counts.

These reports are available by right-clicking a column in the Synonym Editor and selecting **Data Profiling**.
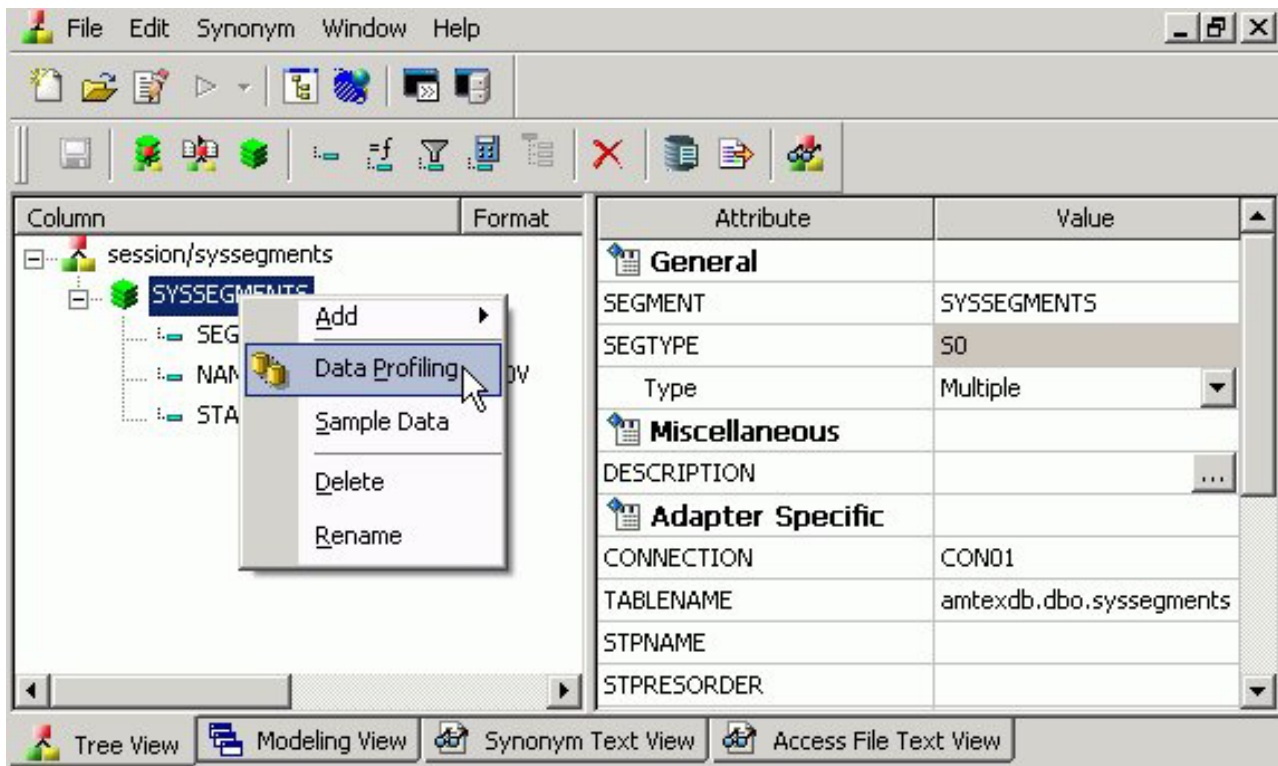
The Patterns report shows the number of patterns found for each alphanumeric column.

	Segment	Name	Format	Distinct Count	Count	Patterns Count	Minimum Length	Maxim
1	STORES01	STORE_CODE	A05	12	12	1	5	
2	STORES01	STORE_NAME	A23	12	12	1	19	
3	STORES01	ADDRESS1	A19	12	12	10	8	
4	STORES01	ADDRESS2	A31	12	12	11	13	
5	STORES01	CITY	A22	12	12	7	6	
6	STORES01	STATE	A02	11	12	1	2	
7	STORES01	ZIP	A06	12	12	1	5	

Clicking on a patterns count displays the actual patterns.

The image below is an example of an address column.

	Pattern	Count	Percent
1	99 AAAAAA AAAA	1	8.33
2	99 AAAAAA AAAAAAA	1	8.33
3	99 AAAAAAA AAAAA	1	8.33
4	99 AAAAAAA AAAAAAAA AAAA	1	8.33
5	99 AAAAAAAA AAAA	2	16.67
6	99 AAAAAAAA AAAAAA	1	8.33
7	9999 AAA AAAA	1	8.33
8	9999 AAA AAAAA AAAA	1	8.33
9	9999 AAA AAAAAA AAAA	1	8.33

For pattern analysis, a "9" represents a digit, an "A" represents any upper case letter, and an "a" represent any lower case letter. All printable special characters are represented by themselves, and unprintable characters are represented by an "X".

View Data Profile Statistics

To view the statistical data profiling information for a single column:

1. From the Projects or Data Servers area, open a synonym by double-clicking a Master File from the Master Files folder.

   The Synonym Editor opens to the Tree View tab.

2. Right-click a column and, select **Statistics** from the Data Profiling submenu.

The Statistical Data Profiling information displays in the workspace.

3. Optionally, you can click a column name or patterns count (for alphanumeric columns) to drill down to the Values or Patterns reports, respectively.

View Data Profile Patterns

Data Profile Patterns shows patterns of letters, digits, and special characters, as well as counts. This is only available for alphanumeric columns.

To view the patterns data profiling information for a single column:

1. From the Projects or Data Servers area, open a synonym by double-clicking a Master File from the Master Files folder.

   The Synonym Editor opens to the Tree View tab.

2. Right-click a column and, select **Patterns** from the Data Profiling submenu.
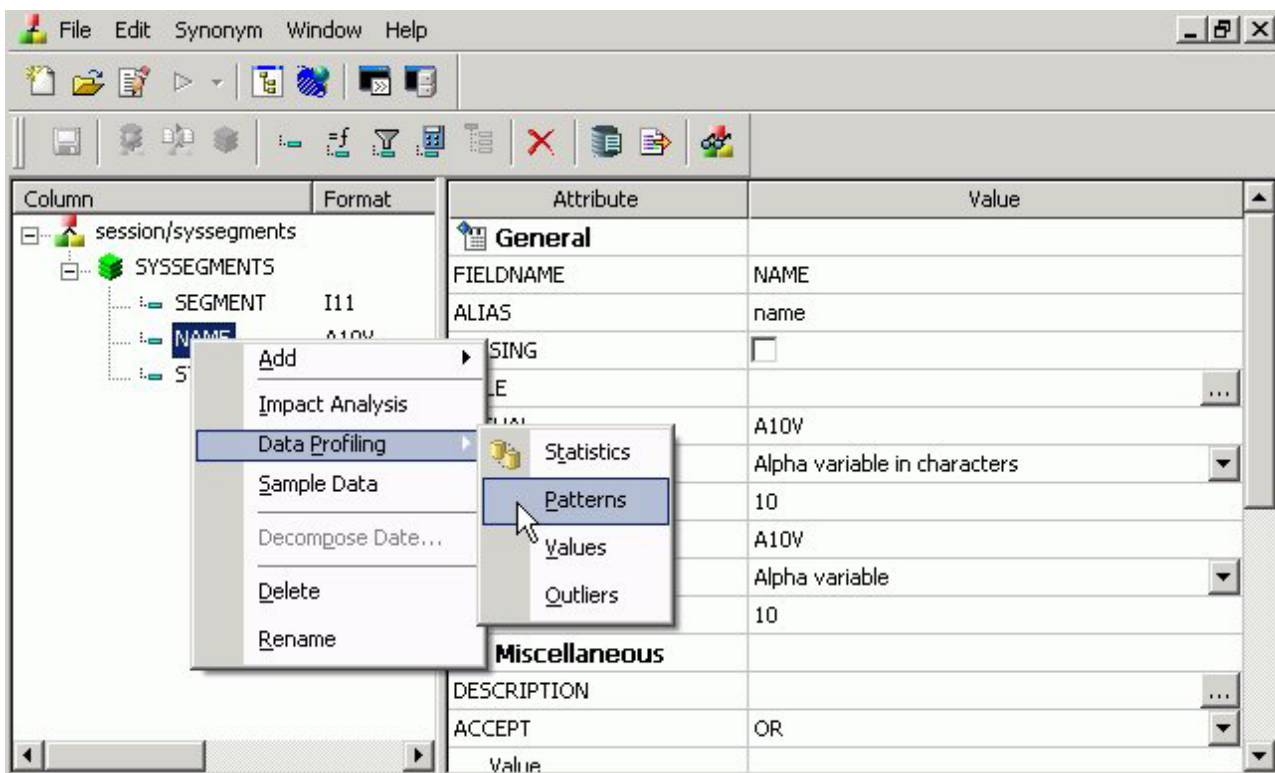
The Patterns Data Profiling information displays.

For pattern analysis, a "9" represents a digit, an "A" represents any upper case letter, and an "a" represent any lower case letter. All printable special characters are represented by themselves, and unprintable characters are represented by an "X".

View Data Profile Values

Data Profile Values shows unique values.

To view the values data profiling information for a single column:
1. From the Projects or Data Servers area, open a synonym by double-clicking a Master File from the Master Files folder.

   The Synonym Editor opens to the Tree View tab.
2. Right-click a column and, select **Values** from the Data Profiling submenu.
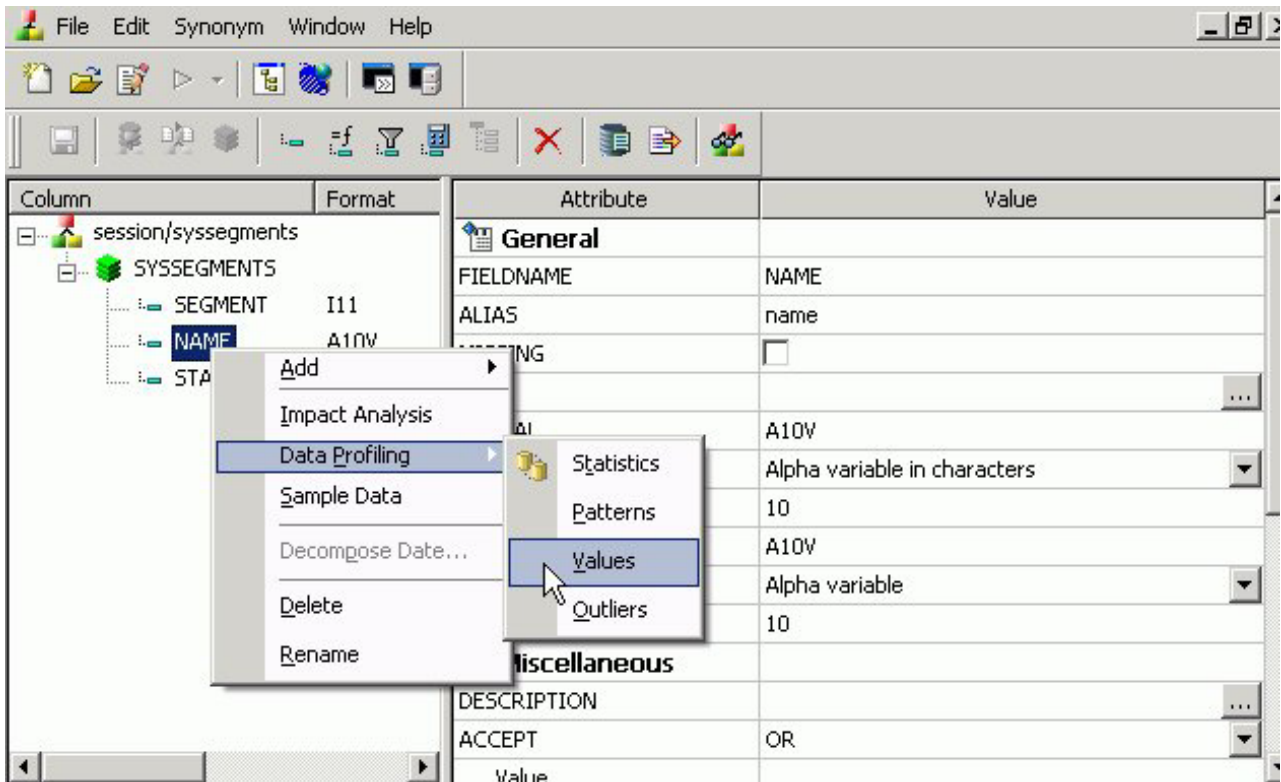
The Values Data Profiling information displays.

View Data Profile Outliers

Data Profile Outliers shows the 10 highest and lowest distinct values.

To view the outliers data profiling information for a single column:

1. From the Projects or Data Servers area, open a synonym by double-clicking a Master File from the Master Files folder.

   The Synonym Editor opens to the Tree View tab.

2. Right-click a column and, select **Outliers** from the Data Profiling submenu.
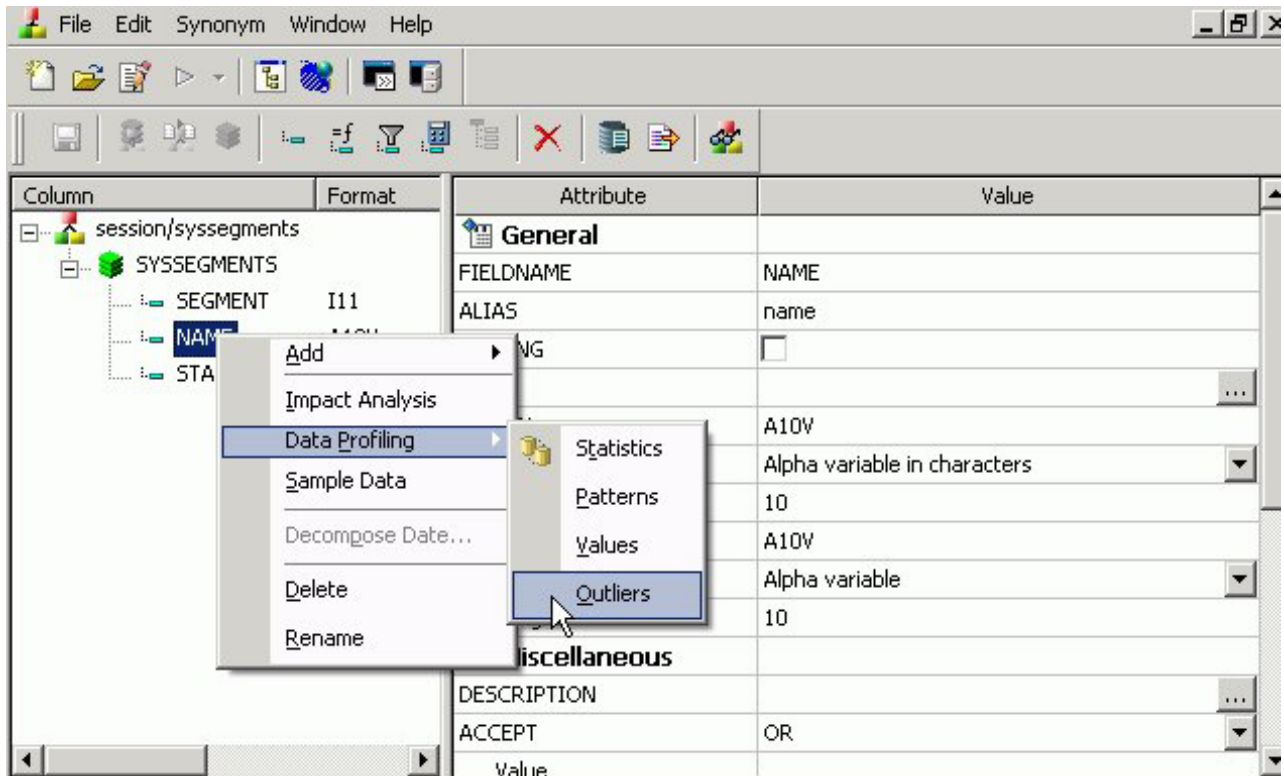
The Outliers Data Profiling information displays.

Note: Outliers produce a maximum of 10 highest and lowest distinct values, if they exist.

Chapter 16. Using Report Broker

Describes how to access and use Report Broker and the Report Broker Scheduler tool.

Report Broker is a report scheduling and distribution tool that provides a central point from which you can automatically distribute essential and current information to select people in an organization.

Through Report Broker, you can schedule a report to run at specific times or intervals and be distributed through either e-mail, to a printer, or to Web Query. You have the option to distribute the report to a single address, or to a group of recipients using a distribution list.

You can distribute an entire report, or you can break a report into sections using the Report Broker burst feature. When you burst a report, you send only the relevant report sections to the users you specify.

The Report Broker Scheduler tool is the interface through which you create schedules. It offers all of the scheduling options you need to define the parameters of a schedule.

In the main Report Broker interface, you can create and maintain Distribution Lists, create, edit, clone, and delete schedules, run log reports, and purge log files. Additionally, you can check the status of scheduled jobs that are in the Report Broker Distribution Server queue.

Preparing Report Broker for Use

Once Report Broker is installed and the table space is created, the Report Broker administrator must:

- Set up the user profile.
- Start the Distribution server.
- Configure the connection information to DB2.
- Test the connection to the repository.
- Create the Repository tables.

The following sections explain how to create the repository and verify configuration settings in the Web Query Administration Console.

Note: Until the repository is created, the Distribution Server will start in Console mode.

Setting Up the User Profile

To enable named users with Report Broker privileges based on the tasks they will perform, add one of the following groups to the user profile. (The user profile can not contain both groups.)

- MRSCHEDULE enables the named user to create Report Broker schedules.
- MRADMIN enables the named user to create schedules and perform administrative tasks, such as setting Report Broker server configuration options.

Note: A named user is a user ID that exists in the License Manager entry for the base Web Query product.

Starting or Stopping the Distribution Server

When you are ready to use the Distribution Server, use the following commands to start and stop the server:

- To start the Distribution Server:

  `STRWEBQRY START(*DSTSRV)`

- To stop the Distribution Server:

  `ENDWEBQRY END(*DSTSRV)`

Configuring the DB2 Connection

Use the following procedure to configure the connection information to DB2, so that Web Query can connect to the Report Broker repository.

Configure the DB2 Connection

1. In left pane of the Web Query Administration Console, click **Repository**, then **Connection**.

   The Report Broker - Connection information appears in the right pane, as shown in the following image.
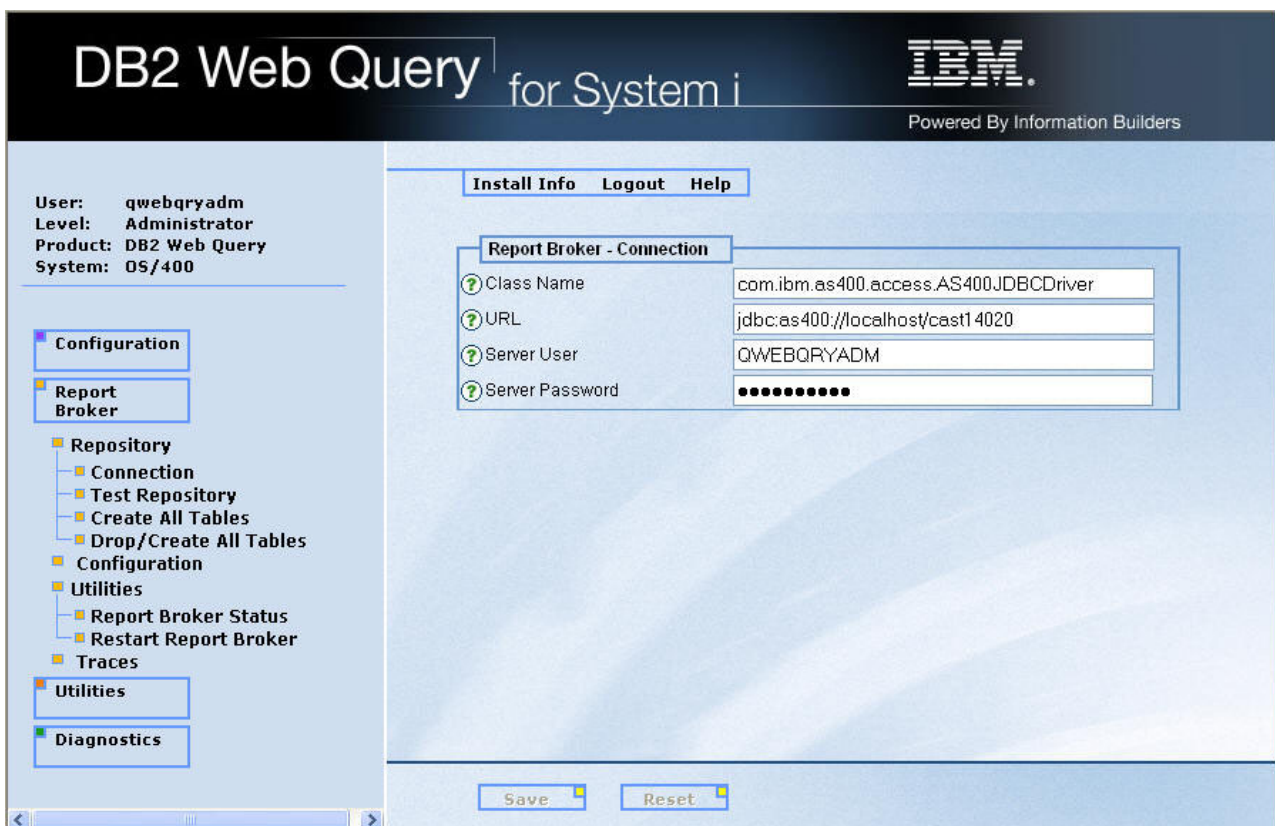
2. In the Server Password field, type the password to connect to DB2, then click **Save**.

   The following message appears:

   `Report Broker Configuration changes have been saved.`

   `Do you want to restart Report Broker?`

Note: The Distribution server must be restarted to implement any changes made to the configuration.

3. Click **OK**.

   The following message appears:

   `Report Broker restart completed.`

4. Click **OK**.

Test the Repository Connection

To verify that DB2 can connect to the Report Broker repository:

In left pane of the Web Query Administration Console, click **Repository**, then **Test Repository**.

The right pane displays the test results. A status of **Pass** indicates the Distribution server can connect to the Report Broker repository.

Creating the Report Broker Repository

Once the connection configuration and testing are complete, you can created the Report Broker repository tables using the following procedure.

Create the Report Broker Repository

1. In the left pane of the Web Query Administration Console, click **Repository**, then **Create All Tables**.

   A confirmation window opens with the following message:

   `Do you wish to create the Report Broker repository tables?`

2. Click **OK**.

   The right pane displays the status of the repository. A status of **Pass** indicates the repository has been created successfully.

The repository tables have been created and Report Broker is ready for use.

Verifying the Report Broker Configuration Settings

This section explains how to access and verify the configuration settings.

