Db2 Web Query for i Developer Workbench

Version 2.2.1

Contents

IBM

Preface

This documentation describes how to install and configure Db2 Web Query for i Developer Workbench on machines running Windows® operating systems.

How This Manual Is Organized

This manual includes the following chapters:

	Chapter/Appendix	Contents
1	Introducing Db2 Web Query Developer Workbench	Introduces Db2 Web Query Developer Workbench.
2	Navigating in Developer Workbench	Identifies the main components of the Developer Workbench interface.
3	Creating Metadata	Describes how to create and edit metadata in App Studio.
4	Creating Reports	Describes how to launch the Report Wizard to create reports, the types of reports you can create with App Studio, and the tabs and panels that are available when you are developing reports in the Report canvas.
5	Creating Charts and Visualizations	Describes how to launch the Chart Wizard to create charts and the types of charts you can create with App Studio.
6	Creating HTML Pages	Describes how to launch the HTML/Document Wizard, and how to create and edit HTML pages.
7	Editing Source Syntax	Describes the Text Editor that is available when you are developing the source syntax for procedures, procedure components, and other types of files.

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) in syntax for a value that you or the system must supply.
<u>underscore</u>	Indicates a default setting.
this typeface	Represents a placeholder (or variable), a cross-reference, or an important term. It may also indicate a button, menu item, or dialog box option you can click or select.
this typeface	Highlights a file name or command.
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 are 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 Db2 Web Query Developer Workbench

Db2 Web Query is a complete, web-ready data access and reporting system that connects users to data. It accesses and processes information located in any format, on any platform, and presents that information to end users through a web browser.

Business intelligence (BI) needs continue to evolve, as organizations seek to make more information from more sources available to more people inside and outside the enterprise. Developers need powerful, yet flexible tools in order to satisfy the increasingly sophisticated BI content demands posed by a growing number of employees, customers, suppliers, partners, and other stakeholders.

Db2 Web Query Developer Workbench is the Windows-based graphical user interface (GUI) environment for creating advanced Report Broker applications. Db2 Web Query Developer Workbench is the most robust BI application development platform on the market today.

In this chapter:

❏ What Is Developer Workbench?

❏ Launching Developer Workbench

❏ Documentation Overview

What Is Developer Workbench?

Building on the industry-familiar Microsoft Office Ribbon Interface, Developer Workbench offers a simplified user experience and workflow, empowering developers to create application content immediately, without a costly and time-consuming learning curve. Additionally, it dramatically increases development efficiency by eliminating the need for developers to utilize multiple tools to piece together a BI application.

Developer Workbench provides:

❏ **Rapid deployment.** Installation and setup are fast and simple.

❏ **Ease of use.** A familiar ribbon interface, which can be customized to individual preferences, allows users to get up to speed quickly.

❑ **Maximum flexibility.** Build and deploy ad hoc or instant queries that go beyond the prescriptions of any particular application. Developers can also add dimensions to data structures at any time without affecting other queries, since the underlying data source is not changed.

❑ **Support for complex requests.** Unlike other BI development environments, which require the use of complex code to handle anything more than the most simple of requests, Developer Workbench allows users to create any request, no matter how sophisticated, using the same intuitive interface.

Developer Workbench eliminates the complex multi-tool paradigm that exists with most other development solutions, providing a single, fully-integrated environment for rapidly designing and creating reports, dashboards, InfoApps, and other types of BI apps and content, including:

❑ Income statements, balance sheets, and other types of tabular or financial reports.

❑ Charts and visualizations. A library of more than 120 HTML5 chart types meet the needs of every information consumer.

❑ Dashboards that draw data from one source, or multiple disparate systems. Dashboards can be viewed online or offline in disconnected mode, with full interactivity and analytical capabilities.

❑ InfoApps, business intelligence applications that are form-driven and extremely simple to use, providing the most effective way to make information accessible to all stakeholders, as well as for custom and SaaS implementations.

❑ Mobile BI applications that run on any device and on any platform.

❑ Responsive designs that conform to the device form factor, maximizing the user experience.

❑ Metadata creation and modeling, such as multifact star schemas, joins, and business views.

Developer Workbench Capabilities

In Developer Workbench, you can do the following:

Access data and descriptions. Using the Metadata canvas, you can create new synonyms, and view or modify existing synonyms, in a graphical user interface. Synonyms enable you to access and interpret data sources for use in reporting applications. Capabilities include metadata design and development, with full visual modeling of schemas.

Create reporting applications. Build reporting procedures in the Report Canvas, Chart Canvas, or HTML Canvas, which can include the following components.

❏ **Reports.** Display your data in a tabular format. You can create stand-alone reporting procedures, or add the report to a document or an HTML page.

❏ **Charts.** Communicate your data in a more illustrative format. You can translate complex data into an easy-to-read and understandable visual format. This often provides a new perspective to the information that users view. You can create stand-alone charting procedures or add the chart to a document or an HTML page.

❏ **HTML pages.** Build webpages for users to launch, view, and analyze data. You can add multiple reports and charting procedures to an HTML page, in addition to the controls that enable you to manipulate the data. You can also apply styling through the use of cascading style sheets (CSS), JavaScript files, and jQuery animations.

❏ **Visualizations.** Create charts, maps, and grids to visually represent your data. You can add multiple visuals to the canvas to create a complete visualization, and you can update, change, or revise the visualization at any time to account for shifts in data needs.

Customize reporting applications. In addition to customizing your reporting applications by applying styling and color, you can add the following components to a procedure.

❏ **Virtual fields (DEFINE or COMPUTE statements).** Create temporary fields that are treated as real fields stored in the data source.

❏ **Define functions.** Create user functions that you can use in components of a procedure. You can retrieve your stored functions from the Functions Arguments dialog box.

❏ **Joins.** Define relationships between two or more data sources so that a report can use the data from all sources at once. You can also merge data sources by creating logical expressions on the Match canvas.

Db2 Web Query Architecture

This topic briefly explains the main Db2 Web Query components and implementations. For a complete description of Db2 Web Query, see the Db2 Web Query documentation.

Db2 Web Query and Your Network

Db2 Web Query integrates into your existing network by connecting your web server to your data. End users access Db2 Web Query applications through a web browser, so they need only the following elements:

❏ **Web browser.** To access Db2 Web Query applications, users need a browser and a TCP/IP connection to a web server.

❏ **Web server.** WebSphere Liberty handles requests by returning files to a browser or by executing processes that provide additional functionality. You can provide Db2 Web Query functionality by connecting to the web server using Java servlet calls.

❏ **Data.** Db2 Web Query can access data from almost anywhere. Once you have configured data access and described that data, you can report on it.

Db2 Web Query Components

There are two main Db2 Web Query components.

❏ **Db2 Web Query Client.** The Db2 Web Query Client resides on the web server and connects Db2 Web Query to the web through Java servlets. When a user makes a request from Developer Workbench or a browser, the Db2 Web Query Client receives and processes the request by passing it to the Reporting Server.

❏ **Db2 Web Query Reporting Server.** The Reporting Server is packaged with Db2 Web Query and resides on the same machine as the Client. The Reporting Server provides data access, number crunching, and report generation functionality.

Developer Workbench Architecture

This topic briefly explains the main Developer Workbench component and implementation.

❏ **Developer Workbench Graphical User Interface (GUI).** Graphical development and code generation features for application development. From Developer Workbench, it is necessary to connect to a Db2 Web Query environment before you begin to manage metadata or develop applications.

Db2 Web Query and Developer Workbench Processing

The following figure illustrates how Db2 Web Query and Developer Workbench process requests. Each step is explained below the figure.
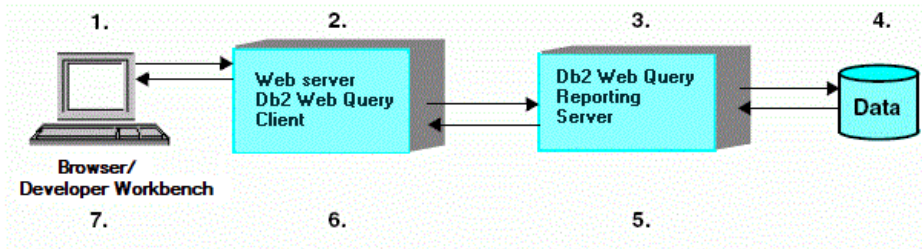
1. A user makes a request and passes parameters by calling a Db2 Web Query Servlet through links and forms on a webpage, or through Developer Workbench.

2. The request and parameters come to the Db2 Web Query Client on the web or application server, which processes the parameters and creates a request for the Reporting Server.

3. The Reporting Server receives the request, processes it, and accesses any necessary data.

4. Data is retrieved from data sources to process the request.

5. The Reporting Server processes the user request using the retrieved data.

6. The response is returned to the Db2 Web Query Client on the web or application server.

7. The response is returned to the user.

Launching Developer Workbench

Once you have installed Developer Workbench, you can launch the product and begin developing applications.

❏ **Launch Developer Workbench and the Developer Workbench Reporting Server at the same time.** From the Start menu, expand the *Db2 Web Query for i* program group, expand the *Db2 Web Query for i Developer Workbench 82* folder, and then double-click *Db2 Web Query for i Workbench*.

Note: If you have previously configured Db2 Web Query environments on your machine, those environments appear when you launch Developer Workbench.

Documentation Overview

The following documentation components are available for Developer Workbench users:

❏ Context-sensitive online Help system

You can access the online Help system through the product. Press F1 on your keyboard or click *Help* throughout the interface to open the Help window. The online Help system is designed to provide you all of the information that you need to develop content in Developer Workbench.

Chapter 2

Navigating in Developer Workbench

Developer Workbench provides an easy-to-use interface for creating applications, such as reports, charts, and HTML pages.

The first time you launch Developer Workbench, a default interface opens. You can modify the default interface by customizing the appearance of the application window and rearranging the placement of the main interface components.

These changes are automatically saved and made available the next time you launch the product.

In this chapter:

Opening the Application Menu of File-Related Commands

The Application button, represented by the Developer Workbench icon , opens the Application menu of file-related commands. You can save an active document and print an active document. The Options button opens the Developer Workbench Options dialog box, where you can set user preferences. The Application menu also lists recently opened documents. The Application button is always available. It is located in the upper-left corner of the Developer Workbench interface.

Tip: Double-click the Application button to exit Developer Workbench. If you exit Developer Workbench in this way, active (open) applications will not be saved.

From the Application menu, you can perform the following actions:

❏ **Save.** Save the active document.

❏ **Save As.** Save the active document with a new name.

❏ **Save All.** Save all open documents with the current file names. There will be no prompting for file names.

❏ **Run.** Run the report, chart, or HTML page. When you click the *Run* menu, the Message Viewer Options submenu opens.

From the Message Viewer Options submenu, you can select from among four Message Viewer Options that affect the Developer Workbench Viewer display when a report, chart, or HTML page is run. The options are:

❏ When you select *Message Viewer OFF* and run a report, the Developer Workbench Viewer window displays the report without any messages.

❏ When you select *Message Viewer ON* and run a report, the Developer Workbench Viewer window displays the report and a message.

❏ When you select *Display command lines,* the Developer Workbench Viewer window displays the report and the command lines.

❏ When you select *Display Dialogue Manager commands,* the Developer Workbench Viewer window displays the report and the Dialogue Manager commands, and the result of their evaluation.

❏ **Print.** Print the active document. When you click the *Print* menu, the Preview and print the document submenu opens.

From the Preview and print the document submenu, you can print the active document using the current print options, preview the active document before printing, and configure your print options.

Note: The Print and Quick Print options are only available when you edit content in the Text Editor, or when in a tab containing plain text.

❏ **Close.** Close the active document.

❏ **Close All.** Close all open documents. If changes were made to a file, you will be prompted to save the changes.

❏ **Options.** Open the Options dialog box, where you can set user preferences.

❏ **Exit.** Close the active document and exit Developer Workbench.

Note:

❏ In the Application menu, the terms *document* and *active document* represent all files which you can create in Developer Workbench.

❏ Spaces are not allowed in file names. When naming a report, chart, or HTML page in their respective wizards or in the Save As dialog box, spaces are replaced with underscores (_). When naming a procedure, spaces and underscores are not permitted, and when you press the Space bar, nothing happens. If the domain is set to display by title, the wizards, Open dialog box, and Save As dialog box will still expect a proper name, with no spaces, to be entered.

Reference: Save As Dialog Box

The Save As dialog box opens when you click *Save As* from the Application menu. You can use this dialog box to save a file to another location or to provide a different file name.

Configured Environments list

Contains a list of currently Configured Environments. You can navigate to different environments, development areas, and project folders to display different files in the file list.

File list

Displays all files of a specific type in a selected project folder.

File name text box

Use this text box to type the name of a file you want to open.

Note: If you are in the Repository area and the Configured Environments tree is set to display by title, the Save As dialog box will only require a name to be entered.

File Type drop-down list

Displays the filter based on applicable file type.

Details Toggle

Toggles between displaying the files in the file list with details (date modified, size, and so on) or in a list. Displaying files with details is the default.

> **Note:** If you choose to display details or not, Developer Workbench remembers your choice and will keep the setting you chose the next time the Open File dialog box is open. For example, if you choose to display file details the first time you invoke the Open File dialog box, then the next time you invoke the Open File dialog box, file details are shown.

Refresh Button

When clicked, refreshes the list of available files.

Setting User Preferences Using the Options Dialog Box

You can customize Developer Workbench by setting specific user preferences, using the following tabs in the Developer Workbench Options dialog box. To open the Options dialog box, click *Options* in the Application menu.

Reference: General Tab

The General tab contains options for starting Developer Workbench, minimizing the main window, and other settings. The following options are available.

Main Window Options

Option	Description
Maximize main window	Maximizes the application window when you begin each session.

Other Settings

Option	Description
Show Welcome Screen	Controls the appearance of the Welcome screen when Developer Workbench is launched.
Confirm close	Displays a prompt for users to confirm that they want to exit Developer Workbench.
Recent file list limit	Allows you to set the number of recently used files that appear in the Recent file lists. The default value is 25.

Option	Description
Ping Interval (minutes)	Sets the frequency at which requests are sent to the web or application server so your connection does not time out. The default is 5.
Reset All Message Boxes	Resets all message boxes to their default state. For example, if you selected the *Don't show this message again* check box in any message box, clicking this button will reset all message boxes. Once you click this button, it becomes inactive until you choose not to display a message box in the product.
Reset All Options to Default	Restores all options in Developer Workbench to the default settings.
Default file editor	Lists file editor options, such as: ❑ Edit in Developer Workbench canvas ❑ Edit in Text Editor ❑ Edit in Windows registered tool **Note:** The Edit in Windows registered tool option is only visible if the file type you are accessing has been associated with a Windows application. The file editor used determines which shortcut menu options are available and controls the default behavior for double-clicking and right-clicking files.

Option	Description
Language	Provides a drop-down list that allows you to change the language of the Developer Workbench screen text. After you change the language, restart Developer Workbench to make the change effective. The languages listed are selected in the Dynamic Language Switch setting in the Db2 Web Query Administration Console.
	The Sync with WebQuery check box is selected, by default. When this check box is selected, the language used for Developer Workbench and Db2 Web Query are synced with the language selected in the Language drop-down list. If this check box is not selected, Developer Workbench uses the language selected in the Language drop-down list and Db2 Web Query uses the language selected in the Environments dialog box. If you make changes, restart Developer Workbench to make the change effective.

Reference: **HTML Page Tab**

The HTML Page tab enables you to set grid settings for an HTML page, preview reports and charts, set single or multiple layers, and alter the frequency in which you refresh thumbnails.

Note: If you make changes to the HTML Page settings in the Developer Workbench Options dialog box (Grid Settings, Preview Settings), you need to press F5 to refresh an HTML page that is currently open in order to see the changes.

The following options are available.

Grid Settings

Option	Description
Show Grid	Enables you to view a grid while you create an HTML page. The Show Grid check box is selected, by default. Clear this check box if you do not want to develop with a grid.
Snap to Grid	Enables you to snap to grid on demand. The Snap to Grid check box is selected, by default. Clear this check box if you do not want to enable the snap to grid option.
Width	Customizes the width of your grid. The default is 10 pixels.
Height	Customizes the height of your grid. The default is 10 pixels.

Form type

Option	Description
None	Specifies no form object in the HTML page.
Single layer	Specifies a single layer form in the HTML page. This is the default setting.
Multiple layer	Specifies a multiple layer form in the HTML page.

Tab Characters

Option	Description
Insert spaces	Indicates that, when the Tab key is pressed, the number of spaces specified in the Tab size option is inserted.
Keep tabs	Indicates that, when the Tab key is pressed, a tab character is inserted.

Option	Description
Tab size	Specifies the number of spaces inserted when you press Tab.

Check boxes

Option	Description
Show 'New Parameters' dialog	Sets the New Parameters dialog box to appear in the HTML page. This is the default setting.
Default caching option	Sets the Default caching option to run in the HTML page. By default, this option is not selected.
Auto Arrange Objects	Sets the objects in the HTML page to arrange automatically. This is the default setting.

Preview settings

Option	Description
Reports and Charts Preview	Enables you to preview reports and charts before saving and deploying the HTML page. This is the default setting.
Simulated Data	Enables you to view the HTML page based on simulated data from Db2 Web Query.
Live Data	Enables you to view reports and charts using live data in the HTML page. This is the default setting.
Record limit for reports	Sets the record limit for reports. Values range from -1 to 999. The default setting is 500.
Record limit for input controls	Sets the record limit for input controls. Values range from -1 to 999. The default setting is 10.
Use Prefix	Sets a prefix to run before each report or chart component on an HTML page.

Option	Description
Refresh thumbnails every seconds	Sets the time interval for refreshing thumbnails. Values range from 0 to 999 seconds. The default setting is 20.
Default Theme	Sets the default theme for an HTML page. The Information Builders theme is the default theme.

Reference: Document Tab

The Document tab enables you to set grid settings for a document, preview reports and charts, and alter the frequency in which you refresh thumbnails. The following options are available.

Grid Settings

Option	Description
Show Grid	Enables you to view a grid while you create a document. The Show Grid check box is selected, by default. Clear this check box if you do not want to develop with a grid.
Snap to Grid	Enables you to snap to a grid on demand. The Snap to Grid check box is selected, by default. Clear this check box if you do not want to enable the snap to grid option.
Width	Customizes the width of your grid. The default is 10 pixels.
Height	Customizes the height of your grid. The default is 10 pixels.

Tab Characters

Option	Description
Insert spaces	Indicates that, when the Tab key is pressed, the number of spaces specified in the Tab size option is inserted.

Option	Description
Keep tabs	Indicates that, when the Tab key is pressed, a tab character is inserted.
Tab size	Specifies the number of spaces inserted when you press Tab.

Preview Settings

Option	Description
Reports and Charts Preview	Enables you to preview reports and charts before saving and deploying the document. This is the default setting.
Simulated Data	View the document based on simulated data from Db2 Web Query.
Live Data	View reports and graphs using live data in the document. This is the default setting.
Record limit for reports	Sets the Record limit for reports. Values range from -1 to 999. The default setting is 500.
Use Prefix	Sets a prefix to run before each report or chart component on a document.
Refresh thumbnails every seconds	Sets the time interval for refreshing thumbnails. Values range from 0 to 999 seconds. The default setting is 20.

IBM

Reference: **Environments Tab**

Note: The Environments tab does not apply to Db2 Web Query. The Environments options are used to control the development areas that are visible in the Configured Environments tree. At least one option must be selected and applied to all configured environments.

The following options are available.

Environments settings

Option	Description
Show Projects area	If selected, displays the Projects area in the Configured Environments tree.
Show Data Servers area	Displays the Data Servers area in the Configured Environments tree.
Show domain area	Displays the domain area in the Configured Environments tree.
Show Web Applications area	Displays the Web Applications area in the Configured Environments tree.

Reference: **Help Configuration Tab**

The Help Configuration tab enables you to configure the location of the web-based Help system for Developer Workbench. It provides the fields in which you can modify the Developer Workbench Help configuration information to access the online Help system from your environment. The following options are available.

Option	Description
Protocol	Specify the protocol, either HTTP or HTTPS.
Host	Type the name of the machine where the Help resides.
Port	Type the port number of the web and/or application server.
Help Context Root	Type the context root for the location where the Help is hosted.

Reference: **Output Viewer Settings Tab**

The Output Viewer Settings tab enables you to set a specific browser to use for output, and to set the navigation options for the output (run in new window, run in same window). The following options are available.

Browser Setup

Option	Description
Browser Setup	Select a browser to use for output. **Note:** ❏ The Browser Setup section is populated with a list of browsers that are installed on the Developer Workbench machine. ❏ Developer Workbench works with Mozilla Firefox®, Google Chrome™, Microsoft Internet Explorer®, and Microsoft Edge® browsers. Microsoft Edge is the default browser for Windows 10 operating systems. Internet Explorer is the default browser for all Windows operating systems prior to Windows 10. ❏ The browser that is highlighted when you close the dialog box is the browser that will be used for execution.
Browser executable path	Sets the location of the browser executable. This field box is populated based on the browser selection.
Web driver location	Sets the location of the web driver for the browser. This field box is populated based on the browser selection.
Test URL	Indicates the test URL for testing the browser setup. The default URL is http://www.ibi.com.
Test browser setup	Tests the browser setup, using the test URL. If the test is successful, the webpage will display in the browser and you will receive a success message. If the test is unsuccessful, you will receive a failed message.

Option	Description
Save selected browser setup	Saves the setup information for the selected browser.

Navigation Options

Option	Description
Run in new window/tab*	Sets the output to run in a new window or tab.
Run in same window	Sets the output to run in the same window.

Reference: **File Extensions Tab**

The File Extensions tab lists the currently supported file extensions. They are all selected, by default, and you can clear the check box for any file types that you do not want retrieved from the server.

Note: You must exit and restart Developer Workbench for your selections to take effect.

Option	Description
File Type	The file extension for a specific type of file.
Display	If selected, displays this file type throughout Developer Workbench.

Accessing Frequently Used Commands

The Quick Access Toolbar provides access to frequently used commands, and the option to customize the toolbar with the commands that you use most often. By default, the Save, Save All, Quick Print, Undo, Redo, Cut, Copy, Paste, Run, and Customizing Quick Access Toolbar buttons appear on the Quick Access Toolbar, as shown in the following image.

Note:

❏ When using the undo and redo commands, consider that these actions may not take effect on options or information that is changed in every panel. In addition, if you change multiple options in one panel, for example, the undo or redo command will reverse all of those indicated changes in one action.

❏ The Cut, Copy, and Paste commands can be used on text, objects on the HTML canvas and Document canvas, and fields on the Report canvas.

The Customize Quick Access Toolbar button (down pointing arrow) appears on the Quick Access Toolbar and opens the Customize Quick Access Toolbar menu. This menu contains a list of the default commands on the toolbar. You can clear these commands to hide them from the toolbar. There are also options to add more commands with the Customize dialog box, move the Quick Access Toolbar below the ribbon, and hide the ribbon.

You can also add commands using the Customize dialog box. To access the Customize dialog box, select *More Commands* from the Customize Quick Access Toolbar menu. From the Customize dialog box, you can choose which commands you want to add to or remove from the Quick Access Toolbar, as well as the order in which the commands appear.

The Quick Access Toolbar is always available. By default, it is located in the upper-left corner of the Developer Workbench interface. To move the Quick Access Toolbar below the ribbon, click the arrow button, and then click *Show Below the Ribbon*.

Reference: Customize Dialog Box

Using the Customize dialog box, you can add or remove commands from the Quick Access Toolbar. By providing access to the Customize Keyboard dialog box, you can also customize the keyboard by setting keyboard shortcuts for commonly performed tasks.

Choose commands from

Opens a list of available areas in Developer Workbench. The Commands section dynamically repopulates with the commands that are associated with your selection.

Commands

A list of commands from a specific area of Developer Workbench. You can select commands and use the Add or Remove buttons to add or remove them from the Quick Access Toolbar. Commands are represented by the icon that displays on the ribbon. Commands that have no icon and a vertical arrow are command groups. Commands that have an icon and a horizontal arrow are commands with submenus. You can add all commands, command groups, and commands with submenus to the Quick Access Toolbar.

Add

Adds a selected command from the Commands list to the Quick Access Toolbar.

Remove

Removes a selected command from the Quick Access Toolbar. If no command is selected, the last command on the Quick Access Toolbar is removed.

Up

Moves a command up in order on the Quick Access Toolbar.

Down

Moves a command down in order on the Quick Access Toolbar.

Reset

Reverts the Quick Access Toolbar to use the default commands (Open, Save, Quick Print, and Run).

Show Quick Access Toolbar below the Ribbon

Displays the Quick Access Toolbar below the ribbon when selected.

Customize

Opens the Customize Keyboard dialog box. For more information on the Customize Keyboard dialog box, see *Customize Keyboard Dialog Box* on page 34.

OK

Applies the changes you made to the Quick Access Toolbar.

Cancel

Exits the Customize dialog box without applying any changes you made to the Quick Access Toolbar.

Help

Opens the Developer Workbench Help.

Procedure: **How to Add Commands to the Quick Access Toolbar**

You can quickly add commands to the Quick Access Toolbar by right-clicking the ribbon, groups, or the individual commands, and selecting *Add to Quick Access Toolbar*, or you can follow the procedure below.

1.   On the Quick Access Toolbar, click the arrow, and then click *More Commands*.

The Customize dialog box opens.

2. In the Choose commands from list, select a command group.

3. Select a command name, and then click *Add*.

    The command name appears in the Quick Access Toolbar list.

    If you select a command group or a command with a submenu, all associated commands are accessible from the Quick Access Toolbar.

    To remove a command name from the Quick Access Toolbar list, select the command, and then click *Remove*.

4. Click the arrow buttons to rearrange the order of command names in the list.

    This also rearranges how the command buttons appear in the Quick Access Toolbar.

5. Click *OK* to save your changes.

Reference: Customize Keyboard Dialog Box

The Customize Keyboard dialog box is used to create and edit keyboard shortcuts to commands. You can access the Customize Keyboard dialog box by clicking the *Customize* button in the Customize dialog box from the Quick Access Toolbar.

Categories

A list of the available areas in Developer Workbench. The commands associated with that area are displayed in the Commands section.

Commands

A list of commands from a specific area of Developer Workbench. You can select commands and then assign a keyboard shortcut to them.

Current Keys

Displays the current keyboard shortcuts associated with the selected command.

Press new shortcut key

Press the keyboard shortcut you want to assign to a specific command. Once the Assign button is clicked, this keyboard shortcut will then be displayed in the Current Keys list for the selected command.

Set Accelerator for

Enables you to set an accelerator for one of the following: Default, HtmlPage, or Document.

Description

Provides a description of the selected command.

Assign

Allows you to assign a keyboard mapping to a command.

Remove

Allows you to remove an assigned keyboard mapping.

Reset All

Resets all keyboard mapping assignments for every command. A warning displays to ensure that want to perform this action.

Close

Exits the Customize Keyboard dialog box and return to the Customize dialog box.

Procedure: **How to Create Keyboard Shortcuts**

1. To open the Customize dialog box, do one of the following:

    ❑ Click the drop-down arrow on the Quick Access Toolbar and then click *More Commands*.

    ❑ Use the shortcut menu on the ribbon or Quick Access Toolbar and then click *Customize Quick Access Toolbar*.

2. Click *Customize*.

    The Customize Keyboard dialog box opens.

3. From the Categories list, select an area that contains the command that you want to create a shortcut for.

4. From the Commands list, select a command.

5. In the Press new shortcut key area, enter the key combination that you want to use with the command you selected.

    You can make a shortcut up to three keystrokes long (For example, Ctrl+Shift+R).

6. Click *Assign*.

    A shortcut is now assigned to the command you selected.

7. Click *Close* to exit the Customize Keyboard dialog box.

Developing Applications in the Canvas Area

You can use Developer Workbench features and functions to develop applications in the canvas area. The canvas is designed to maximize your design-time options. At run time, elements appear in the browser based on the order in which they were added to the canvas, left to right, and top to bottom. Your canvas may have the same appearance as the browser at run time, but it is not required. You can drag elements to areas of the canvas as needed to make it easier to work with them.

The size of your canvas depends on the placement and location of panels around the interface window.

As you develop in Developer Workbench, tabs open in the canvas area. Since you can develop an HTML page at the same time that you create a report, each canvas opens with a different colored tab. The colored tab at the top of each canvas displays the name of the file you are developing.

If you chose to name the file after you develop the content, a default file name, such as Report1, appears on the tab, as shown in the following image.

A second set of tabs opens when you create procedures with multiple components. For example, you can add Set, Define, and Join components to a procedure at any time. These canvases open as tabs below the report or chart procedure, as shown in the following image.
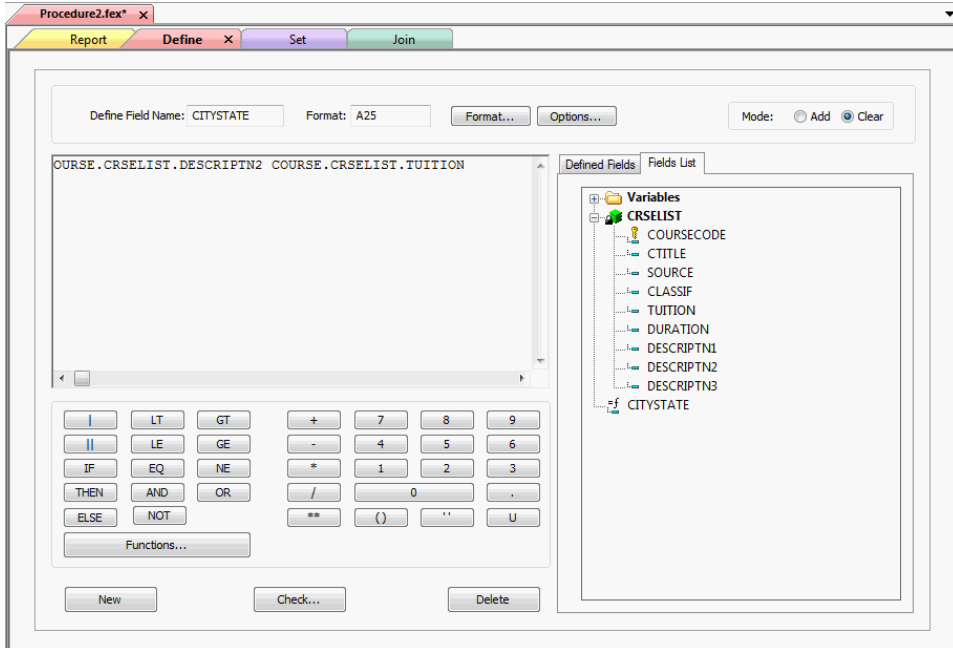
In addition to colored tabs, some canvases also contain View tabs. A View tab appears at the bottom of the canvas and changes the display of information on the canvas. View tabs, which display different views of the same component, are shown in the following image.
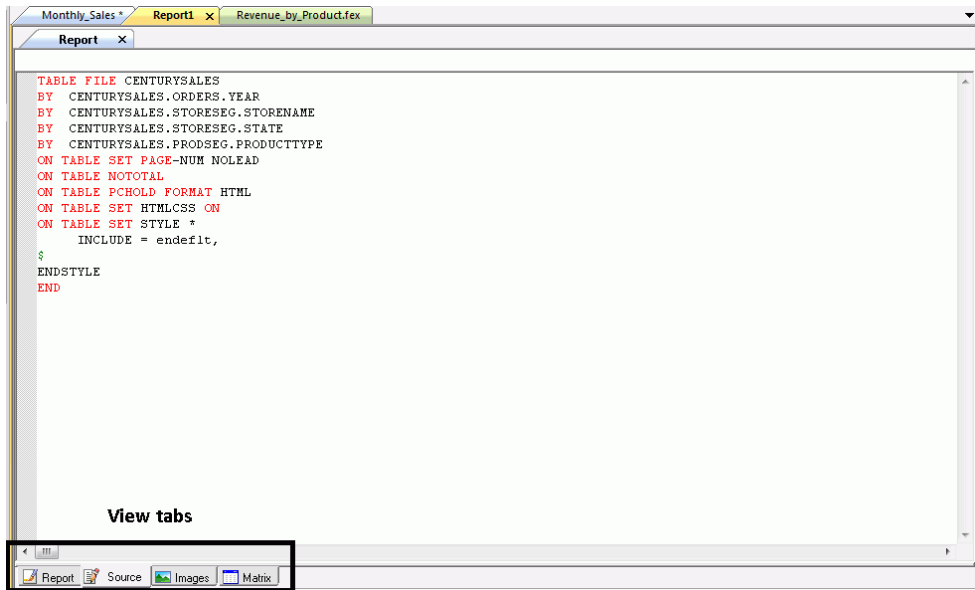
View tabs, such as those on the HTML canvas, also enable additional features and functions that you can use to customize an application. For example, in the HTML canvas, you can add JavaScript functionality to an HTML page through the Embedded JavaScript view tab.

You can develop Developer Workbench content in the following canvases.

Creating Charts Using the Chart Canvas

The Chart canvas enables you to create different types of charts. You can select from a variety of chart types and output formats, and add custom features to a chart. To access the Chart canvas, on the *Home* tab, in the *Content* group, click *Chart*. The Chart Wizard opens.

Creating Cascading Style Sheets Using the Cascading Style Sheet Canvas

The Cascading Style Sheet canvas allows you to create a cascading style sheet. To access the Cascading Style Sheet canvas, right-click any folder or file in the Environments Tree panel and in the shortcut menu click *New*, and then click *Cascading Style Sheet*.

Working With the JavaScript and CSS Canvas

The Embedded JavaScript canvas and the Embedded CSS canvas have a built-in Auto Complete feature to assist in the creation of code segments, similar to functionality offered by Microsoft IntelliSense® and other code editors. When you begin typing a string, array, number, or user-defined object, a list of available JavaScript methods, related to those code segments, displays.

The Auto Complete feature is on by default. To turn it off, right-click in the Cascading Style Sheet canvas and clear *Auto Complete* from the shortcut menu.

Note: If there is a syntax error, the Auto Complete features will not display.

Editing Component Code Using the Text Editor Canvas

The Text Editor canvas allows you to edit component code. To access the Text Editor canvas, right-click a component in the Procedure View panel, and then click *Open in Text Editor*.

Creating HTML Pages Using the HTML Canvas

The HTML canvas enables you to create HTML pages for Db2 Web Query. Designed for business users, you do not need to have experience in HTML, CSS, XML, or JavaScript. The HTML canvas is a graphical interface that is easy to use.

Note: HTML canvas files are intended to be created and edited with the HTML canvas only. You can either modify HTML pages created in the HTML canvas or create HTML pages using a text editor. When these pages are opened in the canvas, a message displays that allows you to decide whether to open the file in the canvas or the text editor.

If you are a web developer and want to write your own code instead of using the HTML canvas, you can use the authoring tool of your choice and also use RESTful Web Services.

Note: When opening an HTML file that was not created in the HTML canvas, you will be prompted to open the file only in a Windows Associated tool.

To access the HTML canvas, on the *Home* tab, in the *Content* group, click *HTML/Document*. When the HTML/Document Wizard opens, select *HTML Page* from the available options. The HTML canvas opens.

The HTML canvas uses HTML 5 Document Type Definitions (DTD).

Creating and Styling Reports Using the Report Canvas

The Report canvas provides you with many features that enable you to create and style reports. To access the Report canvas, on the *Home* tab, in the *Content* group, click *Report*. The Report Wizard opens. This is the InfoAssist+ tool that is available in the Web Query browser interface. Follow the prompts in the Report Wizard to create a report. The Report canvas opens.

Working With Synonyms Using the Metadata Canvas

The Metadata canvas provides a graphical interface that enables you to work with synonyms and perform tasks, such as creating, viewing, and modifying synonyms. To access the Metadata canvas from the Environments Tree panel, right-click a Master File and select *Metadata Canvas*. To access the Metadata canvas from the ribbon, on the *Home* tab, in the *Content* group, click *Data*, and then click *Synonym via Metadata Canvas*.

Customizing Canvases

You can switch between the canvases by clicking the colored tab at the top of the canvas. You can also develop in the canvases side by side.

Procedure: **How to Move Canvases for Side-by-Side Development**

To create side-by-side canvas development, follow these steps:

1. Drag one canvas tab away from its original position.

   A pointer with a document icon appears.

2. Drag the pointer with the document icon to the upper-right corner of the canvas.

   The canvases now appear side by side.

   Tip: Repeat the preceding steps to develop on additional canvases side by side.

Accessing Features With the Ribbon

The Developer Workbench ribbon replaces traditional toolbars with a single command bar that organizes commands and controls into a series of static and contextual tabs. From the ribbon, you can also access the Db2 Web Query for i Administration menu, the Style menu, and the Help menu.

The default ribbon is shown in the following image.
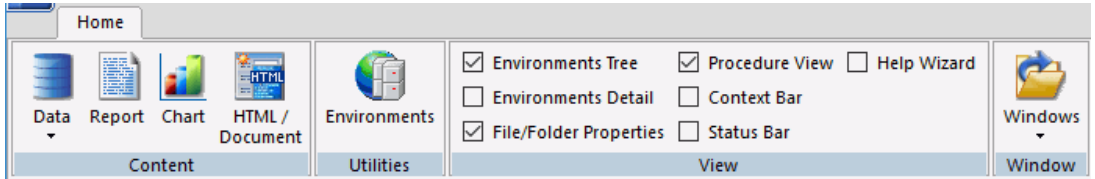
Each tab on the ribbon, whether static or contextual, contains commands and controls that are organized into groups, according to their function or relationship. All tabs are named and all groups are labeled. Icons are used to represent the commands or controls in each group. By default, all icons are labeled.

In addition to single-click commands, some commands on the ribbon contain a submenu of options that opens when you click the arrow associated with the command. These arrows can appear below or next to a label. An example of this is the Data command, which opens a submenu of options, as shown in the following image.
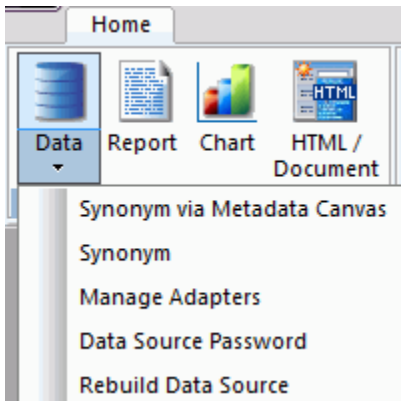
Note: There is no default ribbon size. As your monitor or application window size changes, the commands and controls on the ribbon resize themselves to fit the available space. Large monitors or application windows will display larger icons and entire groups. Smaller monitors or application windows may display groups as a single, labeled icon that opens a submenu.

Building an Application Using the Home Tab

The Home tab enables you to access the features and functions necessary to build an application, create new components, open existing components, run existing components, and edit components. The Home tab contains the Content, Utilities, View, and Window groups. The Home tab is always available and is static. It is located in the upper-left corner of the Developer Workbench interface, below the Quick Access Toolbar, as shown in the following image.
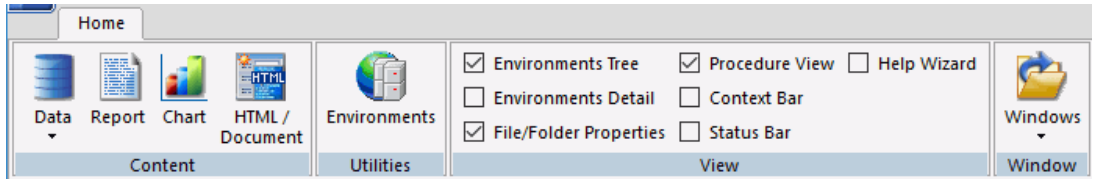
Note: The Home tab is the only static tab in Developer Workbench.

Creating an Application Using the Content Group

The Content group contains the components or tools that you can use to create an application. These include: Data, Report, Chart, and HTML/Document. The Content group is shown in the following image.
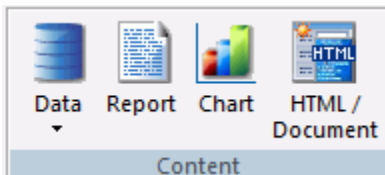
The commands in the Content group are:

Data

Contains the Synonym via Metadata Canvas, Synonym, Manage Adapters, Data Source Password, and Rebuild Data Source commands. These provide you the options you need to create and edit synonyms for existing adapters, configure a new adapter, or add a remote server.

If you click *Synonym via Metadata Canvas*, the Data Source Definition Wizard opens. Navigate to the location of the data source you want to create or edit and click *OK*.

If you click *Synonym*, the Select Server Node dialog box opens. Navigate to the application folder where you want to create your synonym and click *OK*. After the location of the synonym is determined, the Select Adapter dialog box opens. Select the adapter you want to use and click *OK*. You will then be able to edit the synonym details.

If you click *Manage Adapters*, the Select Server Node dialog box opens. Navigate to a valid server location to begin configuring a new adapter or edit the settings of an existing adapter.

If you click *Data Source Password*, the Password Dialog box opens. You can enter the password for the data source you have selected in your Environments Tree panel.

If you click *Rebuild Data Source*, the Rebuild dialog box opens. You can rebuild a disorganized file, index a specific field from a file, or check the integrity of a file.

Note: To index a specific field from a file, the field must have FIELDTYPE=I, specified in the Master File. You can do this from the Metadata canvas using the FIELDTYPE property, I-Index check box.

Report

Opens the Report canvas in report mode after you enter a procedure name and select a location for your report from the Report Wizard.

In report mode, you can create and style simple or complex reports, add data to the Report canvas, and style that data creating a graphical representation of the report page. This allows you to view how the report displays at run time.

Using report mode, you can:

❏ Display and sort data.

❏ Select records.

❏ Include totals, subtotals, column calculations, headings, footings, and images.

❏ Format columns.

❏ Style fonts, colors, and grids.

❏ Add drill downs to detailed reports and URLs.

❏ Save output in many types of formats for display and reuse.

❏ Create active reports.

Chart

Opens the Chart canvas after you enter a procedure name and select a location for your chart from the Chart Wizard.

In chart mode, you can easily transform almost any type of data into effective custom charts. You can create a variety of charts, such as bar charts, line charts, scatter charts, or pie charts, to help users analyze data in a different way. You can include selection criteria by defining parameters for your data. Additionally, you can apply drill-down capabilities and conditional styling to highlight specific data on a chart.

The Chart canvas contains a ribbon of features and options that allow you to add and style the data in your chart. Contextual tabs open within the Chart canvas, rather than on the Developer Workbench ribbon.

HTML/Document

Opens the HTML/Document Wizard. You can choose the type of page that you want to create from the following options: HTML Page, Guided Report Page, or Document. Depending on your selection, the corresponding canvas will open.

HTML mode provides the features you need to create HTML pages so that end users can interact with your data. You can build and customize an HTML page in the HTML canvas, using the contextual tabs on the ribbon to add objects to the page. HTML mode is fully integrated with JavaScript and cascading style sheets (CSS).

In the HTML canvas, you can:

❏ Build an HTML launch page.

❏ Add push buttons, hyperlinks, and other controls to launch Db2 Web Query reports in your application.

❏ Create a launch page for one or more reports that contain parameters.

❏ Create a complete HTML page by adding multiple reports and charts.

❏ Create an advanced report layout by including images, frames, and other web objects.

❏ Modify the location, size, and properties of all objects in your page layout.

❏ Set background, font, and other page properties.

In the Document canvas, you can coordinate and distribute layouts made up of multiple reports and charts in a single document. You can position reports and charts anywhere on a single page or combine a series of layouts within a single document. When creating compound reports in the Document canvas, PDF, HTML, active reports, PowerPoint, and Excel are available as output formats.

Creating a New Environment and Accessing the Command Console Using the Utilities Group

The Utilities group enables you to create a new environment. The Utilities group is shown in the following image.

The command in the Utilities group is:

Environments

> Opens the Environments List dialog box. A Db2 Web Query environment consists of a web server, a Db2 Web Query Client, and a Reporting Server. Adding Db2 Web Query environments allows you to create and manage multiple environments, such as development, test, and production instances. You can set a new development environment, add a new environment, edit an existing environment, or remove an environment.

Controlling the Display of the Screen Using the View Group

The View Group enables you to control what displays on your screen. For example, you can choose whether to display or hide the Environments Tree or Help Wizard. The View group is shown in the following image.
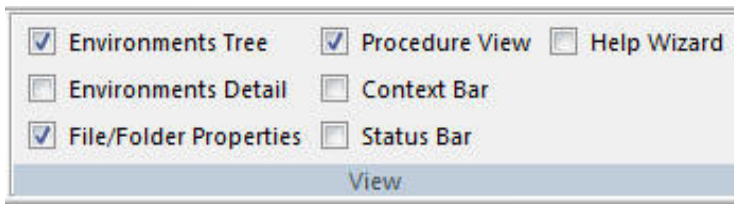
The commands in the View group are:

Environments Tree

> Displays or hides the Environments Tree panel, which displays on the left side of the screen and displays environment information.

Environments Detail

> Displays or hides the Environments Detail panel.

File/Folder Properties

Displays or hides the File/Folder Properties panel.

Procedure View

Displays or hides the Procedure View panel.

Context Bar

Displays or hides the Context Bar. The Context Bar displays under the ribbon.

Status Bar

Displays or hides the Status Bar. The Status Bar displays along the bottom of the screen.

Help Wizard

Displays or hides the Help Wizard. By default, the Help Wizard panel opens in the bottom-right corner of the interface. The Help Wizard content dynamically changes, depending on where you are in the interface.

Viewing and Manipulating Windows Using the Window Group

You can view and manipulate all opened windows. The Window group is shown in the following image.

The commands in the Window group are:

New Window

Opens another view of the same document, enabling you to have multiple canvas views of the same document. You can edit in any one of multiple views, each reflecting the changes made to the document. This option is only available when using the Text Editor. This option is not enabled for all other canvas tools.

Windows

Manipulates all opened windows. You can view and manipulate the windows using the Windows dialog box. The Windows dialog box enables you to Activate or Save an open window.

Using the Db2 Web Query for i Administration Menu

From the Db2 Web Query for i Administration menu, you can access the Db2 Web Query for iBI Portal, the Report Broker Console, and the Session Viewer. The Db2 Web Query for i Administration menu is shown in the following image.
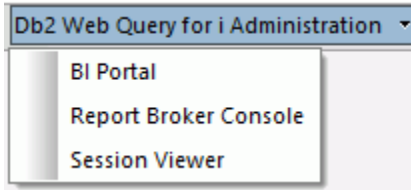
The Db2 Web Query for i Administration menu is only available when you select an environment in the Environments Tree panel.

Note: Developer Workbench always uses Microsoft Internet Explorer for all the tools available on the Db2 Web Query for i Administration menu. This is true even if you have selected another browser as the default viewer.

BI Portal

Opens the Db2 Web Query BI Portal. Here you can create dynamic websites using the content you create in Developer Workbench.

Report Broker Console

Opens the Report Broker Console. Here you can schedule and distribute Db2 Web Query reports, the content of URLs, and files.

Session Viewer

Enables you to review system events that took place during recent work sessions and export them to system administrators or customer support staff. Traces of system events and error messages captured by the Session Viewer provide a clear picture of system operations, and enable you to investigate the causes of system disruptions or performance issues.