To access the configuration settings, from the Web Query Administration Console, click **Report Broker**, then **Configuration**. The right pane displays the configuration settings, as shown in the following image.
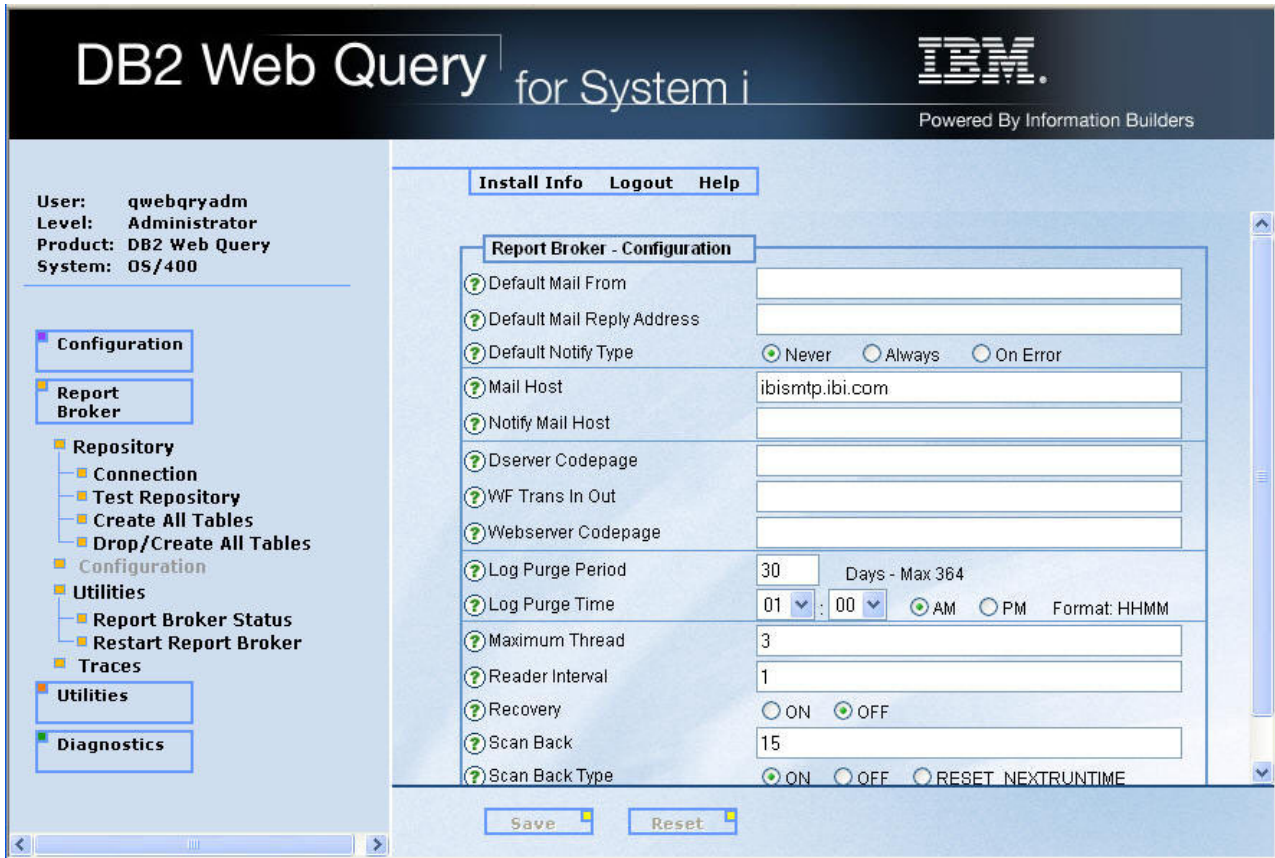
Change the Mail Host setting to your mail host server. Review the other settings and make any appropriate changes. When you have completed all changes, click **Save**.

The following message appears:

```
Report Broker Configuration changes have been saved.
Do you want to restart Report Broker?
```

Click **OK** to restart the Report Broker Distribution server. A message appears when the restart is complete. In the message window, click **OK**.

Report Broker Configuration Settings

The Report Broker configuration settings are:

Default Mail From

> Default value for the From field in a schedule using e-mail distribution. This can be any value.

Default Mail Reply Address

> Default value for the Reply Address field in a schedule using e-mail distribution.

Default Notify Type

> Specifies whether to send notification of the schedule status to a specified e-mail address. Possible values are:

`Never` - This is the default value. Report Broker does not send notification of the schedule status under any circumstance.

`Always` - The specified user is always notified when the schedule runs.

`On Error` - The specified users are notified when errors are encountered while running the schedule. We recommend using the On Error notification option.

Mail Host

Name of the default mail server used for a schedule using e-mail distribution.

You can also specify a port for the Mail Host using *hostname*[:*port*]. If you do not specify a port or the port you specify is not present, the default port is used.

Notify Mail Host

Name of the mail server that handles notification e-mail. If left blank, the mail server specified in the Mail Host setting is used as the notification mail server.

You can also specify a port for the Notify Mail Host using *hostname*[:*port*]. If you do not specify a port or the port you specify is not present, the default port is used.

Tip: We recommend using different mail servers for notification and e-mail distribution in case there is a problem with the default mail server (notification e-mails are still sent). Having separate mail servers ensures you are informed if the default mail server falters.

Dserver Codepage

The code page of the platform, where the Distribution Server is running, is passed to the Reporting Server to enable its communication back to the Distribution Server. Used for National Language Support (NLS).

WF Trans In Out

Configures the Servlet WFTRANSINOUT plug-in. This is a custom-written program that, when configured, is called by the Distribution Server as the last step prior to sending each request to the Reporting Server and again as the first step when receiving output from the Reporting Server.

In the WFTransInOut field, type the fully qualified name of the package and class that implements the WFTransInOut interface.

To configure the exit jar file and classpath:
1. Copy the jar that contains the implementation of the WFTransInOut exit to your_Distribution_Server_root/lib directory.
2. Edit the claspath.bat file to add this jar file to the list of existing jar files.
3. For the Report Broker service, make the same change to the classpath in the Registry Key (HKEY_LOCAL_MACHINE\SOFTWARE\ Information Builders\ReportCaster\76x\CLASSPATH).
4. Restart the Distribution Server.

Webserver Codepage

Code page of the platform where the Web server is installed.

Log Purge Period

Automatically purges individual log reports when they are older than a set number of days. The default value is 30.

Log Purge Time

Time at which log purging occurs. The default value is 1:00 A.M.

Maximum Thread

Controls how many simultaneous connections (threads) the Distribution Server can utilize to process scheduled jobs.

Reader Interval

Polling interval (in minutes) for the Distribution Server to check for scheduled jobs. The default value is 1 minute.

Recovery

Recovers scheduled jobs. Possible values are:

ON - During startup, the Distribution Server recovers scheduled jobs that were processed but not completed.

OFF - This is the default value. During startup, the Distribution Server does not recover any scheduled jobs.

Scan Back

If the Distribution Server is unavailable for a period of time, any jobs scheduled during that period are not run. By default, when the Distribution Server is restarted, it searches for and runs all jobs with a next run time that is less than the current time, then resets the next run time according to the job's next scheduled run time. The Scan Back value and Scan Back Type settings allow you to control the period of time the Distribution Server will look for jobs and whether or not it will run them.

The Scan Back value is an integer value that represents the number of 24 hour periods, beginning when the Distribution Server is restarted, that the Distribution Server scans back to look for, and run, jobs that have not yet run. The maximum integer value for Scan Back is 365. The default value is 15 (periods of 24 hours).

Scan Back Type

Possible Scan Back Type values are:

ON - This is the default value. Turns on Scan Back functionality. The Distribution Server searches for all jobs with a next run time that is less than the current time, runs only those jobs found within the time period set in the Scan Back setting, and resets the next run time of all of the jobs initially found.

OFF - Turns off Scan Back functionality, which results in the Distribution Server following its default behavior to locate, run, and reset all jobs with a next run time that is less than the current time.

RESET_NEXTRUNTIME - Searches for all jobs with a next run time less than the current time and resets the next run time to the job's next scheduled run time. A Scan Back value of zero defaults to this Scan Back Type.

Zip Encoding

Enables Report Broker administrators to specify an encoding other than the default encoding of the Distribution Server platform. It must match the encoding used by WinZip or any other Zip utility installed on the Distribution Server.

Accessing the Report Broker User Interface

To access the Report Broker user interface, click the Report Broker tab in DB2 Web Query.

The Report Broker user interface opens, displaying all of the schedules you have created. The following image shows the options and data fields available when first accessing the interface.

From this interface you can:

- Create and maintain a Distribution List that you can then assign to scheduled reports. For more information, see Creating and Maintaining a Distribution List.
- Check the status of a scheduled job. For more information, see Checking the Status of a Scheduled Job.
- Globally update certain schedule values, such as the schedule owner or the mail server. See Globally Updating Schedule Field Values.
- Edit, clone, or delete schedules that you own. For more information, see Maintaining a Schedule.
- Run a log report to obtain information about a schedule and purge the log file to conserve space. For more information, see Viewing a Log Report and Purging the Log File.
- Move data from one Report Broker repository to another using the Change Management tool. For more information, see Report Broker Change Management.
- View the dates on which schedules cannot run or be set to run. For more information, see Schedule Blackout Dates.
- Open the online Help.

Considerations When Using Report Broker

The following are important considerations when using Report Broker:

- Although Report Broker end users can only view and perform actions on their own objects (schedules, Distribution Lists, and log files), Report Broker Administrators can view and perform actions on all user's objects.

- When you select a schedule in the Report Broker interface, the Open, Log, Delete, and Copy options are enabled. As an alternative to the toolbar icons, you can use Alt-O for Open, Alt-L for Log, Alt-D for Delete, and Alt-C for Clone.

Specifying Multiple E-mail Addresses

When creating a schedule or distribution list, you can specify multiple e‐mail addresses within a single field, row, or record.

When creating a schedule or distribution list, you can separate each e-mail address with a comma or a semicolon. The multiple e-mail addresses will appear in the To: line of a single e-mail when the scheduled output is distributed.

Note: To distribute separate e-mails for each address, specify the e-mail addresses on separate lines within the distribution list.

Specifying Multiple Burst E-mail Addresses

If you are using the default configuration (Packet Email = YES), one e-mail is distributed for multiple burst values specified for the same e-mail address. The e-mail address values specified on each row are treated as a string that is a key. If there are multiple rows with the same address value (key), one e-mail is distributed with all the burst values. For example, consider the following distribution list:

Burst Value	Address
A	user1@abcd.com;user2@abcd.com
B	user1@abcd.com
C	user1@abcd.com

In this example, user1@abcd.com receives two e-mails when the scheduled output is distributed. In the first e-mail, user1@abcd.com; user2@abcd.com appears in the To: line and one attachment is distributed for burst value A. In the second e-mail, user1@abcd.com appears in the To: line and two attachments are distributed, one for burst value B and one for burst value C.

If you are using the configuration that specifies to distribute a single e-mail for each row (Packet Email = NO), then the following behavior occurs for the above example. Three separate e-mails are distributed. In the first e-mail, user1@abcd.com; user2@abcd.com appears on the To: line and one attachment is distributed for burst value A. The second e-mail is sent to user1@abcd.com with one attachment for burst value B. The third e-mail is sent to user1@abcd.com with one attachment for burst value C.

Another consideration is when using the default configuration (Packet Email = YES) and the same burst value is specified multiple times for the same Address (key) value. For example, consider the following Distribution List:

Burst Value	Address
A	user1@abcd.com;user2@abcd.com
B	user1@abcd.com
B	user1@abcd.com

In this Distribution List, only two e-mails are distributed for user1@abcd.com. In the first e-mail, user1@abcd.com; user2@abcd.com appears in the To: line and the attachment is for Burst Value A. In the second e-mail, user1@abcd.com appears in the To: line and the attachment is for Burst Value B. The third row in the Distribution List is ignored since it contains the same key and the same burst value, B, as the second row.

As a best practice, be sure to review your distribution information to make sure you have not duplicated the same burst and address value pairs multiple times.

Specifying Multiple Non-Burst E-mail Addresses

Consider the following sample Distribution List, which does not contain any burst values:

Address

user1@abcd.com;user2@abcd.com

user1@abcd.com

user2@abcd.com

user3@abcd.com

user1@abcd.com;user2@abcd.com;user3@abcd.com

In this Distribution List, an e-mail is distributed for each address line regardless of whether Packet Email is set to YES or NO. This is because each address value is unique. For the first e-mail, user1@abcd.com; user2@abcd.com appears in the To: line and the attachment is for the full report. The second e-mail is distributed to user1@abcd.com, and so on.

If one of the address lines is repeated in the Distribution List (for example, if user3@abcd.com is added as the sixth line in the above list), the behavior would work as follows. If Packet Email = YES, only one e-mail is distributed for user3@abcd.com. However, if Packet Email = NO, two separate e-mails are distributed to user3@abcd.com.

Creating and Maintaining a Distribution List

When creating a schedule, you can distribute report output to a single recipient or several recipients. If you are creating a schedule to be distributed to several recipients, you may want to create a Distribution List consisting of multiple recipients. The Distribution List may then be assigned to any schedule.

If the entire report is not relevant to those receiving it, you can specify sections of the report to be sent using the burst option. Each person on your Distribution List can receive different sections of the report depending on the individual burst values you specify. For more information see Bursting a Report.

Accessing the Distribution List Interface

From the Report Broker interface, click the Distribution List option. All Distribution Lists to which you have access (public lists and your own private lists) appear. Each Distribution List contains columns specifying its properties. Sorting is available for each column by clicking the column heading. In addition, the interface includes toolbar icons that enable you to perform functions on each Distribution List.

The following image shows the available options and data fields in the Distribution List pane of the Report Broker interface.
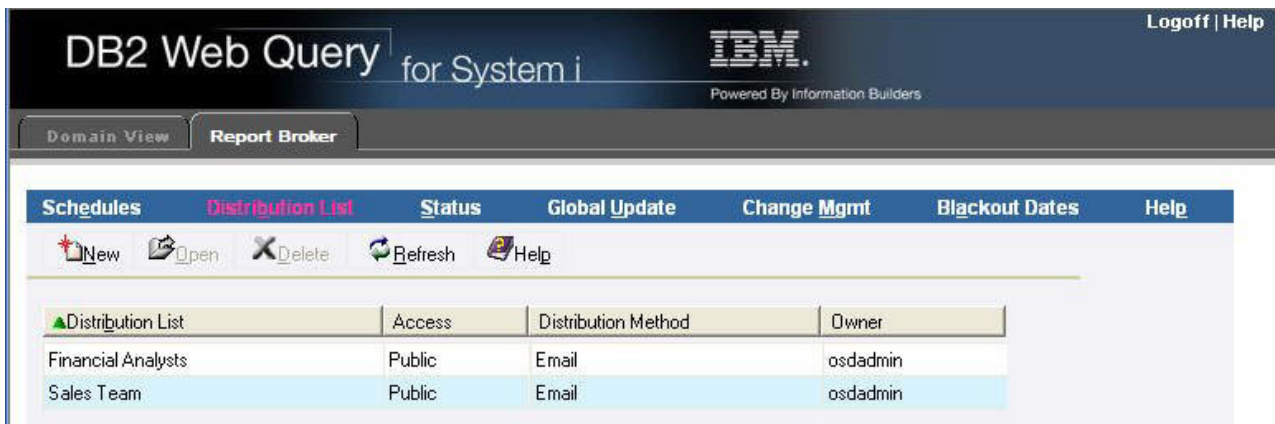
From the Distribution List pane, you can:

- Create a new Distribution List. For more information, see Creating a Distribution List.
- Edit the properties of a Distribution List. For more information, see Editing a Distribution List.
- Delete a Distribution List. For more information, see Deleting a Distribution List.
- Refresh the window so that it contains the latest Distribution List information.
- Open the online Help.

Creating a Distribution List

When you create a Distribution List, you specify a name for the Distribution List, the distribution method (e-mail or printer), the destinations to which the report is distributed, the optional burst values, and whether public or private access is applied.

Create a Distribution List

To create a Distribution List:

1. From the Distribution List pane, click **New**.

   The options, fields, and drop-down lists that enable you to specify the properties of the Distribution List appear, as shown in the following image.

2. In the Distribution List field, type a name for your Distribution List (for example, Sales Team).

3. From the Access drop-down list, select **Public** (default) or **Private**.

    Only the owner and Report Broker Administrators can view a Private Distribution List, whereas every Report Broker user can view a Public Distribution List.

4. From the Distribution Method drop-down list, select **Email** (default) or **Printer**.

5. If you are bursting a report, type individual sort values in the Burst Value field. The burst value is case-sensitive and may be a maximum of 75 characters.

    Note: The burst value for a tabular report is the first BY field, which is the primary sort field. The burst value for a graph report is the second BY field. The burst value specified must exist in the data source that the scheduled job reports against.

    For more information about bursting a report, see Bursting a Report.

6. In the Destination field, type the destinations to which the report is distributed.

    • If you selected Email as the distribution method, type the e-mail addresses of the recipients (for example, chuck_hill@ibi.com). Be careful typing this information because there is no edit checking.

    • If you selected Printer as the distribution method, use the following format to specify the printer:

        queue@printserver

        where:

        queue

            Is the name of the printer queue

        printserver

            Is the host name or IP address of the printer.

Report Broker can differentiate between the printer queue and the printer host name/IP address due to the presence of the '@' separator. We recommend specifying both the printer queue and host name/IP address when distributing Report Broker output to a printer. However, Report Broker supports specifying only the host name or IP address of the printer.

You can use a maximum of 800 characters in a single Destination line.

Note:

- If you specified burst values, be sure to match the specified e-mail or printer destinations with the appropriate burst values.

- The maximum number of e-mail addresses or printers you can specify in a Distribution List is 9999.

- If you are specifying more than ten e-mail addresses or printers in your Distribution List, click **Insert** to insert additional rows.

- You can specify multiple e-mail addresses within a single Destination field. For more information, see Specifying Multiple E-mail Addresses.

- For more information about creating a printer Distribution List, see Considerations When Creating a Printer Distribution List.

7. Click **Save** to save the Distribution List and return to the Distribution List pane.

   The Distribution List you created (for example, Sales Team) is added to the list of available Distribution Lists, as shown in the following image.

Considerations When Creating a Printer Distribution List

When printing on Windows and UNIX®, Report Broker uses the lp (line printer) protocol, which runs on top of TCP/IP to communicate with printers. lp printing with the Novell Client is not supported. For printing on z/OS®, Report Broker uses the lpr (line printer remote) UNIX command, which has special options on the mainframe for sysout dest and class.

When creating a printer Distribution List on z/OS, the printer should be a SYSOUT class (such as A) that has been routed to a printer. For example, you can indicate that scheduled report output be distributed to a specific network printer by typing the following printer identification

```
A DEST printserver
```

where:

A

   Is the SYSOUT class to be assigned to a network printer.

printserver

   Is the printer host name (for example, IBVM.P24E2).

Note: Before creating a Distribution List, we recommend verifying that you can connect to your printer by test printing a document outside of Report Broker.

Editing a Distribution List
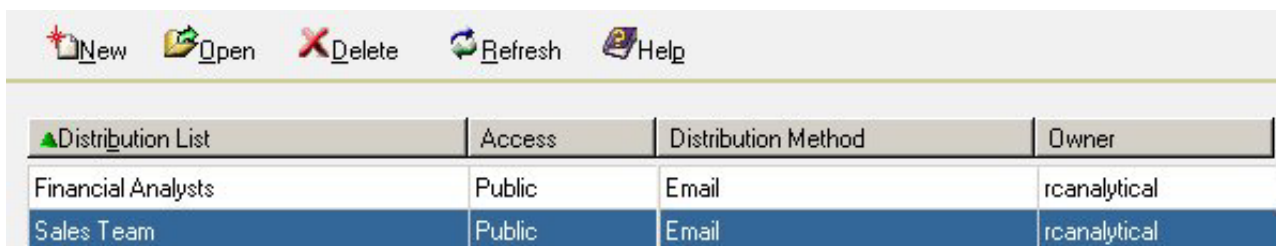
From the Distribution List pane in the Report Broker HTML User Interface, you can edit your own previously created Distribution Lists at any time by performing the following steps.

Edit a Distribution List

To edit a Distribution List:

1. Select the Distribution List you want to edit.

   The following image shows a selected Distribution List named Sales Team.

2. Click **Open**.

   The following image shows the properties of the selected Distribution List.

3. Make the necessary edits. You can:
   - Copy the Distribution List by changing its name. The original Distribution List remains unaltered.
   - Change the values of already existing entries. For example, you can make the Distribution List Private instead of Public, or you can change the destinations.
   - Insert additional Burst values and Destinations by clicking **Insert**. A new row appears where your cursor is positioned, enabling you to insert additional entries.
   - Delete Burst Values and Destinations by selecting the row and clicking **Delete**.

4. Click **Save** to save the changes.

Deleting a Distribution List
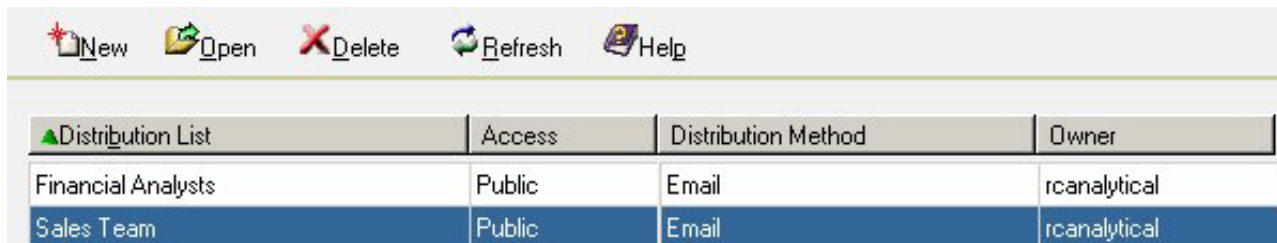
From the Distribution List pane in the Report Broker interface, you can delete your own Distribution Lists at any time.

Delete a Distribution List

To delete a distribution list:

1. Select the Distribution List you want to delete.

   The following image shows a selected Distribution List named Sales Team.

2. Click **Delete**.

   A message appears asking for confirmation that you want to delete the list.
3. Click **OK** to confirm that you want to delete the Distribution List.

Bursting a Report

Instead of distributing an entire report, you can use the Report Broker burst feature to break a report into sections to be distributed separately. Bursting enables you to target relevant sections of a report to individual users. Each report section is saved as a separate file.
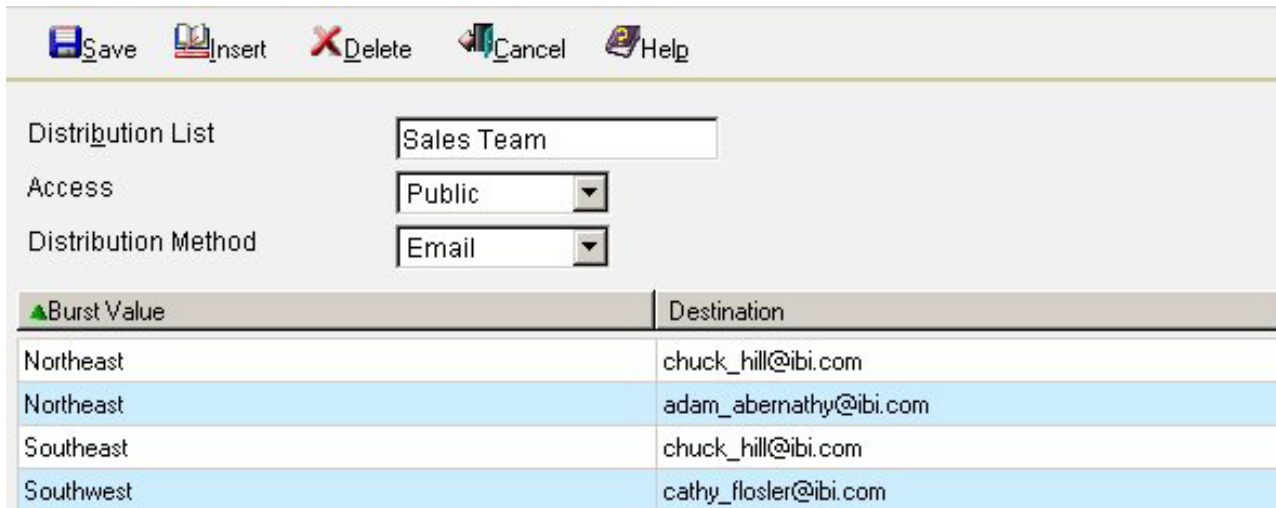
If you are distributing a burst tabular report, the burst value is determined by the first BY field. If you are distributing a burst graph report, the burst value is determined by the second BY field. The burst value is automatically determined by the internal matrix, which is a memory area that stores each database field value and calculates values referenced by the TABLE or GRAPH request.

You can send several report sections to one recipient by specifying that recipient's destination (e⌂mail address or printer) for each section you want to send. You can send several report sections to one destination, or you can send one report section to several destinations. The burst values you specify in the Distribution List must exist in the data source you are reporting against.

If you want to burst a report, you must select the Enable Report Bursting check box when you create the schedule. The burst values specified in the Burst Value column in the Distribution List pane are ignored unless the schedule specifies to burst the report. For more information, see Creating a Schedule.