Customizing the Appearance of the Interface Using the Style Menu

You can customize the appearance of the Developer Workbench interface from the Style menu. From this menu, you can select a theme with which to customize the Developer Workbench interface. The themes available to you are determined by the themes that you have installed for Microsoft® Office. The Style menu is always available. It is located in the upper-right corner of the Developer Workbench interface, as shown in the following image.
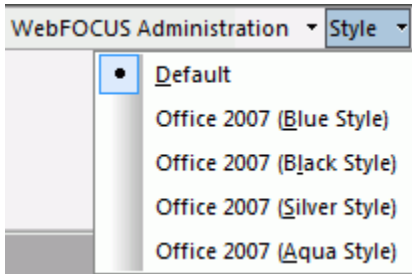
Note:

❏ When you close Developer Workbench, the theme that you have selected is saved and used the next time you launch the product.

❏ When you change to the Office 2007 styles, the drop-down arrow is removed from the panels. You can still access the menu by right-clicking the top frame of the panel.

Accessing Developer Workbench Help Content

You can access the Developer Workbench Help content, *Db2 Web Query for i Forums*, Developer Workbench licensing information, and other product options from the Help menu. The Help button is always available. It is located in the upper-right corner of the Developer Workbench interface, as shown in the following image.
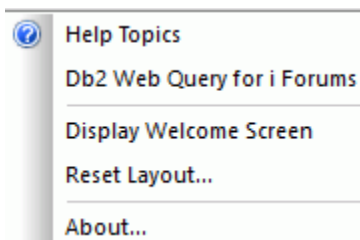
From the Help menu, you can do the following:

❏ Access the online Help system. Click *Help Topics* to open the Developer Workbench Help window.

❏ Access the IBM developerWorks® forums. Click *Db2 Web Query for i Forums* to access our online developer center and more than a message board. It is an interactive network of developers from almost every profession and industry, collaborating on solutions and sharing tips and techniques.

❏ Reopen the Welcome screen after it has been closed. Click *Display Welcome Screen* to reopen the Welcome screen.

❏ Reset the interface layout. Click *Reset Layout* to return the Developer Workbench interface settings, such as the ribbon, Quick Access Toolbar, panels, and styling, to their default. This change will not occur until Developer Workbench is restarted.

❏ View license and product information. Click *About* to open the About Db2 Web Query for i Developer Workbench dialog box.

Working With the Environments Tree Panel

The Environments Tree panel is the primary navigational aide in Developer Workbench. Using it, you can always stay oriented to the Repository or web application that you are working in. You can use the panel to move or copy objects between folders, share content with other Developer Workbench users, or hide content.

Note:

❏ Do not copy and paste Content folders from one environment to another. The use of Change Management packages is recommended to move Content folders from one environment to another in Developer Workbench.

❏ When you copy and paste from the Repository area to the Data Servers area, the IBFS properties (metadata) are lost and InfoAssist+ cannot function. The same issue occurs if you copy and paste from the Domains area to your desktop, and then copy and paste from your desktop back to the Data Servers area.

From the Environments Tree panel, you can:

❏ Customize the appearance of the panel.

❏ Filter content listed under the Db2 Web Query environments in the Configured Environments tree.

❏ Access configured Db2 Web Query environments to create new content or modify existing content.

❏ Refresh an application or folder in the Configured Environments tree, to view new files and updated file information, and omit deleted files.

The Environments Tree panel is shown in the following image.
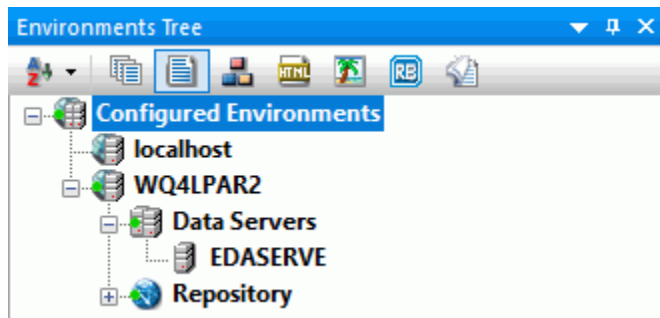
The Environments Tree panel opens with Developer Workbench.

Reference: Environments Tree Toolbar

The Environments Tree toolbar contains the buttons and menus that you need to navigate, filter, and sort the information that appears in the Configured Environments tree.

View Options

Presents options for viewing items (for example, alphabetical sorting or grouping), as described in the following table.

Option	Name	Function
	View items sorted in Alphabetical order	Sorts the files alphabetically from A to Z.
	View items sorted in reverse Alphabetical order	Sorts the files alphabetically from Z to A.
	View items sorted in Chronological order	Sorts the files by the last saved time, in descending order.
	View items sorted in reverse Chronological order	Sorts the files by the saved time, in ascending order.
	View Items grouped by File Type	Sorts the files by file type.
	Respect Sort Order Property	Respects the Sort Order property value specified in the File/Folder Properties panel. This is the default.

Option	Name	Function
☑	View items by Title	Displays the files by title and sorts the files by title. If an item does not have a title, its name is shown.
☑	View items by Name	Displays the files by name and sorts the files by name. If an item does not have a title, its name is shown.
	Refresh View	Refreshes all of the files and folders that you see in the Configured Environments tree. *Refresh View* also shows the dependencies of the file or folder that you have selected.
	Show All Content files	Filters the tree to show all content file types, except for Master Files. To view Master Files, click *Show only Master files*.
	Show only Procedure files	Filters the tree to show only this file type.
	Show only Master files	Filters the tree to show only this file type.
	Show only HTML files	Filters the tree to show only this file type.
	Show only Image files	Filters the tree to show only this file type.
	Show only Report Broker files	Filters the tree to show only this file type.
	Other files	Filters the tree to show other file types.

Viewing Nodes in the Configured Environments Tree

The Configured Environments tree displays the Db2 Web Query Environments , Repository, and Data Servers nodes. These are the development areas where you can create content.

If a node under the Configured Environments Tree has been populated and contains content, a green dot is shown on the node icon. If a node has been populated and does not contain any content, a red dot is shown on the node icon. If a node has not been populated, Developer Workbench does not know if there is any content in that node and no dot is shown on the node icon.

Note: By default, the Configured Environments tree sorts files and folders by title. If you change the sorting options to *View Items by Name*, the names of your files and folders will display. Whether you display content by title or by name, the Repository and Data Servers nodes do not display differently.

You can expand the nodes on the Configured Environments tree to view your configured environments, folders, and application files. Use the filtering commands on the toolbar to sort files alphabetically, display by name or title, or include the associated paths.

From the Configured Environments tree, you can manage content for each of your Db2 Web Query environments. This includes creating new folders or files, opening existing files, and copying files. Right-click a node, folder, or file to view the shortcut menu of options. You can also open files by double-clicking them.

You can also use the Refresh Descendants option to refresh a specific application or folder. Right-click the application or folder that you want to refresh, and then click *Refresh Descendants*.

Note:

❏ You can use the shortcut menu on any file, select *New*, and click the type of content you want to create, to create new content in the same location.

You can copy files from your desktop to the Configured Environments Tree.

From the Configured Environments node, you can access the following shortcut menu option:

❏ **Add.** Opens the Db2 Web Query Environments Properties dialog box where you can add additional environments.

From your Db2 Web Query environment, in the Environments Tree panel, you can access the following shortcut menu options:

❏ **Sign In.** Allows you to sign in to your Db2 Web Query environment. This option is only available if you are not already signed in to your Db2 Web Query environment.

❏ **Sign Out.** Allows you to sign out of your Db2 Web Query environment. This option is only available if you are already signed in to your Db2 Web Query environment.

Note: You can copy files that are not in Developer Workbench and paste them in the Configured Environments tree to use those files when creating your application.

Reference: Data Servers

This node expands to display the files that you can use to develop Db2 Web Query applications successfully.

The process of populating the application paths of the files in the Data Servers node is done on a background thread. While this background thread is running, you can navigate on the tree, open a folder, and create files. However, you will be unable to save a new file, or do anything that requires a path list, such as create a new style sheet, until the background thread completes. This section of the Configured Environments tree, which is visible, will populate.

From a data server, you can access the following shortcut menu options:

❏ **Manage Adapters.** Opens the Select Server Node dialog box from which you can configure a new adapter or edit the settings of an existing adapter.

❏ **Refresh Descendants.** Refreshes the contents of the application or folder in the Configured Environments tree to omit deleted files, and show new files and file information, such as date, time, and size.

From the Applications folder, you can access the following shortcut menu options:

❏ **New Application.** Creates a new application folder.

❏ **Refresh Descendants.** Refreshes the contents of the application or folder in the Configured Environments tree to omit deleted files, and show new files and updated file information, such as date, time, and size.

From an application folder or a file, you can access the following shortcut menu options:

❏ **New.** Creates a new folder or file. The following shortcut menu options are available when pointing to *New*:

  ❏ **Synonym.** Creates a new Master File after you have selected a configured adapter, or configure a new adapter.

  ❏ **Synonym via Metadata Canvas.** Creates a new Master File in the Metadata canvas.

  ❏ **Application.** Creates a new application folder in your current application folder.

❏ **Duplicate.** Creates a copy of the selected file in the Data Servers node.

❏ **Copy.** Copies the selected file to the clipboard where you can paste it to a different node.

❏ **Paste.** Pastes the cut or copied file.

❏ **Rename.** Changes the name of the selected folder or file.

❏ **Delete.** Deletes the selected folder or file. You will be prompted to confirm your deletion. Click *No* if you do not want to delete an item, or click *Cancel* to abort the delete operation. If you choose to abort the delete operation while deleting a list of items, Cancel will prevent all subsequent items from being deleted.

❏ **Properties.** Displays the folder properties in the Data Servers node.

❏ **Refresh Descendants.** Refreshes the contents of the application or folder in the Configured Environments tree to omit deleted files, and show new files and updated file information, such as date, time, and size.

Reference: Repository

The Repository node is storage space for data or information, and allows you to use Developer Workbench to administer and develop against a Db2 Web Query environment. The Repository node lets you manage resources and applications on remote servers, as well as on your local machine if you have performed a full installation of Developer Workbench. You can create and edit application files on all remote servers from one easily accessible interface. You can also create and administer reports from a Windows application rather than a web browser.

From a Db2 Web Query node, you can access the following shortcut menu options:

❏ **Impact Analysis.** Enables you to generate a list that identifies the procedures that access a specific Master File or field within a Master File.

❏ **New Folder.** Creates a new folder.

❏ **Mode Manager.** Allows you to edit private files, if you have the proper authorization setting.

❏ **Physical View** or **Shared View.** Enables you to toggle between the two. Shared View displays all folders (including empty folders) in the hierarchy that includes shared content. Physical View suppresses empty folders in the hierarchy that includes shared content. Physical View displays all folders that contain shared content.

❏ **Report Broker Explorer.** Opens the Report Broker Explorer.

❏ **Refresh Descendants.** Refreshes the contents of the application or folder in the Configured Environments tree to omit deleted files, and show new files and updated file information, such as date, time, and size.

From a folder or file, in the Repository node, you can access the following shortcut menu options:

❏ **Impact Analysis.** Enables you to generate a list that identifies the procedures that access a specific Master File or field within a Master File.

❏ **New.** Creates a new folder or file. The following shortcut menu options are available when pointing to *New*:

    ❏ **Procedure Using InfoAssist.** Creates a new procedure folder or file in your current application folder.

❏ **Procedure via Text Editor.** Accesses the Text Editor that you can use to create, view, and edit the source syntax for procedures, procedure components, and other types of files required by your applications. For more information, see *Editing Source Syntax* on page 361.

❏ **Report.** Creates a new report.

❏ **SQL Report.** Creates a new SQL report.

❏ **Chart.** Creates a new chart.

❏ **SQL Chart.** Creates a new SQL chart.

❏ **HTML File** Creates a new HTML page.

❏ **Visualization.** Creates a new chart, map, or grid.

❏ **URL.** Creates a new URL.

❏ **Cascading Style Sheet.** Creates a new cascading style sheet.

❏ **Db2 Web Query StyleSheet.** Creates a new Db2 Web Query StyleSheet.

❏ **Text Document.** Creates a new text document.

❏ **Schedule.** Creates a new schedule.

❏ **Distribution List.** Creates a new distribution list.

❏ **Folder.** Creates a new folder.

❏ **Publish/Unpublish.** Allows or suppresses public access to every file within the selected folder. Multiple folders can be selected at one time by pressing the Ctrl key while selecting the desired folders.

❏ **Hide/Show.** Hides or shows every file in the selected folder, to users who are permitted to access the folder. Multiple folders can be selected at one time by pressing the Ctrl key while selecting the desired folders. You can specify which users can see this content by changing the *Do not show on user's list* property in the File/Folder Properties panel.

❏ **Security.** Enables you to control user permissions, which limit what development areas a user can access. The options are:

❏ **Rules.** Opens the Security Rules dialog box, where you set up simple and advanced security rules for groups and users.

❏ **Rules on this Resource.** Opens the Rules on this Resource dialog box, where you can view and run reports on the security rules defined for the domain.

❑ **Effective Policy.** Opens the Effective Policy dialog box, where you can view the effective policy by privilege and user.

❑ **Owner.** Opens the Set Resource Owner dialog box, where you can set the portal user, set the group to which the user belongs, and publish the portal. Since ownership is set at the Domain level, this option is not available for the folders and files under a Domain.

❑ **Duplicate.** Creates a copy of the selected file in the Domains node.

❑ **Copy/Paste.** Copies or pastes a folder or file.

❑ **Rename.** Changes the name of the selected folder or file.

To ensure that all instances of the original name and references are updated, it is recommended that you use the Save As option to rename an HTML page.

❑ **Delete.** Deletes the selected folder or file. You will be prompted to confirm your deletion. Click *No* if you do not want to delete an item, or click *Cancel* to abort the delete operation. If you choose to abort the delete operation while deleting a list of items, Cancel will prevent all subsequent items from being deleted.

❑ **Report Broker Explorer.** Opens the Report Broker Explorer.

❑ **Properties.** Displays the folder properties in the Repository node.

❑ **Refresh Descendants.** Refreshes the contents of the application or folder in the Configured Environments tree to omit deleted files, and show new files and updated file information, such as date, time, and size.

Reference: Scheduling

Scheduling allows you to indicate the method and time at which the procedure should run. You can schedule a procedure using one of the following options:

❑ Email

❑ FTP

❑ Printer

❑ Managed Reporting

Reference: **Node Population**

Population is a check to see what content, if any, is contained within a node in the Configured Environments tree. Node population occurs when you select a node in the Configured Environments tree. The node icon will be different depending on whether population has occurred. If a node contains files after population a green dot appears on the node icon. If a node does not contain files after population, a red dot appears on the node icon.

If a node has been populated and contains content, deleting that content will change the green dot to a red dot. Conversely, if a node has been populated and contains no content, creating content in that node will change the red dot to a green dot.

Reference: **Master File Shortcut Commands**

The following shortcut menu options are available for a Master File:

❏ **Open.** Opens the Master File in the Metadata canvas.

❏ **Create Report.** Opens the Report Wizard, where you can create a new report.

❏ **Create Chart.** Opens the Chart Wizard, where you can create a new chart.

❏ **Open in Text Editor.** Opens the Master File in the Text Editor canvas.

❏ **Edit in Windows Associated Tool.** Opens the Master File in your Windows associated tool. Note that this only appears if you have a Windows associated tool in which to open the file.

❏ **Refresh Synonym.** Regenerates the synonym. Use this option if the underlying object has been altered.

❏ **Sample Data.** Displays sample output for the Master File.

❏ **Impact Analysis.** Enables you to generate a report that identifies the procedures that access a specific Master File or field within a Master File.

❏ **Print.** Prints the Master File.

❏ **Check.** Tests the validity of the Master File.

❏ **Edit Access File as Text.** Enables you to view and manually edit the Access File synonym.

❏ **New.** Creates a new folder or file. The following shortcut menu options are available when pointing to *New*:

  ❏ **Synonym.** Creates a new Master File after you have selected a configured adapter, or configure a new adapter.

❏ **Synonym via Metadata Canvas.** Creates a new Master File in the Metadata canvas.

❏ **Application.** Creates a new application folder in your current application folder.

❏ **Duplicate.** Creates a copy of the selected file.

❏ **Copy/Paste.** Copies the selected file to the clipboard where you can paste it to a different node.

❏ **Rename.** Changes the name of the selected file.

❏ **Delete.** Deletes the selected file. You will be prompted to confirm your deletion. Click *No* if you do not want to delete an item, or click *Cancel* to abort the delete operation. If you choose to abort the delete operation while deleting a list of items, Cancel will prevent all subsequent items from being deleted.

❏ **Properties.** Displays the folder properties in the File/Folder Properties panel.

Note: When a Master File is opened in the Text Editor canvas or the Metadata canvas, all other open options from the shortcut menu are unavailable.

Opening Files in the Configured Environments Tree

In addition to opening files through the Application menu, or from the Quick Access Toolbar, you can also open files from the development areas in the Configured Environments tree.

❏ To open a file in its native canvas, double-click the file or right-click the file, and then click *Open*.

❏ To open a file in the Text Editor canvas, right-click the file and then click *Open in Text Editor*.

❏ To open a file in a Windows Associated Tool, right-click the file and then click *Edit in Windows Associated Tool*. This option appears only if a Windows associated tool is available. When you access this functionality, the file will open in a separate dialog box using the relevant Windows tool.

Note: Files can only be opened in one area of Developer Workbench at a time to ensure that your changes are not overwritten by another opened version of the file. However, you are able to open a file in Developer Workbench and in a Windows Associated Tool at the same time. To ensure that your changes are saved properly, you should only modify one opened version of the file at a time.

Setting Up Access to Db2 Web Query Environments

As you set up access to Db2 Web Query environments, your settings are retained in a file named wfscom.xml. Db2 Web Query environment settings are typically stored in the following locations.

Windows 7:

drive:\Users*user_id*\AppData\Roaming\Information Builders\wfscom.xml

where:

user_id

> Is your Windows user ID.

Note:

❏ This file and directory might not be visible, by default. To see this directory, open the Windows Control Panel, click *Folder options*, and then click the *View* tab. Select *Show hidden files, folders, and drives*, and click *OK*.

❏ User IDs and passwords stored in wfscom.xml are encrypted to keep them confidential.

❏ Db2 Web Query environment properties apply to your current Developer Workbench configuration.

Procedure: **How to Add a Db2 Web Query Environment**

A Db2 Web Query environment consists of a web server, a Db2 Web Query Client, and a Reporting Server. Adding Db2 Web Query environments lets you create and manage multiple environments, such as development, test, and production instances.

To add a Db2 Web Query environment, do the following:

❏ Specify a web server that includes a host name and port number.

❏ Configure the HTML alias/context root to access the Db2 Web Query Client.

❏ Provide the appropriate user credentials if web server authentication is required.

The following procedure provides the information you need to add a Db2 Web Query Environment.

1. On the *Home* tab, in the *Utilities* group, click *Environments*.

   The Environments List dialog box opens. This dialog box lists all Db2 Web Query environments defined for Developer Workbench.

2. Click *Add*.

    The Db2 Web Query Environment Properties dialog box opens, as shown in the following image.
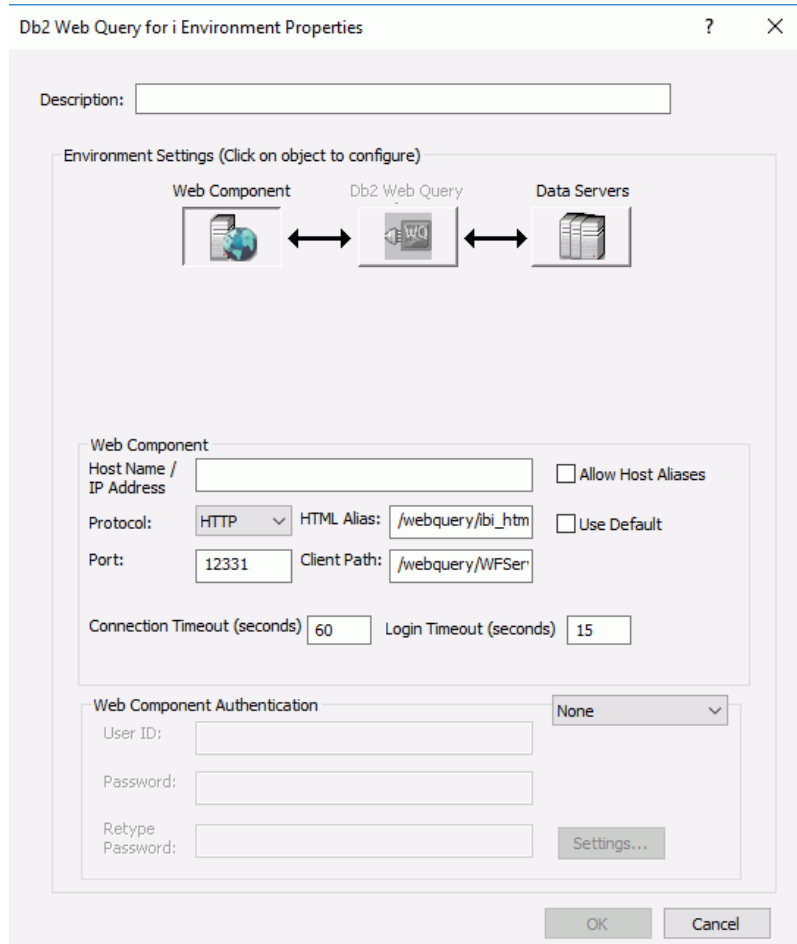
    At the top of the dialog box is the Description field, followed by the Environment Settings area. The Environment Settings area contains a series of buttons that represent components in a Db2 Web Query environment, and the required parameter fields to configure that environment. When you configure an environment, the areas below the buttons can change according to the parameters that are required.

3. In the Description field, type a description for the Db2 Web Query environment. This description appears in the Configured Environments tree.

> **Note:** The description cannot contain special characters, such as ., /, \, ?, *, and others. A message with characters not allowed appears if one of the restricted characters is detected during validation.

4. In the Web Component area, specify the web server information.

   For some environments, once you specify the web server, all other settings default. If the Db2 Web Query environment you are accessing does not use default settings, or the components require authentication, click the appropriate button in the Environment Settings area to provide the parameters. The areas that follow explain the parameters available for each component.

   > **Note:** Db2 Web Query environment properties must be supplied in a specific order. For example, if web server security is enabled, you cannot set the Db2 Web Query Client script name until you have provided valid web server credentials. Similarly, you cannot retrieve a list of Reporting Servers until you have provided a validDb2 Web Query Client Path.

   As you select a component button in the Db2 Web Query Environment Properties dialog box, Developer Workbench ensures that it has the necessary information before it displays the properties of that component in the lower part of the dialog box. If the required information is not available, you cannot proceed to the next component.

5. Click *OK*.

   The Db2 Web Query Logon - Db2 Web Query dialog box opens and prompts you for a Db2 Web Query ID and password.

   There is a check box option on the logon dialog box for remembering the ID or password. By default, it is not selected. If you select this box, your credentials are stored and encrypted in the wfscom.xml file, the local configuration file that stores information processed by the Developer Workbench communication layer.

   To clear stored credentials, open the Db2 Web Query Environment Properties dialog box and select the environment that users need to make changes. Click *Db2 Web Query Client*. Under Db2 Web Query Credentials, delete the User ID and Password information, and then clear the Supply Credentials check box.

   > **Note:** A logon dialog box can also open for connection to the web server, application server, or Reporting Server, depending on the security implemented in theDb2 Web Query environment that is being accessed.

6. Type the appropriate user name and password, and then click *Logon*.
   The Db2 Web Query Environment Properties dialog box closes.

7. In the Environments List dialog box, click *OK*.

Specifying Web Component Properties

The *Web Components* button is typically selected by default. Web Component properties specify how Developer Workbench accesses the web server. The web server must be identified before any other components.

The following properties are available:

❏ **Host Name/IP Address.** Specifies the host name or IP address where your web server is running. This field is required and has no default value.

❏ **Allow Host Aliases.** If checked, enables you to configure multiple Db2 Web Query instances that have been installed on the same host machine.

❏ **Protocol.** The protocol to use for accessing the web server (HTTP or HTTPS).

❏ **Port.** The TCP/IP port for accessing the web server. Port 8080 is the default.

❏ **HTML Alias.** Identifies the alias used to access content from the Db2 Web Query environment directory, ibi_html. In Developer Workbench, there is no need to configure an ibi_html alias. The ibi_html content is accessed through the webquery alias. The configuration of this alias is:

```
/webquery/ibi_html
```

where:

```
webquery
```

Is customizable.

```
ibi_html
```

Is constant.

❏ **Client Path.** Specifies how calls are made from Developer Workbench to the web server. By default, when you add a new Db2 Web Query environment, it is set to use the Db2 Web Query Servlet with the default webquery context path:

```
/webquery/WFServlet
```

where:

```
webquery
```

Is customizable.

```
WFServlet
```

Is constant.

If the Db2 Web Query environment uses a non-default context path, clear the *Use Default* check box and provide the correct Client Path. For example:

`/webquery/WFServlet`

If the Client Path is incorrect for the environment, you receive an error when you click the *Db2 Web Query* button at the top of the page, or when you click *OK* to exit and save your changes.

If you do not know your path, ask your Db2 Web Query Administrator or check the Db2 Web Query Administration Console of the environment to which you want to connect. The Client Path settings for the environment are located under *Utilities* and *Client Selection*.

❑ **Use Default.** Specifies that the default ibi_html alias is used. Keep this option selected unless you change the HTML Alias value.

Note:

❑ If the Client Path field is empty, and the Use Default check box is selected, there is a problem connecting to theDb2 Web Query Client. Ensure your web server is started and that you typed the correct properties on the Web Components page. If you cannot connect, contact your Db2 Web Query Administrator.

❑ If while providing a custom HTML Alias and Client Path, your connection fails because of incorrect information and the Client Path is deleted, you can clear *Use Default* to restore the values you entered. Click the *Use Default* check box to edit the HTML Alias and Client Path, and type the correct information. If a custom webquery alias or context path is used, it needs to be entered in both the HTML Alias and Client Path fields. For example:

HTML Alias:

`/webquery/ibi_html`

Client Path:

`/webquery/WFServlet`

❑ **Connection Timeout.** The default time to connect to a Db2 Web Query environment is 60 seconds. However, in some slow systems this needs to be increased to avoid connection failures when the timeout period is reached. If timeout issues are experienced, please increase this to 240 or higher. Also, increasing the timeout can address issues with slow environments or environments that have many resources (Applications/files) during development.

❏ **Login Timeout.** Specifies how long Developer Workbench should wait for the login credentials to be validated. The default time is 15 seconds. You can set this to a higher value, if necessary. You can configure this setting for each environment.

❏ **Web Component Authentication.** Specifies whether authentication is required on the web server. To set security, select *Basic* in the drop-down list, and type a web server user ID and password. If this is set to *None*, the web server allows anonymous access.

Db2 Web Query Properties

When you click the *Db2 Web Query* button, Developer Workbench makes a connection to your web server to retrieve information about the Db2 Web Query environment. Therefore, you have to first specify Web Component properties, and your web server must be running.

You are prompted to sign in to Db2 Web Query to verify your configuration. You will also be prompted to sign in to Db2 Web Query if you click *OK* to exit the dialog box.

The following properties are available:

❏ **Client Path.** Is specified in the Db2 Web Query Component properties area, and only appears in this pane for reference. This field is read-only and can only be changed in the Db2 Web Query Component properties area.

❏ **Select Language.** Specifies the language of the Db2 Web Query Client.

❏ **Supply Credentials.** Specifies whether to supply and store credentials when you connect to Db2 Web Query and access the environment.

When the correct Db2 Web Query Client Path is provided, you can specify properties for the remaining component.

Data Server Properties

You can set authentication and view available Reporting Servers by clicking the *Data Servers* button. When you select *Data Servers*, Developer Workbench connects to the Db2 Web Query Client and retrieves a list of servers from its communication configuration file (odin.cfg).

The following property is available:

❏ **Supply Credentials.** If selected, you can type a Db2 Web Query Server ID and password for the server highlighted in the list. Clicking *Set* stores the credentials with the environment properties, and the ID entered is shown next to the server in the list. The credentials are checked the next time you use a feature on that server, not when you click *Set*.

Procedure: **How to Edit an Existing Db2 Web Query Environment**

1. On the *Home* tab, in the *Utilities* group, click *Environments.*

   The Environments List dialog box opens.

2. Select the environment that you want to edit.

3. Click *Properties*.

   The Db2 Web Query Environment Properties dialog box opens.

4. Edit the Db2 Web Query environment accordingly, and then click *OK* to accept your changes.

5. In the Environments List dialog box, click *OK*.

Mode Manager

If you have the *manage private resources* permission, you can use Mode Manager to view and edit private files. To enable Mode Manager, right-click on the Repository node and click *Mode Manager*. Your view refreshes, displaying all private files in the repository. You can open and make changes to these private files as needed. When you are finished, right-click on the Repository node and click *Mode Normal*. Your view refreshes, hiding the private files in the Repository node.

Note: When a file in the repository is marked as shared, and you do not have permission to save the original file, you will only be allowed to use the *Save As* command to save the file. This will allow you to save the file under a different name, leaving the original file untouched.

Working With the Environments Detail Panel

The Environments Detail panel is an optional panel, which contains panel customization commands, the Environments Detail toolbar, the Configured Environments tree, and the Details pane. To display the Environments Detail panel, on the *Home* tab, in the *View* group, select the *Environments Detail* check box.

The Environments Detail panel is similar to the Environments Tree panel except that content is displayed in the Details pane rather than under the Configured Environments tree.

From the Environments Detail panel, you can:

❏ Customize the appearance of the panel.

❏ Filter content listed under the Db2 Web Query environments in the Configured Environments tree.

❏ Access configured Db2 Web Query environments to create new content or modify existing content.

❑ Refresh an application or folder in the Configured Environments tree, to view new files and updated file information, and omit deleted files.

When you select a folder in the Configured Environments tree, the content within that folder is shown in the Details pane.

Reference: Environments Detail Toolbar

The Environments Detail toolbar contains the buttons and menus that you need to navigate and filter the information that appears in the Configured Environments tree.

View Options

Presents options for viewing items (for example, alphabetical sorting or grouping), as described in *Environments Tree Toolbar* on page 50. Note that the available icons for the Environments Detail toolbar and the Environments Tree toolbar are the same.

Note: To sort content in the Environments Detail panel, you do not use the View Options command. Instead, you would click the columns at the top of the Details pane to sort content by name, size, type, last modified, or location.

Viewing File and Folder Properties

The File/Folder Properties panel shows overview properties of a folder or file selected in the Environments Tree panel. For example, if you highlight a folder in the Environments Tree panel, the File/Folder Properties panel shows information like the name and title of the folder and the date it was last modified. If you highlight a report file in the Environments Tree panel, the File/Folder Properties panel shows things like whether the report is enabled for Auto Link or Auto Drill.

The File/Folder Properties panel is shown in the following image.
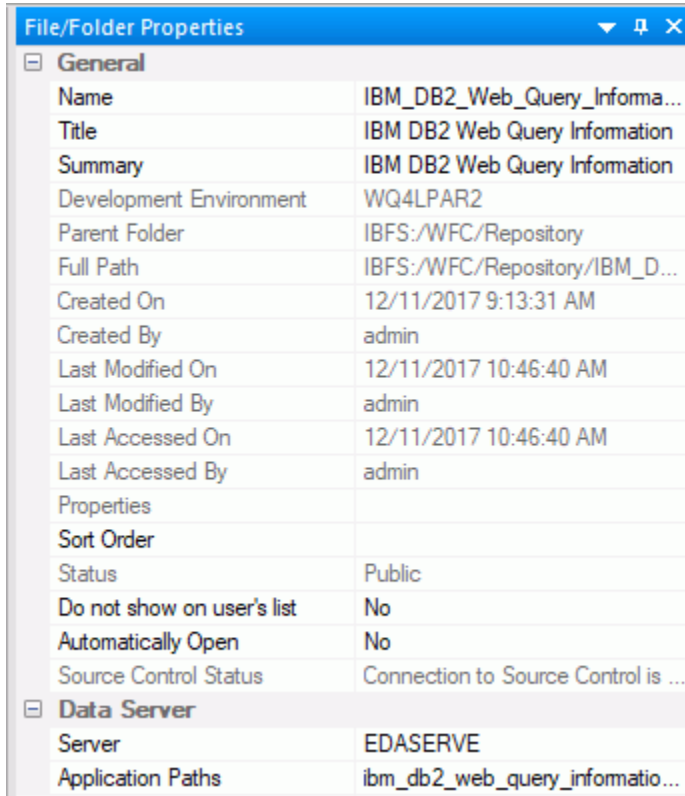
The options within this panel change according to the type of file or folder you select. You can view the properties of a file or folder in the Data Servers and repository areas of the Configured Environments tree.

Copying Property Values in the File/Folder Properties Panel

You can copy a property value from the File/Folder Properties panel. Right-click a property value and click *Copy* to copy it to the clipboard. This is particularly useful if you want to capture the full path of an open file.

Editing Properties in the File/Folder Properties Panel

The following table defines the properties that you can change for each node within the File/Folder Properties panel.

Note: The changes that you make in the File/Folder Properties panel are not reflected in the Configured Environments tree until you click outside of the panel.

Data Servers

Element	Editable Panel Properties
Node	❏ **Title.** Changes the title of the Data Server node in the Configured Environments tree. ❏ **Server Path.** Indicates the fully-qualified path to the server.
Primary Folder	❏ **Title.** Changes the title of the primary folder in the Configured Environments tree. ❏ **Server Path.** Indicates the fully-qualified path to the server.
Subfolder	❏ **Name.** Changes the name of the selected subfolder in the Configured Environments tree.
File	❏ **Name.** Changes the name of the selected file in the Configured Environments tree.

Repository

Element	Editable Panel Properties
Node	❏ **Title.** Changes the title of the Repository node in the Configured Environments tree. ❏ **Server Path.** Indicates the fully-qualified path to the server.

IBM

Element	Editable Panel Properties
Folder	❑ **Name.** Changes the name of the folder in the Configured Environments tree.
	❑ **Title.** Changes the title of the folder in the Configured Environments tree.
	❑ **Summary.** Adds summary information to the folder properties.
	❑ **Sort Order.** Specifies the order in which to list the folder or item in the tree.
	❑ **Do not show on user's list.** This value is set to No, by default. If Yes is selected, this folder will display on the user's list.
	❑ **Automatically Open.** This value is set to No, by default. If Yes is selected, this folder will automatically be open in the next session of Developer Workbench.
	❑ **Server.** This value defaults to the server that is currently in use. You can use the drop-down list to affiliate a different server with the folder.
	❑ **Application Paths.** This value defaults to the application path indicated for the selected folder. You can modify this value as needed.

Element	Editable Panel Properties
File	❑ **Name.** Changes the name of the file in the Configured Environments tree. ❑ **Title.** Changes the title of the file in the Configured Environments tree. ❑ **Summary.** Adds summary information to the folder properties. ❑ **Sort Order.** Specifies the order in which to list the file in the tree. ❑ **Do not show on user's list.** This value is set to No by default. If Yes is selected, this file will display on the user's list. ❑ **Prompt for parameters.** This value is set to Yes, by default. If No is selected, you will not be prompted for parameters. ❑ **Only run as deferred report.** This value is set to No, by default. If you select Yes, the report or procedure will run only as a deferred report. ❑ **Use Title for Deferred Report Description.** This value is set to No, by default. If you select Yes, the title is used instead of the name, as the description for a deferred report. ❑ **Schedule only.** This value is set to No, by default. If Yes is selected, you will only be able to schedule the report or procedure. ❑ **Enable Auto Linking.** This value is set to No, by default. If Yes is selected, Auto Linking enables you to develop and maintain numerous referenceable reports and charts that can be linked together. ❑ **Auto Link Target.** Indicates whether the file is set as a target for an Auto Link drill down. This value is set to No, by default.

Element	Editable Panel Properties
File	❏ **Enable Auto Drill.** This value is set to No, by default. If Yes is selected, Auto Drill allows you to navigate through different levels within the dimension hierarchy of your data source. ❏ **Server.** This value defaults to the server that is currently in use. You can use the drop-down list to affiliate a different server with the file. ❏ **Application Paths.** This value defaults to the application path indicated for the selected file. You can modify this value as needed.

Application Paths Dialog Box

The Applications Paths of a Db2 Web Query folder display in the File/Folder Properties panel when you select the folder.. You can add, remove, or reorder the associated application paths in the Application Paths dialog box.

Open the Application Paths dialog box by clicking the ellipsis button next to the Associated Paths information. The Application Paths dialog box displays a list of applications that you can associate with your project and a list of projects that are currently associated with your project.

Adding or Removing Applications

To add an application to the project, double-click the name of the application in the Available list. Click *OK* to view your changes in the File/Folder Properties panel.

To remove an application from the project, double-click the name of the application in Selected list. Click *OK* to view your changes in the File/Folder Properties panel.

Reordering Applications

To reorder an application in the Available list, click the name of the Application and then click the up or down arrows. Click *OK* to view your changes in the File/Folder Properties panel.

Editing Areas of Functionality Using the View Tabs

You can use the view tabs to edit different areas of functionality for the same document. These tabs show information that would be inaccessible from any other view tab.

Designing the Layout of a Page Using the Design View Tab

The Design View tab is accessible when you are creating an HTML page. From here, you can design the layout of your page by adding controls and components to the canvas. This is tab is selected by default when you are creating an HTML page.

Creating and Modifying Parameter Values Using the Parameters View Tab

The Parameters View tab is accessible when you are creating an HTML page. From here, you can create and modify parameter values, input controls, and customize parameter conditions. You can also bind parameters to controls and chain controls to one another.

The Parameters tab consists of the following components:

❏ Input control objects.

You may select the input control object to view and edit the Properties and settings of the control.

 ❏ Creating an input control from the Design view prompts you to create a bound parameter on the Parameters tab.

 ❏ Editing an input control, which is inserted when setting input controls for new parameters.

❏ Add new parameters.

Right-click anywhere on the Parameters tab to add a new parameter.

Note: Manually adding a parameter creates an unbound parameter.

❏ Refresh unresolved parameters.

All parameters on the parameters tab are parsed every two minutes to check if any are unresolved. If there are any unresolved parameters, their surrounding polygon is colored red. If you want to check for unresolved parameters on demand, right-click and select *Refresh unresolved*.

❏ Binding controls and parameters.

Input controls and parameters can be bound and unbound from the Parameters tab.

You may bind a parameter to an input control, or bind an input control to a parameter.

 ❏ Binding a parameter to a control makes it an incoming parameter that will populate the control. Drag a parameter object to a control object on the Parameters tab.

 ❏ Binding a control to a parameter will populate the parameter. Drag a control object to a parameter object on the Parameters tab.

❏ Chain one control to another.

Chaining will populate controls based on the selected value from the prior control in the chain. You can chain static and dynamic controls, link or unlink parts of a chain, and create conditions on links in a chain. Chains are represented by lines connecting control objects on the Parameters tab. By clicking the arrow head in a link of a chain, the Properties and settings dialog box enables you to modify and set properties and conditions of the chain.

Note: Chaining is applicable only for controls, not parameters.

Creating JavaScript and Cascading Style Sheets Using the JavaScript and CSS View Tabs

The Embedded JavaScript and Embedded CSS View tabs are accessible when you are creating an HTML page. From here, you can create JavaScript code and cascading style sheets (CSS) for use in your HTML page. You can reference existing cascading style sheet files and JavaScript files by typing the URL in the URL/Find File area of the Settings panel. You can also search for these files by using the CSS or JavaScript commands.

Embedded code is used only in the host HTML file. External code is available to be used by multiple files.

Procedure: **How to Customize Titles Using jQuery Tooltips**

The following procedure describes how to customize titles, using jQuery and Internal CSS.

1. Create an HTML page.

2. Add a component or control to the page.

3. Navigate to the Properties panel.

4. Type a value for the Title property, under the Core attributes section.

5. Click the canvas to select the *DOCUMENT* object.

6. On the Properties panel, click the lightning bolt icon on the toolbar to show the events.

7. For the Load event, click the column on the right to display the ellipsis button, as shown in the following image.
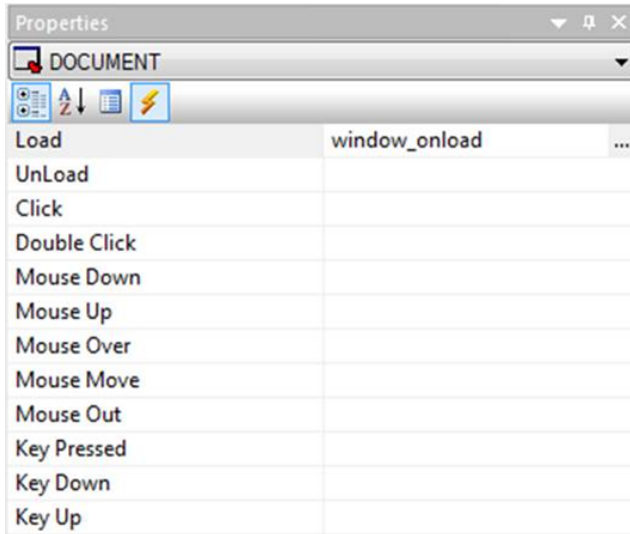
8. Click the ellipsis button, to navigate to the Embedded JavaScript/CSS view tab (Release 8.1.05M or earlier) or the Embedded JavaScript view tab (Release 8.2.01 or higher) of the HTML canvas.

9. After the comment lines, add a new line, and type the following code, as shown in the following image.

```
$(document).tooltip();
```

```
//Begin function window_onload
function window_onload() {

    UpdateData();

    // TODO: Add your event handler code here
    //add onInitialUpdate() function to make changes before initial run of the reports
    $(document).tooltip();
}
//End function window_onload
```

10. From the Embedded JavaScript/CSS view tab (Version 2.2.0 or earlier), go to the Settings panel, and select the *Internal CSS* radio button, to activate the internal CSS editor. For Version 2.2.1 or higher, navigate to the Embedded CSS tab, as shown in the following image.
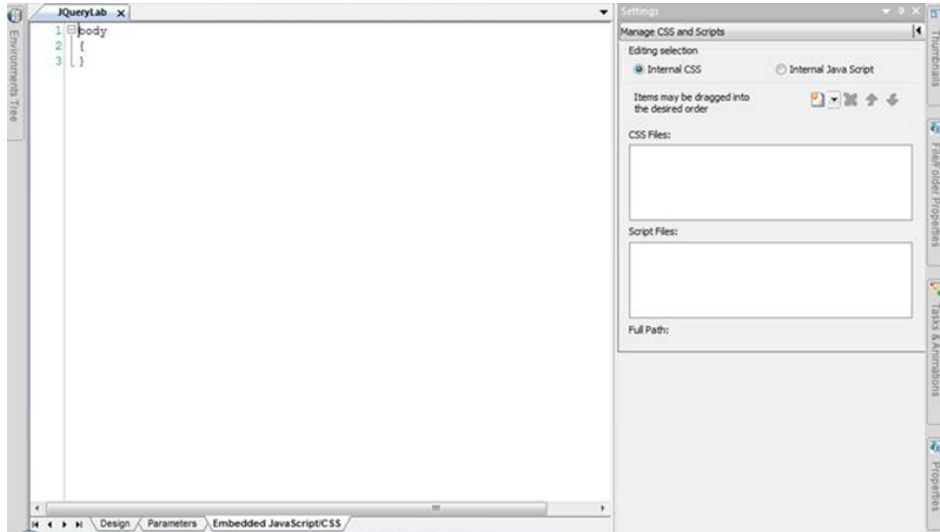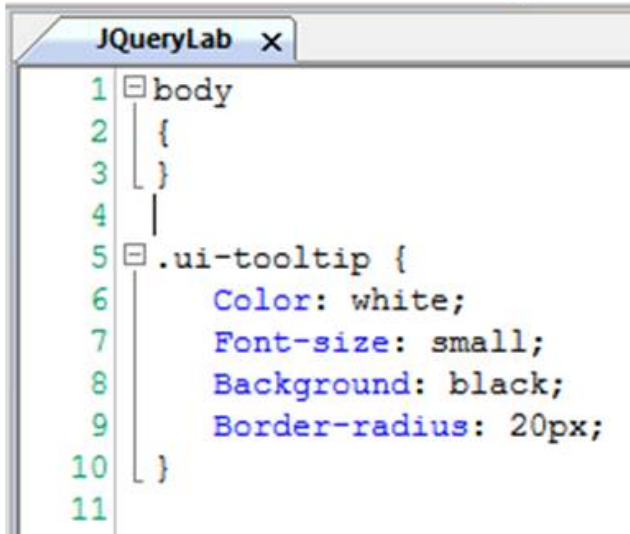
11. Enter a new line after the body class and type the following as the new class name.

    `.ui-tooltip`

12. Add the following CSS Styles, as shown in the following image.

    ❏ Color: white;

    ❏ Font-size: small;

    ❏ Background: black;

❏ Border-radius: 20px;

Note: You can add any styling that you want.

13. Save and run the HTML page. You can hover the mouse over objects to show the styled tooltips.

Customizing the Panels

Similar to the ribbon tabs, Developer Workbench features open panels that provide additional options to assist in the development of your application. Not all panels open at the same time, nor do they provide the same options. When you launch Developer Workbench for the first time, only the Environments Tree panel appears docked to the edge of the canvas. The File/ Folder Properties panel also appears, but is auto hidden.

By default, all panels, except the Environments Tree panel, are auto hidden when they first open. You can use the panel customization options, which are located in the upper-right corner of the panel, to change the appearance and location of the panels around the canvas.

Procedure: **How to Rearrange Panels on the Canvas**

To rearrange a panel on your canvas using the panel properties menu, follow these steps:

1. Click the *Window Position* button ▾.

   The panel properties menu opens.

2. Select an option from the panel properties menu.

The options available are:

❏ **Floating.** Undocks the panel and allows you to move it freely around the canvas.

❏ **Docking.** Docks the panel onto the edge of your canvas where it last appeared.

❏ **Tabbed Document.** Opens the panel in the canvas area, as a tab. This option is unavailable in Developer Workbench.

❏ **Auto Hide.** Hides the panel as a tab on the edge of your canvas. When you pause over the tab, the panel reopens.

❏ **Hide.** Closes the panel.

Note: You can also open the panel properties menu if you right-click the top of the panel.

Procedure: **How to Pin and Unpin Panels to the Canvas**

You can pin a panel to the canvas to keep it open while you develop your application. When you unpin the panel, it becomes auto hidden again, and appears as a tab on the edge of the canvas. When you hover the mouse over this tab, the panel reopens.

❏ To pin a panel to your canvas, click the *Auto Hide* horizontal pin button .

❏ To unpin a panel to your canvas, click the *Auto Hide* vertical pin button .

Procedure: **How to Close and Reopen Panels**

❏ To close a panel, click *Close* in the upper-right corner of the panel.

❏ To reopen the panel, go to the *Home* tab, and in the *View* group, and select the appropriate check box.

❏ To reopen a panel in the HTML canvas, go the *Utilities* tab, and in the *View* group, select the appropriate check box.

Using Handles

You can dock floating panels within the canvas to ease accessibility. When you drag a panel across the Developer Workbench canvas, a set of handles appears. These handles guide the placement of the panel, and automatically dock the panel to the edge of the canvas upon which you rest the mouse.