Bursting a Sales Report

You can specify sort field Burst Values and Destinations (e-mail addresses or printers) when creating or editing a Distribution List. The following image shows burst values and e-mail addresses specified in the Distribution List pane.

Using the primary sort field values (Northeast, Southeast, and Southwest), each representative's e-mail address is associated with the relevant sales report data. Since Adam Abernathy needs only the data for the Northeast branch, the sort value Northeast is listed in the Burst Value column opposite his e-mail address which is in the Destination column.

However, Chuck Hill works in both the Northeast and Southeast regions. Since he requires data for both regions, his e-mail address is listed in the Distribution column twice, with a Burst Value column entry for each region opposite his e-mail address.

Tip: You can specify multiple e-mail addresses on a single Address line. For details, see Specifying Multiple E-mail Addresses.

Creating a Schedule

This section explains how to schedule a report using the Scheduler tool, and describes all of the options available in the tool.

Accessing and Using the Report Broker Scheduler Tool

To create a new schedule for a report, in DB2 Web Query, click the Domain View tab, and expand the Reports folder to display your reports. Right-click the report you want to schedule and select **Schedule**, as shown in the following image.
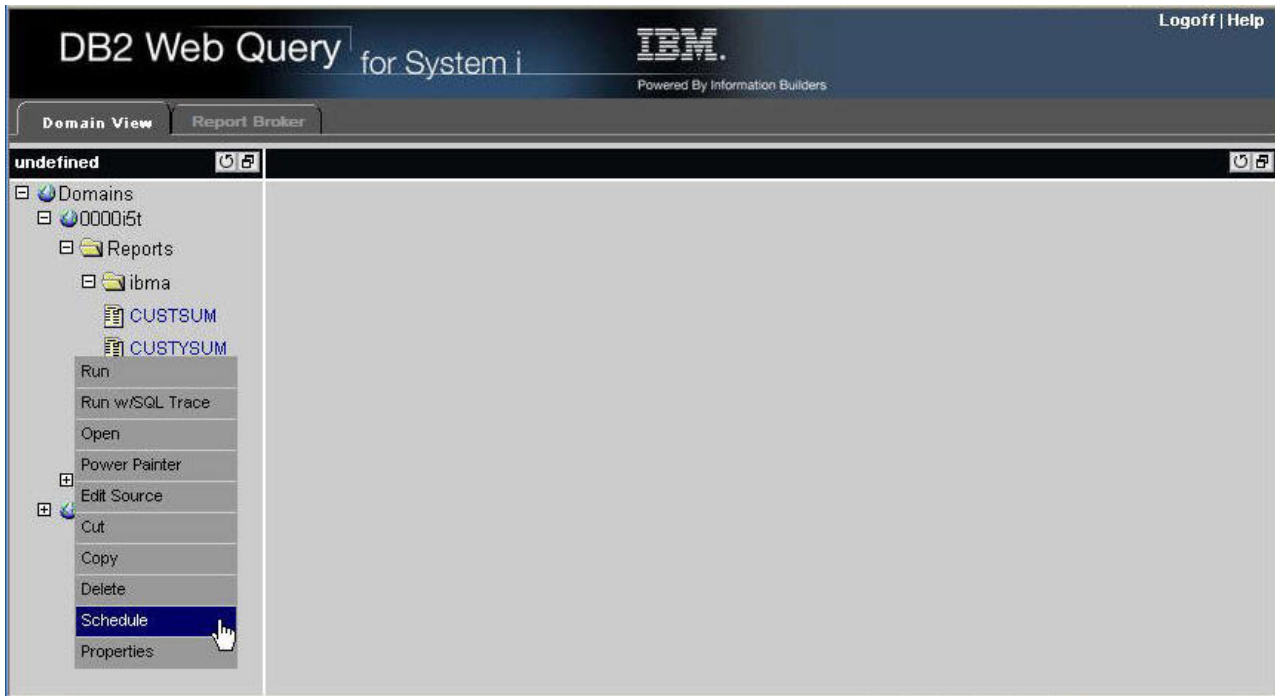
The Scheduler tool opens, as shown in the following image.
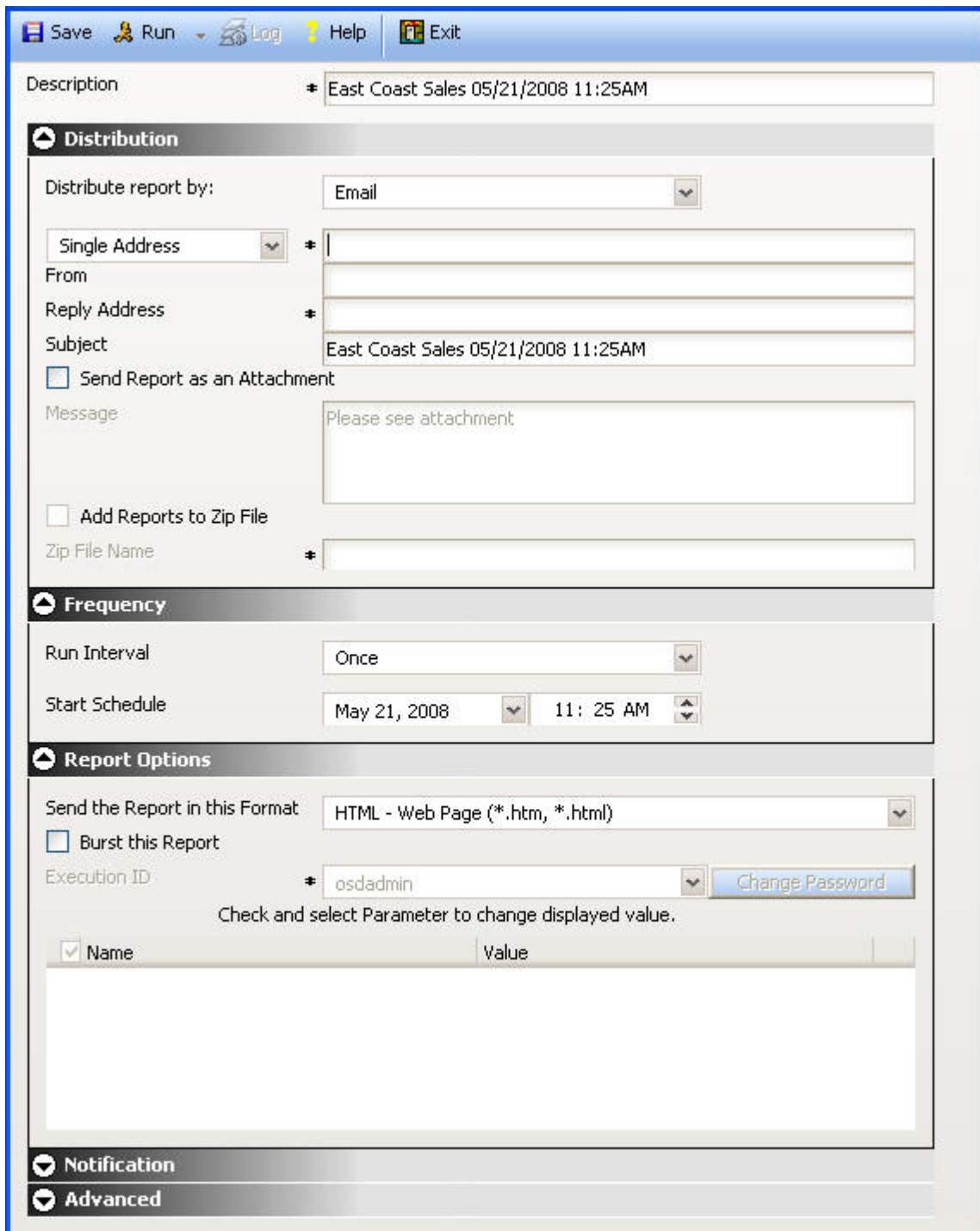
The Scheduler Tool is partitioned into the following sections:

- **Distribution.** Provides a drop-down list of distribution methods and corresponding distribution options.
- **Frequency.** Provides the options related to the time interval to run the schedule.
- **Report Options.** Provides options for the report format, burst option, and report parameters.
- **Notification.** Provides the options to set up notification of the schedule status.

- **Advanced.** Provides options to set the schedule priority level, and selections to enable or delete the schedule.

An asterisk indicates a required field.

When working in the Scheduler tool, you can expand or collapse sections, as needed. For example, the following image shows the Frequency, Report Options, and Notification sections collapsed.

Creating a Schedule Using the Scheduler Tool

A report schedule directs the execution of a report. It defines the format that the report output will take and how, when, and where it will be distributed. All options associated with scheduling a report are available in the Scheduler tool.

This section provides the overall procedure to create a new schedule for a report. Some steps in the procedure contain details on the associated options, while other steps direct you to a separate section that contains detailed descriptions of the options and additional information, such as tips in making a selection.

Create a Schedule

1. Open the Scheduler tool as explained in "Accessing and Using the Report Broker Scheduler Tool" on page 701.

2. In the Description field at the top of the tool, type a name for the schedule. This field is required.

3. In the Distribution section, select the method to distribute the report from the Distribute report by: drop-down list. The distribution methods are:
   - Email
   - Printer
   - Web Query

   For options related to each distribution method, see Distribution Options.

4. Make the following selections in the Frequency section.

   a. From the Run Interval drop-down list, select the time interval that the schedule will use to run the report.

   You can set the interval to run the schedule once, or every specified minutes, hours, days, weeks, months, and years.

   b. From the Start Schedule options, select the date (from the drop⌂down calendar) and time you want the schedule to begin running.

   Note: To change the time setting, select either the hour or minutes and use the arrows to increase or decrease the value.

   c. If applicable, from the End Schedule options, select the date and time you want the schedule to stop running.

   d. If applicable, from the Every option, use the up and down arrows to set the number of times you want the schedule to run for the selected interval, for example, 3 times each month.

   e. Select the remaining options associated with the chosen run interval. For a description of the run interval options, see Frequency Options.

5. In the Report Options section:

   a. Select the format of the report. The valid formats are AHTML, DHTML, DOC, EXL2K, EXL2K FORMULA, EXL97, HTML, JPG, PDF, PNG, PS, SVG, and WP. For information on these formats, see Chapter 18, "Report Broker Formats for Scheduled Output," on page 765.

   b. Select whether or not you want to burst the report.

   For details on bursting reports, see Bursting a Report.

   c. If needed, change the value of an existing report parameter

   For details on changing a report parameter, see Report Parameters.

6. In the Notification section, select whether or not you want to send a notification when the schedule runs and under what conditions to send it. The notification options are:
   - **Never.**
   - **Always.** Send a notification each time the schedule runs.
   - **On Error.** Only send a notification when there is an error running the schedule.

7. In the Advanced Options section, select the following options:
   - **Priority Level for the Job.** Use the drop-down list to select the priority for running the job, with 1 being the highest priority and 5 the lowest priority. The default priority is 3.

The Report Broker Distribution Server queue sorts scheduled jobs by priority and then by time. If multiple jobs share the same priority and time, Report Broker arbitrarily schedules the jobs.

- **Enabled.** Select this option to activate the schedule to run.
- **Delete this schedule if it is not scheduled to run again.** This option is useful for test purposes.

8. To save the schedule, click **Save** at the top of the Scheduler tool.

The Scheduler tool validates the fields and displays a status message in a separate window. If the schedule was successfully created, the status message, `Your Schedule has been created.`, appears, as shown in the following image.
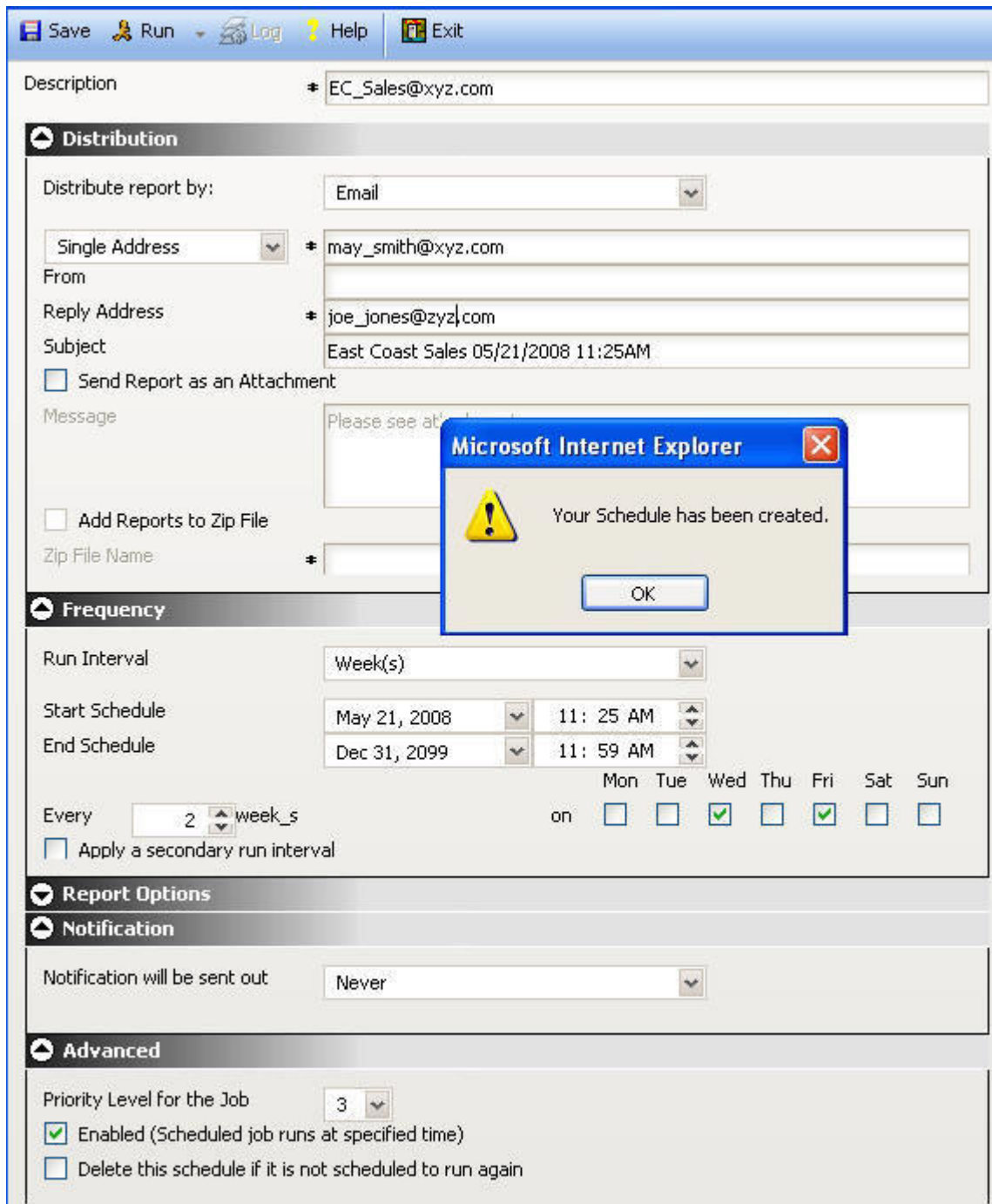
If the Scheduler could not create your schedule, the following message appears in a separate window:

```
Please enter valid information in all required fields.
```

An alert icon appears next to the field that is invalid, and the field name appears in red. The following image shows an invalid schedule with the Reply Address option appearing in red to indicate this value is missing.
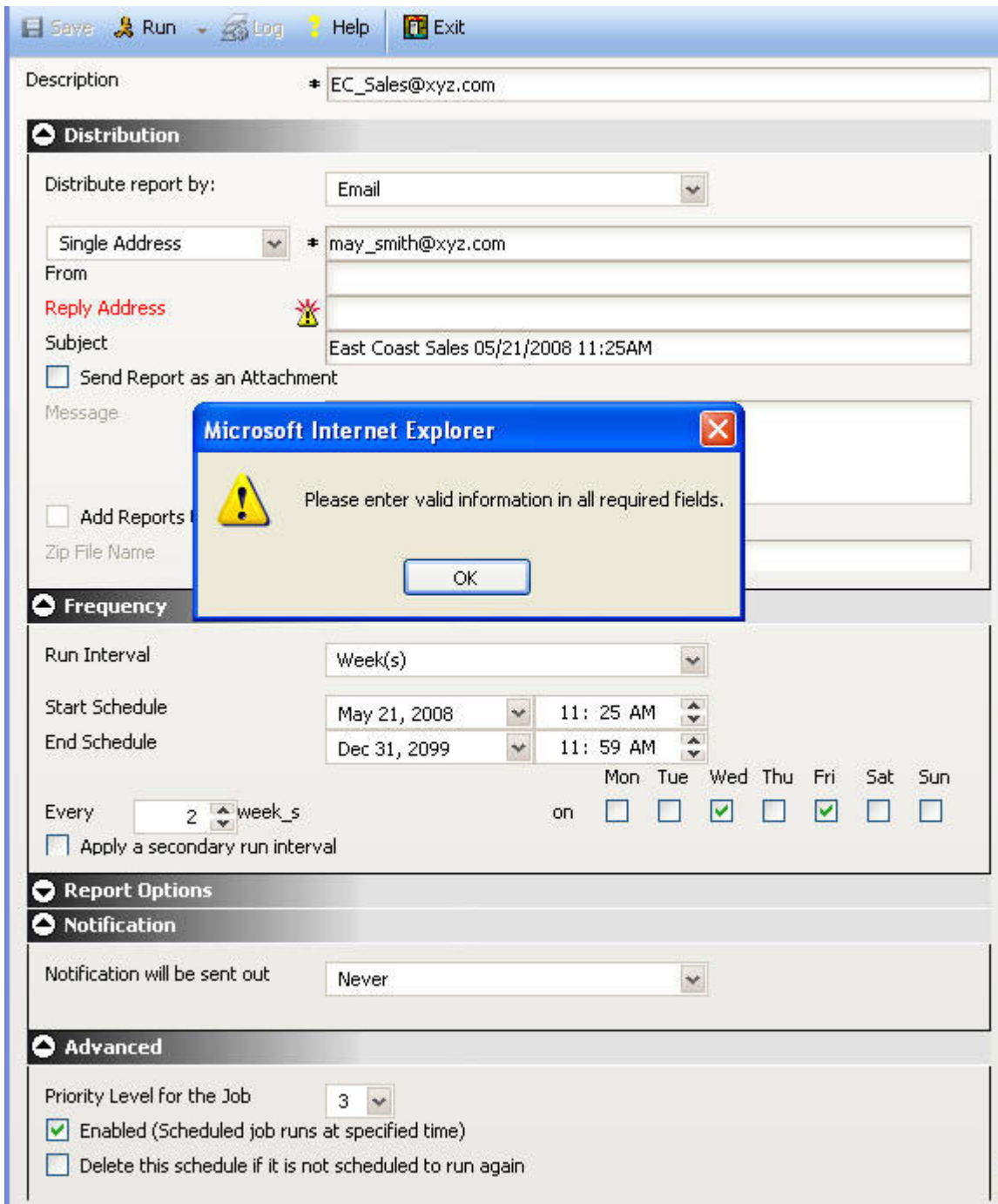
Re-enter the information in the fields that have alerts and click **Save**.

Running the Schedule from the Scheduler Tool

To run the schedule you just created from the Scheduler tool, click **Run** at the top of the tool.

Distribution Options

The Distribution section of the Scheduler tool provides the options available for distributing the scheduled report. You can distribute a report using one of the following methods:

- Email
- Printer
- Web Query

This section describes the options available for each distribution method.

Distributing a Report Using E-mail

When you distribute a report through e-mail, you can include the report in the body of the e-mail (known as an inline e-mail message) or send it as an attachment. Distributing a report as an inline e-mail message is particularly useful when the report is distributed to mobile devices, fax machines, or through e-mail systems that do not support attachments. You can also distribute a report to a Fax machine, as explained later in this section.

The following image shows the e-mail distribution options in the Distribution Options section of the Scheduler tool.

Distribute a Report Using E-mail:

1. From the Distribute report by: drop-down list, select **Email**.
2. From the drop-down list under Distribute report by:, select the form of the e-mail address and type or select the accompanying value in the text box next to the selection. The options are:
   - **Single Address.** The report will be sent to a single e-mail address. Type the e-mail address in the accompanying text box. Report Broker cannot validate the e-mail address. An incorrect or unresolved address may not be noted in the log file. This is dependent upon the ability of the SMTP mail server to validate the e-mail address. The SMTP mail server returns undeliverable e-mail messages to the reply address you specify for the schedule. The burst option is not supported with this distribution option.
     Tips:

- You can specify multiple e-mail addresses within a Single Address. Separate each e-mail address with a comma or a semicolon. The e-mail addresses will appear in the To: line of a single e-mail when the scheduled output is distributed. Each individual e-mail address can be a maximum of 130 characters according to the SMTP specification. The total maximum length of this field is 800 characters.

- Additionally, you can use group mail lists (defined on your mail server) with the Single Address option. Group mail lists enable you to distribute a report or notification to multiple recipients without having to maintain multiple e-mail addresses in the Report Broker Repository. The format of the group mail list is dependent upon the mail server being used. For example, if you are using a Microsoft Exchange Server and your group mail list is defined as #group1, you would enter group1@*listdomain* in the Single Address field. If the group mail list contains a space within its name, it must be enclosed within quotation marks. For more information, see your mail server administrator.

- **Distribution List.** The report will be sent to all e-mail addresses on the selected distribution list. Select a distribution list from the drop-down list, which is activated when you select the Distribution List option. For information on creating a distribution list, see Creating and Maintaining a Distribution List.

3. In the From field, type any value (for example, the name of the person creating the schedule). This field is not required by Report Broker, but may be required by your e-mail system.

4. In the Reply Address, type a valid e-mail address. If recipients reply to the e-mail, their messages are sent to this address. If your e-mail system is unable to deliver the content, the undeliverable output message is also returned to this address. This field is required by Report Broker.

5. In the Subject field, type the text you want to appear in the message subject line. This information is not required by Report Broker, but may be needed by your e-mail system. The value you entered in the schedule Description field is used as the default Subject value.

   When creating a schedule, you can place multiple parameters and burst values from the scheduled procedure in an e-mail subject line. This allows the subject to be dynamically created in order to personalize e-mails to recipients. Parameters referenced in the subject line must be stored with the schedule information in the Report Broker tables.

   Parameters must be specified in the format '&*parmname*' (where *parmname* is the name of the parameter). Burst values must be specified using the syntax '%BURST'.

   Note:

   - The parameter name cannot also be the name of an existing report procedure. You may specify an unlimited number of parameters.

   - If you have multiple burst values in a Distribution List, only the first value sent to the Report Broker Distribution Server is included in the subject.

   For an example of specifying parameters and burst values in an e-mail subject line, see E-Mail Distribution Examples.

6. Specify whether or not you want the report to be sent as an e-mail attachment by selecting or de-selecting the Send Report as an Attachment option.

   If you select the Send Report as an Attachment option, you can also:

   a. Type a message to appear in the e-mail body.

b. Select the option Add the Reports to Zip File, and provide a name for the file in the Zip File Name field.

Consideration When Distributing a Report Using E-mail:

Report Broker transfers e-mail asynchronously to your e-mail system. The delivery time depends upon your e-mail system.

Exchange Mail Server Considerations:

- If Report Broker end users do not see a From value that is specified in the schedule information for the e-mail they receive in the e-mail client inbox, this is due to their mail server configuration. For more information on how to control this setting for a Microsoft Exchange Server, see the following URLs:
  - http://support.microsoft.com/default.aspx?scid=kb;en-us;288635
  - http://support.microsoft.com/kb/174755/EN-US/
  - http://support.microsoft.com/default.aspx?scid=kb;en-us;242301

  Alternatively, you can contact Microsoft customer support for assistance. If you are using a mail server other than a Microsoft Exchange Server, refer to that vendor's documentation or contact their customer support for assistance.

- When distributing a report to an Exchange Server, attachment names have a .txt extension added to WP and DOC format output. This occurs because WP and DOC formats are not mapped to a standard MIME type. The Exchange Server interprets WP and DOC output as a text/plain MIME type and adds the .txt extension to the end of the file name. For example, *hold.wp.txt* or *hold.doc.txt*.

- Exchange Server individual recipient names (*person@company.com*) and Exchange Server-defined distribution lists (*list@company.com*) can be specified in Report Broker Distribution Lists. However, Exchange Server user-defined distribution lists cannot be referenced in a Report Broker Distribution List. They are defined internally on the user's machine. The Report Broker Distribution List is parsed by the Exchange Server and not the user's mail client. Since a Report Broker Distribution List is parsed by the Exchange Server, when an external server submits an Exchange Server distribution list, it must contain the proper prefix and the proper domain (which is prefaced by an @ sign). Consult your Exchange Server Administrator for these values.

CC:Mail Considerations:

When you distribute Report Broker generated e-mail attachments greater than 20K in size to users of cc:Mail, cc:Mail renames the attachment textitm.txt, ignoring the file name and extension supplied by the user. This change affects DOC, HTML, and WP formats on UNIX and Windows platforms. However, despite the naming convention issue, the attachments contain the correct output and can be viewed if the attachment is saved to disk with the correct extension.

E-Mail Distribution Examples:

This section provides an example of using burst values and multiple parameters when sending the report in an e-mail, and distributing a report to a Fax machine using e-mail.

Specifying Parameters and Burst Values in an E-mail Subject Field

This schedule specifies burst values and multiple parameters in the e-mail subject line:

The following image shows an example of a resulting e-mail subject line.
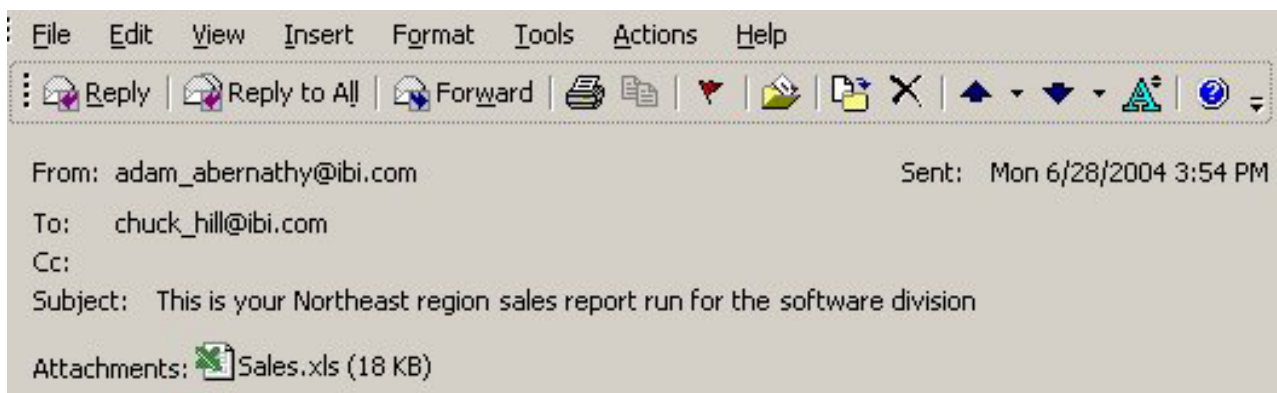
Distributing a Report to a Fax Machine Using E-mail

To distribute an inline e-mail message to a fax machine, you must register your e-mail address with a third-party e-mail distribution provider. The features offered by providers, (such as supported area codes and file formats), in addition to requirements on the structure of e-mail parameter values, may vary. It is important that you select a provider whose features are compatible with Report Broker.

Note: You cannot distribute an e-mail attachment to a Fax machine.

The following example shows how to distribute a report directly to a fax machine. The e-mail address, john_doe@xyz.com, was registered with the e-mail distribution provider called *emfax.com*. During the processing of the request, Report Broker generates the scheduled report output and then distributes it using the e-mail address of *emfax.com*. The reply address specified in Report Broker is the registered e-mail address that is validated by *emfax.com*. If the e-mail address is valid, *emfax.com* distributes the report to the fax number 12129999999. The validation of the registered e-mail address is performed by *emfax.com*, not by Report Broker.

Note: The syntax used in this illustration is specific to this example. The required syntax for your provider may be different.

The following image shows the Distribute section of the Scheduler tool, which contains the name of the report to be distributed, fields, drop-down lists, and option buttons that enable you to specify the destination(s) for the scheduled report, and an e-mail address to which report recipients can reply.

1. From the Distribute report by drop-down list, select **Email**.
2. From the mail drop-down list, select **Single Address**.

   In the accompanying text box, type the e-mail address to be used by the e-mail provider according to their requirements. In this example, it is *phone-number@emfax.com* or 12129999999@*emfax.com* (where *emfax.com* is the name of your e-mail provider).

   Note: You can also select a Distribution List. However, be sure to use the syntax required by your provider.
3. In the From field, type any value (for example, the name of the person creating the schedule). This field is not required by Report Broker, but may be required by your e-mail system or by the e-mail provider.
4. In the Reply Address field, type your registered e-mail address. If your e-mail system is unable to deliver the content, the undeliverable output message is returned to this address. This field is required by Report Broker.
5. In the Subject field, type the text you want to appear in the message subject line. This information is not required by Report Broker, but may be needed by your e-mail system or e-mail provider.
6. Continue selecting other schedule options, then click **Save** to save the schedule.

Distributing a Report to a Printer

The following image shows the printer distribution options in the Distribution Options section of the Scheduler tool.

Distribute a Report to a Printer:

1. From the Distribute report by drop-down list, select **Printer**.
2. From the drop-down list that appears under the Distribute report by option, select the form of the e-mail address and type or select the accompanying value in the text box next to the selection. The options are:
   - **Single Printer.** Specify the printer using the following format.
     `queue@printserver`
     where:

   queue

   > Is the name of the printer queue.

   printserver

   > Is the host name or IP address of the printer.
   >
   > Report Broker can differentiate between the printer queue and the printer host name/IP address due to the presence of the '@' separator. Although Report Broker supports specifying only the host name or IP address of the printer, Information Builders recommends specifying both the printer queue and host name/IP address when distributing Report Broker output to a printer. The maximum length of this field is 800 characters.

   - **Distribution List.**

     The field next to Distribution List provides a drop-down list of all printer Distribution Lists that you are the owner of, and all public printer Distribution Lists. For more information about creating a Distribution List, see Creating and Maintaining a Distribution List.

   Note: If the printer you specify is unrecognized, the "Cannot connect to specified printer" message is recorded in the log file when the Report Broker Distribution Server tries to distribute the report.

Distributing a Report to Web Query

The following image shows the Web Query distribution options in the Distribution Options section of the Scheduler tool.
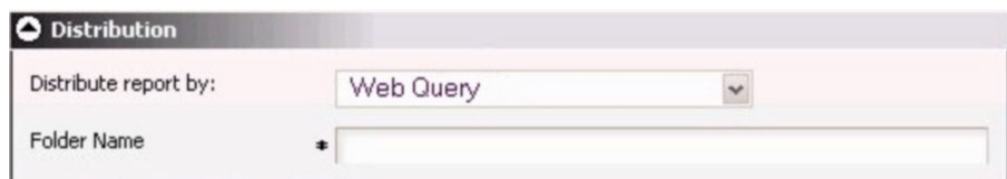
Distribute a Report to Web Query:

To distribute a report to Web Query:
1. From the Distribute report by drop-down list, select **Web Query**.
2. In the Folder Name field, type a name for the folder on Web Query to which you will distribute the report.

   This description must not exceed 90 characters.

Considerations When Distributing a Report to Web Query:

When distributing a report to Web Query, consider the following:
- You cannot distribute Standard Reports back to Web Query.

- Multiple Web Query reports are distributed to the same folder if they both run at the same time. We recommend assigning each Web Query report a different start time.
- When distributing a report to Web Query, you can only drill-down to another report if it is a procedure on the Reporting Server. There is currently no support for drill-down functionality to Web Query procedures.
- If you are distributing a large report to Web Query, take into consideration the amount of memory configured on your application server. The report output returned to Web Query will be stored in memory by the Client (Servlet) and then written to the Web Query Repository. If there is not enough memory available on your application server, the Report Broker servlet trace file will contain the message "return code not found. See your application server trace for information". The log report will include the message "Unable to distribute to Web Query: return code not found". The problem can be resolved by one or more of the following:
  - Increase the memory available on your application server. Contact your Report Broker Administrator.
  - Modify your report so that it returns a smaller report or less output.
  - Limit the number of scheduled jobs distributing to Web Query during the same time period.

Frequency Options

You can schedule a report to run just once or repeatedly, for example, twice a week or the last Thursday of every month. This section describes the options available in the Scheduler tool Frequency section when the schedule is set to the following run intervals:

- Once
- Minute(s)
- Hour(s)
- Day(s)
- Week(s)
- Month(s)
- Year(s)

Note: Report Broker Administrators can define dates on which schedules cannot run or be set to run. These are known as schedule blackout dates and can be viewed from the DB2 Web Query Report Broker tab. For more information, see Schedule Blackout Dates.

The Once Run Interval

The Once option in the Run Interval drop-down list sets the job to execute immediately. You can modify the date or time if you do not want the schedule to run immediately. You can specify the date and time you want the schedule to run using the Start Schedule options, as shown in the following image.

To select a date, choose a date from the drop-down date calendar. To select a time, select either the hour or minutes and use the up and down arrows to increase or decrease the value.

The Minutes Run Interval

The Minute(s) option in the Run Interval drop-down list, sets the schedule to run every n minutes.

In the Every minute(s) field, type or select the minutes interval (1 to 59), check the days of the week on which you want to run the schedule, and select the Start Schedule and End Schedule date and time to define the time period in which the schedule will run. For example, the following schedule will run every 30 minutes on Mondays and Fridays beginning at noon May 16, 2008 and ending 6:00 PM October 30, 2010.

Tip: Selecting this option may affect system performance if you choose to run the schedule every 5 minutes or less. We recommend specifying a minimum of 30 minutes. The minute interval option is primarily useful for alert schedules.

The Hourly Run Interval

The Hour(s) option in the Run Interval drop-down list, sets the schedule to run every n hours.

In the Every hour(s) field, type or select the hours interval (1 to 24), check the days of the week on which you want to run the schedule, and select the Start Schedule and End Schedule date and time to define the time period in which the schedule will run. For example, the schedule shown in the following image will run every 3 hours on Mondays and Fridays beginning at noon May 16, 2008 and ending 6:00 PM October 30, 2010.

The Daily Run Interval

The Day(s) option in the Run Interval drop-down list, sets the schedule to run every *n* days. In the Every day(s) field, type or select the days interval to run the schedule and select the Start Schedule and End Schedule date and time to define the time period in which the schedule will run. For example, the schedule shown in the following image will run every 5 days beginning at noon May 16, 2008 and ending 6:00 PM October 30, 2010.

You can also set a secondary run interval. For information about this setting, see Applying a Secondary Run Interval.

The Weekly Run Interval

The Week(s) option in the Run Interval drop-down list, sets the schedule to run every *n* weeks.

In the Every week(s) field, type or select the weekly interval to run the schedule, check the days of the week on which you want to run the schedule, and select the Start Schedule and End Schedule date and time to define the time period in which the schedule will run. The following schedule will run every two weeks on both Monday and Friday beginning at noon May 16, 2008 and ending 6:00 PM October 30, 2010.

Important: When selecting the Week(s) interval, set the Start Schedule to the date of the first day (current or future) of the week you want the schedule to run. If you select the current date, then you must make sure that the Start Schedule time is later than the current time when you save the schedule. If the Start Schedule time is less than or equal to the current time, the calculation for the next run time results in the schedule not running on the current date.

You can also set a secondary run interval. For information about this setting, see Applying a Secondary Run Interval.

The Monthly Run Interval

The Month(s) option in the Run Interval drop-down list, sets the schedule to run every n months. You can then refine the monthly interval with one of the following options (Note that these options are mutually exclusive.):

- Every first, second, third, fourth, or last n day of the week (where n is Monday, Tuesday, Wednesday, Thursday, Friday, Saturday, or Sunday) every n months.
- Specific days every n months.

Also select the Start Schedule and End Schedule date and time to define the time period in which the schedule will run. The following image shows a schedule set to run on the first Monday of every month beginning at noon May 16, 2008 and ending 6:00 PM October 30, 2010.

The following image shows a schedule set to run on the 2nd, 9th, 16th, 23rd, and 30th of every month, regardless of the day of the week those dates fall on.

You can also select the **Last Day of the Month** option at the end of the calendar to run the schedule on the last day of the month.

Note: When selecting the Month(s) interval, set the Start Schedule to the date of the first day (current or future) of the month you want the schedule to run. If you select the current date, then you must make sure that the Start Schedule time is later than the current time when you save the schedule. If the Start Schedule time is less than or equal to the current time, the calculation for the next run time results in the schedule not running on the current date.

You can also set a secondary run interval. For information about this setting, see Applying a Secondary Run Interval.

The Yearly Run Interval

The Year(s) option in the Run Interval drop-down list, sets the schedule to run every *n* years during a specific time period. The following image shows a schedule set to run every 2 years beginning at noon May 16, 2008 and ending 6:00 PM October 30, 2010.
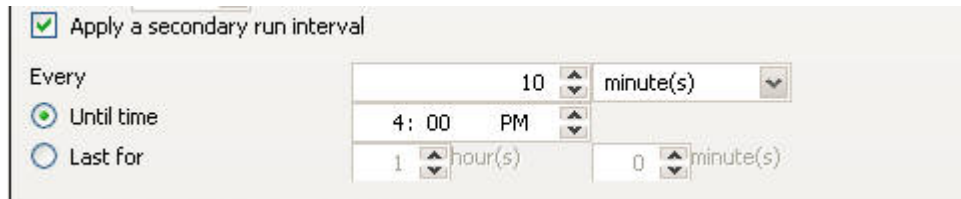
You can also set a secondary run interval. For information about this setting, see Applying a Secondary Run Interval.

Applying a Secondary Run Interval

The Apply secondary run interval option allows you to create a secondary run interval within the day the schedule runs. You can apply the secondary run interval every *n* minutes or hours for a specified number of hours/minutes or until a specified time. This option is available for schedules that run every day(s), week(s), month(s), or year(s).

When you select Apply a secondary run interval, the options related to this setting appear in the Frequency section. The following image shows an example of set secondary run interval options.

The Apply secondary run interval options are:

- **Every *n* [minutes | hours].** Applies the secondary run interval every *n* minutes or hours (in this example every 10 minutes) within the day the schedule runs.
- **Until Time.** The time up until which the secondary run interval will be applied. In this example, the schedule will be rerun every 10 minutes until 4:00 P.M.
- **Lasts for *n* hour(s) *n* minute(s).** The duration, specified in hours and minutes, during which the secondary run interval will be applied. This option and the Until time option are mutually exclusive, so this option is inactive in this example.

Note: The secondary run interval will not be validated when the schedule is created. Instead, validation is performed every time the schedule's next run time is calculated when running within the secondary run interval. The secondary run interval cannot exceed the next run time for the primary run interval. For example, a daily schedule cannot have a secondary run interval greater than Every 1 day(s). If a secondary run interval is scheduled to run after the schedule's next primary run interval, the secondary run interval is stopped and an error message is displayed to the user and written to the log file.

Report Options

This section describes the options available in the Report Options section of the Scheduler tool. You can set these options to send the report in a specific format (for example (HTML, EXL2K, and so on), whether or not you want to burst the report and if so, the burst values the report will use, and specify parameter values to pass to the report when it runs.

The following images shows the Report Options section of the Scheduler tool.

Report Formats

Select the format of the report from the Send the Report in this Format drop-down list of the Scheduler, Report Options area. The formats available to a scheduled report are:

- AHTML
- DHTML
- DOC
- EXL2K
- EXL2K FORMULA
- EKL97
- HTML (This is the default value.)
- JPG
- PDF
- PS
- PNG
- SVG
- WF

There are limitations on what formats are valid for certain options. Not all formats are supported for bursting or for printing. If the report format statement in the coded procedure is a styled report format, then you must select a styled report format (for example, AHTML, EXL2K, EXL97, HTML, PDF). The report format specified in this field overrides the format statement in the procedure.

If the report format statement in the coded procedure is a specialized report format, such as EXL2K FORMULA, or if it is Coordinated Compound PDF Report, it is required that you select the same report format that is specified in the procedure.

If a TABLE request containing an unstyled binary format is scheduled for distribution as a styled ASCII format (for example, EXL2K, HTML, or PDF), the report is not created by the Reporting Server and the Report Broker log report

contains a "No destination found for this distribution" message. This also occurs if a TABLE request containing a styled ASCII format is scheduled for distribution as an unstyled binary format.

For details on each format, see Chapter 18, "Report Broker Formats for Scheduled Output," on page 765.

Bursting a Report

The burst feature enables you to break a report into sections and distribute relevant sections to individual users. To burst the report, select the **Burst this Report** check box. Burst values may be specified in a Distribution List.

Bursting Guidelines and Limitations:

Each section of a burst report is saved as a separate file. If you are distributing a burst tabular report, the burst value is determined by the first BY field. If you are distributing a burst graph report, the burst value is determined by the second BY field. The burst value is automatically determined by the internal matrix of the Reporting Server. The internal matrix is a memory area that stores each database field value and calculates values referenced by the TABLE or GRAPH request.

When a report is burst, all data values generated for each burst section are returned to the Report Broker Distribution Server.

- For the Web Query distribution methods, each burst section is distributed to Web Query. Each burst section may be viewed by the owner of the schedule.
- For the e-mail and printer distribution methods, specific burst sections are distributed based on the burst values specified when creating the Distribution List, single location, or dynamic address list used by the schedule.

The following are guidelines and limitations that apply to the Report Broker burst feature:

- **Case.** Burst values are case-sensitive.
- **Formats.** AHTML, DHTML, EXL2K, EXL97, EXL2K FORMULA, HTML, JPG, PDF, PNG, and SVG support bursting. Each burst section of the report will be named *burstvalue.format* (for example, Northeast.pdf).
- **ACROSS command.** This command is not evaluated as a primary sort field. To burst a report, you must also include a BY field. Bursting occurs on the BY field.
- **FML reports.** Bursting an FML report is supported only if the request has a BY field.
- **Coordinated Compound Reports.** Report Broker can burst and distribute coordinated compound reports developed using the PDF Layout Painter in Developer Studio in DHTML and PDF formats. Compound reports that are not coordinated cannot be burst.
- **TABLEF.** No internal sort processing is performed. The specification of a BY field requires that the data already be sorted in the data source.
- **ON TABLE SUBHEAD/ON TABLE SUBFOOT.** Creates a SUBHEAD for only the first page of the report, and a SUBFOOT for only the last page of the report. When bursting a report, the SUBHEAD and SUBFOOT should occur for each sort break. Therefore, specify the primary sort field in place of TABLE in the ON command. For example:

```
ON primarysortfield SUBHEAD
```

- **AnV field types.** Bursting is not supported on a field with the AnV (where n is an integer value) field type.

Report Parameters

If the report you are scheduling contains parameters, they are listed in the Report Options section of the Scheduler tool. The following images shows the Report Options section for a report with parameters.

The report parameter name automatically appears in the Name column. The value or possible values of the parameter available to you in the Reports Options section depends on the report procedure. The report procedure either assigns no value to a parameter, assigns a default value, or provides a selection of values from which you can choose. For considerations when specifying parameter values for a procedure and for examples of the report procedure code that determines the available values, see Considerations When Specifying Parameter Values.

The possible presentation of report parameters in the Report Options section are:

- no value, where you assign the value in the schedule.
- A default value.
- A static list of values from which you can select one value.
- A dynamic list (generated from the database the report is using) of values from which you can select one value.
- A static list of values from which you can select multiple values.
- A dynamic list (generated from the database the report is using) of values from which you can select multiple values.

When you access the report Scheduler tool, parameters with default values have a check next to their name and parameters with a choice of values have a grayed-out check mark next to their name. The following image shows a parameter with a default value (Enter a Region) and parameters that provide a choice of values (Store Code and Product name).
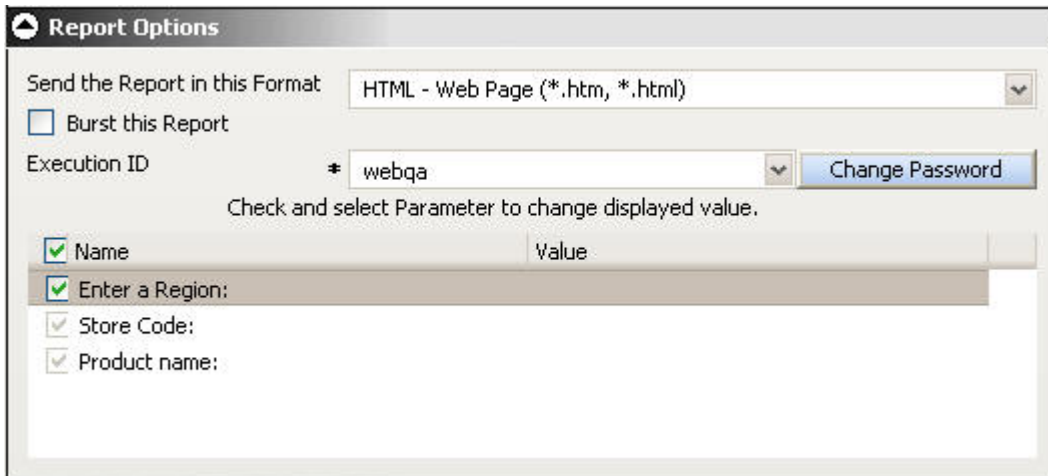
To select the default value for a parameter, uncheck the box next to the parameter name. If you want all parameters in the list to use their default values, uncheck the **Name** option. The default value for the unchecked parameters appear in the Value column. The following image shows the Enter a Region parameter unchecked and its default value displayed in the Value column.
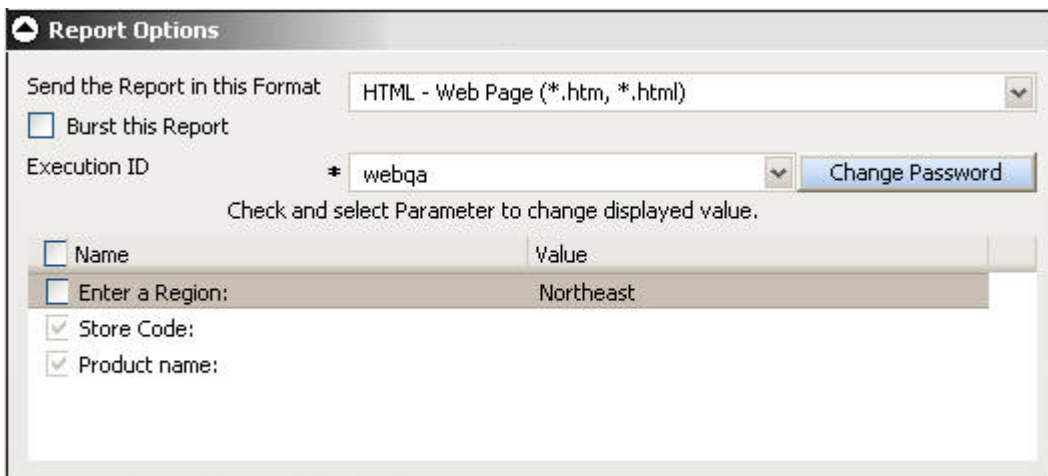
Define a Parameter Value in the Scheduler Tool:

Parameters with an active check mark next to their name have a default value or no value assigned. To change a default value or assign a new value for a report parameter:

1. In the Name column, click the parameter name that you want to assign a value to. The Parameter Editor window opens, as shown in the following image.