Procedure: How to Dock Panels Around the Canvas Using Handles

1. From the panel customization toolbar, click the arrow, and then click *Floating*.

2. Drag the panel across the canvas and rest the mouse on a handle.

   When you rest the mouse over a handle, a shaded area appears. This shaded area provides a preview of where the panel will dock if you release the mouse, as shown in the following image.
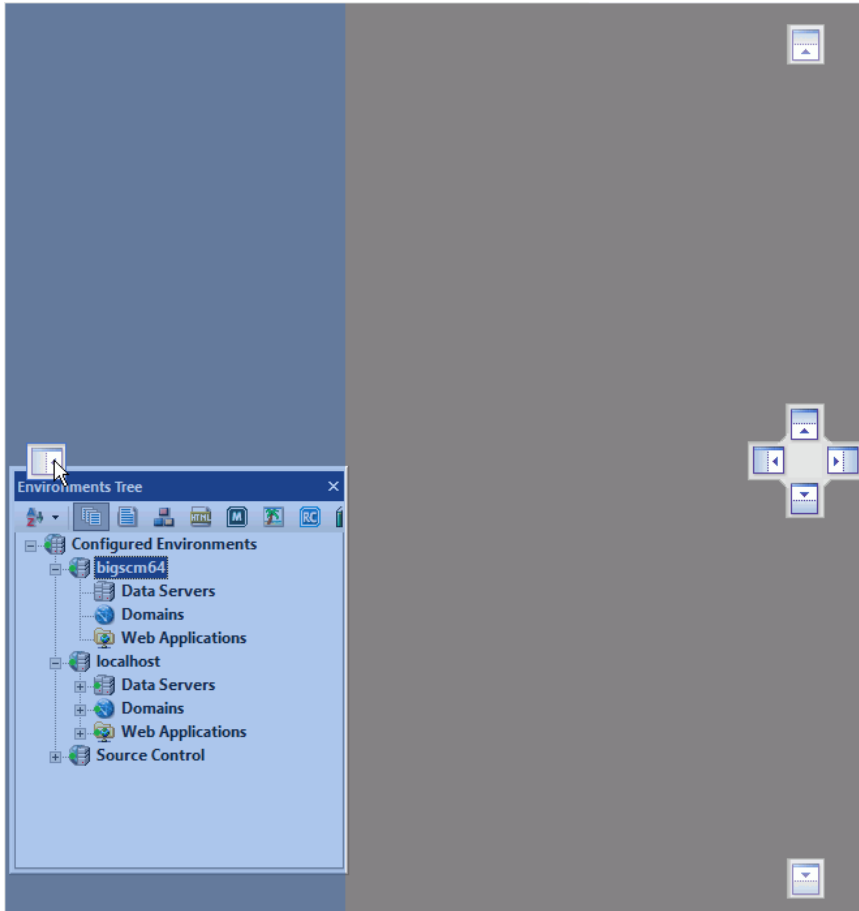
3. Release the mouse when you are satisfied with the previewed location of the panel.

   The panel automatically docks to the edge of the canvas.

Additional panels open as you develop applications throughout the product. To conserve space around the canvas, you can combine these panels. When you mouse over a docked panel, a tabbed handle option appears.

This handle option combines the panels into a single container with tabs. You can switch between the panels by clicking the appropriate tab at the bottom of the container. An example of the Environments Tree tab and File/Folder Properties tab, in a combined panel container, are shown in the following image.

Chapter 3

Creating Metadata

Defined simply, metadata is *data about data*. Metadata describes the characteristics of your data sources to Developer Workbench and gives it the information it needs to access those data sources. This insulates you from the need to know how to optimally access the data, so that you can concentrate on using the data for reporting, charting, visualizations, and applications. Examples of metadata components are data type, data location, and data content and information about how to connect to it. Metadata can be generated whenever data is created, acquired, added to, deleted from, or updated in any data source within an enterprise.

In this chapter:

❏ Understanding Data Terminology

❏ Accessing Data

❏ Generating a Synonym

❏ Changing Views in the Metadata Canvas

❏ Viewing and Editing Synonym Attributes

❏ Viewing Data Profiling Characteristics

❏ Enhancing a Master File with Expressions

❏ Describing Group Fields and Repeating Fields in a Synonym

❏ Parameterizing a Synonym with Variables

❏ Adding Indexes to a Synonym

❏ Creating Cluster Joins

❏ Defining a Business View of a Master File

❏ Applying Database Administrator (DBA) Security

❏ Implementing DBA Security Using Db2 Web Query Language

❏ Analyzing Procedures Using Impact Analysis

Understanding Data Terminology

The smallest meaningful element of data in a data source is referred to as a column (for relational data sources) or a field (for non-relational data sources). Column and field are used interchangeably throughout this content. Every column or field has several characteristics, such as the data type (character, date, integer, or some other type) and length or scale. Field characteristics can also include output display options, such as comma inclusion, currency symbol, date display, and so on. For consistency, all fields should be included in the metadata, so all users get the same view of the data. However, you can create a logical view of the data in which only a subset of the fields are available.

Certain fields in a data source may have a one-to-one relationship with each other and describe a group of related characteristics. For example, each unique ID number represents only one employee. These fields can be grouped together into a segment or, in relational terms, a table. You can relate segments to each other by creating a multi-segment data source or by joining segments together. Segments are the building blocks of larger data structures.

Fields can be categorized as measures (facts) or dimensions based on how they will be used in a request.

A measure or fact is a numeric value, such as Gross Profit or Cost of Goods Sold, that you may want to aggregate. All numeric values that can logically be summed are measures. Numeric fields that cannot be summed, such as product number and order ID, are not treated as measures. Instead, they may be used in the same way as dimension fields to analyze measures. It is up to you to understand your data and determine whether each numeric field should be summed. Related measures can be organized into measure groups. For example, Gross Profit and Cost of Goods Sold can be part of a Sales measure group.

A dimension is a way to categorize data. You can use a dimension to analyze and compare measures. Generally, any field that is not a measure, usually an alphanumeric field such as product, is a dimension. Dimensions can be organized into hierarchies to define the relationships between the fields in the hierarchies. For example, a Geography hierarchy can contain the Continent, Country, State, and City dimensions. You can also define dimension fields that are not part of a dimension hierarchy.

An attribute is a collection of related objects in a dimension. For example, in the Model dimension, there are attributes for Brand, Brand Type, and Product Name. Attributes are specific to a dimension and are typically more specific than general in nature.

Accessing Data

When your Developer Workbench application accesses data, Developer Workbench needs to know how to interpret the data that it finds. A Master File, which is the primary component of metadata, is a map of the segments in the data source and all of the fields in each segment. The Master File also identifies the name of the data source and the type of data source it is.

For some types of data sources, the Master File is supplemented by an Access 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. You require one Master File and, for some data sources, one Access File to describe a data source. These metadata files (Master File and Access File) are referred to as a synonym.

Synonyms are stored separately, apart from the associated data source. Your Developer Workbench application uses a synonym to interpret a data source as follows:

1. It identifies the data sources available and their respective fields.

2. It examines the security rules, if Information Builders data source security (DBA) has been specified for the data source, and ensures that user access is allowed based on any DBA security specified.

3. It identifies the Access File for the data source, if that data source requires an Access File.

The data source content is interpreted based on the information in the Master File and, if applicable, the Access File.

Working With Data Adapters

With the appropriate Db2 Web Query Data Adapters, you can access data from many different sources, including the following:

❏ Relational data sources, such as Db2 and MySQL.

❏ Sequential data sources, both fixed-format and delimited format.

❏ ERP data sources, such as JD Edwards.

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. The adapter manages the communication between the data interface and the data source and returns either answer sets or messages to the requestor. 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.

Generating a Synonym

A synonym defines a unique logical name (also known as an alias) for each web services operation. Synonyms are useful because they insulate client applications from changes to the location and identity of a request. You can move or rename a request without modifying the client applications that use it. You need to make only one change, redefining the request synonym on the server. Synonyms provide support for the extended metadata features of the server, such as virtual fields and security mechanisms. Creating a synonym generates a Master File and an Access File.

Often, a synonym already exists on the server, and reporting can begin at once. However, if the synonym you require does not exist on the server, an authorized server or application administrator can create it directly using the Metadata canvas in Developer Workbench.

To begin creating a synonym, you use the Data Source Definition Wizard, which guides you through the steps that you need to follow to open a recently used synonym, create a new synonym, or edit an existing synonym. You use the Metadata canvas, and the available tab and panels, to develop your synonym. This enables you to explore DBMS catalogs and select the objects for which you wish to create synonyms. The Metadata canvas prompts for the information it needs to create a synonym for a particular data source and stores the generated synonym on the server.

In order to generate a synonym, 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. When you begin to create a synonym, Developer Workbench opens the Select adapter to configure or Select connection to create synonym window. The option to create the synonym becomes available only after the adapter is successfully configured.

Once you have generated a synonym, you can report against the synonym. 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 the Metadata canvas.

When you use the Metadata canvas, there is no need to know the data description language. All viewable and editable attributes of the synonym components display. If you make changes to the generated synonym, the Metadata canvas validates your entries and displays messages if they violate the underlying syntax of the data description language.

Enhancing a Synonym

The following are some of the attributes you might want to add to the synonym to enhance your data access and reporting capabilities. You can:

❏ Apply security rules for fields and values to ensure that user access is based on Information Builders data source security (DBA) specifications.

❏ Add virtual columns (DEFINE fields) and columns for aggregated values (COMPUTE fields).

❏ 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 alphanumeric field or the format of a date field).

❏ Create a cluster join view by linking available synonyms to create a multisegment (multitable) file for reporting.

❏ 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 and child hierarchies for cube data sources.

Procedure: **How to Access the Metadata Canvas**

In Developer Workbench, you create new synonyms and edit existing synonyms using the Metadata canvas. You can access the Metadata canvas through the Data command, in the *Content* group, on the *Home* tab.

This launches the Data Source Definition Wizard, where you can open a recently used Master File, create a new Master File, or edit an existing Master File. The Data Source Definition Wizard is shown in the following image.
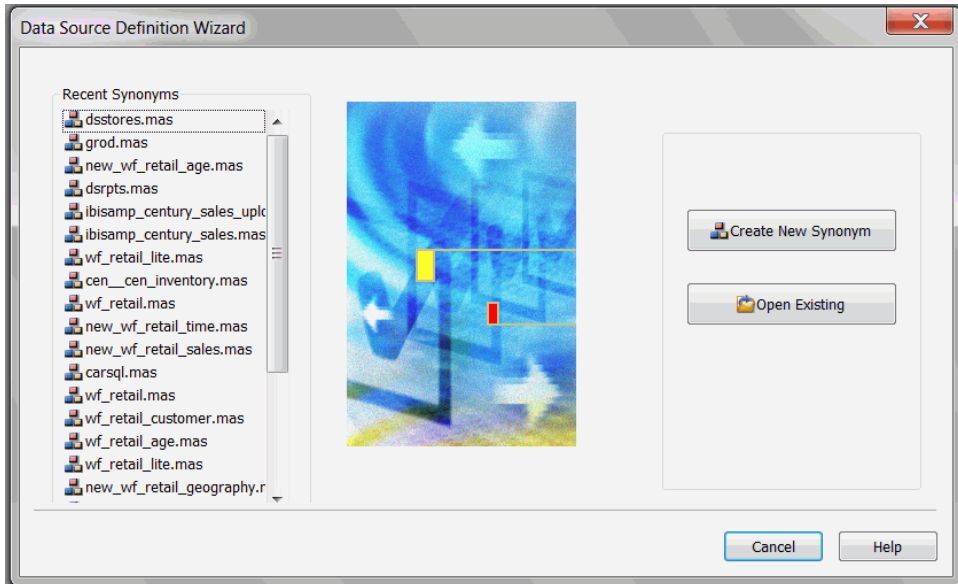
❑ Click *Create New Synonym*, which opens the Select Application pane of the Data Source Wizard. Select a location to create your synonym and click *Finish* to open the Metadata canvas.

❑ Click *Open Existing*, which opens the Select Data Source pane of the Data Source Wizard. Select a data source to edit and click *Finish* to open the Metadata canvas.

The Metadata canvas opens, as shown in the following image.
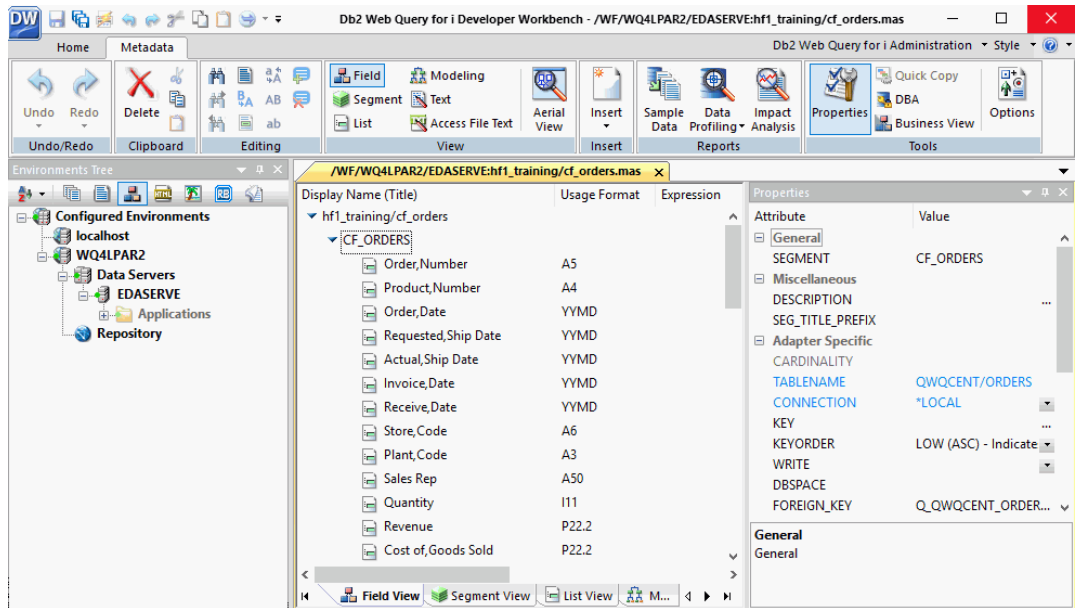
Note: The examples in this document use a multi-fact structure containing multiple Fact tables that have corresponding Dimensions described in the Master File.

Building a Synonym Using the Metadata Tab

The Metadata canvas displays the Metadata tab, which contains all the commands necessary to build your synonym.

The Metadata tab contains groups and commands that provide quick access to commonly performed functions while working in the Metadata canvas. You can use the Metadata tab to undo or redo actions, manipulate selections of data, find and edit text, switch to different view tabs, insert notes, run different reports, and switch between different view modes. The behavior of the command is determined by the selected object. Therefore, certain tab components may be inactive. The Metadata tab is shown in the following image.
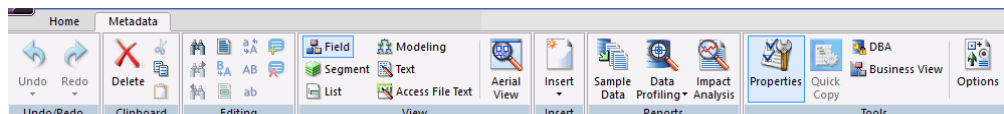
Restoring Actions Using the Undo/Redo Group

You can undo and redo actions. The Undo/Redo group is shown in the following image.

The commands are:

Undo

Undoes the last action.

Redo

Restores a previously undone action.

Deleting, Cutting, Copying, and Pasting Using the Clipboard Group

You can delete, cut, copy, and paste sections from the canvas. The Clipboard group is shown in the following image.

The commands are:

Delete

Deletes your selection.

Cut

Cuts your selection.

Copy

Copies your selection.

Paste

Pastes from your clipboard to the designated area.

Searching and Revising Text Using the Editing Group

You can find specific text, select the entire canvas, replace specific text, move to a specific point in the canvas, apply case formatting to text, and add comments. The Editing group is shown in the following image.
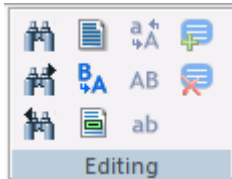
The commands are:

Find

Finds specified text.

Find Next

Finds the next instance of specified text.

Find Previous

Finds the previous instance of specified text.

Select All

Selects the entire canvas.

Replace

Replaces the specified text with different text.

Go To

Moves your view of the canvas to the specified area.

Invert Case

Inverts the case of the selected text.

Upper Case

Upper case the selected text.

Lower Case

Lower case the selected text.

Comment Selection

Comment the selected text.

Uncomment Selection

Uncomment the selected text.

Switching Between Views and Aerial View Using the View Group

You can switch between the different views and switch Aerial view on or off. The View group is shown in the following image.
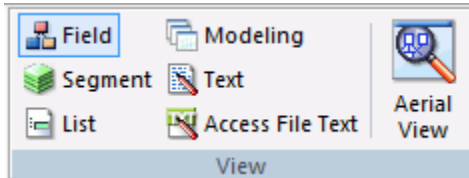
The commands are:

Field

Switches to the Field View. The Field View shows a list of objects on the left, with the attributes and values of the selected item on the right.

Segment

Switches to the Segment View. The Segment View shows the segments that the synonym contains. The Segment View shows a list of objects on the left, with the attributes and values of the selected item on the right.

List

Switches to the List View. The List View shows a list of objects on the left, with the attributes and values of the selected item on the right.

When you right-click a column heading in the List View, a shortcut menu provides options to sort the display based on ascending or descending column values. The display can be sorted by any column.

Modeling

Switches to the Modeling View.

Text

Switches to the Text View tab.

You can print the Master File code by selecting *Print* from the Application menu, performing search operations, and making changes, if necessary.

Access File Text

Switches to the Access File Text View tab.

The Access File Text View shows the description of the Access File for a synonym, which is used to access the database.

Note: You can print the Access File code by selecting *Print* from the Application menu, performing search operations, and making changes, if necessary.

Aerial View

Switches between showing and hiding Aerial view. This view is only available when in the Modeling View tab.

Inserting Nodes Using the Insert Group

You can insert a node. The Insert group is shown in the following image.

The commands are:

Insert

Enables you to insert one of the following:

Reference to Existing Synonym as Child

The reference is a pointer to the source synonym using the synonym as a child. If you subsequently make changes to the source synonym, reopening the current synonym will reflect those changes.

Reference to Existing Synonym as Root

The reference is a pointer to the source synonym using the synonym as a root. If you subsequently make changes to the source synonym, reopening the current synonym will reflect those changes.

Copy of Existing Synonym

Adds a static copy of an existing synonym to the current synonym. If you subsequently make changes to the source synonym, the current synonym will not reflect those changes.

Segment via Metadata Import

Enables you to create and add a new synonym to the current synonym using the Create Synonym dialog box.

Segment Manually

Adds a synonym that must be coded manually.

DBA

Adds DBA security to the segment.

Field

Adds a general column to the segment.

Define

Adds a virtual or defined column to the segment.

Filter

Adds a filter to the segment.

Compute

Adds a calculated value to the file.

Sort Object

Adds a Sort Objects folder and a sort object. This option is available only if *Support extended options* is selected.

Variable

Adds a Variables folder and a variable to the segment.

Style

Adds a Styles folder and a style object. This option is available only if *Support extended options* is selected.

Subquery

Adds a subquery to the fields list.

Running a Sample Data, Data Profiling, or Impact Analysis Report Using the Reports Group

You can run a Sample Data report, Data Profiling report, or Impact Analysis report. The Reports group is shown in the following image.
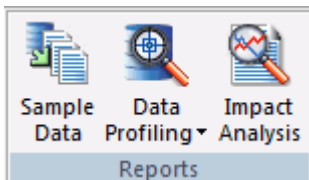
The commands are:

Sample Data

Enables you to view and refresh sample data for the selected segment or synonym.

Data Profiling

Runs a Data Profiling report, which provides the data characteristics for synonym columns. The Statistics, Count, and Key Analysis options are available.

Impact Analysis

Runs an Impact Analysis report for the particular column in the workspace. An Impact Analysis report identifies the procedures that access a Master File or column within a Master File.

Showing Properties, Data Flow, and View Panels Using the Tools Group

You can switch between showing or hiding the Properties, DBA, and Business View panels. You can also recreate the default Business View structure and edit the user options. The Tools group is shown in the following image.
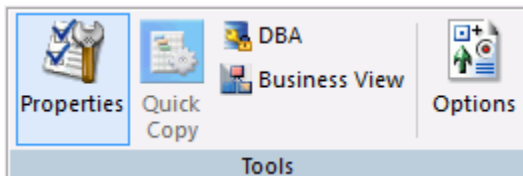
The commands are:

Properties

Switches between showing and hiding the Properties panel.

Quick Copy

Generates a simple Data Flow without using the Data Flow designer. For the selected tables, all rows and columns are copied to new tables.

Although no transformations or where conditions are generated, the Data Flow can still be opened as such and additional criteria can be added.

DBA

Switches between showing and hiding the DBA panel.

Generating a Synonym

Business View

Switches between showing and hiding the Business View panel. Enables you to create a Business View and a custom Master File that can use selected columns from the original synonym. In addition, you can customize field names, titles, and descriptions.

Options

Allows you to edit the user options. When you click *Options* from the Tools Group, the Options dialog box appears. It enables you to set preferences and customize the look of the Metadata canvas.

Setting Preferences for the Metadata Canvas Using the Options Dialog Box

Click the *Options* button in the *Tools* group to open the Options dialog box.

Reference: **Synonym Editor Options Settings**

When you click *Options* from the Tools group, the options dialog box appears. It enables you to set preferences and customize the look of the Metadata Canvas.

The Synonym Editor settings page has the following fields and options:

Use application directory name with synonym

If this check box is selected, an application directory name is used when you select a synonym name for both referencing an existing synonym and a transformation with db_lookup.

Undo/Redo Limit

Specifies the maximum number of undo and redo operations allowed in the Synonym Editor.

Support extended options

If this check box is selected, both the Sort objects and Styles folders appear when you edit a synonym.

Enable insert copy of existing synonym

If this check box is selected, this option will be available in the shortcut menu when a synonym or segment name is selected.

Insert child segment with snowflakes

Inserts a reference to the selected table and any tables that the base synonym references through foreign keys.

Automatically arrange segments/folders in Modeling View

If you select or drag multiple or individual tables while in Modeling View, they will be automatically arranged for you.

Show parent segments in Join Editor

If this check box is selected, it controls whether or not columns in parent segments are displayed in the Join Editor for a cluster join or a synonym that references or includes, other synonyms.

Automatically detect entry segment

This information is not yet available.

Default Join Type

Sets the default Join type.

❏ **Unique.** This join 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. This value is the default.

❏ **Multiple.** This join 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.

Auto Calculate Missing

Allows you to adjust the MISSING attribute.

❏ **Prompt.** Prompts before adjusting the MISSING attribute.

❏ **ON.** Adjusts the MISSING attribute without prompting.

❏ **OFF.** Does not adjust the MISSING attribute.

Modeling View Line Colors

Allows you to set colors for connector lines in the Modeling View.

Default. Changes the color lines in the Modeling View.

Highlighted. Changes the color of the highlighted lines in the Modeling View.

No keys. Changes the color of lines with no keys in the modeling view.

Reset Colors

Restores the default colors.

Reference: **Synonym Editor Format Options Settings**

The Format pane is available from the Options dialog box. To open the Options dialog box, select *Options* from the Tools group. Expand the *General* node, and select *Format*.

The Fonts section has the following fields and options:

Fonts Category

Allows you to set the font for text in the process flow workspace, reports, text views, and log views.

Font

Launches a dialog box for specifying font settings.

Reset Font to Defaults

Restores the default fonts.

Reference: **Synonym Editor Column Management Settings**

The Column Management pane is available from the Options dialog box. To open the Options dialog box, select *Options* from the Tools group. Expand the *General* node, and select *Column Management*.