2. Type a value for the parameter.
3. Click **OK**.

   The value appears in the Value column next to the parameter, as shown in the following image.
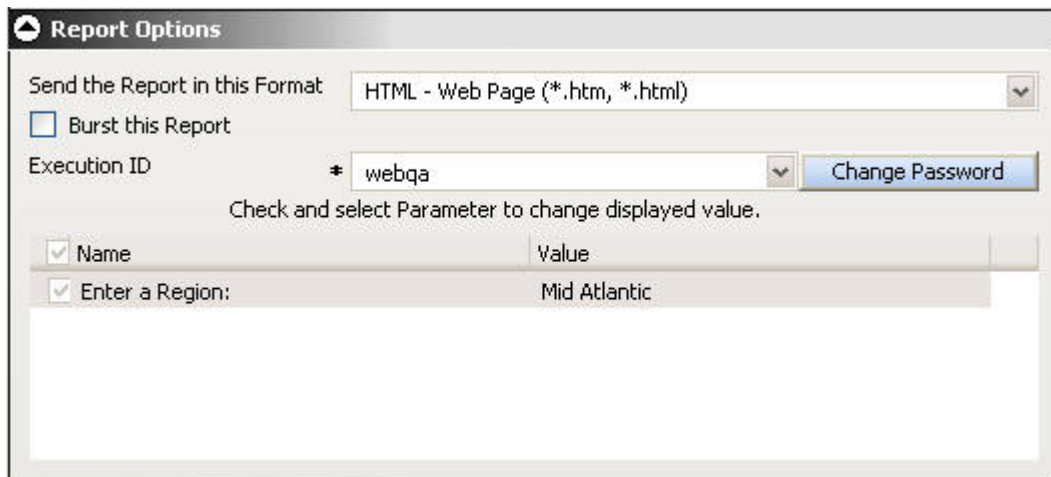
Select Parameter Values From a List:

Parameters with a grayed-out check mark next to the parameter name provide a list of values from which you can choose.

1. In the Name column, click the parameter name that you want to assign a value to.
2. From the Parameter Editor window, select the values you want, as follows. Use standard Windows methods to select individual or contiguous items.
   - From the single-select list, select the value for the parameter. The following image shows the Value drop-down list from which you can choose the value.

- From the multi-select list, select either No Selection or one or more values, as shown in the following image.

- From the multi-select list, select either Select All or one or more values, as shown in the following image.
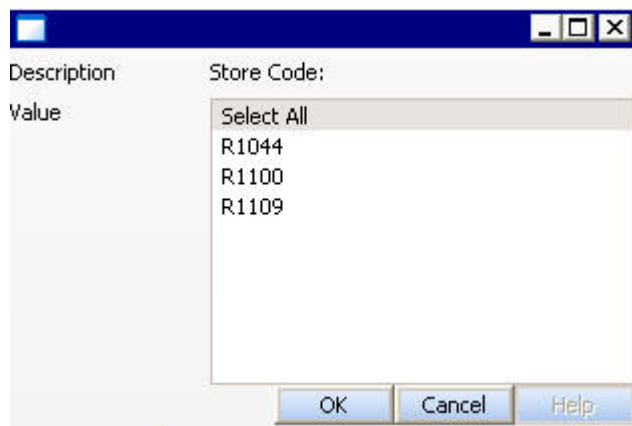
Considerations When Specifying Parameter Values:

The following are considerations when specifying parameter values for a procedure:

- The maximum number of characters for each individual parameter value is 3200. You can store multiple values for a single parameter. Multiple values for a parameter is stored as one entry which must not exceed the 3200 maximum character limit.
- Report Broker displays the description for the parameter when it is specified in the procedure. Otherwise, Report Broker displays the parameter name.
- Report Broker performs field validation for numeric and alpha formats, and range value validation.
- Report Broker displays default variable values, as well as static or dynamic single-select and multi-select lists.

  Note:

  – The 'Select All' option is displayed for static multi-select lists. When selected, this option includes every value for the selected parameter. The 'No Selection'

option is displayed for dynamic multi-select lists. When selected, this option does not perform any data selection test on that field.

– Using the -HTMLFORM command to create a dynamic selectable list of parameter values is not supported in Report Broker.

– As an alternate method of creating a dynamic selectable list of parameter values, you can use the schedule technique in the HTML Layout Painter.

- The Specify All Parameter Values check box is always enabled if there is any parameters available for the report (Standard Report, My Report, or WF Server Procedure). You can check or uncheck this option to quickly enable or disable the check boxes for each individual parameter. If the check box for an individual parameter is unchecked, the input field for the parameter value is disabled, the default parameter value(s) are displayed (but cannot be changed), and no parameter data is saved to the Report Broker Repository.

- Report Broker will not prompt for variables with defaults set by the -DEFAULTH command. The purpose of the -DEFAULTH command is to be able to assign amper variables a default value and not be dynamically prompted for that variable.

- When specifying parameters with special characters (for example, %, &, |):

  If you are specifying the entire WHERE condition as the parameter value, you must enclose the value within two single quotation marks rather than a double quotation mark. For example: ''WHERE CAR NOT LIKE MOTO%''

  If you are only specifying a value as the parameter value, you do not need to enclose the parameter value within quotation marks. For example: O&DINFO

Note: You can only schedule saved procedures using Report Broker; HTML forms cannot be scheduled using Report Broker. Parameter prompting is not available in pre or post processing procedures.

Notification Options

The Notification area of the Scheduler tool provides the options to send a notification of the schedule status to specific e-mail recipients. You can send notification:

- **Never.** Report Broker will not send a notification of the schedule status under any circumstances. This is the default value.

- **On Error.** The specified users are notified when errors are encountered while running the schedule. We recommends using the On Error notification option.

- **Always.** The specified users are always notified when the schedule runs.

Setting On Error and Always Notification

When you select the On Error or Always notification option, additional options become available, as shown in the following image.

The On Error and Always notification options are:

- **Type.** Select if you want a brief or detailed notification. follows.
  - **Full Notification.** A full notification sends a complete log report as an e-mail attachment.
  - **Brief Notification.** A brief notification sends the ID and job description of a schedule, as well as messages about the schedule (such as "Completed Successfully").
  - **Full & Brief Notification.** Sends both types of notifications.

  Tips:
  - We recommend using the Brief Notification option when you are sending notification to devices that have limited memory, such as pagers and cell phones. If you want to notify multiple recipients, you can use group mail lists defined on your mail server provided that you append an @ sign followed by a valid domain.

- **Full Notification.** When the notification type is Full Notification or Full & Brief Notification, type the e-mail address to which you want a full notification sent. There is no syntax error checking for this field.

- **Brief Notification.** When the notification type is Brief Notification or Full & Brief Notification, type the e-mail address to which you want a full notification sent. There is no syntax error checking for this field.

- **Reply Address.** Type the sender's e-mail address. If report recipients reply to the report sender, their messages are sent to this address. If your e-mail system is unable to deliver a report, the undeliverable report message is also returned to this address.

- **Subject.** Type the text you want to display in the subject line of the e-mail notification. There is a limit of 255 alphanumeric characters. By default, this field contains the report name and data and time stamp.

Notification and Log Information for Unavailable Options

When schedules with unavailable Task Types and/or distribution methods are not permitted to run, error notification is triggered. The full and brief notifications and the log report informs you that the Report Broker Administrator or the owner of the schedule must change the unavailable Task Types and/or distribution methods in the schedule.

When schedules with unavailable Task Types and/or distribution methods are permitted to run, normal job execution occurs and a message appears in the log report indicating that your Report Broker Administrator is allowing existing schedules using the unavailable Task Types and/or distribution methods to run.

Running a Schedule from Control Language (CL)

Once you have created a schedule in Report Broker, you can run it at any time from the Web Query Report Broker application. There are times when a business user is not logged into Web Query, but may want to run a schedule as part of a business workflow process. The Report Broker CL program allows you to run schedules from outside of Web Query.

If you are a Web Query named licensed user or developer, you can run the schedules that you own. If you are a Report Broker administrator, you can run any schedule.

To run a Report Broker schedule outside of Web Query, type the following in a 5250 emulation session CL command line:

RUNBRSCHED

```
                   Run a Report Broker Schedule (RUNBRSCHED)

Type choices, press Enter.

Schedule ID  . . . . . . . . . .   _____   Character value
Run with User ID . . . . . . . .   *CURRENT__     Character value


















                                                                      Bottom
F3=Exit      F4=Prompt     F5=Refresh    F12=Cancel    F13=How to use this display
F24=More keys
```

At the prompt, type the ID of the schedule you want to run, and the user ID for the job being submitted.

Note: The Schedule ID is a unique 12-alphanumeric string assigned to a schedule when it is created. The Schedule ID is found in the schedule list of the Web Query's Report Broker tab.

A message informs you that the job was submitted.

Maintaining a Schedule
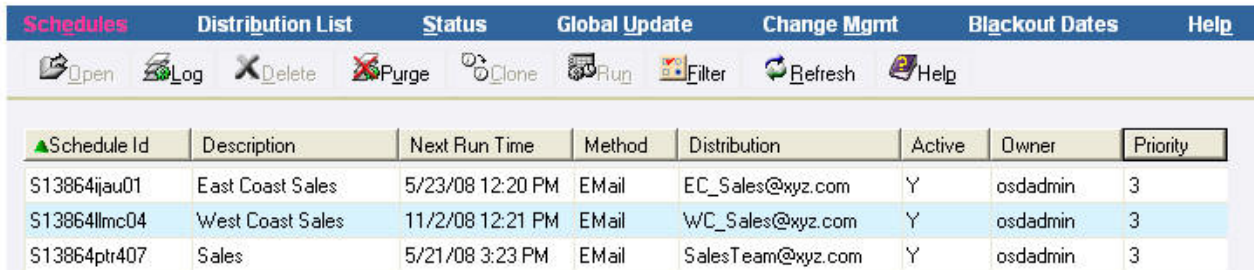
This section explains how to edit the properties of an existing schedule and to clone, delete, and check the status of a schedule. It also explains how to run a log report to obtain information about a schedule, and to globally update schedule properties, such as the schedule owner.

Maintenance Options in the Report Broker Interface

After you create a schedule, you can edit its properties at any time or delete the schedule when it is no longer needed. If a schedule contains properties you want to use in a new schedule, the clone option creates a template with those properties for the new schedule. You can also check the status of a schedule and run a log report to obtain detailed information about the schedule. Additionally, you can purge log records to conserve space in the log file and view the dates on which schedules cannot run or be set to run.

The maintenance options are available in the Report Broker interface. To view a list of your schedules, click **Schedules**.

The Schedules pane opens and displays a list of your schedules along with information about the schedule, such as its next run time. The following image shows how each scheduled job contains information displayed under column headings for Schedule Id, Description, Next Run Time, Method, Distribution, Active, Owner, and Priority. The pane includes toolbar of icons that enable you to perform functions on each scheduled job. Sorting is available for each column by clicking the column heading.

If you want a list of schedules with specific properties, such as those with an e-mail distribution, use the Filter options. For details, see Filtering the Schedule List.

From the Schedule pane, you can:
- Open the Scheduler tool to edit the properties of a selected schedule. For more information, see Editing a Schedule.
- View a log report for one or more selected schedules. For more information, see Viewing a Log Report.
- Delete one or more selected schedules. For more information, see Deleting a Schedule.
- Purge log file information for one or more selected schedules or for all schedules in your list. For more information, see Purging the Log File.
- Clone a selected schedule. For more information see, Cloning a Schedule.
- Run one or more selected schedules.
- View the dates on which schedules cannot run or be set to run. For more information, see Schedule Blackout Dates.
- Refresh the current schedule list with any newly created schedules.
- Open the online Help.

Filtering the Schedule List

To focus the types of schedules that appear in the Schedule list, for example to list only those schedules with a certain priority level or distribution method, use the filter option.

Use the Schedule Filter:

To focus the list of schedules:

1. From the Report Broker interface Schedule pane, click **Filter**.

   The Schedule Chooser dialog box opens, as shown in the following image.
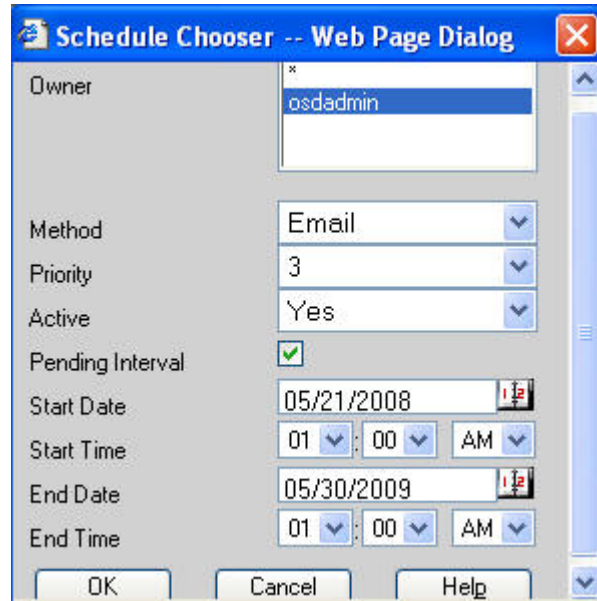
2. Select the filter parameters as needed. the parameters are:
   - **Owner.** List all schedules by the selected owner.
   - **Method.** List all schedules by this report distribution method.
   - **Priority.** List all schedules by this schedule priority level.
   - **Pending Interval.** List all schedules using these date and time parameters: Start Date, Start Time, End Date, and End Time. To set the Start Date and End Date, click the calendar icon next to the date and select the date you want in the calendar dialog box.

Editing a Schedule
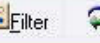
You can edit the schedules that you own at any time by using the following procedure.

Edit a Schedule

To edit a schedule:

1. Select the schedule you want to edit.

   When you select a schedule, the schedule function icons become available. The following image shows the Sales schedule and the Open icon selected in the Schedule pane.

2. Click **Open** or double-click the schedule.

 The Scheduler tool opens. The following image shows the Scheduler tool with all information previously entered for the Sales schedule.

3. Make the necessary changes to the schedule.

   For more information about navigating through the Scheduler tool, see Creating a Schedule.

4. Click **Save**.

   A window opens with a message that the schedule has been created.

5. Click **OK**.

Considerations When Editing a Schedule

The following are considerations when editing a schedule using the Scheduling Wizard:

- If a schedule already exists and your Report Broker Administrator changes the available options, the existing schedule runs as previously configured, regardless of the changes. However, if you attempt to specify unavailable options (Task Types, distribution methods, or report formats) when editing a schedule, a

message is displayed informing you that the options available for scheduling have been changed by your Report Broker Administrator. Information is then displayed that describes the change(s) that you must make for the schedule to use available options. Changes to the schedule cannot be saved until the schedule uses available options.

- If you selected once for the run interval, the schedule runs immediately unless you change the Start Time to a time later than the current time. All other run intervals run at the next run time of the schedule.

- If you want your selected schedule(s) to run immediately, click **Run**. A new schedule ID is created for the job.

Cloning a Schedule

From the Schedule window in the Report Broker interface, you can clone existing schedules that you own using the following procedure.

Clone a Schedule

To clone a schedule:

1. In the Schedule pane, select the schedule you want to clone.

    The following image shows the Sales schedule and Clone icon selected in the Schedule pane.

2. Click **Clone**.

    The Clone selected schedule -- Web Page Dialog box that opens, as shown in the following image. This dialog box prompts you to type a description for the cloned schedule.
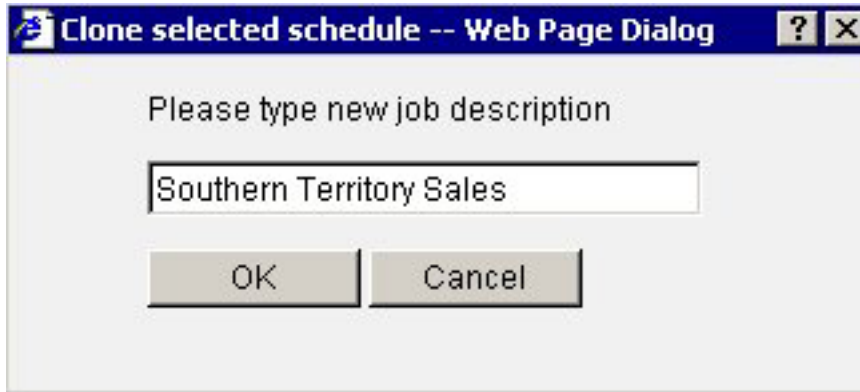
3. Type a description for the new schedule (for example, Southern Territory Sales) and click **OK**.

   The following image shows the new cloned schedule in the Schedule window. Note that the cloned schedule is disabled by default, as specified in the Next Run Time column.

4. To edit the settings and enable the schedule, perform the following steps:

   a. Click **Open** or by double-clicking the schedule.

      The Scheduler tool opens, displaying the schedule settings.

   b. Make the necessary changes to make this schedule unique from the original schedule.

   c. In the Advanced area, check the **Enabled** check box.

   d. Click **Save**.

      The Your schedule has been created message appears.

   e. Click **OK**.

   f. In the Report Broker interface, click **Refresh**.

      The following image shows that the Southern Territory Sales schedule is active, as noted in the Next Run Time column of the Schedule pane. If the only change you made was setting the schedule to enabled, it contains the same properties as the schedule from which it was cloned.

Deleting a Schedule

You can delete a schedule you own at any time using the following procedure.

Delete a Schedule

To delete one or more schedules:

1. Select the schedule(s) you want to delete.

   Note: To select multiple schedules, use the Shift key and Control (Ctrl) key as in a standard Windows interface.

   The following image shows the West Coast Sale and Southern Territory Sales schedules selected in the Schedule pane.

2. Click **Delete**. A message appears asking for confirmation that you want to delete the selected schedule(s).
3. Click **OK** to delete the schedule(s).

Viewing a Log Report

A log report provides information about the schedule date, time, execution status, and recipients of a distributed report. It also includes whether or not the job executed successfully, when and in what format the report output was distributed, and the method of distribution. Log reports are stylized HTML format, and appear in a separate browser window. You can search, print, or save the log report.

View a Log Report

From the Schedule pane in the Report Broker interface, you can view one or more log reports for a schedule using the following procedure.

1. Select the schedule(s) for which you want to view the log report(s).

   Note: To select multiple schedules, use the Shift key and Control (Ctrl) key as in a standard Windows interface.

   The following image shows the East Coast Sales and Sales schedules selected in the Schedule pane.

2. Click **Log**.

   The following image shows the Report Broker Web Page Dialog box that opens, with the Selected Schedule check box active and checked and the Last Executed option button selected.
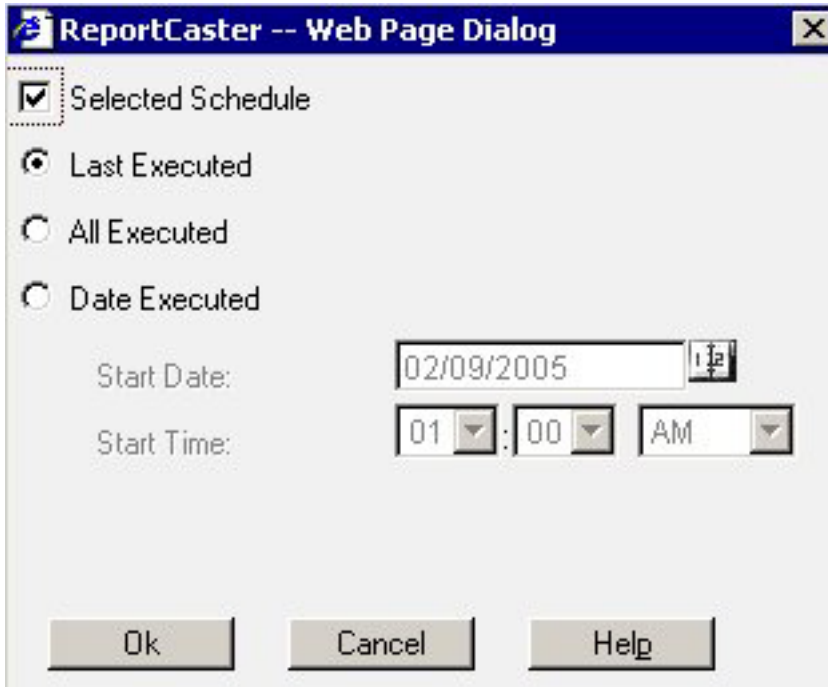
To switch from the schedule you selected to viewing information about all of your schedules, uncheck the **Selected Schedule** check box. If you want to change your selection criteria, click **Cancel** and reselect another schedule from the list.

3. Select one of the following option buttons:

   - **Last Executed.** Produces a log report containing the most currently run process for the selected schedule(s) or for all schedules (if you did not select a schedule). This is the default option.

   - **All Executed.** Produces a log report containing all run processes for the selected schedule(s) or for all schedules (if you did not select a schedule).

   - **Date Executed.** Activates the Start Date and Start Time fields.

   If you have selected the Date Executed option, proceed to the following step. Otherwise, proceed to step 6.

4. In the Start Date field, specify the date on which you want the log report to begin. The report displays all processes for the selected schedule (or schedules) that were run on or after the specified Start Date. You can select a Start Date from the pop-up calendar, or you can accept the default Start Date, which is the current date.

5. In the Start Time field, specify a start time for the Start Date by using the drop-down lists. 1:00 AM is the default value for the Start Time field.

6. Click **OK** to view the log report.

The Job Process Log Report opens.

Reading a Log Report

The log report displays information according to your specifications in a separate browser window. One log record is produced for each scheduled job run in the specified time frame. The following image provides an example of a typical log report.

Job Description: Sales

User: rcanalytical	Starting worker thread
Procedure: J0u6hggbsi07	Starting task: rcanalytical/Sales
Schedule ID: S0u6hgfjun07	Task type: MRE My Report
Start Time: 2004-11-20 02:59:02	Retrieving MRE report: rcanalytical/Sales
End Time: 2004-11-20 02:59:02	Connecting to server EDASERVE with static execution id
	Executing focexec.
	Task finished.
	salesreport.htm distributed to chuck_hill@ibi.com

If you chose to view log reports for multiple schedules, the Job Process Log Report contains a log record for each schedule you selected. The following images shows an example of a log report for multiple schedules.

Job Description: East Coast Sales2

User: writer1	Starting worker thread
Procedure: J1253h0hs70b	Starting task: writer1/myreport1
Schedule ID: S1253gvdkj0b	Task type: MR My Report
Start Time: 2007-03-12 12:52:01 PM	Retrieving MR report: writer1/myreport1
End Time: 2007-03-12 12:52:01 PM	Connecting to server EDASERVE with execution id Writer1
	Executing focexec.
	0 HOLDING HTML FILE ON PC DISK ...
	Task finished.
	distributed to Report Library

Job Description: Sales

User: writer1	Starting worker thread
Procedure: J1253hbhe90d	Starting task: writer1/myreport1
Schedule ID: S1253h1rsd0c	Task type: MR My Report
Start Time: 2007-03-12 12:58:01 PM	Retrieving MR report: writer1/myreport1
End Time: 2007-03-12 12:58:01 PM	Connecting to server EDASERVE with execution id Writer1
	Executing focexec.
	0 HOLDING HTML FILE ON PC DISK ...
	Task finished.
	Sale 2006.htm distributed to SalesTeam@ibi.com

The log report first lists the job description for the record, which is the unique description identifier that you specified when you created the schedule. Underneath the Job Description, the left column of the log report includes the following information:

- **User.** Report Broker user ID, indicating the owner of the schedule.

- **Procedure.** Unique key generated by Report Broker that identifies a specific execution of a scheduled job.

- **Schedule ID.** Unique key generated by Report Broker that was assigned to the job when it was scheduled.

- **Start Time.** Date and time the job started running.

- **End Time.** Date and time the job finished running.

In the second column, the log report specifies messages consisting of the following:

- General information, such as the method of distribution for a particular job (for example, e-mail distribution).
- Processing information, indicating that the request started, distribution was successful, and the request was completed. Processing information also includes reasons why a request failed, such as the unavailability of a data source.

&ECHO and -TYPE Support in Log Reports

Values from &ECHO variables and –TYPE commands in report procedures appear in log reports. The &ECHO variable displays command lines as they execute in order to test and debug procedures. The -TYPE command enables you to comment and evaluate your code for informational and debugging purposes. For example, if the following procedure is scheduled using Report Broker, it may produce a log report similar to the example that follows this procedure:

```
-SET &ECHO=ALL;
-TYPE Country Sales and Growth
-TYPE Parameter RATE is passed into report to forecast potential sales growth
TABLE FILE CAR
HEADING
"Sales Growth Forecast using Rate: &RATE "
SUM SALES AS 'Sales'
COMPUTE GROWTH/D12.2 = (SALES * &RATE) + SALES; AS 'Sales, Forecast'
BY COUNTRY AS 'Country'
END
```

The following image shows an example of the log report.

Job Process Log Report

Job Description: Sample of &ECHO and -TYPE in Log Report

User:	rcadmin	Starting worker thread
Procedure:	J0vhb5jgco03	Starting task: Scheduled FEX with &ECHO and -TYPE
Schedule ID:	S0vhb59o8201	Task type: MRE Standard Report
Start Time:	2004-05-05 02:08:01	Procedure name: salesgrowth
End Time:	2004-05-05 02:08:02	Connecting to server EDASERVE with execution id wfuser

Executing focexec.

-TYPE Country Sales and Growth

Country Sales and Growth

-TYPE Parameter RATE is passed into report to forecast potential sales growth

Parameter RATE is passed into report to forecast potential sales growth

TABLE FILE CAR

HEADING

"Sales Growth Forecast using Rate: .05 "

SUM SALES AS Sales

COMPUTE GROWTH/D12.2 = (SALES * .05)+ SALES; AS Sales, Forecast

BY COUNTRY AS Country

END

SET DISTRIBUTE=OFF

0 HOLDING HTML FILE ON PC DISK ...

Task finished.

salesgrowth.htm distributed to chuck_hill@ibi.com

salesgrowth.htm distributed to casey_barnes@ibi.com

Considerations When Viewing a Log Report

When viewing a log report, be aware of the following considerations.

E-mail Addresses:

Report Broker cannot validate e-mail addresses since e-mail validation is performed by the mail server. The log report will include any e-mail addresses validated by the mail server and returned to Report Broker.

Burst Reports:

- If a valid burst value is omitted in a Distribution List or Dynamic Address List, Report Broker treats the blank value as if it is a valid burst value, and no entries indicating a blank burst value appear in the log file. This will significantly reduce the size of the log file, particularly when the database contains many values for the primary field and only a small subset of those values are burst.
- If a burst value is specified in a Distribution List or Dynamic Address List and it is not found in the database, the following message appears in the log file:

  `Burst Value: value is not in the database.`
- When a report is successfully burst, the log file will include the following message for each burst value:

  `FILE filename SUCCESSFULLY DISTRIBUTED TO destination FOR burst value.`

Unavailable Options:

When schedules with unavailable Task Types and/or distribution methods are not permitted to run, error notification is triggered and information is included in full and brief notifications and in the log report that your Report Broker Administrator or the owner of the schedule must change the unavailable Task Types and/or distribution methods in the schedule.

When schedules with unavailable Task Types and/or distribution methods are permitted to run, normal job execution occurs and a message appears in the log report indicating that your Report Broker Administrator is allowing existing schedules using the unavailable Task Types and/or distribution methods to run.

Troubleshooting Report Broker Log Reports

If you do not receive a log report because there is an insufficient amount of memory available, this may be because the report is too large or you have too many windows open. We recommend closing all windows and attempting to run the log report again. If you are still unsuccessful, rerun a schedule that successfully created a log report. If you are successful in running the log report, this confirms that the original log report that did not run was too large to be processed. Contact your Report Broker Administrator to help you troubleshoot this issue.

Purging the Log File

The log file accumulates information and can become difficult to navigate. we recommend that you periodically purge log records to conserve space.

There is a configuration setting that Report Broker Administrators can configure that automatically purges log reports when they are older than a set number of days. To find out how long your log reports will be available, see your Report Broker Administrator.

You can purge the log file in one of two ways:
- **Purge log file information for specific schedules.** Select one or more schedules and then click **Purge**. The Purge Log Web Page Dialog box opens.

  The Purge Log Web Page Dialog box opens with the Selected Schedule check box active and checked, and the Delete Items through check box inactive and unchecked, as shown in the following image.
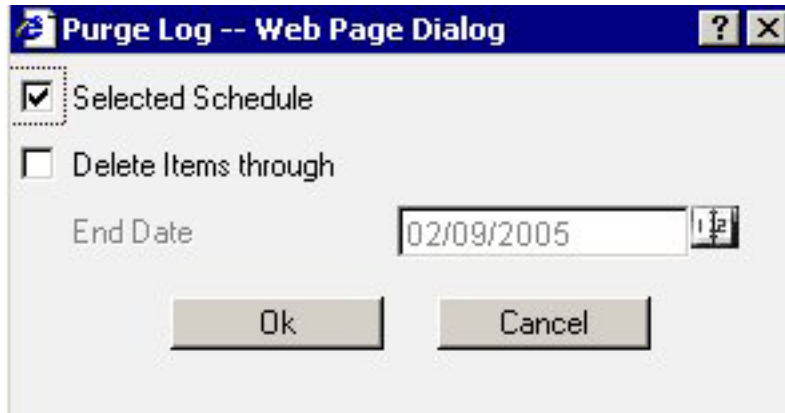
To switch from purging the selected schedule(s) to purging all schedules, uncheck the **Selected Schedule** check box.

- **Purge log file information for all schedules.** Without selecting a schedule, click **Purge**. The Purge Log Web Page Dialog box opens.

  The Purge Log Web Page Dialog box opens with the Selected Schedule check box inactive and unchecked, and the Delete Items through check box active and checked, as shown in the following image.
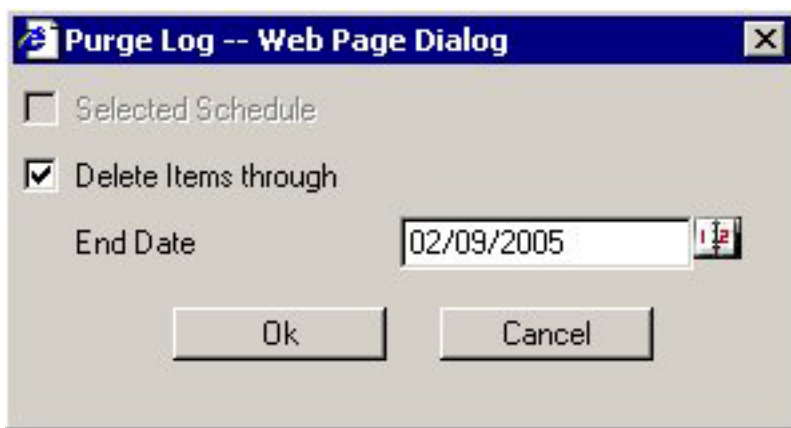
Selecting the Delete Items through check box activates the End Date field, where you can specify the date through which you want to purge the log records. The current date is the default value for the End Date. To change the End Date, click the calendar to the right of the End Date field. The Calendar Web Page Dialog box opens.

The following image shows the Calendar Web Page Dialog box, where you can select the month and year using drop-down lists, and a day by clicking a day on the calendar.
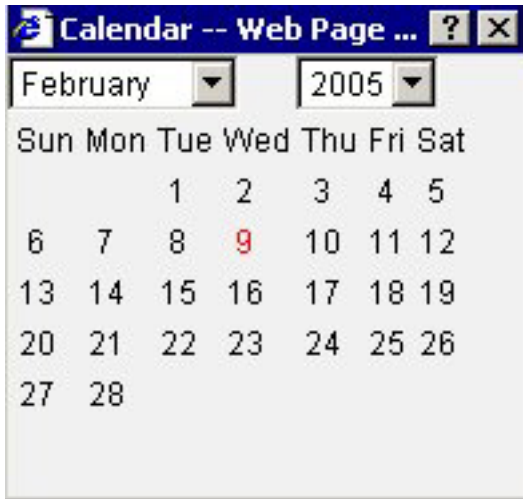
Click **OK** to purge the log file, or click **Cancel** to cancel the purge request.

Note: To verify that the specified log files have been deleted, you may want to run a log report again to note the new log output. The deletion of log files is immediate.

Schedule Blackout Dates

Schedule blackout dates are those dates on which schedules will not run and cannot be set to run. Only Report Broker Administrators and Web Query Group Administrators can define, update, and delete schedule blackout dates.

To view schedule blackout dates, click the **Blackout Dates** option in the Report Broker interface Schedule pane. The Schedule Blackout Dates interface opens to a calendar of the current year with the blackout dates shaded in gray. An example of the Schedule Blackout Dates interface is shown in the following image.
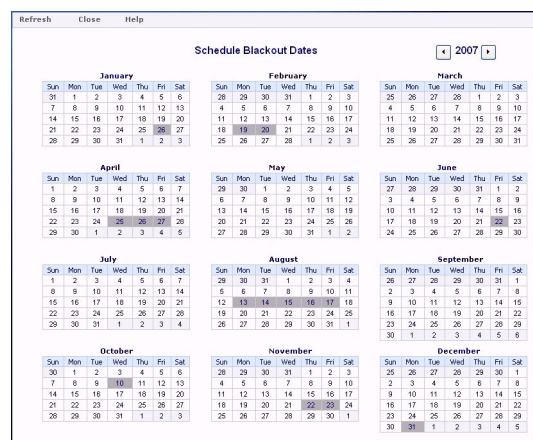
You can change the year using the arrows at the top right corner of the screen. Click **Refresh** to load the latest blackout dates. Dates only appear as available or unavailable.

Click **Close** to exit the Schedule Blackout Dates interface.

Schedule Behavior for Blackout Dates

Blackout dates are enforced during schedule creation and at runtime:

- When creating or editing one of the date fields in a schedule, Report Broker dynamically checks the first upcoming date on which the schedule will run to ensure that this date has not been blacked out. If there is a conflict with the date, an error message appears and the schedule cannot be saved until the conflict is resolved by either changing the schedule date or by not blacking out the date.

- At run time, each schedule is checked against the list of blackout dates. If a blackout date has been defined for the scheduled date, the schedule will not run. If notification is enabled, a notification is distributed indicating that the schedule did not run because of a defined blackout date.

Checking the Status of a Scheduled Job

From the Report Broker interface, click the **Status** option to generate a list of your scheduled jobs that are currently running or in the Report Broker Distribution Server queue.

The following image shows how each scheduled job contains information displayed under column headings for Schedule Id, Description, Priority, Start Time, Owner, and Status. Sorting is available for each column by clicking the column heading.

The Status column contains one of the following values:

- **Run.** The scheduled job is currently running.
- **Queue.** The scheduled job is waiting for a thread to become available to run the request.

From the list of scheduled jobs, you can:

- Delete a scheduled job. To do this, select a scheduled job with a status of Queue and click **Remove**. The schedule is removed from the queue if it is still in the queue, and the Status list is refreshed.

- Change the priority level of a scheduled job. To do this, select a scheduled job with a status of Queue to enable the Update Priority button. In the New Priority field, select a priority level from the drop▵down list and click **Update Priority**.
- Refresh the current list of schedules with any new jobs that are in the Report Broker Distribution Server queue by clicking **Refresh**.
- Automate the refresh process by selecting the time period from the Auto Refresh drop▵down list in which you want the schedule job list to be automatically refreshed (for example, 10 Seconds).
- Click **Help** to open the online Help.

Chapter 17. Report Broker Administration

Provides information to Report Broker Administrators on how to configure for Report Broker access, schedule for blackout dates, move data between environments, and globally update schedule properties.

This is intended for Report Broker Administrators. Before Report Broker can be used, the administrator must confirm the connection to DB2 and create the Report Broker repository tables, as described in Preparing Report Broker for Use. As an administrator, you can also set schedule blackout dates (dates on which reports cannot run or be scheduled to run), move Report Broker data between Report Broker environments, and globally update schedule properties, such as the schedule owner.

Setting Schedule Blackout Dates

Schedule blackout dates are those dates on which schedules cannot be run or be set to run. Only Report Broker administrator can set up or change schedule blackout dates.

When managing schedule blackout dates, you can add new blackout dates, delete and replace the description of existing blackout dates, and extract existing blackout dates to a file for future use. Report Broker provides the following ways for you to access these management options:

- Report Broker user interface.
- Schedule Blackout Dates Import and Extract Utilities. This method provides an alternative to using the Report Broker user interfaceDevelopment and Administration GUI, and unlike the interface, does not require that the Distribution server be running.

  These utilities are available for Windows and UNIX systems, as follows:

  – In Windows, the utilities are provided in the **rcbdimport.bat** and **rcbdextract.bat** files, located in the ibi\ReportCaster76\bin directory on the drive and under the directory where Web Query is installed.

    If you do a full Web Query client installation, the import and extract options are available from the Windows Start Programs menu.

    You can also run the utilities from a command line.

  – In UNIX, the utilities are provided in the **rcbdimport** and **rcbdextract** scripts, located in the ibi/ReportCaster76/bin directory under the directory where Web Query is installed. The scripts are run from a command line.

  The Import and Extract Utilities are run in an interactive mode.

Schedule Behavior for Blackout Dates

Blackout dates are enforced during schedule creation and at run time:

- When creating or editing one of the date fields in a schedule, Report Broker dynamically checks the first upcoming date on which the schedule will run to ensure that this date has not been blacked out. If there is a conflict with the date, an error message appears and the schedule cannot be saved until the conflict is resolved by either changing the schedule date or by not blacking out the date.

- At runtime, each schedule is checked against the list of blackout dates. If a blackout date has been defined for the scheduled date, the schedule will not run. If notification is enabled, a notification is distributed indicating that the schedule did not run because of a defined blackout date.

About the Schedule Blackout Dates Import Utility

The Import Utility allows you to import a file that contains the desired schedule blackout dates, and can also include instructions to insert, delete, or replace date ranges and descriptions of blackout dates. You can import a file that you created or a file that was created using the Extract Utility. For information about extract files, see "About the Schedule Blackout Dates Extract Utility" on page 749. The import of blackout dates can be done through the Report Broker Schedule Blackout Dates user interface or the Import Utility.

To create an import file, use a standard text editor. This section provides an example of an import file.

At a minimum, the import file need only consist of the dates (following the format below) you want to black out. You can also include comments about the file or specific dates, and add flags that specify how to process individual dates or a range of dates.

Use the following rules to create the content of a Schedule Blackout Dates import file:

- Start a comment line with the pound sign (#). Comment lines are optional.
- Enter dates in the YYYYMMDD format. For example, May 1, 2007 = 20070501.
- Each date must be on a separate line. Dates do not have to be in chronological order.
- To add a comment about a specific date, add a space after the date entry, then type your comment. For example, 200070101 New Years Day. Comments are optional.
- Include the following instruction flags, as needed. Flags can be used multiple times in a file.
    - $INSERT
    - $DELETE
    - $REPLACE
    - $RANGE BEGIN and $RANGE END

      The flag before $RANGE BEGIN determines the action performed on the date range. The $RANGE END flag must be the next flag after $RANGE BEGIN.

Note: Use the flags when you want to perform different tasks on different dates within a single import file. Otherwise, you can choose to add or delete all dates specified in the import file while using the import utility, described later in this section.

The following is an example of an import file that includes a comment, an insert, and a date range:

```
#Schedule Blackout Dates Update - Import File
$INSERT
20070101 New Years Day
20070115 Martin Luther King Day
$INSERT
```

```
$RANGE BEGIN
20070226 Winter School Break Begins
$RANGE END
20070302 Winter School Break Ends
```

Trace information is written to the Report Broker trace file.

About the Schedule Blackout Dates Extract Utility

The extract option allows you to save existing schedule blackout dates in a text file to use as reference documentation or as a schedule blackout dates import file. The extract utility generates an extract file that includes a comment line with information about the extract followed by a list of the extracted blackout dates. The date format is the same as that in the import file (YYYYMMDD). Extracting blackout dates can be done through the Report Broker Schedule Blackout Dates user interface or the Extract Utility.

The default name of an extract file is rcbdextract_YYMMDD_HHMMSS.txt, which includes the date and time stamp, as follows:

- YY is the year.
- MM is the month.
- DD is the day.
- HH is the hour.
- MM is the minute.
- SS is the second.

The following is an example of an extract file:

```
#Report Broker Schedule Blackout Date Extract created
#070208 180618 by admin for date range:(All)
#Dates extracted from Group: Sales_NE
20070101
20070115 Martin Luther King Day
20070125
20070219
20070220
20070221
20070222
20070223
20070226 Winter School Break Begins
20070227
20070228
20070301
20070302 Winter School Break Ends
20070425
```

Trace information is written to the Report Broker trace file.

Managing Schedule Blackout Dates From the Report Broker User Interface

To access the Schedule Blackout Dates interface, click **Blackout Dates** in the DB2 Web Query Report Broker tab.

The Schedule Blackout Dates interface opens to a calendar of the current year with the blackout dates shaded in gray. The following image shows an example of an initial view of the Schedule Blackout Dates interface.

Schedule Blackout Dates   ◄ 2008 ►

January

Sun	Mon	Tue	Wed	Thu	Fri	Sat
30	31	1	2	3	4	5
6	7	8	9	10	11	12
13	14	15	16	17	18	19
20	21	22	23	24	25	26
27	28	29	30	31	1	2

February

Sun	Mon	Tue	Wed	Thu	Fri	Sat
27	28	29	30	31	1	2
3	4	5	6	7	8	9
10	11	12	13	14	15	16
17	18	19	20	21	22	23
24	25	26	27	28	29	1

March

Sun	Mon	Tue	Wed	Thu	Fri	Sat
24	25	26	27	28	29	1
2	3	4	5	6	7	8
9	10	11	12	13	14	15
16	17	18	19	20	21	22
23	24	25	26	27	28	29
30	31	1	2	3	4	5

April

Sun	Mon	Tue	Wed	Thu	Fri	Sat
30	31	1	2	3	4	5
6	7	8	9	10	11	12
13	14	15	16	17	18	19
20	21	22	23	24	25	26
27	28	29	30	1	2	3

May

Sun	Mon	Tue	Wed	Thu	Fri	Sat
27	28	29	30	1	2	3
4	5	6	7	8	9	10
11	12	13	14	15	16	17
18	19	20	21	22	23	24
25	26	27	28	29	30	31

June

Sun	Mon	Tue	Wed	Thu	Fri	Sat
1	2	3	4	5	6	7
8	9	10	11	12	13	14
15	16	17	18	19	20	21
22	23	24	25	26	27	28
29	30	1	2	3	4	5

July

Sun	Mon	Tue	Wed	Thu	Fri	Sat
29	30	1	2	3	4	5
6	7	8	9	10	11	12
13	14	15	16	17	18	19
20	21	22	23	24	25	26
27	28	29	30	31	1	2

August

Sun	Mon	Tue	Wed	Thu	Fri	Sat
27	28	29	30	31	1	2
3	4	5	6	7	8	9
10	11	12	13	14	15	16
17	18	19	20	21	22	23
24	25	26	27	28	29	30
31	1	2	3	4	5	6

September

Sun	Mon	Tue	Wed	Thu	Fri	Sat
31	1	2	3	4	5	6
7	8	9	10	11	12	13
14	15	16	17	18	19	20
21	22	23	24	25	26	27
28	29	30	1	2	3	4

October

Sun	Mon	Tue	Wed	Thu	Fri	Sat
28	29	30	1	2	3	4
5	6	7	8	9	10	11
12	13	14	15	16	17	18
19	20	21	22	23	24	25
26	27	28	29	30	31	1

November

Sun	Mon	Tue	Wed	Thu	Fri	Sat
26	27	28	29	30	31	1
2	3	4	5	6	7	8
9	10	11	12	13	14	15
16	17	18	19	20	21	22
23	24	25	26	27	28	29
30	1	2	3	4	5	6

December

Sun	Mon	Tue	Wed	Thu	Fri	Sat
30	1	2	3	4	5	6
7	8	9	10	11	12	13
14	15	16	17	18	19	20
21	22	23	24	25	26	27
28	29	30	31	1	2	3

When defining schedule blackout dates, you can only flag a date as available or unavailable. By default, all dates are available. Calendars are only applicable for the year they are defined. If you define dates only for 2007, when 2008 arrives you must set the blackout dates for the new year.

If a blackout date was defined through a file import and that date included a description, you can view that description by positioning the cursor over the date in the calendar. Updates to a description are made in an import file, which is brought into the Schedule Blackout Dates through the Import Utility.

Note: Non-administrators can view the information in the Schedule Blackout Dates interface, but they cannot define blackout dates.

The options available from the Schedule Blackout Dates user interface are:

* **Year Display.** Appears in the top-right corner. Allows you to select a calendar year.
* **Save.** Saves changes to blackout date selections.
* **Refresh.** Refreshes the view to show the most current blackout dates.
* **Import.** Opens the Import Blackout Dates utility from which you can import a file that contains blackout dates.
* **Extract.** Opens the Extract Blackout Dates utility from which you can extract all or selections of the existing blackout dates and save them in a file.
* **Close.** Closes the Schedule Blackout Dates interface.

The following procedures explain how to define, import, and extract schedule blackout dates using the Schedule Blackout Dates interface.

Define Blackout Dates Using the Interface Calendar

To define blackout dates from the Report Broker Schedule Blackout Dates calendar:

1. Select the year using the arrows next to the year display that appears in the top right corner.
2. In the calendar, click the dates you want to black out.
3. Click **Save**.

   A confirmation window opens to tell you that the dates have been saved.
4. Click **OK**.

The blackout dates you saved appear in the Schedule Blackout Dates calendar.

Import Schedule Blackout Dates in the User Interface

To import schedule blackout dates in the user interface:

1. In the Schedule Blackout Dates interface, select **Import** from the options that appear in the top left corner.

   The Import Blackout Dates dialog box opens, as shown in the following image.

2. In the File Name field, type the full path to the file, or click **Browse** and navigate to the file you want to import.
3. Select an Import Action.
   - **Add.** Add the dates specified in the import file to the blackout dates.
   - **Remove.** Delete the dates specified in the import file from the blackout dates.
   - **Replace.** Updates the description of the specified dates.
   - **Use the actions specified in the file.** Use this option when the import file contains a combination of commands (add, remove, replace) and date ranges.
4. Click **Import**.

The following confirmation message is displayed:

```
Blackout Dates have been successfully imported.
Select Save on the Schedule Blackout Dates toolbar to save the changes.
```

5. Click **OK**.

6. In the Schedule Blackout Dates option menu, click **Save**.

   Note: The imported blackout date changes do not take effect until you save them.

7. Required: In the save confirmation window, click **OK**.
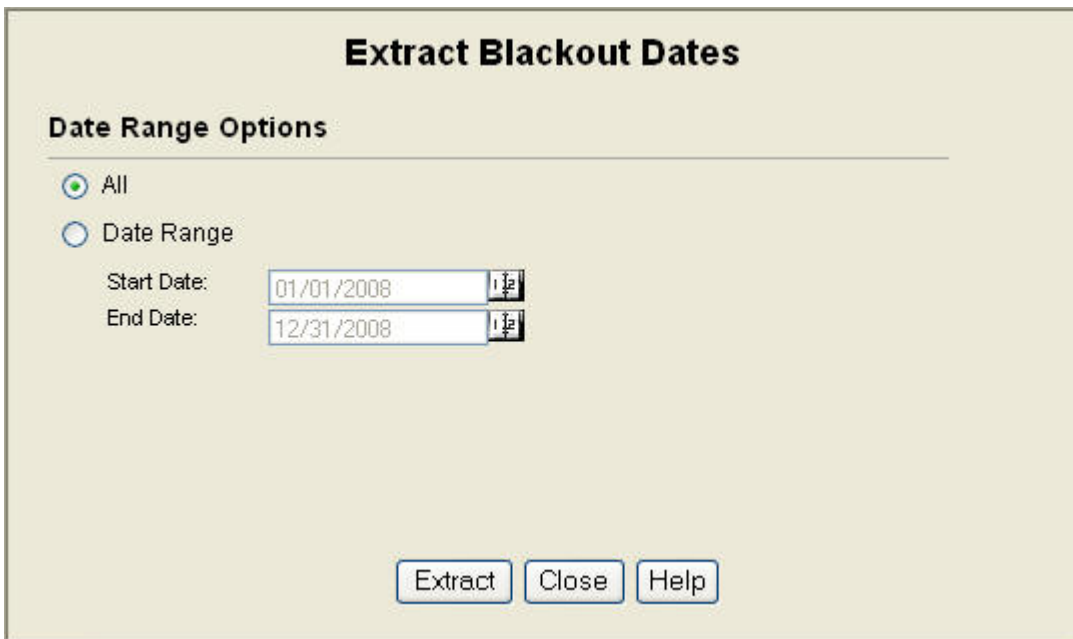
The Schedule Blackout Dates calendar is updated with the blackout date changes.