Column Management user preferences enable you to choose which columns to display on the grids for transformations, column selection, joins, and sorts. You can also set the column display order. The settings apply to all column-related dialog boxes.

The Column Management pane has the following fields and options:

Customize column display

Lists the column-related dialog boxes that can be customized. Expanding a folder will display check boxes that can be used to add columns to each dialog box.

Reset to default

Restores the default values.

Column name display strategy

Controls the information that appears in trees and grids. The available options are Name, Title, Description, and Alias. If no Title, Description, or Alias exists, the display will default to the Name.

Note: It is recommended that you use Title when working with Business Views.

Function display strategy

Controls how functions are identified on the functions tab in the calculators. By default, the function syntax is displayed. It can be changed to a short description, which displays a brief description of what the function does instead.

Use segment to qualify field reference

Indicates whether to qualify field names with the segment name in which they reside for duplicate field names only or always.

Expand any of the available Customize column display options to see default settings. Not all columns are on every grid. The following columns can be added:

Alias

Assigns an alternative name for a column or the real column name for a DBMS synonym.

Application

Indicates the application where the synonym resides.

Belongs To Segment

Shows the parent segment in the base synonym.

Connection

Indicates the adapter connection name used.

Data Origin

Indicates the date the synonym was created.

Date Modified

Indicates when the synonym was last modified.

Description

Is a description or comments about the column.

Expression

Is the expression for the column.

Extension

Indicates the suffix (data source type) of the synonym.

Field Type

Indicates that a column is an index (I) or is read-only.

Format

Is the type and length of a column data as stored.

Function

Indicates the name of the function.

Geographic Role

Describes the geographic role of the column.

Has Foreign Keys

Indicates that the synonym includes foreign keys.

Index

Indicates an index column.

Join Condition

Indicates the condition for the join.

Join Parent

Indicates the parent of the join.

Join Status

Out of the candidate list, indicates the strongest candidate or candidates for insertion.

Join Strategy

Indicates the strategy for the join.

Join To

Indicates whether you can join to the column.

Keys

Indicates the keys in the synonym.

Length

Is the column length.

Name

Indicates the column name.

Nulls

Indicates whether or not the column can contain null data.

3. Creating Metadata

Number of Segments

Indicates the number of segments in the synonym.

Order

Indicates the order of the column in the segment.

Parent(s)

Indicates the parent or parents on the active Metadata canvas.

Prefix

Indicates a prefix for the column.

Primary Key Tables

Indicates the primary key for the synonym.

Property

Indicates whether the column is an attribute or a measure.

Real Table Name

Indicates the actual name of the table or the physical file name.

Reference

Indicate a reference for the column to an index column.

Scale

Is the maximum number of digits to the right of the decimal.

SCD Type

Used for processing slowly changing dimensions.

Segment

Indicates the segment name.

Sequence Number in Key

Indicates the sequence number of the field within the key.

Size

Indicates the size of the synonym.

Source

Indicates the source of the synonym.

SQL Conversion Notes

Indicates how SQL is converted.

Table

Is the synonym that contains the column.

Title

Supplies a title to replace the column name normally used in reports.

Type

Is the type of object in an application directory.

Reference: Synonym Editor Run Options Settings

Run Options user preferences include determining the number of rows and columns to retrieve, as well as the default format when sampling data.

The Run Options settings pane has the following fields and options:

Maximum number of rows for test reports

Sets the number of rows retrieved to produce sample data when testing transformations or SQL. The default is 50.

Maximum number of columns for test reports

Sets the number of columns retrieved to produce sample data when testing transformations or SQL. The default is to retrieve all columns (with a highest value setting of 999999).

Test reports default format

Sets the format of reports for retrieving sample data when testing transformations or SQL. The default is Default. The following report formats are available:

❏ **Default.** Formats numeric and date columns based on edit options in the synonym.

❏ **HTML.** Produces the report in HTML format.

❏ **HTML - plain text.** Produces the report in plain text format.

❏ **active report.** Produces an HTML active report designed for offline analysis.

❏ **PDF.** Produces the report in PDF format.

❏ **Unformatted.** Does not apply formatting to numeric and date columns.

Stop after DBMS error

Sets the number of DBMS-related errors allowed before the server stops running the procedure.

Changing Views in the Metadata Canvas

The Metadata canvas displays Field View, Segment View, Modeling View, Text View, and Access File Text View tabs. You can switch to different views in the Metadata canvas to edit different aspects of your data source more efficiently.

Displaying Fields in a Synonym Using the Field View Tab

The Field View tab is accessible when you are creating or editing a synonym. From this tab, you can see a table with all available fields. As shown in the following image, the Field View tab shows a hierarchy of segments and columns on the left, with the attributes and values of the selected item in the Properties window, which displays, by default, on the right side of the canvas. This is the default view tab when you are creating or editing a synonym. The Properties panel can be resized, positioned in different areas, or hidden.

Note: The attributes available depend on the type of synonym.

The following image is an example of a multi-fact structure with a key column selected.
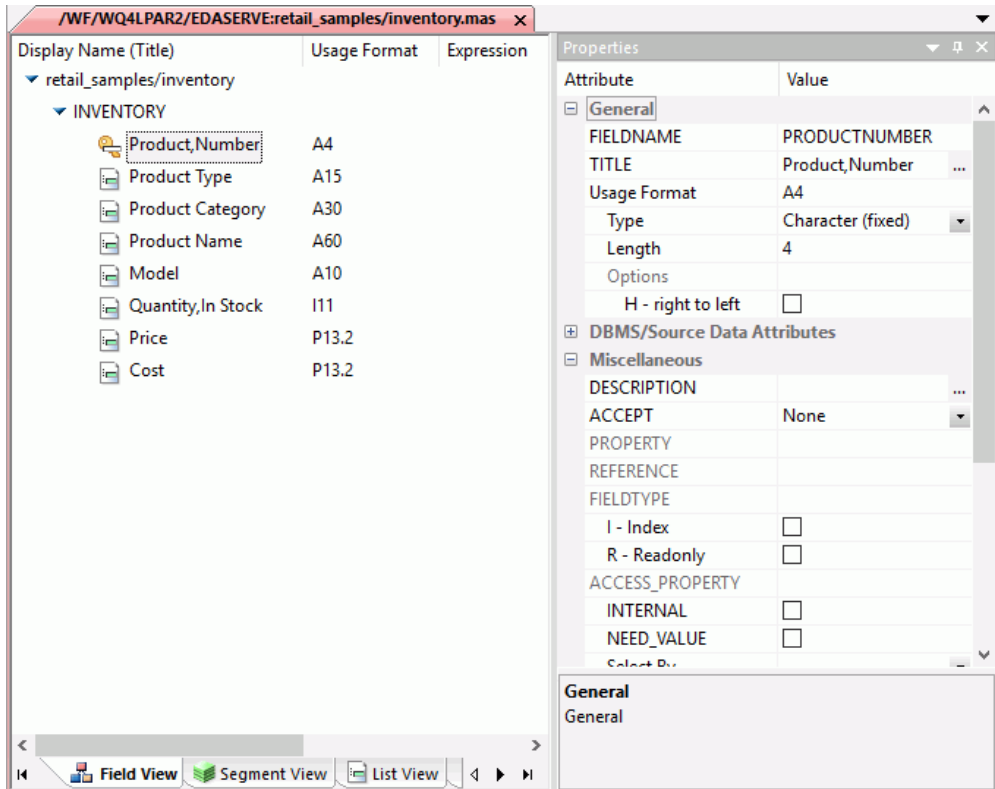
Note: Information about the attribute that has focus is displayed at the bottom of the attribute list. In this case, an explanation of the FIELDNAME attribute appears.

Displaying Segments in a Synonym Using the Segment View Tab

The Segment View tab is accessible when you are creating or editing a synonym. As shown in the following image, the Segment View tab shows a hierarchy of segments that the synonym contains on the left, with the attributes and values of the selected item in the Properties window, which displays, by default, on the right side of the canvas. The Properties panel can be resized, positioned in different areas, or hidden.
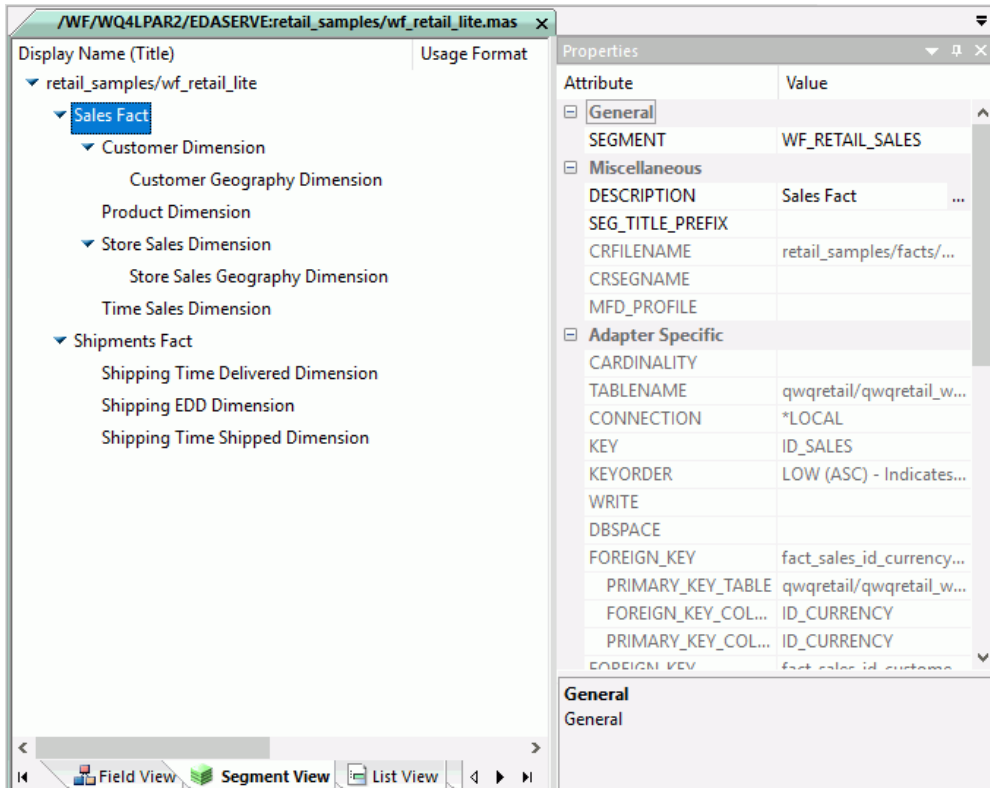
Note: Information about the attribute that has focus is displayed at the bottom of the attribute list. In this case, an explanation of the SEGMENT attribute appears.

Reference: **Segment Shortcut Menu**

When you right-click a segment in the Metadata canvas, the following options are available:

Properties

Opens a pane showing the properties of the selected segment.

Insert

Enables you to insert one of the following:

Reference to Existing Synonym as Child

The reference is a pointer to the source synonym using the synonym as a child. If you subsequently make changes to the source synonym, reopening the current synonym will reflect those changes.

Reference to Existing Synonym as Root

The reference is a pointer to the source synonym using the synonym as a root. If you subsequently make changes to the source synonym, reopening the current synonym will reflect those changes.

Copy of Existing Synonym

Adds a static copy of an existing synonym to the current synonym. If you subsequently make changes to the source synonym, the current synonym will not reflect those changes.

Segment via Metadata Import

Enables you to create and add a new synonym to the current synonym using the Create Synonym dialog box.

Segment Manually

Adds a synonym that must be coded manually.

Field

Adds a general column to the segment.

Define

Adds a virtual or defined column to the segment.

Filter

Adds a filter to the segment.

Compute

Adds a calculated value to the file.

Sort Object

Adds a Sort Objects folder and a sort object. This option is available only if *Support extended options* is selected.

Variable

Adds a Variables folder and a variable to the segment.

Style

Adds a Styles folder and a style object. This option is available only if *Support extended options* is selected.

Subquery

Adds a subquery to the fields list.

Index

Add an Index folder and an index.

Add/Remove Parent Links

If applicable, enables you to add or remove segments in the current cluster synonym.

Data Profiling

Provides the characteristics of the data for a segment. The following options are available:

Statistics

Shows statistic values for the selected segment only.

Count

Shows a count of the number of rows in the selected source table. For a multi-segment synonym, it shows the count for each segment.

Statistics with parent segment

This option is available only when viewing a lower segment of a hierarchy (cluster join).

Key Analysis

Allows you to select the columns that you would like to analyze as potential keys to the source table.

Values

Shows unique values and their percentages.

Duplicate Values

Shows identical values and their percentages.

Sample Data

Displays sample data in the workspace for the selected segment or synonym.

Sample Data with parent key

Displays sample data for the selected segment joined to the parent segment. This option is only available when a child segment is selected in a multi segment synonym.

Delete

Deletes the segment.

Rename

Allows you to rename the segment.

Edit All Properties

Allows you to edit adapter specific properties, such as the delimiter value in a delimited flat file.

Open Base Synonym

Opens the base synonym for the current synonym.

Edit Parent Links

Indicates the parent segment or segments of the selected child that you can edit.

Assign SCD Columns

Allows you to assign columns to a SCD type by using drag-and-drop.

Reference: **Column/Field Shortcut Menu**

When you right-click a column in the Metadata canvas, the following options are available.

Note: If you multi-select columns and right-click them, an abridged version of the shortcut menu opens.

Properties

Opens a pane showing the properties of the selected column.

Insert

Enables you to insert one of the following:

Field

Adds a general column to the synonym.

Character Function

Provides a choice of preconfigured character functions that open in the Function Assist dialog box. The selected functions will be saved in the Constant Defines/Computes folder for the synonym.

Conversion Function

Provides a choice of preconfigured conversion functions that open in the Function Assist dialog box. The selected functions will be saved in the Constant Defines/Computes folder for the synonym.

Geography Function

Provides a choice of preconfigured geography functions that open in the Function Assist dialog box. The selected functions will be saved in the Constant Defines/Computes folder for the synonym.

Define

Adds a virtual or defined column to the synonym.

Filter

Adds a filter to the segment.

Compute

Adds a calculated value to the file.

Grouping

Opens the Group values for dialog box, where you can configure group values.

Aggregation Function

Provides a choice of preconfigured aggregation functions that open in the Function Assist dialog box. The selected functions will be saved in the Constant Defines/Computes folder for the synonym.

Analytic Function

Provides a choice of preconfigured analytic functions that open in the Function Assist dialog box. The selected functions will be saved in the Constant Defines/Computes folder for the synonym.

Sort Object

Adds a Sort Objects folder and a sort object. This option is available only if *Support extended options* is selected and is not used with DataMigrator.

Variable

Adds a Variables Folder if needed.

Style

Adds a Styles folder and a style object. This option is available only if *Support extended options* is selected and is not used with DataMigrator.

Subquery

Adds a subquery to the fields list.

Pivot

The following options are available:

Multiple columns to rows

Allows you to turn (or pivot) columns into rows or a single column into multiple columns, for retrieval.

Multiple values to rows

Allows you to turn (or pivot) values into rows or a single value into multiple values, for retrieval.

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.

Data Profiling

Provides the characteristics of the data for a column.

Statistics

Shows statistic values for the selected segment only.

Key Analysis

Allows you to select the columns that you would like to analyze as potential keys to the source table.

Patterns

Shows a patterns report for the selected column.

Hex View

Shows the Hexadecimal value of the selected field.

Values

Shows a values report for the selected column.

Values Graph

Displays a graph for both alpha and numeric field types.

Values Pie Graph

Displays a pie graph.

Duplicate Values

Shows identical values and their percents.

Outliers

Shows outlier report for the selected column.

Sample Data

Displays sample data in the workspace.

Delete

Deletes the column.

Rename

Allows you to rename the column.

Displaying Fields and Properties in a Synonym Using the List View Tab

The List View tab is accessible when you are creating or editing a synonym. As shown in the following image, the List View tab shows the fields and properties displayed next to their name, with the attributes and values of the selected item in the Properties panel, which displays, by default, on the right side of the canvas. The Properties panel can be resized, positioned in different areas, or hidden. The List View tab is accessible when you are creating or editing a 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.

Showing a Graphical Representation of a Synonym Using the Modeling View Tab

The Modeling View tab is accessible when you are creating or editing a synonym. As shown in the following image, the Modeling View tab shows a graphical representation of the synonym. You can use the Modeling View tab to view join properties, create cluster joins, and add or edit segments. The following image shows the Sales Dimension of a multi-fact data source.
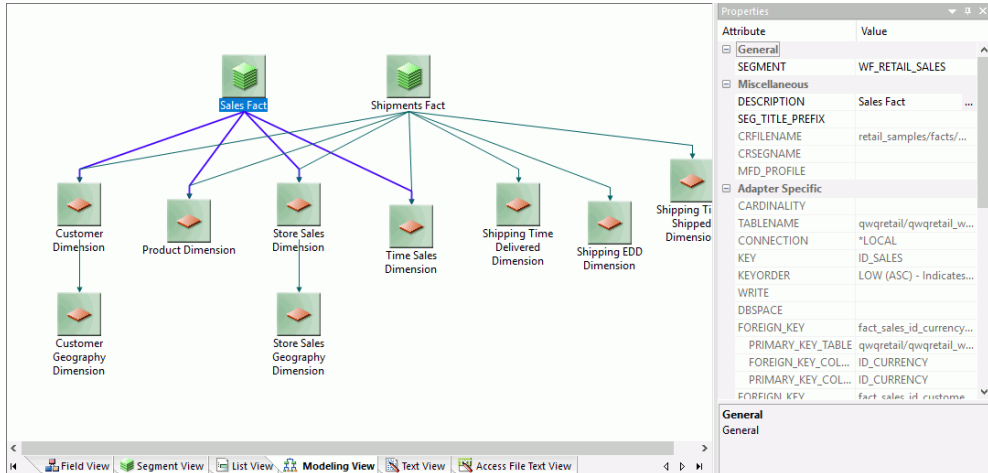
Reference: **Modeling View Shortcut Menu**

When you right-click a synonym or segment in the Metadata canvas Modeling View tab, the following options are available:

Collapse/Expand

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.

Properties

Opens a pane showing the properties of the selected segment.

Insert

- ❑ **Reference to Existing Synonym as Child.** Enables you to reference an existing synonym using the synonym as a child.

- ❑ **Reference to Existing Synonym as Root.** Enables you to reference an existing synonym using the synonym as a root.

❏ **Segment via Metadata Import.** Enables you to create a new synonym through the Create Synonym tool and add it to the current synonym.

❏ **Segment Manually.** Inserts a segment to the current synonym, that must by coded manually.

Add/Remove Parent Links

If applicable, enables you to add or remove segments in the current cluster synonym.

Data Profiling

Provides the characteristics of the data for a segment.

Sample Data

Displays sample data in the workspace.

Sample Data with parent key

Displays sample data with parent key in the workspace.

Delete

Deletes the segment.

Rename

Enables you to rename the segment.

Open Base Synonym

Opens the selected synonym or segment in Modeling View.

Manage Parent Links

If applicable, enables you to edit parent links in the Join Editor.

Assign SCD Columns

Allows you to assign columns to a SCD type by using drag-and-drop.

Displaying the Contents of a Master File Using the Text View Tab

The Text View tab is accessible when you are creating or editing a synonym. The Text View tab shows the Master File of the underlying table in a text editor. You can print the Master File syntax by selecting *Print* from the Application Menu.

The following image is an example of a multi-fact structure in the Text View tab.

Displaying the Contents of an Access File Using the Access File Text View Tab

The Access File Text View tab is accessible when you are creating or editing a synonym. The Access File Text View tab shows the Access File for a synonym, which is used to access the data source. You can print the Access File syntax by selecting *Print* from the Application Menu.

The following image shows an example of an SQL data source in the Access File Text View tab.

```
1    SEGNAME=WF_RETAIL_SALES,
2      TABLENAME=wrd_wf_retail_sales,
3      CONNECTION=mssql2008,
4      KEY=ID_SALES, $
5    FOREIGN_KEY=fact_sales_id_currency_fk,
6        PRIMARY_KEY_TABLE=wrd_wf_retail_currency,
7        FOREIGN_KEY_COLUMN=ID_CURRENCY,
8        PRIMARY_KEY_COLUMN=ID_CURRENCY, $
9    FOREIGN_KEY=fact_sales_id_customer_fk,
10       PRIMARY_KEY_TABLE=wrd_wf_retail_customer,
11       FOREIGN_KEY_COLUMN=ID_CUSTOMER,
12       PRIMARY_KEY_COLUMN=ID_CUSTOMER, $
13   FOREIGN_KEY=fact_sales_id_discount_fk,
14       PRIMARY_KEY_TABLE=wrd_wf_retail_discount,
15       FOREIGN_KEY_COLUMN=ID_DISCOUNT,
16       PRIMARY_KEY_COLUMN=ID_DISCOUNT, $
17   FOREIGN_KEY=fact_sales_id_product_fk,
18       PRIMARY_KEY_TABLE=wrd_wf_retail_product,
19       FOREIGN_KEY_COLUMN=ID_PRODUCT,
20       PRIMARY_KEY_COLUMN=ID_PRODUCT, $
21   FOREIGN_KEY=fact_sales_id_store_fk,
22       PRIMARY_KEY_TABLE=wrd_wf_retail_store,
23       FOREIGN_KEY_COLUMN=ID_STORE,
24       PRIMARY_KEY_COLUMN=ID_STORE, $
25   FOREIGN_KEY=fact_sales_id_time_fk,
26       PRIMARY_KEY_TABLE=wrd_wf_retail_time,
27       FOREIGN_KEY_COLUMN=ID_TIME,
28       PRIMARY_KEY_COLUMN=ID_TIME, $
```

Note: The text views are read-only. You cannot edit the underlying description from these tabs. However, you can right-click a synonym from the navigation pane and click *Edit in Windows Associated Tool* (for relational synonyms).

Viewing and Editing Synonym Attributes

The Metadata canvas enables you to view and edit the field attributes in a synonym. For example, a geographical field may have a role attribute that you want to change. You can do this using the Metadata canvas. Then, all subsequent reports derived from the synonym will include the edited geographical role.

Procedure: How to View and Edit Synonym Attributes

To view and edit synonym attributes:

1. Double-click a synonym, or right-click it and select *Open*.

   The Metadata canvas opens to the Field 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.

   The attributes available depend on the type of synonym.

   The values for Display Name (Title), Format, Expression, Description, and Nulls are viewable in the hierarchy with the columns. To edit these values, use the corresponding attribute fields in the Properties panel on the right-hand side of the Metadata canvas.

2. You can change the attribute values by typing in new values or by using the drop-down menus and check boxes.

   The Properties panel does not let you make any changes that would render the synonym unusable. Therefore, you cannot edit any value field that is highlighted gray. In addition, if a change does not have proper syntax or format applied, the field may appear in red text. Messages and warnings appear if you try to save a file that contains an error.

3. Save changes by clicking *Save* from the Quick Access Toolbar.

   Note: You can multi-select fields to change attributes for multiple fields in a single operation. The display changes to show only those attributes that can be changed.

4. Close the Metadata canvas.

Note: If you close the Metadata canvas without saving your changes, you are prompted to do so.

Reference: File Attributes Summary

Synonyms can have the following file attributes:

General

SUFFIX

Identifies the type of synonym or data source.

MFD_PROFILE

Identifies the name of the FOCEXEC that will be executed before running a request based on the Master File.

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 an 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 an 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.

DATASET

Identifies the location of the data source to be used in the file name, including the extension and the location of the data file.

DV_INCLUDE

Controls the base file DB info inheritance in a cluster.

BV_NAMESPACE

Controls the Business View run-time resolution strategy.

Note: The attributes available depend on the type of synonym.

Reference: Segment Attributes Summary

If a synonym segment is selected, the attributes listed below are available.

Note: Information about the attribute that has focus is displayed at the bottom of the attribute list.

General

SEGMENT

Is the name of the segment.