Extract Schedule Blackout Dates in the User Interface

To extract schedule blackout dates from existing blackout dates in the user interface:

1. In the Schedule Blackout Dates interface, select **Extract** from the options that appear in the top left corner.

   The Extract Blackout Dates dialog box opens, as shown in the following image.

2. Select **All** to extract all currently saved blackout dates, or select **Date Range** to specify the range of dates you want to extract.

   If you select Date Range:

   a. Click the calendar icon to the right of the Start Date field.

      A calendar dialog box opens, as shown in the following image.

b. Choose the date that begins your date range by selecting the month and year from the drop-down menus, then clicking the day in the calendar.

   The Start Date calendar closes and your selection appears in the Start Date field.

c. Click the calendar icon to the right of the End Date field.

d. Choose the date that ends your date range by selecting the month and year from the drop-down menus, then clicking the day in the calendar.

   The End Date calendar closes and your selection appears in the End Date field.

3. Click **Extract**.

   A window opens asking if you want to open or save the file.

   If you choose to save the file:

   a. Click **Save**.

   b. In the Save As window, browse to the directory where you want to save the extract file, and click **Save**.

Note: The default extract file name is `rcbdextract_YYMMDD_HHMMSS.txt`, where YYMMDD and HHMMSS are the date (year, month, day) and time (hour, minute, second) that the file was created.

Using the Import and Extract Utility

You can run the Import and Extract Utility in an interactive mode in a Windows or UNIX environment using a customized .bat file or UNIX script. The following procedures describe how to import and extract schedule blackout dates using the utility in interactive mode.

Use the Import Utility

Use the following procedure to run the Schedule Blackout Dates Import Utility for both Windows and UNIX systems. The images shown in this procedure are from the Windows version of the utility. Although Windows and UNIX interactive dialog boxes may vary in the look and feel, they contain the same options and buttons described here.

1. Run the Import Utility, as follows:

- In Windows, do one of the following:

  From the Start menu, select **Information Builders**, **ReportCaster76**, **Database Management and Utilities**, then **Import Schedule Blackout Dates**.

  In a command line window, navigate to the ibi\ReportCaster76\bin directory, then type **rcbdimport**.

  In Windows Explorer, navigate to the ibi\ReportCaster76\bin directory and double-click the **rcbdimport.bat** file.

- In UNIX, on the command line, run the following script:

  ```
  rcbdimport
  ```

  The Import Blackout Dates dialog box opens, as shown in the following image.
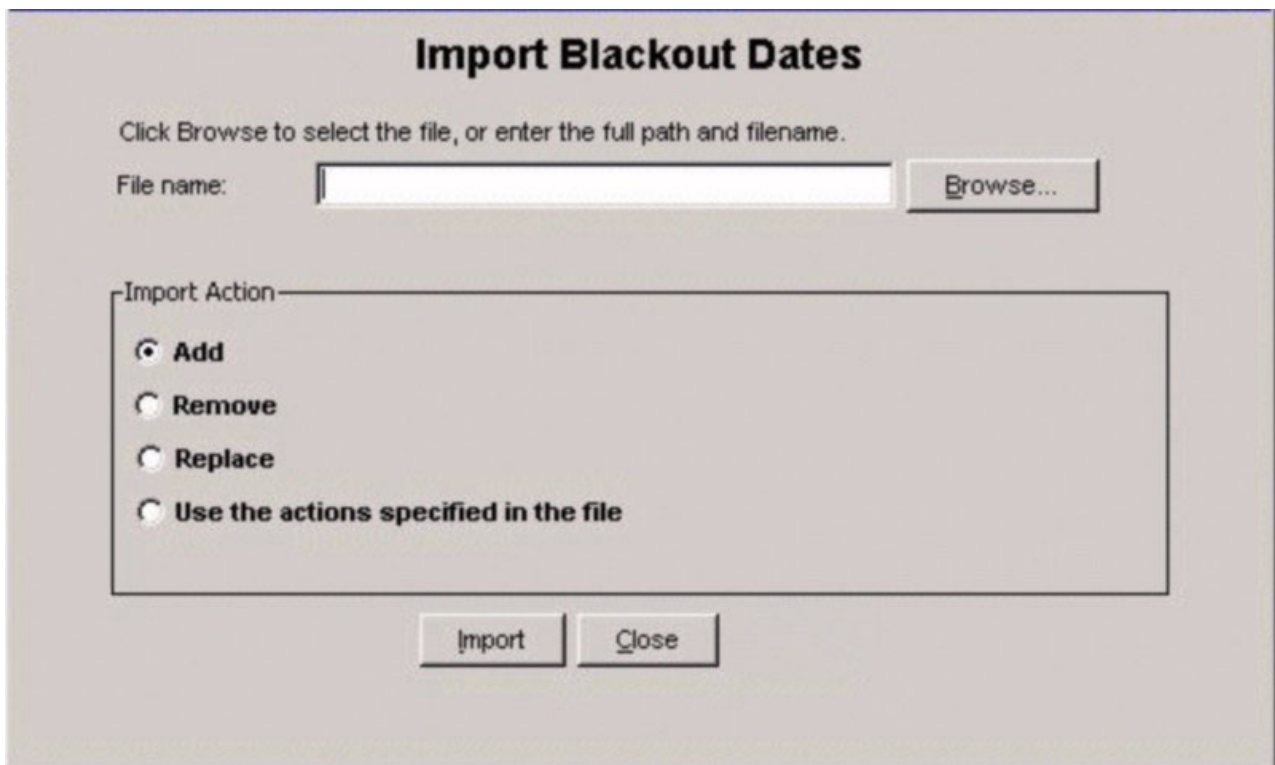
2. In the File name field, type the path to the file you want to import, or click **Browse** to navigate to the file.
3. Select one of the import actions:
   - **Add.** Inserts the blackout dates noted in the import file.
   - **Remove.** Deletes the blackout dates noted in the import file.
   - **Replace.** This option allows you to add or edit a description for an existing blackout date.
   - **Use the action specified in the file.** Act on the blackout dates according to the instructions in the import file. This option allows you to use add, remove, and replace in one file, and to use date ranges.
4. Click **Import**.

   The Report Broker Schedule Blackout Dates Import Log window opens, displaying information about the import, such as the name of the imported file, and the time and date of the import. The following image shows an example of an import log file.

```
ReportCaster Schedule Blackout Date Import Log                    [x]
Action

Import Output

20070521 14:53:20  rcbdextract_default_group_070517_145924.txt
Group selected in GUI:******GLOBAL******
Schedule blackout date information successfully read from import file.
Schedule blackout date information successfully read from DataBase.
Action selected in GUI:******Add******
20070116 is processed.
20070214 is processed.
20070315 is processed.
20070416 is processed.
20070514 is processed.
20070619 is processed.
20070718 is processed.
20070815 is processed.
20070911 is processed.
Blackout dates have been successfully imported.

                              Close      Save
```

5. In the log window, click **Close**.

   You are returned to the Import Blackout Dates dialog box.

6. Click **Close** to exit the Import Blackout Dates utility.

Use the Extract Utility

Use the following procedure to run the Schedule Blackout Dates Extract Utility for both Windows and UNIX systems. The images shown in this procedure are from the Windows version of the utility. Although Windows and UNIX interactive dialog boxes may vary in the look and feel, they contain the same options and buttons described here.

1. Run the Extract Utility, as follows:

   • In Windows, do one of the following:

     From the Start menu, select **Information Builders**, **ReportCaster76**, **Database Management and Utilities**, then **Extract Schedule Blackout Dates**.

     In a command line window, navigate to the ibi\ReportCaster76\bin directory, then type **rcbdextract**.

     In Windows Explorer, navigate to the ibi\ReportCaster76\bin directory and double-click the **rcbdextract.bat** file.

   • In UNIX, in the command line, navigate to the ibi\ReportCaster76\bin directory, then run the following script:

     rcbdextract

     The Extract Blackout Dates dialog box opens, as shown in the following image.

Extract Blackout Dates

Date Range Options

(•) All
( ) Date Range

Start Date: Monday, January 1, 2007

End Date: Monday, December 31, 2007

[Extract]   [Close]

2. Select either All or Date Range.

All. Extract all currently saved schedule blackout dates.

Date Range. Extract the blackout dates within the specified date range.

If you select Date Range:

a. Click the arrow next to the Start Date field.

The Extract Blackout Dates Calendar opens, as shown in the following image.

Extract Blackout Dates

Date Range Options

- ○ All
- ● Date Range

Start Date: Monday, January 1, 2007 ▼

End Date: Monc

January ⬍	2007 ⬍

Sun	Mon	Tue	Wed	Thu	Fri	Sat
31	1	2	3	4	5	6
7	8	9	10	11	12	13
14	15	16	17	18	19	20
21	22	23	24	25	26	27
28	29	30	31	*1*	*2*	*3*
4	*5*	*6*	*7*	*8*	*9*	*10*

Today is Thu, May 17, 2007
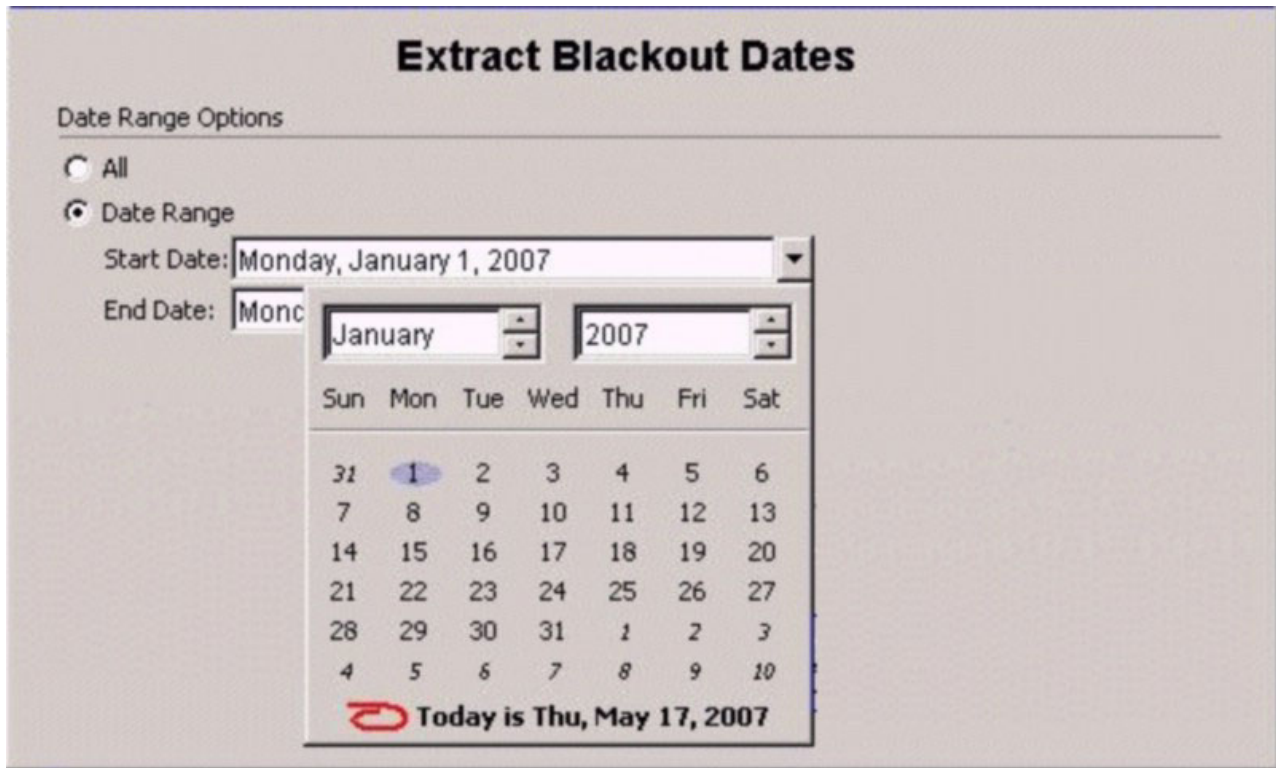
    b. In the calendar, click the date you want to begin the date range.

    c. Click the arrow next to the End Date.

       The Extract Blackout Dates Calendar opens.

    d. In the calendar, click the date you want to end the date range.

3. Click **Extract**.

   The following message appears:

   `Extract of schedule blackout dates successfully created.`

4. Click **OK**.

A file containing the extracted blackout dates with the following file name format is saved in the \ReportCaster76\log directory:

`rcbdextract_YYMMDD_HHMMSS.txt`

Where YYMMDD is the year, month, and day the file was created, and HHMMSS is the hour, minute, and second the file was created.

Report Broker Change Management

The Report Broker Change Management user interface provides a way for you to select specific Report Broker Repository data belonging to specific users and move that data to another Report Broker environment. The Repository objects you can move are schedules and schedule distribution lists. Typically, you would move data between:

- Development and testing environments.
- Test and production environments.

Using the Change Management Interface

This section describes how to use the Report Broker Change Management user interface to transfer Report Broker data to another Repository. The interface allows you to move only one type of repository object for one user at a time.

Move Data Between Report Broker Environments

To move Report Broker data to another Report Broker Repository:

1. Log onto Report Broker in the target environment (the environment that will receive the data).

2. Access the Change Management interface by clicking **Change Mgmt** in the Report Broker tab.

   The Change Management Source Properties window opens. The following image shows the Source Properties window populated with example values.

The Change Management Source Properties window displays the information needed to connect to the source Report Broker Repository. When you access Change Management for the first time, you must set the source Repository properties. For instructions on setting or changing the source connection information, see Setting the Connection Properties.

3. From the Repository Object drop-down list, select the type of data you want to move. The selections are:

   - Schedules
   - Distribution Lists

4. Click **OK**.

   The Report Broker interface displays a list of users found in the source repository, as well as a series of headings related to the type of Repository object you selected.

   The following image shows the Change Management interface when you select Schedules as the Repository Object.
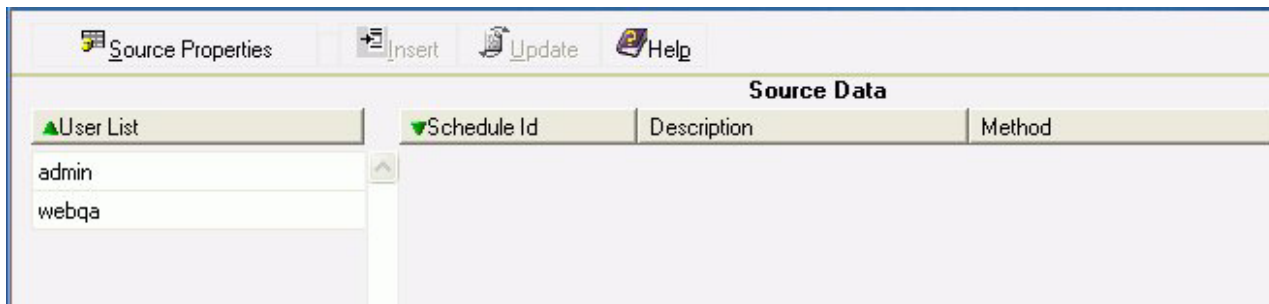
The following image shows the Change Management interface when you select Distribution Lists as the Repository Object.
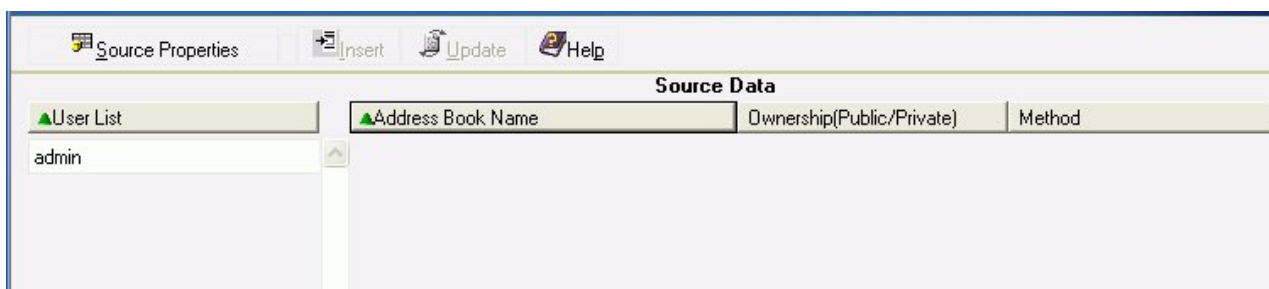
5. In the left pane, select a user whose repository data you want to move.

   Note: You can select only one user for each transfer of data.

   The right pane displays the related repository objects owned by the selected user. The following image is an example of a list of schedules for a specific user.
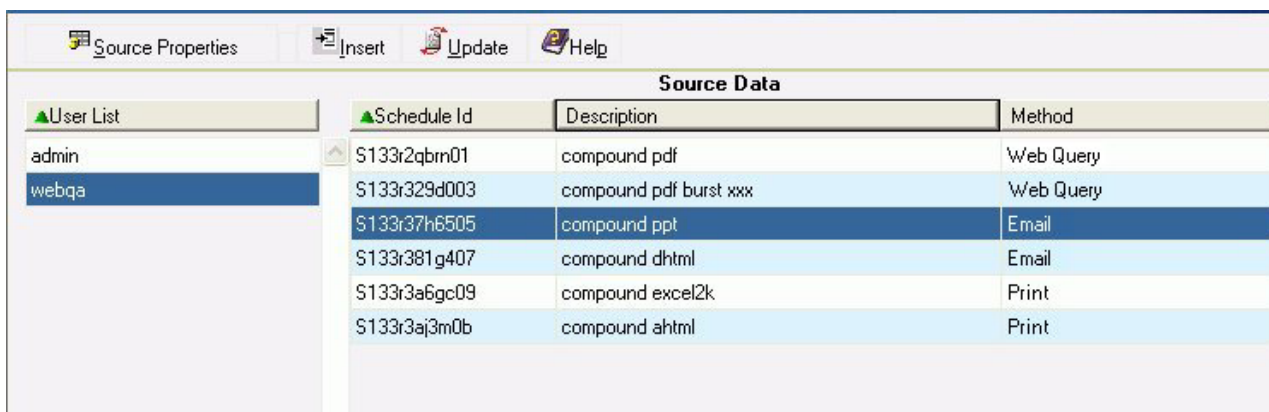
6. Select the items you want to move to the target repository. (Select multiple items using the Shift and Ctrl keys.)

7. Click one of the following:

   • **Insert** - to add the selected source records to the target repository if that record does not already exists.

   • **Update** - to add the selected source records to the target repository and overwrites any corresponding records that exist in the target repository.

   Note: Duplicate records cannot exist in the Report Broker Repository.

A window opens and displays the status of and summary of operations on each repository table that was involved in the data transfer. The information includes the operation (insert or update), whether or not the operation was successful, and if it failed, a brief explanation. See Change Management Logs for examples of this window.

This information is saved in a log file (changemgmt.log) on the Distribution Server in `ibi\ReportCaster76\log`. A new log is created each time you click Insert or Update, so the latest log file is time stamped and renamed with a .bak file extension.

8. To continue transferring the selected Repository object for another user, select a user in the left pane. If you want to transfer a different Repository object, click **Source Properties** in the Change Management interface toolbar to open the Source Properties window, and follow Steps 3 through 7 .

Change Management Logs

This section provides examples of the Change Management log files that indicate:

- A successful transfer of Report Broker data to the target Repository.
- An error in the transfer because the data being transferred already exists in the target repository.

Log of Successful Data Transfer

The following is an example of a log file indicating a successful transfer of records. It provides details for each repository table involved in the transfer.

Load BOTSCHED table data into database ....

SCHEDULEID	OPERATION	STATUS	ERROR REASON
S1333I12mj05	REPLACE	Success	
S1333I96u407	REPLACE	Success	
S1333Ia8h809	REPLACE	Success	

Summary :

OPERATION	TOTAL SUCCESS	TOTAL FAIL
INSERT	0	0
REPLACE	3	0
DELETE	0	0

Load BOTPACK table data into database ....

PACKETID	OPERATION	STATUS	ERROR REASON
P1333I12mj06	REPLACE	Success	
P1333I96u408	REPLACE	Success	
P1333Ia8h80a	REPLACE	Success	

Summary :

OPERATION	TOTAL SUCCESS	TOTAL FAIL
INSERT	0	0
REPLACE	3	0
DELETE	0	0

Log of Failed Data Transfer

The following is an example a log file indicating that a record already exists in the target Repository.

Load BOTSCHED table data into database ....

SCHEDULEID	OPERATION	STATUS	ERROR REASON				
S133p2p5g801	INSERT	Fail	Duplicate	entry	'S133p2p5g801'	for	key

Summary :

OPERATION	TOTAL SUCCESS	TOTAL FAIL
INSERT	0	1
REPLACE	0	0
DELETE	0	0

Load BOTPACK table data into database ....

PACKETID	OPERATION	STATUS	ERROR REASON				
P133p2p5g802	INSERT	Fail	Duplicate	entry	'P133p2p5g802'	for	key

Summary :

OPERATION	TOTAL SUCCESS	TOTAL FAIL
INSERT	0	1
REPLACE	0	0
DELETE	0	0

Setting the Connection Properties

The first time you access the Report Broker Change Management interface, the Change Management Source Properties window opens and displays the target Repository information, and blank fields related to the source Repository information. You must provide the source Repository information needed to make the connection between the two Repositories.

Once this connection information is saved, you will always connect to the repository designated in the source fields when you access the Change Management interface. Once in the Change Management interface, you can change the source connection information whenever you want to connect to a different source repository.

Set Connection Properties

Use the following procedure to set or change the connection properties to the source Report Broker Repository.

1. If the Change Management Source Properties window is not already open, click **Source Properties** in the Change Management interface.

   The Report Broker Change Management Source Properties window automatically displays the target Repository information and if previously saved, the source Repository information. The following image is an example of the Source Properties window that opens when using the Change Management interface for the first time from the target environment.

Information about the Report Broker Repository you are currently logged into (Target URL and Target Class Name) automatically displays in the Source Properties window. Once you save the source Repository information (Source URL, Source Class Name, User ID, and Password) you will enter in this procedure, that will also be displayed whenever you open the Source Properties window.

2. Required: From the Version drop-down list, select the release version of the source Report Broker Repository data.

   Note: Currently, the only available release version is 7.6.x.

3. In the Source URL field, type the URL to the source Report Broker Repository.

4. In the Source ClassName field, type the name of the Java™ class to connect to the Report Broker database.

5. Required: Click **OK**.

   The connection properties are saved and you can now work in the Change Management interface.

Globally Updating Schedule Field Values

You can update the following schedule field values using the Global Update options available in the Web Query Report Broker tab.

- **Mail Server.** Replaces the mail server name in every schedule that is using the specified e-mail server.
- **Printer.** Replaces the printer name in every occurrence of any Distribution List or schedule that is using the specified printer.
- **EMail Address.** Replaces the e-mail address in every occurrence of any Distribution List or schedule with e-mail distribution using the e-mail address. Notification e-mail addresses are not changed.
- **Mail From.** Replaces the optional From field in every schedule that uses e-mail.

- **Owner.** Replaces the owner user ID in every occurrence of any Distribution List or schedule that is owned by the user ID. This option does not change the owner for log reports.

Globally Update Schedule Properties

1. In the Web Query Report Broker tab, click **Global Update**.

   The Global Schedule Update pane opens, as shown in the following image.

2. Click in the Old Value field column next to the property you want to update, and type the current value.
3. Click in the New Value field column next to the property you want to update, and type the new value for the property.
4. Click **Update**.

Chapter 18. Report Broker Formats for Scheduled Output

Provides descriptions, suggested uses, and considerations about each Report Broker output format.

When you create a schedule, you specify the format for the scheduled output. The following topics describe the different types of available Report Broker formats. They also suggest the best usage for each Report Broker format, and provides considerations that you should be aware of when distributing each format type using Report Broker.

AHTML

Format: AHTML (.htm)

Description: Provides customizable options for creating HTML formatted reports that enable users to experience features normally found in Excel workbooks.

Suggested Uses: e-mail for display in a Web browser

Considerations: Can only be distributed as an e-mail attachment. Inline e-mail messages are not supported.

DHTML

Format: DHTML

Description: Supports hyperlinks and other World Wide Web features. Retains StyleSheet formatting.

Along with the features of HTML format, DHTML supports the Web archive format (.mht). An .mht file can contain multiple reports and graphs and is utilized for Coordinated Compound Reports.

Suggested Uses: e-mail for display in a Web browser

Considerations: Can be distributed as an e-mail attachment and be sent as an inline e-mail message when the output is .htm, but not when the output is .mht.

Will return two possible formats:
- The scheduled procedure (FOCEXEC) outputs an HTML file when the request does not contain the SET HTMLARCHIVE=ON command.
- The scheduled procedure (FOCEXEC) outputs a Web archive file (.mht) when the SET HTMLARCHIVE=ON is specified.

DOC

Format: DOC (.txt)

Description: Scheduled output opens as a plain-text word processing document. Text can be opened by any word processing application. Retains ASCII form feed characters to correctly display page output.

Suggested Uses: Word Processing applications, printing unformatted reports, e-mail.

Considerations: Does not retain most formatting. Does not support hyperlinks or alerts.

Can be distributed as an e-mail attachment or as an inline e-mail message.

Bursting is supported.

EXL2K

Format: EXL2K (.xls)

Description: Scheduled output opens within Excel 2000 or higher.

Supports most StyleSheet attributes, allowing for full report formatting.

Suggested Uses: E-mail

Considerations: Microsoft Excel 2000 or higher must be installed. The format is ASCII.

All EXL2K output with an .xht extension is dynamically changed to .xls for e-mail. You must edit your Web server MIME table so that the .xls extension is ascii/application data instead of binary.

Bursting is supported.

EXL2K FORMULA

Format: EXL2K FORMULA (.xls)

Description: Scheduled output opens within Excel 2000 or higher.

Contains Excel formulas that calculate and display the results of any type of summed information (such as column totals, row totals, and sub-totals).

Suggested Uses: E-mail

Considerations: Microsoft Excel 2000 or higher must be installed. The format is ASCII.

All EXL2K output with an .xht extension is dynamically changed to .xls for e-mail. You must edit your Web server MIME table so that the .xls extension is ascii/application data instead of binary.

Bursting is not supported.

EXL97

Format: EXL97 (.xls)

Description: Scheduled output opens as an Excel97 spreadsheet file, an HTML-based display format that supports report formatting and drill-downs.

Suggested Uses: E-mail

Considerations: Microsoft Excel 97 or higher must be installed.

Bursting is supported.

HTML

Format: HTML (.htm)

Description: Supports hyperlinks and other World Wide Web features. Retains StyleSheet formatting.

Suggested Uses: e-mail for display in a Web browser

Considerations: Can be distributed as an e-mail attachment or as an inline e-mail message.

JPG

Format: JPG (.jpg)

Description: Scheduled output opens as a graph image in JPG format.

Suggested Uses: E-mail

Considerations: Only works with procedures that contain GRAPH FILE syntax.

Bursting is supported and is performed on the second BY field in the GRAPH FILE request.

Drill downs are not supported since the JPG format creates a static image.

PDF

Format: PDF (.pdf)

Description: Appearance of the scheduled output is preserved in an electronic document when printed using Adobe Acrobat. Retains all relevant StyleSheet formatting.

Suggested Uses: E-mail

Considerations: Does not support hyperlinks in e-mail attachments. Recipient must have an Adobe Acrobat application to view.

The z/OS USS (UNIX System Services) platform requires the Web server configuration file to have the MIME setting for PDF set to binary in mime.wfs.

Bursting is supported.

When Report Broker distributes PDF reports created with a TABLE request containing BY HIGHEST *primarysortfield* syntax, the report contains page breaks on each primary sort field value.

When distributing a PDF report to a Web Query Repository on z/OS, the PDF report cannot be viewed in Web Query. As a workaround, save the PDF file to your local machine and then view the PDF using Adobe Acrobat. This limitation is due to z/OS being an EBCDIC-based platform and PDF being an ASCII-based format.

The PDF Drill Through feature is supported.

PNG

Format: PNG (.png)

Description: Scheduled output opens as a graph image. These graphs are bitmapped and can support 16 million colors. In addition, PNG graphs have lossless compression (data is decompressed 100% back to the original). Therefore, saving, altering, and re△saving a PNG does not degrade its overall quality.

Suggested Uses: E-mail

Considerations: Only works with procedures that contain GRAPH FILE syntax.

Bursting is supported and is performed on the second BY field in the GRAPH FILE request.

Drill downs are not supported, as this format creates a static image.

PS

Format: PS (.ps)

Description: Appearance of the scheduled output is preserved in an electronic document when printed using PostScript. Retains all relevant StyleSheet formatting.

Suggested Uses: Printing

Considerations: Does not support hyperlinks. Printers must support PostScript. Recipient must have an application (for example, GhostView) that supports PostScript.

Distribution to Web Query is not supported.

Bursting is supported.

SVG

Format: SVG (.svg)

Description: Scheduled output opens as a graph image. This file format, based on Extensible Markup Language (XML), presents powerful, interactive images.

Suggested Uses: E-mail

Considerations: Recipient must have an SVG viewer, such as Adobe SVG Viewer for Windows. To download Adobe SVG Viewer for Windows, go to http://www.adobe.com.

Only works with procedures that contain GRAPH FILE syntax.

Bursting is supported and is performed on the second BY field in the GRAPH FILE request.

Drill downs are not supported, as this format creates a static image.

SVG graphs delivered by e-mail or to Web Query from a z/OS Report Broker Distribution Server cannot be opened. To resolve this issue, edit the Report Broker Distribution Server sendmodes.dita file (located in the /cfg directory) and change the <mode> value from binary to text. For example:

```
<sendmode>     <extension>svg</extension >     <mode>text</mode>
<mime>image/svg+xml</mime > </sendmode>
```

WP

Format: WP (.txt)

Description: Scheduled output opens as a plain-text word processing document in the Web browser. Text can be opened by any word processing application.

Suggested Uses: Word processing applications, printing unformatted reports, e-mail.

Considerations: Does not retain page breaks or most formatting. Does not support hyperlinks or alerts.

Can be distributed as an e-mail attachment or as an inline e-mail message.

Bursting is supported.

Appendix. Running Web Query Reports Using the Java Batch Run Utility

Describes how to run Web Query Reports using the Java Batch Run Utility.

The Java Batch Run utility (RUNWEBQRY) enables you to run a Web Query report from the command line so that the request can be submitted to a batch queue, without having to physically log on to DB2 Web Query. The utility accepts a single report procedure (FOCEXEC) that is in the DB2 Web Query installation and executes the FOCEXEC via a Java program.

Java Batch Run Utility Prerequisites

The following are prerequisites for running the utility:
- DB2 Web Query must be installed and running on the system where the reporting and application servers are running.
- FOCEXECs which exist in the DB2 Web Query environment.
- An available user ID that is licensed for DB2 Web Query.

Invoke the Java Batch Run

To invoke the utility from the command line:
1. Log in to the i5 system via a 5250 terminal emulator.
2. At the command line, execute the following command:

   `QWEBQRY76/RUNWEBQRY`
3. Press the **F4** key.

   The Run Java batch in WebQuery (RUNWEBQRY) screen opens as shown in the following image.

```
                Run Java batch in WebQuery (RUNWEBQRY)

Type choices, press Enter.

Domain href  . . . . . . . . . .    UNTITLED      Character value
Folder href  . . . . . . . . . .    _____    Character value
File Name (fex)  . . . . . . . .    _____
_____
Output Format  . . . . . . . . .    WP_           DOC, WP
Output Queue . . . . . . . . . .    QPRINT____    Character value
Run with User ID . . . . . . . .    *CURRENT__    Character value




                                                                    Bottom
F3=Exit    F4=Prompt    F5=Refresh    F12=Cancel    F13=How to use this display
F24=More keys
```

4. On the screen, type values for the following input parameters:

    Domain href

    > Is the domain name in Dashboard or an href. An href is an internal
    > name given to an object. The href can be displayed via the Properties
    > option in the Dashboard menu, which is available by right-clicking the
    > mouse. You do not need to use the href if the name is exactly 8
    > characters. You need to use the href if the name is less than 8 characters
    > or more than 8 characters. This input parameter can be retrieved from
    > the Properties page by right-clicking on the report name and choosing
    > **Properties** from the drop-down menu.

    Folder href

    > Is the folder name in Dashboard or an href. An href is an internal name
    > given to an object. The href can be displayed via the Properties option
    > in the Dashboard menu, which is available by right-clicking the mouse.
    > You do not need to use the href if the name is exactly 12 characters.
    > You need to use the href if the name is less than 12 characters or more
    > than 12 characters. This input parameter can be retrieved from the
    > Properties page by right-clicking on the report name and choosing
    > **Properties** from the drop-down menu.

    File Name (fex)

Is the name of the report FOCEXEC (fex) as displayed in the Dashboard. This input parameter can be retrieved from the Properties page by right-clicking on the report name and choosing **Properties** from the drop-down menu.

Output Format

Is the report format. Possible values are DOC or WP. WP is the default value.

Output Queue

Is the name of the outq on the i5 system where the report will be sent. QPRINT is the default value.

Run with User ID

Enables you to submit the job using another user ID. The default value is *CURRENT which means that the current user ID will be used to submit the job.

Note: The RUNWEBQRY command can be part of a job stream using SBMJOB to run multiple requests.

Retrieving Input Parameters for the RUNQUERY Command

The following is an example of a Properties page for retrieving parameter information. This page is accessed by right-clicking on a report name in a report folder, and choosing **Properties** from the drop-down menu.
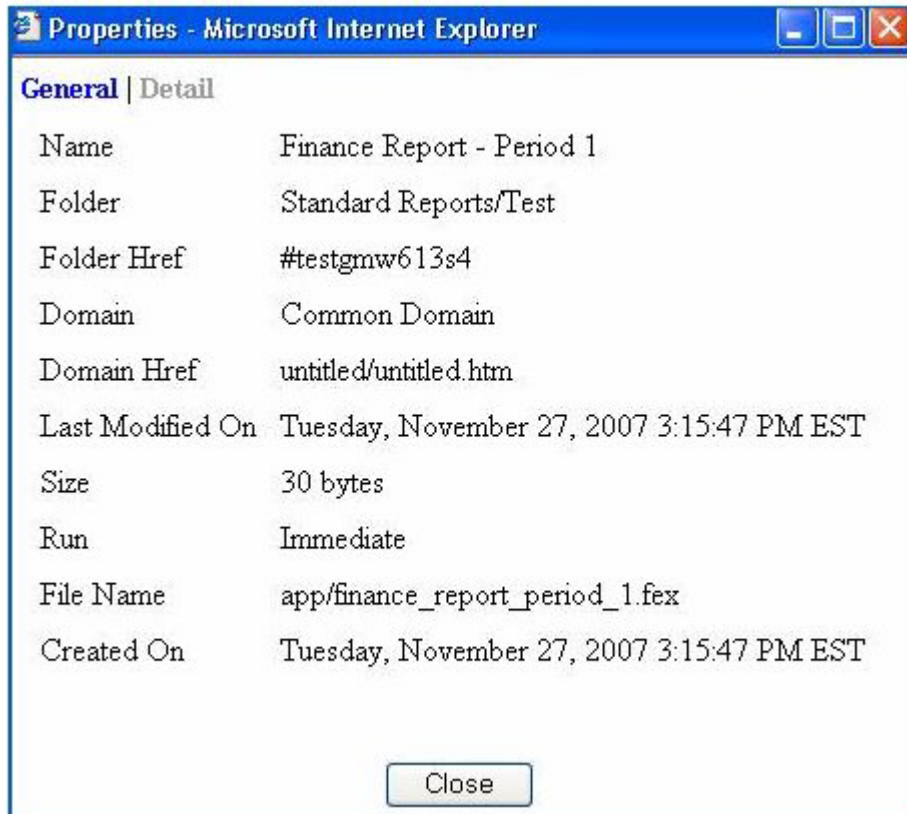
The following image shows the input parameters populated from the example properties page.

```
                    Run Java batch in WebQuery (RUNWEBQRY)

Type choices, press Enter.

Domain href  . . . . . . . . . .   untitled    Character value
Folder href  . . . . . . . . . .   testgmw613s4  Character value
File Name (fex)  . . . . . . . .   finance report period 1_____
____
Output Format  . . . . . . . . .   WP_         DOC, WP
Output Queue . . . . . . . . . .   QPRINT      Character value
Run with User ID . . . . . . . .   *CURRENT    Character value




                                                                      Bottom
F3=Exit    F4=Prompt   F5=Refresh   F12=Cancel   F13=How to use this display
F24=More keys
```

Notices

This information was developed for products and services offered in the U.S.A.

IBM may not offer the products, services, or features discussed in this document in other countries. Consult your local IBM representative for information on the products and services currently available in your area. Any reference to an IBM product, program, or service is not intended to state or imply that only that IBM product, program, or service may be used. Any functionally equivalent product, program, or service that does not infringe any IBM intellectual property right may be used instead. However, it is the user's responsibility to evaluate and verify the operation of any non-IBM product, program, or service.

IBM may have patents or pending patent applications covering subject matter described in this document. The furnishing of this document does not grant you any license to these patents. You can send license inquiries, in writing, to:

IBM Director of Licensing
IBM Corporation
North Castle Drive
Armonk, NY   10504-1785
U.S.A.

For license inquiries regarding double-byte (DBCS) information, contact the IBM Intellectual Property Department in your country or send inquiries, in writing, to:

IBM World Trade Asia Corporation
Licensing
2-31 Roppongi 3-chome, Minato-ku
Tokyo 106-0032, Japan

The following paragraph does not apply to the United Kingdom or any other country where such provisions are inconsistent with local law: INTERNATIONAL BUSINESS MACHINES CORPORATION PROVIDES THIS PUBLICATION "AS IS" WITHOUT WARRANTY OF ANY KIND, EITHER EXPRESS OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF NON-INFRINGEMENT, MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE. Some states do not allow disclaimer of express or implied warranties in certain transactions, therefore, this statement may not apply to you.

This information could include technical inaccuracies or typographical errors. Changes are periodically made to the information herein; these changes will be incorporated in new editions of the publication. IBM may make improvements and/or changes in the product(s) and/or the program(s) described in this publication at any time without notice.

Any references in this information to non-IBM Web sites are provided for convenience only and do not in any manner serve as an endorsement of those Web sites. The materials at those Web sites are not part of the materials for this IBM product and use of those Web sites is at your own risk.

IBM may use or distribute any of the information you supply in any way it believes appropriate without incurring any obligation to you.

Licensees of this program who wish to have information about it for the purpose of enabling: (i) the exchange of information between independently created programs and other programs (including this one) and (ii) the mutual use of the information which has been exchanged, should contact: IBM Corporation _Department number/Building number_ _Site mailing address_ _City, State; Zip Code_ _U.S.A. (or appropriate country)

Such information may be available, subject to appropriate terms and conditions, including in some cases, payment of a fee.

The licensed program described in this document and all licensed material available for it are provided by IBM under terms of the IBM Customer Agreement, IBM International Program License Agreement or any equivalent agreement between us.

Any performance data contained herein was determined in a controlled environment. Therefore, the results obtained in other operating environments may vary significantly. Some measurements may have been made on development-level systems and there is no guarantee that these measurements will be the same on generally available systems. Furthermore, some measurements may have been estimated through extrapolation. Actual results may vary. Users of this document should verify the applicable data for their specific environment.

Information concerning non-IBM products was obtained from the suppliers of those products, their published announcements or other publicly available sources. IBM has not tested those products and cannot confirm the accuracy of performance, compatibility or any other claims related to non-IBM products. Questions on the capabilities of non-IBM products should be addressed to the suppliers of those products.

All statements regarding IBM's future direction or intent are subject to change or withdrawal without notice, and represent goals and objectives only.

All IBM prices shown are IBM's suggested retail prices, are current and are subject to change without notice. Dealer prices may vary.

This information is for planning purposes only. The information herein is subject to change before the products described become available.

This information contains examples of data and reports used in daily business operations. To illustrate them as completely as possible, the examples include the names of individuals, companies, brands, and products. All of these names are fictitious and any similarity to the names and addresses used by an actual business enterprise is entirely coincidental.

COPYRIGHT LICENSE:

This information contains sample application programs in source language, which illustrate programming techniques on various operating platforms. You may copy, modify, and distribute these sample programs in any form without payment to IBM, for the purposes of developing, using, marketing or distributing application programs conforming to the application programming interface for the operating platform for which the sample programs are written. These examples have not been thoroughly tested under all conditions. IBM, therefore, cannot guarantee or imply reliability, serviceability, or function of these programs.

Each copy or any portion of these sample programs or any derivative work, must include a copyright notice as follows:

© (your company name) (year). Portions of this code are derived from IBM Corp. Sample Programs. © Copyright IBM Corp. _enter the year or years_. All rights reserved.

If you are viewing this information softcopy, the photographs and color illustrations may not appear.

Trademarks

The following terms are trademarks of International Business Machines Corporation in the United States, other countries, or both:

```
1-2-3
CICS
DB2
i5/OS
IBM
IMS
Informix
iSeries
Lotus
Lotus Notes
System i
z/OS
```

Adobe, Acrobat, Portable Document Format (PDF), PostScript, and all Adobe-based trademarks are either registered trademarks or trademarks of Adobe Systems Incorporated in the United States, other countries, or both.

Java and all Java-based trademarks are trademarks of Sun Microsystems, Inc. in the United States, other countries, or both.

Microsoft, Windows, Windows NT®, and the Windows logo are trademarks of Microsoft Corporation in the United States, other countries, or both.

UNIX is a registered trademark of The Open Group in the United States and other countries.

Other company, product, or service names may be trademarks or service marks of others.

Terms and conditions

Permissions for the use of these publications is granted subject to the following terms and conditions.

Personal Use: You may reproduce these publications for your personal, noncommercial use provided that all proprietary notices are preserved. You may not distribute, display or make derivative works of these publications, or any portion thereof, without the express consent of IBM.

Commercial Use: You may reproduce, distribute and display these publications solely within your enterprise provided that all proprietary notices are preserved. You may not make derivative works of these publications, or reproduce, distribute or display these publications or any portion thereof outside your enterprise, without the express consent of IBM.

Except as expressly granted in this permission, no other permissions, licenses or rights are granted, either express or implied, to the publications or any information, data, software or other intellectual property contained therein.

IBM reserves the right to withdraw the permissions granted herein whenever, in its discretion, the use of the publications is detrimental to its interest or, as determined by IBM, the above instructions are not being properly followed.

You may not download, export or re-export this information except in full compliance with all applicable laws and regulations, including all United States export laws and regulations.

IBM MAKES NO GUARANTEE ABOUT THE CONTENT OF THESE PUBLICATIONS. THE PUBLICATIONS ARE PROVIDED "AS-IS" AND WITHOUT WARRANTY OF ANY KIND, EITHER EXPRESSED OR IMPLIED, INCLUDING BUT NOT LIMITED TO IMPLIED WARRANTIES OF MERCHANTABILITY, NON-INFRINGEMENT, AND FITNESS FOR A PARTICULAR PURPOSE.

Index

Special characters

-TYPE commands   740
&ECHO variables   740

Numerics

3D graphs   279

A

About link   7
ABS function   416
absolute value   416
ACCEPT list   228
Access File Text View tab   616
Access Files   605, 607
ACROSS fields   79, 81, 305
ACROSS fields:selecting   190
ACROSS fields:selecting;sorting in OLAP
  reports   192
Across tab   296
active report control   589
active report control list   581
Active Report control values   581
active report controls   581, 585
active report properties   579
Active Reports   130
Active Reports:formatting   130
ad hoc reports   449
add/disable schedule button   598
adding a computed field (COMPUTE) in
  virtual fields   659
adding color to graphs   305, 313
adding date-time values   382, 383
adding existing field to group field   670
adding fields   310
adding fields:to footings   310, 313
adding fields:to headings   310, 313
adding footings to graphs   310, 312
adding group fields   668
adding headings to graphs   310, 312
adding labels to graphs   310, 313
adding new field to group field   669
adding objects to HTML pages   463
adding objects to HTML pages:form
  controls   497
adding objects to HTML
  pages:frames   468
adding objects to HTML
  pages:graphs   463
adding objects to HTML pages:group
  boxes   481
adding objects to HTML pages:HTML
  templates   563
adding objects to HTML
  pages:hyperlinks   479
adding objects to HTML
  pages:images   476
adding objects to HTML pages:lines   463,
  474, 475

adding objects to HTML pages:push
  buttons   482
adding objects to HTML
  pages:reports   463
adding objects to HTML pages:reset
  buttons   486
adding objects to HTML pages:tab
  control   487
adding objects to HTML pages:text   478
additional tabs   491
adjusting work area   68
administering Report Broker   747
AHTML format   765
all columns   676, 678
ALL. prefix   92, 93
alphanumeric argument   363
alphanumeric fields   357
alphanumeric formats   86, 400, 405, 412
alphanumeric strings   410
Alternate Comment for Hyperlink   306
amper variable parameters   529
amper variable parameters:setting form
  controls   529
amper variables   306, 307
And/Or keyword   114, 294
applying selections criteria in
  OLAP;limiting data;limiting data:in
  OLAP reports   224
area graphs   279
area graphs:styling   351, 353
ARGLEN function   364
argument formats   363
argument types   362
arithmetic operations   92
arranging graphic elements   283
ascending order   93
assigning prefix operators   288
associating components   497
ATODBL function   410
Auto Prompt button   113, 293
AutoDrill   176
AutoDrill;measures:drilling down
  on;dimensions:drilling down on;drilling
  down on dimensions;drilling down on
  measures;column values in OLAP
  reports   184
automatic drill down   126, 176
automatic field population   556
AYM function   382
AYMD function   383

B

background images and properties   495
banner links   7
bar charts   416
BAR function   416
bar graphs   279, 427
bar graphs:associating with columns   427
bar graphs:associating with dimensions
  and measures   429, 432, 433, 436

bar graphs:attributes   429
bar graphs:deleting from measures   433
bar graphs:horizontal   427
bar graphs:styling   351, 353
bar graphs:vertical   427
bipolar graphs   279
BITSON function   365
blackout dates for schedules   744, 747
browser settings   3
btnBackColor   150
btnbold   150
btnCopy   150
btnCut   150
btnFldListList   151
btnFldListTree   150
btnForeColor   150
btnitalic   150
btnJustifyCenter   150
btnJustifyRight   150
btnLeftJustify   150
btnModDefine   150
btnPaste   150
btnRemoveFormat   150
btnSelectAll   150
btnunderline   150
burst values   694, 697, 700
burst values:specifying in an e-mail
  subject line   711
bursting reports   694, 700, 701
bursting reports:guidelines and
  limitations   722
Business View Master File   651
Business Views   646, 647, 648, 650, 651
Business Views menu options   648
Button Properties window   485
BY fields   74, 79
BY fields:OLAP and;ACROSS
  fields;OLAP and;drilling
  down on measures;ACROSS
  fields:sorting with;measures:drilling
  down on   190
BY fields:selecting   192
BY fields:selecting;ACROSS
  fields:selecting; dimensions;measures:in
  different dimensions   193
BYTVAL function   365

C

calculated values   83, 88
calculating remainders   418
calendar   522
Cascading Style Sheets (CSS)   105, 123,
  124
Cascading Style Sheets (CSS):generating
  internal CSS code   124
case sensitive searches   441, 442
cc mail considerations   711
centering character strings   368
chain feature   550
chained parameters   551

W

X

Y

Z

IBM ®

Printed in USA