ENCRYPT

Check this box to scramble field values in the current segment in order to protect it from unauthorized examination.

SEGTYPE

Specifies the type of relationship that a segment has to its parent, and indicates which of the segment 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 names 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.

Miscellaneous

DESCRIPTION

Contains a description or comments about the segment.

SEG_TITLE_PREFIX

An optional attribute that enables you to provide the title prefix for the segment fields that appear in reports. You can split the text across up to five separate title lines by separating the lines with a comma.

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 Db2 Web Query data sources.

OCCURS

Describes repeating fields or groups of fields in a data source. OCCURS can be based on an integer value showing the number of occurrences (from 1 to 4095), a field name, which names a field in the parent segment or a virtual field in an ancestor segment whose integer value contains the number of occurrences of the descendant segment, or a variable, which indicates that the number of occurrences varies from record to record.

POSITION

Specifies the name of the field in the parent segment that specifies the starting position and overall length of the multiply occurring fields.

Adapter Specific

Adapter Specific fields are shown if an Access File component has been generated with the synonym.

CARDINALITY

Defines how many members of a dimension can be retrieved for a report.

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. This value may be a variable.

CONNECTION

Indicates the host server or data source for synonyms. This value may be a variable.

KEY

Identifies the names of the columns that 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.

FOREIGN_KEY

Specifies the type of table persistence and related table properties. This is optional for database management systems that support volatile tables, and required otherwise.

DBSPACE

Identifies the storage area in which the table resides.

PERSISTENCE

Specifies the type of table persistence and related table properties. This is optional for database management systems that support volatile tables, and required otherwise.

Note: The attributes available depend on the type of synonym.

Reference: Column/Field Attribute Summary

If a column or field in a synonym is selected, the attributes listed below are available. The available attributes depend on the type of synonym. Information about the attribute that has focus is displayed at the bottom of the attribute list.

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.

For file data sources, a special reserved ALIAS of INSTANCE means the associated FIELD will show the name of the file when it is read.

When a FIELDNAME of RECTYPE is used the ALIAS contains the value that identifies the record type.

KEY COMPONENT

Indicates whether the field is part of the primary key for the table.

KEY SEQUENCE NUMBER

Indicates the sequence number of the field in the primary key for the table.

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.

ACTUAL

Describes the type and length of data as it is actually stored in the data source.

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 or field.

Miscellaneous

DESCRIPTION

Contains a description or comments about the column or field.

ACCEPT

Specifies criteria for validating data.

- ❏ **OR** enables you to specify an acceptable value.

- ❏ **SYNONYM** enables you to specify a synonym as the source of the accept list.

❏ **FOCEXEC** enables you to specify a FOCEXEC as the source of the accept list.

❏ **FROM-TO** enables you to specify a range of acceptable value fields.

❏ **FIND** enables you to supply file and field names to instruct 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 Db2 Web Query data source where the field can be found, and the field name of the target data field that contains the validation criteria.

FIND is only available for Db2 Web Query data sources and does not apply to OLAP-enabled synonyms.

WITHIN

Contains the name of a field to be included in a dimension.

PROPERTY

Describes which hierarchy property the column represents.

REFERENCE

Points to the field that is the unique identifier for the hierarchy.

FIELDTYPE

❏ **I - Index** identifies an indexed column in a Db2 Web Query data source.

❏ **R - Readonly** indicates a read-only column. This setting is useful for columns that are automatically assigned a value by the RDBMS.

ACCESS_PROPERTY

Specifies access options for the column 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 multi-values.

❏ **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.

This attribute is not used by Db2 Web Query.

GEOGRAPHIC_ROLE

Describes the geographic role of the column.

TEMPORAL_PROPERTY

Inserts virtual fields in the Master File to show how selected totals progressed through a time period for specified intervals.

USE_STYLE

Are the styles (for example, font and color) to apply to the field.

The attributes available depend on the type of synonym.

Adapter Specific

AUTOINCREMENT

Indicates that the field is automatically incremented by the relational database when rows are inserted. This option is only available for selected relational databases. When selected, the field must also be identified with a field type of read-only.

The following attributes are only used if AUTOINCREMENT is checked.

❏ **START.** Indicates the starting value for this AUTOINCREMENT field. If not specified, 1 is used.

❏ **INCREMENT.** Indicates the increment value for this field. If not specified, 1 is used.

❏ **SEQUENCE.** Is the name of the corresponding database SEQUENCE object for this field. This option is only available for ORACLE tables.

Setting Up Multilingual Titles and Descriptions

You can open a synonym in the Metadata canvas 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 Metadata canvas.

Note: The attributes available depend on the type of synonym.

Procedure: How to Set Up Multilingual Titles and Descriptions

1. From the Data Servers area, double-click the Master File.

    The Master File opens to the Field View tab in the Metadata canvas.

2. To add multilingual text:

    ❏ For *Remarks*, click the root level of the Master File (application/filename) on the left.

    ❏ For *Title*, click a column from the Master File hierarchy of columns 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 (...) button at the end of the value field for Remarks, Title, or Description.

    The Multilingual dialog box opens.

4. From the Specify for different languages drop-down list, choose the language in which you want the titles or remarks (descriptions) 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 a description or title in the Text field.

7. You may add, edit, or delete additional titles or descriptions.

    To add an additional language:

    a. Select another language from the drop-down list.

    b. Click *Add*.

    c. Type a title or description for the field.

    To edit an existing specified title or description:

    a. Select the title or description and click *Edit*.

> You may also double-click the title.

   b.   Manually type a title or description name.

To delete a specified title:

   a.   Select the title or description and language to be deleted.

   b.   Click *Delete*.

8.   Click *OK* to close the Multilingual Remarks dialog box.

9.   Click *Save* from the File menu to save the synonym.

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.

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 values, and number of nulls.

Data Profiling for an individual column provides access to Statistics, Patterns, Values, and Outliers reports.

Procedure: **How to View Data Profiling for a Synonym or Segment**

To view the Data Profiling information for a synonym or segment:

1.   From the Data Servers area, open a synonym by double-clicking a Master File.

    By default, the Metadata canvas opens to the last view used.

2.   Right-click the synonym or segment name, select *Data Profiling*, and then *Statistics*.

    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.

For pattern analysis, a *9* represents a digit, an *A* represents any uppercase letter, and an *a* represents any lowercase letter. All printable special characters are represented by themselves and unprintable characters are represented by an *X*.

Viewing Data Profiling Columns

Data Profiling can be done for all the columns in a synonym or segment (press the Shift or Ctrl key while selecting multiple columns), or for an individual column. Data Profiling for an individual column provides access to the following 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 values, and number of nulls.

❏ **Hex View.** Shows values in Hexadecimal format.

❏ **Key Analysis.** Allows you to select the columns that you would like to analyze as potential keys to the source table.

❏ **Patterns.** Only available for alphanumeric columns, shows patterns of letters, digits, and special characters, as well as counts and their percents.

❏ **Values.** Shows unique values and their percents.

❏ **Values Graph.** Displays a graph for alphanumeric field types.

❏ **Values Pie Graph.** Displays a pie graph for alphanumeric field types.

❏ **Duplicate Values.** Shows identical 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 Metadata canvas and selecting *Data Profiling*.

Procedure: **How to View Data Profile Statistics**

To view the Statistical Data Profiling information for a single column:

1. From the Data Servers area, open a synonym by double-clicking a Master File.

   By default, the Metadata canvas opens to the last view used.

2. Right-click a column, select *Data Profiling*, and then select *Statistics*.

   The Statistical Data Profiling information opens in the workspace, as shown in the following image.

	Segment	Name	Format	Count	Distinct Count	Distinct Percent	Minimum	Maximum	Average
1	WF_RETAIL_SALES	ID_SALES	I9	2488361	2488361	100.00	1	2506705	1252063

3. Optionally, you can click a column name or patterns count (for alphanumeric columns) to drill down to the Values or Patterns reports, respectively.

Procedure: **How to View Data Profile Key Analysis**

To view the Key Analysis Data Profiling information for a single column:

1. From the Data Servers area, open a synonym by double-clicking a Master File.

   By default, the Metadata canvas opens to the last view used.

2. Right-click a column, select *Data Profiling*, and then select *Key Analysis*.

   The Key Analysis Data Profiling information displays, as shown in the following image.

/WF/WQ4LPAR2/EDASERVE: Key Analysis for "ID Store" (retail_samples/facts/wf_retail_s...

	Segment	Name	Format	Elements	Count	Distinct Count	Distinct Percent	Duplicate Count	Duplicate Percent
1	WF_RETAIL_SALES	ID_STORE	I9	1	2488361	87	.00	2488274	100.00
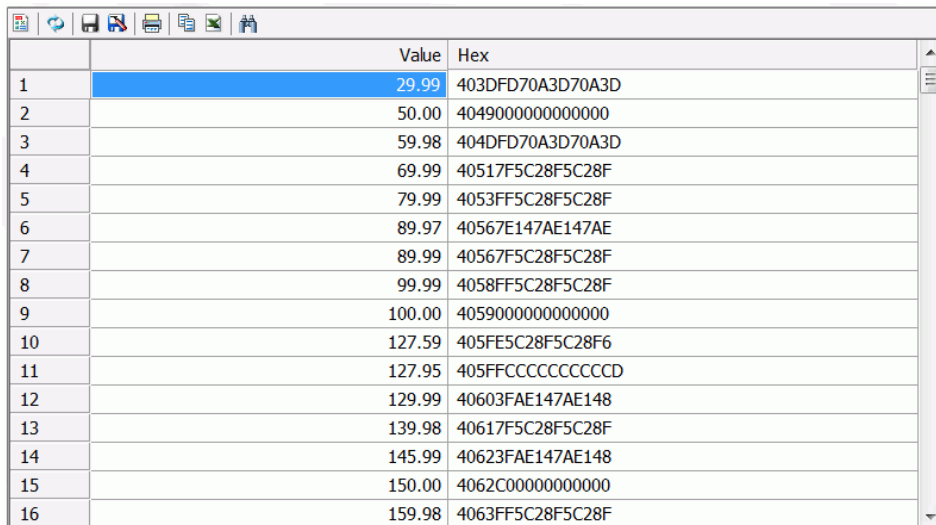
Procedure: **How to View Data Profile Hex View**

Data Profile Hex View shows values in Hexadecimal format.

To view the Hex View Data Profiling information for a single column:

1. From the Data Servers area, open a synonym by double-clicking a Master File.

   By default, the Metadata canvas opens to the last view used.

2. Right-click a column, select *Data Profiling*, and then select *Hex View*.

The Patterns Data Profiling information displays, as shown in the following image.

	Value	Hex
1	29.99	403DFD70A3D70A3D
2	50.00	4049000000000000
3	59.98	404DFD70A3D70A3D
4	69.99	40517F5C28F5C28F
5	79.99	4053FF5C28F5C28F
6	89.97	40567E147AE147AE
7	89.99	40567F5C28F5C28F
8	99.99	4058FF5C28F5C28F
9	100.00	4059000000000000
10	127.59	405FE5C28F5C28F6
11	127.95	405FFCCCCCCCCCCD
12	129.99	40603FAE147AE148
13	139.98	40617F5C28F5C28F
14	145.99	40623FAE147AE148
15	150.00	4062C00000000000
16	159.98	4063FF5C28F5C28F
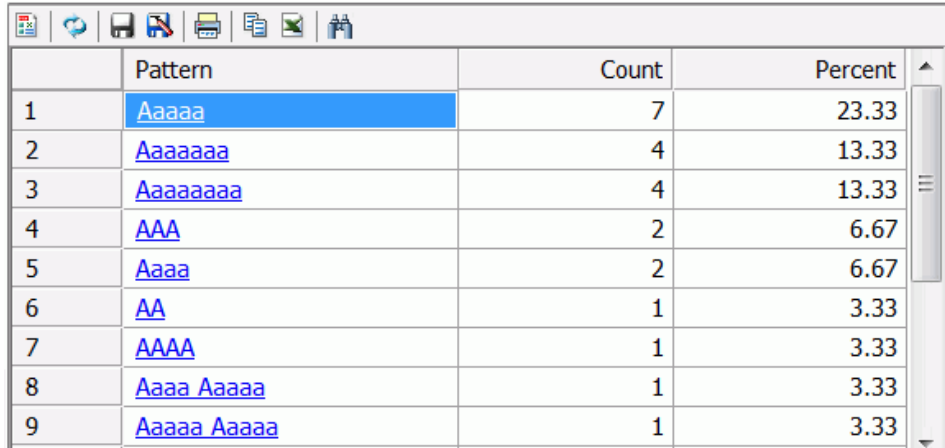
Procedure: **How to View Data Profile Patterns**

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 Data Servers area, open a synonym by double-clicking a Master File.

   By default, the Metadata canvas opens to the last view used.

2. Right-click a column, select *Data Profiling*, and then select *Patterns*.

The Patterns Data Profiling information displays, as shown in the following image.

	Pattern	Count	Percent
1	Aaaaa	7	23.33
2	Aaaaaaa	4	13.33
3	Aaaaaaaa	4	13.33
4	AAA	2	6.67
5	Aaaa	2	6.67
6	AA	1	3.33
7	AAAA	1	3.33
8	Aaaa Aaaaa	1	3.33
9	Aaaaa Aaaaa	1	3.33

For pattern analysis, a 9 represents a digit, an *A* represents any uppercase letter, and an *a* represents any lowercase letter. All printable special characters are represented by themselves and unprintable characters are represented by an *X*.
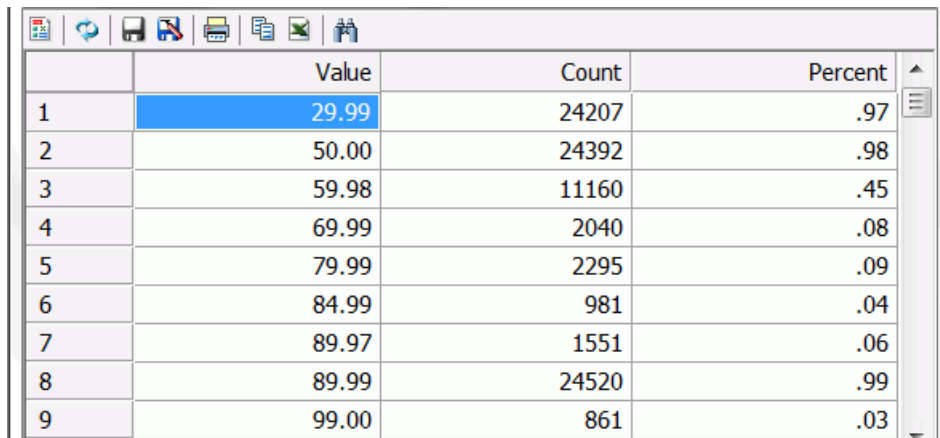
Procedure: How to View Data Profile Values

Data Profile Values shows unique values.

To view the Values Data Profiling information for a single column:

1. From the Data Servers area, open a synonym by double-clicking a Master File.

    By default, the Metadata canvas opens to the last view used.

2. Right-click a column, select *Data Profiling*, and then select *Values*.

The Values Data Profiling information displays, as shown in the following image.

	Value	Count	Percent
1	29.99	24207	.97
2	50.00	24392	.98
3	59.98	11160	.45
4	69.99	2040	.08
5	79.99	2295	.09
6	84.99	981	.04
7	89.97	1551	.06
8	89.99	24520	.99
9	99.00	861	.03

Procedure: **How to View the Data Profile Values Graph**

The Data Profile Values Graph displays values as a bar graph.

To view the Values Graph Data Profiling information for a single column:

1. From the Data Servers area, open a synonym by double-clicking a Master File.

   By default, the Metadata canvas opens to the last view used.

2. Right-click a column, select *Data Profiling*, and then select *Values Graph*.

The Values Graph Data Profiling information displays, as shown in the following image.
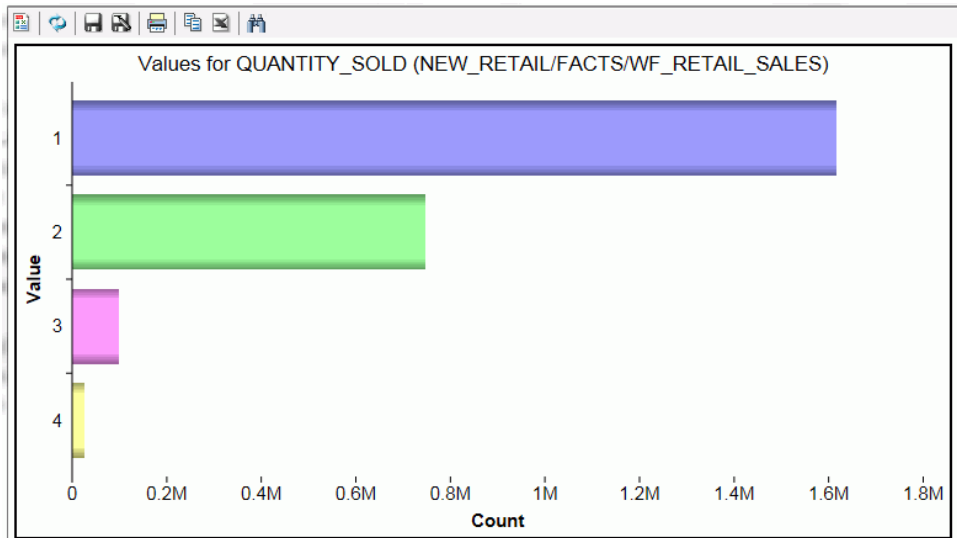
Procedure: **How to View the Data Profile Values Pie Graph**

The Data Profile Values Pie Graph displays values as a pie graph.

To view the Values Pie Graph Data Profiling information for a single column:

1. From the Data Servers area, open a synonym by double-clicking a Master File.

   By default, the Metadata canvas opens to the last view used.

2. Right-click a column, select *Data Profiling*, and then select *Values Pie Graph*.

The Values Pie Graph Data Profiling information displays, as shown in the following image.
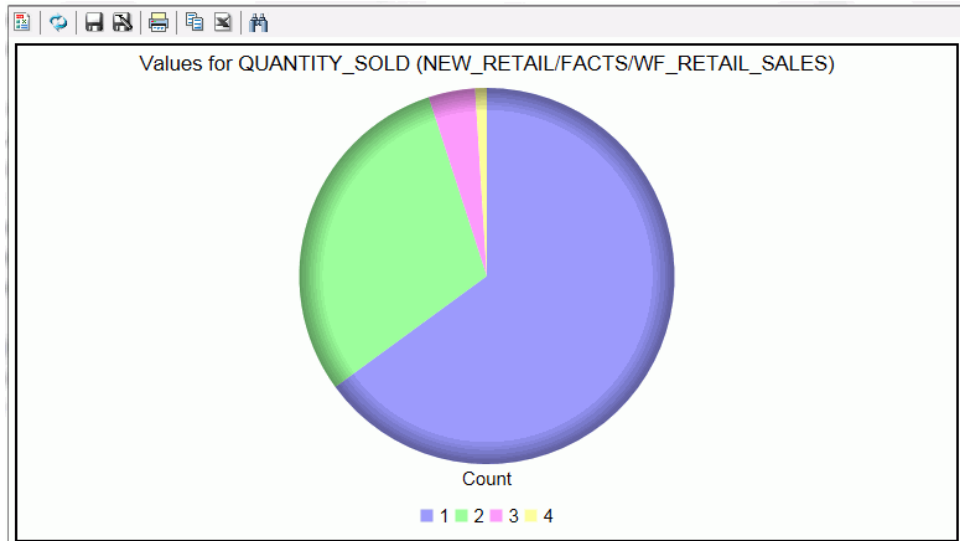
Procedure: How to View Data Profile Duplicate Values

Data Profile Duplicate Values shows identical values.

To view the Duplicate Values Data Profiling information for a single column:

1. From the Data Servers area, open a synonym by double-clicking a Master File.

   By default, the Metadata canvas opens to the last view used.

2. Right-click a column, select *Data Profiling*, and then select *Duplicate Values*.

   The Duplicate Values Data Profiling information displays, as shown in the following image.

	Value	Count	Percent
1	100	1833462	73.68
2	101	252236	10.14
3	102	176451	7.09
4	103	75401	3.03
5	104	75239	3.02
6	105	50120	2.01
7	106	25452	1.02
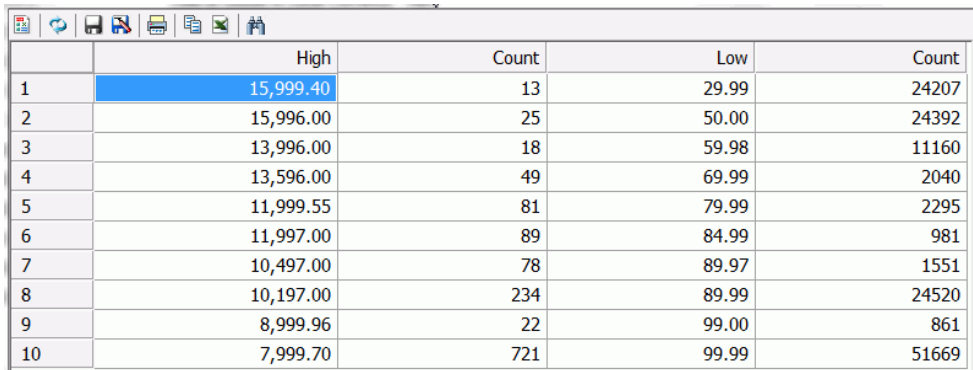
Procedure: **How to View Data Profile Outliers**

Data Profile Outliers shows the ten highest and ten lowest distinct values.

To view the Outliers Data Profiling information for a single column:

1. From the Data Servers area, open a synonym by double-clicking a Master File.

    By default, the Metadata canvas opens to the last view used.

2. Right-click a column, select *Data Profiling*, and then select *Outliers*.

    The Outliers Data Profiling information displays, as shown in the following image.

	High	Count	Low	Count
1	15,999.40	13	29.99	24207
2	15,996.00	25	50.00	24392
3	13,996.00	18	59.98	11160
4	13,596.00	49	69.99	2040
5	11,999.55	81	79.99	2295
6	11,997.00	89	84.99	981
7	10,497.00	78	89.97	1551
8	10,197.00	234	89.99	24520
9	8,999.96	22	99.00	861
10	7,999.70	721	99.99	51669

Note: Outliers produce a maximum of the ten highest and ten lowest distinct values, if they exist.

Enhancing a Master File with Expressions

Using an expression, you can create virtual columns and store calculations and selection criteria in a synonym. For example, using the Revenue measure, you can create a calculation for Revenue per Square Foot. This is saved in the synonym as an object and is available for use in all subsequent reports derived from the synonym.

Adding Virtual Columns (DEFINE) in a Synonym

You may create a DEFINE field as a custom field in a synonym. A custom field can be used in a request as though it is a real data source field. Virtual columns (DEFINE fields) are available when the data source is used for reporting.

A virtual column can contain an expression, a constant, or a column name.

Procedure: **How to 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.

    The Metadata canvas opens.

2. Right-click a segment (or column), point to *Insert*, and then click *Define*.

    The Define Calculator opens, as shown in the following image.
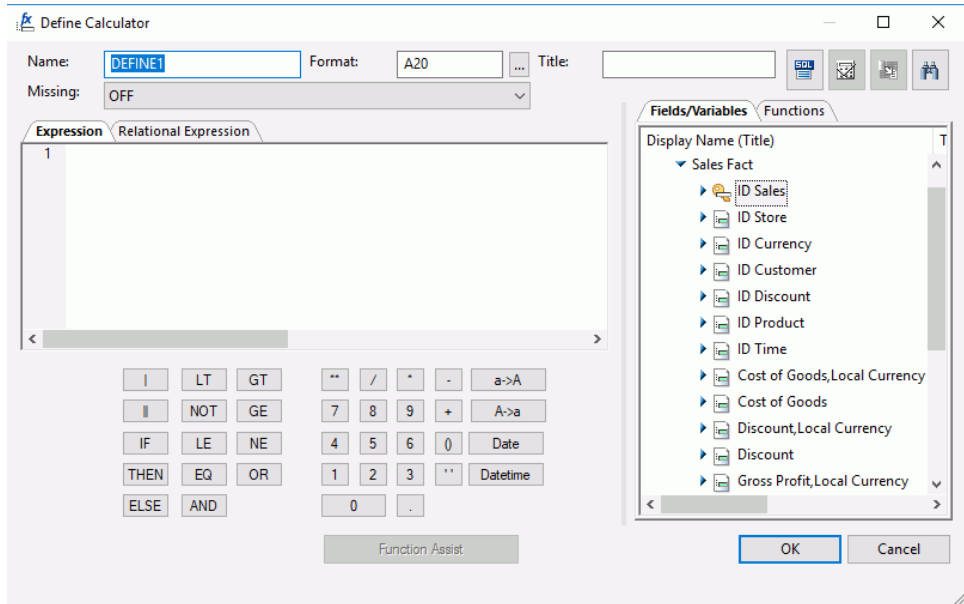
3. From the Define Calculator, type a name for the column in the Name input field, or use the default define name.

4. Type a format for the column in the Format input field, or click the browse (...) button at the right of the Format input field to launch the Properties dialog box to enter the information.

5. You may enter a descriptive title for the virtual column (DEFINE) in the Title input field.

    Tip: From the Metadata canvas, click the browse (...) button at the right of the TITLE and DESCRIPTION value fields to specify multiple language titles.

6. Specify the Missing Data option for columns that allow null data. You can allow all missing data.

7. Use the Expression tab and the calculator buttons to build the expression for the virtual column (DEFINE).

   or

   Use the Relational Expression tab to build the expression.

   a. From the Relational Expression 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 values and use the arrows to add or remove values.

   d. Click *OK* to close the Value Selection dialog box and return to the Relational Expression 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 Virtual Column Calculator.
      The filter is added to the Relational Expression tab, where you can add the expression value.

   f. To delete an expression, click in the row number column of the expression that you are deleting, right-click, and select *Delete Selected Row(s)*. The expression is removed from the Relational Expression tab.

8. 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.

9. Click *OK* to close the Define Calculator and return to the Metadata canvas.

   Note: To edit the Define, Title, or Expression, you may do so directly from the Metadata canvas, or you may click the browse *(…)* button at the right of the EXPRESSION value field to relaunch the Define Calculator.

10. 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 (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.

11. Click *Save* from the Quick Access Toolbar to save the synonym.

Adding Computed Fields (COMPUTE) in a Synonym

You may create a Computed field as a custom field in the Metadata canvas. 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.

Procedure: **How to Create a Computed Field in the Metadata Canvas**

To create a computed column in a synonym

1. From the Data Servers area, open a synonym by double-clicking a Master File.

    The Metadata canvas opens.

2. Right-click a segment (or column), point to *Insert*, and click *Compute*.

    The Compute Calculator opens.

3. From the Compute Calculator, type a name for the column in the Column input field or use the default compute name.

4. Type a format for the column in the Format input field, or click the browse *(...)* button at the right of the Format input field to launch the Properties dialog box to enter the information.

5. You may enter a descriptive title for the computed field (COMPUTE) in the Title input field.

    Tip: From the Metadata canvas, click the browse (...) button at the right of the TITLE and DESCRIPTION value fields to specify multiple language titles.

6. Specify the Missing Data options for columns that allow null data. You can allow all missing data.

7. Use the Expression tab and the calculator buttons to build the expression for the computed field (COMPUTE).

    or

    Use the Relational Expression tab to build the expression.

    a. From the Relational Expression tab, click the *Add New Row* button and 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.

    c. Select from the available values and use the arrows to add or remove values.

    d. Click *OK* to close the Value Selection dialog box and return to the Relational Expression 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 Relational Expression tab, where you can add the expression value.

f. To delete an expression, select the number or field column of the expression and click the *Delete* button, or right-click it and select *Delete Selected Row(s)*. The expression is removed from the Relational Expression tab.

8. You may click 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.

9. Click *OK* to close the Compute Calculator and return to the Metadata canvas.

Note: To edit the Compute, Title, or Expression, you may do so directly from the Properties section of the Metadata canvas or you may click the browse (...) button at the right of the EXPRESSION value field to relaunch the Compute Calculator.

10. Click *Save* from the File menu to save the synonym.

For more information about expressions and COMPUTE attributes, see *Defining Attributes and Creating Expressions for Custom Fields* on page 138.

Creating Filters in a Synonym

Filters are created in the Master File through the Metadata canvas and can be used in a Business View file or in reporting tools. 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.

Procedure: **How to Create Filters in a Synonym**

1. From the Data Servers area, open a synonym by double-clicking a Master File.

    The Metadata canvas opens.

2. Right-click a segment or field, point to *Insert*, and click *Filter*.

    The Filter Calculator opens, as shown in the following image.
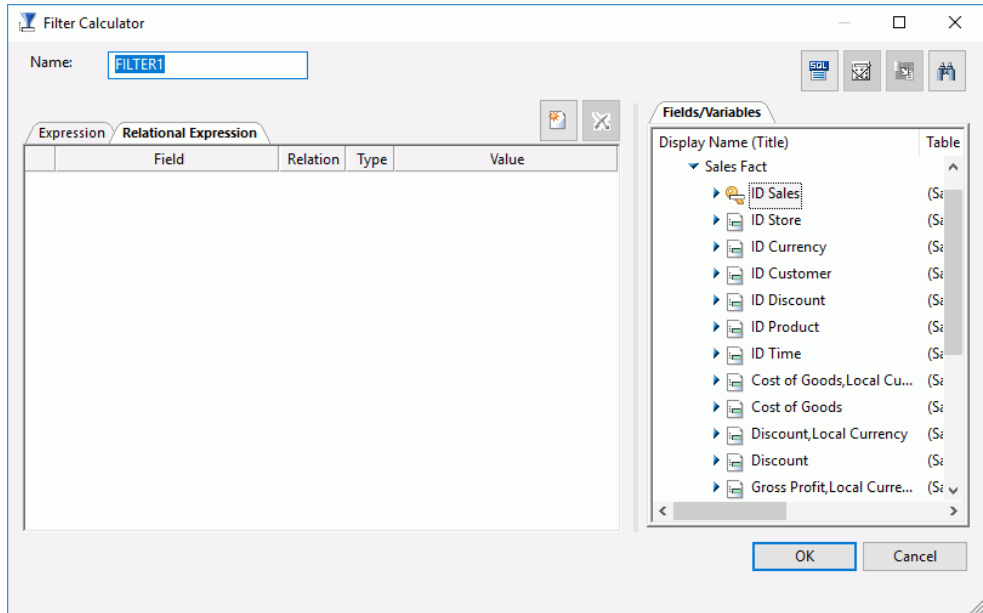
3. From the Filter Calculator, type a name for the filter in the Name input field or use the default filter name.

    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 Properties panel in the Metadata canvas, 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 Relational Expression tab to build the expression.

a. From the Relational Expression tab, click the *Add New Row* button and 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 values and use the arrows to add or remove values.

d. Click *OK* to close the Value Selection dialog box and return to the Relational Expression tab.

   The expression is added to the value field.

e. To add another filter, double-click a column or variable from the Fields/Variables section on the right side of the Filter Calculator.

   The filter is added to the Relational Expression tab where you can add the expression value.

f. To delete an expression, select the number or field column of the expression and click the *Delete* button, or right-click it and select *Delete selected row(s)*.

   The expression is removed from the Relational Expression tab.

7. You may click 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 and return to the Metadata canvas.

   Note: To edit the Filter, Title, or Expression, you may do so directly from the Properties panel in the Metadata canvas or you may click the browse (...) button at the right of the EXPRESSION value field to relaunch the Filter Calculator.

9. To create a Filter without segment association, use the WITH drop-down list in the Properties panel to select a blank segment.

   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 you are 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 (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.

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), a Computed Field (COMPUTE), and a Master File filter (FILTER). 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 using the Metadata canvas.

Reference: Custom Field Attributes

The following attributes may be available for custom fields (DEFINE, COMPUTE, and FILTER) in the Metadata canvas.

Note: The attributes available depend on the type of synonym and the type of custom field selected.

Custom fields (DEFINE, COMPUTE, and FILTER) 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.

COMPUTE

Is the name of the computed field.

Note: This attribute only appears when a virtual computed field is selected.

FILTER

Is the name of the Master File Filter field.

Note: This attribute only appears when a virtual filter field is selected.

EXPRESSION

Is the expression 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.

MISSING

Allows missing data. If not, the transaction value is supplied.

Note: This attribute only appears for DEFINE and COMPUTE custom fields.

Miscellaneous

REDEFINES

A DEFINE expression may not contain qualified field names. REDEFINES enables you to redefine or recompute a column whose name exists in more than one segment.

Note: This attribute only appears for DEFINE custom fields and cannot be used by Db2 Web Query.

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 (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.

ACCESS_PROPERTY

Specifies access options for the column 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.

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.

GEOGRAPHIC_ROLE

Defines the geographic role of the 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 Db2 Web Query files and relational tables.

Note: This attribute only appears for DEFINE and FILTER custom fields.

TEMPORAL_PROPERTY

Inserts virtual fields in the Master File to show how selected totals progressed through a time period for specified intervals.

USE_STYLE

Are the styles (for example, font and color) to apply to the field.

Note: The attributes available depend on the type of synonym.

Reference: **Calculators for Custom Fields**

The Define Calculator, Compute Calculator, and Filter Calculator have the following fields and options:

Name

Is the name of the object being created (virtual field (DEFINE), filter, computed field).

Format

Is the field format.

Title

Is the title of a new target table created by a data flow. The title will be included in the synonym that is created.

Missing

Indicates if missing (null) values are allowed in this field. The following options are available:

❏ **OFF.** The result is never missing.

❏ **ON.** The result is missing only if all missable fields are missing.

❏ **ON ALL.** The result is missing if any missable fields are missing.

Expression 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.

Relational Expression tab

Displays the expression building window from which you can add and delete columns, choose the relation and type, and select values for your filter.

Fields/Variables tab

Displays a hierarchical list of available source columns and System Variable 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

Enables you to 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        SMITHSONJ
ANE            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 the quotation marks.

a->A

Converts selected text to uppercase.

A->a

Converts selected text to lowercase.

Date

Opens the Date dialog box, where you can change the format for the Date field.

Datetime

Opens the Date Time dialog box, where you can change the format for the Datetime field.

Check expression button

Verifies the validity of the expression.

Sample data button

Produces sample data for the expression.

Describing Group Fields and Repeating 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.

Procedure: **How to Add a Group Field to 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 Metadata canvas opens.

2. Right-click a segment (or column), select *Insert*, and click *Group*.

   Note: The group 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.

3. Type a name for the group in the *GROUP* field.

4. Select the USAGE *Type* value for the group. Specify the length, the decimal places (if applicable) and any display options for the group.

5. Select the *I - Index* check box if you want the group to be indexed.

6. Select the ACCESS_PROPERTY values to specify access options for the group data.

   ❏ INTERNAL defines a column that will not appear in sample data or in the list of available fields.

   ❏ NEED_VALUE defines a column that requires a value to access the data.

   ❏ Select By defines a column by value, range, or multivalues.

7. Optionally, you can specify the TITLE and DESCRIPTION display options.

8. Click *Save* from the File menu to save the synonym.

Note: When you add a group, a field is automatically added to the group.

Procedure: **How to Add a New Field to a Group Field**

1. Right-click the group field to which you want to add a new field, select *Insert*, and click *Field*.

    A field is added to the group and its attributes and values appear on the right.

2. Supply the required information for the group field. For more information about field attributes, see *Viewing and Editing Synonym Attributes* on page 114.

Procedure: **How to 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.

Procedure: **How to Delete a Group Field From a Segment**

1. Right-click the group and select *Delete*.

    A confirmation appears stating that all columns (fields) within the group will be deleted.

2. 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 Confirm dialog box and return to the Metadata canvas.

Procedure: **How to Delete a Field From a Group Field**

Right-click the field, then select *Delete*.

Reference: **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 Metadata canvas 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 the fields that make it up.

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 that are 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 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.

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.

USE_STYLE

Are the styles (for example, font and color) to apply to the field.

Note: The attributes available depend on the type of synonym.

Storing the Number of Repetitions of a Repeating Field in a Virtual Field

The OCCURS attribute in a Master File describes repeating fields or groups of fields in a data source. The repeating group of fields is described as a descendent segment in the Master File, and the OCCURS attribute for that segment specifies how to determine the number of repetitions.

The number of repetitions does not have to be the same for every record instance. Sometimes, the number of repetitions can be derived from a field in the data source. In that case, you can create a virtual field in the Master File that indicates the number of repetitions for each record and use that virtual field as the value of the OCCURS attribute.

Syntax: **How to Specify an OCCURS Segment Using a Virtual Field**

```
SEGNAME = parent, SEGTYPE = segtype,$
   .
   .
   .
 DEFINE definefield/In = expression;
 SEGNAME = osegname, SEGTYPE=S0, PARENT = parent,
    OCCURS = definefield ,$
 FIELDNAME = rfield, ALIAS = ralias,
    USAGE = rufmt, ACTUAL = rafmt,$
   .
   .
   .
[FIELDNAME = orderfield, ALIAS = ORDER,
    USAGE = In, ACTUAL = I4,$]
```

where:

parent

Is the name of the parent segment.

segtype

Is the SEGTYPE of the parent segment.

definefield

Is the virtual field that indicates the number of repetitions of the repeating field or group of fields. This field must be defined in a segment that is an ancestor of the segment containing the repeating fields.

n

Is the format of the virtual field that describes the number of repetitions. It must be an integer format.

expression

Is a valid expression that derives the number of repetitions for each record instance.

osegname

Is the name of the descendent OCCURS segment.

rfield

Is the name of a repeating field in the OCCURS segment.

ralias

Is the alias of a repeating field in the OCCURS segment.

rufmt

Is the display format for a repeating field in the OCCURS segment.

rafmt

Is the actual format for a repeating field in the OCCURS segment.

orderfield

Is the name of an internal counter field that you can specify as the last field in the OCCURS segment. The ORDER field associates a sequence number with each occurrence and is useful when the order of the repeating data is significant. For example, the values may represent monthly or quarterly data, but the record itself may not explicitly specify the month or quarter to which the data applies. The USAGE format must be integer and the ACTUAL format is I4.

IBM

Note: The virtual field used as the OCCURS value cannot be redefined inside or outside of the Master File.

Example: **Using a Virtual Field With an OCCURS Segment**

The following request against the EMPLOYEE data source creates a fixed-format sequential file with a repeating field. The request:

❏ Counts the number of FICA deductions for each employee.

❏ Creates a calculated field that contains the length of all FICA deduction fields for each employee.

❏ Creates a HOLD file in which each record contains the calculated length of the deduction fields for the employee, the identifying information for the employee, and all FICA deductions for the employee.

Note that the number of deductions will vary for each employee. The part of the record that contains the deductions will constitute the OCCURS segment. The number of repetitions will have to be derived from the length field created in the TABLE request.

The procedure to create the file with the repeating deduction field follows:

```
DEFINE FILE EMPLOYEE
  CTR/I5 WITH DED_AMT = IF EMP_ID NE LAST EMP_ID THEN 1 ELSE LAST CTR + 1;
END
TABLE FILE EMPLOYEE
  SUM CNT.DED_AMT NOPRINT EMP_ID LAST_NAME FIRST_NAME CURR_SAL
  COMPUTE DEDLEN/I5 = 12 * CNT.DED_AMT;
  BY EMP_ID NOPRINT
  SUM DED_AMT
  BY EMP_ID NOPRINT
  ACROSS CTR NOPRINT
  WHERE DED_CODE EQ 'FICA'
  ON TABLE SET HOLDLIST PRINTONLY
  ON TABLE HOLD AS OCCURS1 FORMAT ALPHA
END
```

The OCCURS1 file has one record per employee with a variable number of DED_AMT fields. The total length of the number of actual instances of DED_AMT is stored in the field named DEDLEN. The Master File generated by the HOLD command lists 10 DED_AMT fields:

```
FILENAME=OCCURS1 , SUFFIX=FIX      , $
  SEGMENT=OCCURS1, SEGTYPE=S0, $
    FIELDNAME=EMP_ID, ALIAS=E01, USAGE=A9, ACTUAL=A09, $
    FIELDNAME=LAST_NAME, ALIAS=E02, USAGE=A15, ACTUAL=A15, $
    FIELDNAME=FIRST_NAME, ALIAS=E03, USAGE=A10, ACTUAL=A10, $
    FIELDNAME=CURR_SAL, ALIAS=E04, USAGE=D12.2M, ACTUAL=A12, $
    FIELDNAME=DEDLEN, ALIAS=E05, USAGE=I5, ACTUAL=A05, $
    FIELDNAME=DED_AMT, ALIAS=E06, USAGE=D12.2M, ACTUAL=A12, $
    FIELDNAME=DED_AMT, ALIAS=E07, USAGE=D12.2M, ACTUAL=A12, $
    FIELDNAME=DED_AMT, ALIAS=E08, USAGE=D12.2M, ACTUAL=A12, $
    FIELDNAME=DED_AMT, ALIAS=E09, USAGE=D12.2M, ACTUAL=A12, $
    FIELDNAME=DED_AMT, ALIAS=E10, USAGE=D12.2M, ACTUAL=A12, $
    FIELDNAME=DED_AMT, ALIAS=E11, USAGE=D12.2M, ACTUAL=A12, $
    FIELDNAME=DED_AMT, ALIAS=E12, USAGE=D12.2M, ACTUAL=A12, $
    FIELDNAME=DED_AMT, ALIAS=E13, USAGE=D12.2M, ACTUAL=A12, $
    FIELDNAME=DED_AMT, ALIAS=E14, USAGE=D12.2M, ACTUAL=A12, $
    FIELDNAME=DED_AMT, ALIAS=E15, USAGE=D12.2M, ACTUAL=A12, $
```

You can edit the Master File to describe these repeating DED_AMT fields with an OCCURS segment. The DEFINE field named NUMOCC derives the number of occurrences from the DEDLEN field. The ORDER field is not actually in the file. It is an internal counter populated by Developer Workbench.

```
FILENAME=OCCURS1 , SUFFIX=FIX      , $
  SEGMENT=OCCURS1, SEGTYPE=S0, $
    FIELDNAME=EMP_ID, ALIAS=E01, USAGE=A9, ACTUAL=A09, $
    FIELDNAME=LAST_NAME, ALIAS=E02, USAGE=A15, ACTUAL=A15, $
    FIELDNAME=FIRST_NAME, ALIAS=E03, USAGE=A10, ACTUAL=A10, $
    FIELDNAME=CURR_SAL, ALIAS=E04, USAGE=D12.2M, ACTUAL=A12, $
    FIELDNAME=DEDLEN, ALIAS=E05, USAGE=I5, ACTUAL=A05, $
    DEFINE NUMOCC/I2 = DEDLEN/12;,$
  SEGNAME=DEDUCTION, SEGTYPE=S0, PARENT=OCCURS1, OCCURS=NUMOCC,$
    FIELDNAME=DED_AMT, ALIAS=E06,    USAGE=D12.2M, ACTUAL=A12, $
    FIELDNAME=ORDER,    ALIAS=ORDER, USAGE=I2     , ACTUAL=I4 , $
```

The following request uses the ORDER field to select and print the first occurrence of the repeating field for each employee. Since every employee has at least one deduction, every employee is represented on the report output:

```
TABLE FILE OCCURS1
  PRINT NUMOCC LAST_NAME CURR_SAL DED_AMT
  WHERE ORDER EQ 1
END
```

The output is:

```
NUMOCC   LAST_NAME                       CURR_SAL            DED_AMT
------   ---------                       --------            -------
    10   STEVENS                      $11,000.00             $64.17
     8   SMITH                        $13,200.00            $100.10
     4   JONES                        $18,480.00            $247.94
     8   SMITH                         $9,500.00             $60.96
     1   BANNING                      $29,700.00            $519.75
     8   IRVING                       $26,862.00            $626.78
     4   ROMANS                       $21,120.00            $317.62
     1   MCCOY                        $18,480.00            $161.70
     5   BLACKWOOD                    $21,780.00            $444.67
     7   MCKNIGHT                     $16,100.00            $187.88
     4   GREENSPAN                     $9,000.00             $52.50
    10   CROSS                        $27,062.00            $631.40
```

If you print the tenth occurrence of the repeating field, only two employees are displayed on the report output:

```
TABLE FILE OCCURS1
  PRINT NUMOCC LAST_NAME CURR_SAL DED_AMT
  WHERE ORDER EQ 10
END
```

The output is:

```
NUMOCC   LAST_NAME                       CURR_SAL            DED_AMT
------   ---------                       --------            -------
    10   STEVENS                      $11,000.00             $58.33
    10   CROSS                        $27,062.00            $526.20
```

Parameterizing a Synonym with Variables

You can define variables in a synonym in order to specify synonym attributes, such as the connection name or the table name. The variable is saved in the synonym as an object and is available for use in all subsequent reports derived from the synonym.

Procedure: **How to Add a Variable to a Synonym**

1. Right-click an object in the Metadata canvas, point to *Insert*, and then click *Variable*.

2. A new folder named Variables is added to the object tree, with &&VARIABLE1 as an entry. The Properties panel shows the attributes of the variable.

3. If desired, change the name of the variable and enter a default value in the appropriate fields.

Reference: ## Variables Attribute Summary

Variables in a synonym can have the following attributes:

Variable Name

The name of the variable. A double ampersand (&&) is automatically inserted in the name to indicate a global variable.

Prompt Caption

A description of the variable that replaces the variable name in the prompt.

Default Value

The default value for the variable. If no value is set, the default value is used.

Prompt Values

Provides a drop-down menu of optional prompt values for the variable. The options are Input box, Static list of values, Static list of values/captions, Synonym based dynamic list of values, and Procedure based dynamic list of values.

Quoted

When selected, single quotation marks are added around the assigned string for the variable. A single quotation mark within the string is converted into two single quotation marks.

Usage Format

The data type. The following options are available:

❏ Integer

❏ Character (fixed)

❏ Decimal

Procedure: ## How to Use a Variable in a Synonym

Variables in a synonym are commonly used to represent a table name or connection name.

1. In a synonym for a relational database where variables have been established, click a segment to open the Properties panel that shows the Adapter Specific attributes.

2. In the fields for TABLENAME and CONNECTION, enter the names of the variables to be used.

3. Click *Save* to save your changes.

Adding Indexes to a Synonym

Indexes are used to quickly locate data when a database table is queried, providing efficient access to ordered records. Indexes can be created using one or more columns of a database table.

When you use *New*, *Synonym* to create a synonym for a relational database table with more indexes than just the primary key, they will be included in the synonym.

For Db2 Web Query files, additional indexes are useful when you anticipate using a column in a JOIN operation, a WHERE condition, or as an argument to DB_LOOKUP. You can add multiple indexes. Each index can be on one or more columns. You can specify if the index is sorted, and whether it is a unique index.

Example: **Adding an Index to a Column**

1. Right-click the field name and click *Properties*.

2. In the Properties panel, select the I - Index check box.

3. Select *Rebuild Data Source* from the *Data* button on the *Home* tab to rebuild the data source.

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 or structure. You can create cluster joins by using the Metadata canvas Modeling View.

The Master File 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 and the Join information. It also shows how the tables are linked.

The total number of tables that you can add to the tool is 512 (using 511 joins), which results in a new Master File that has a maximum of 512 segments.

The Cluster Join tool enables you to create a Star Schema which consists of a fact table referencing a number of dimension tables. Optionally, you can also create a view that has more than one fact table.

Procedure: **How to Create a Cluster Join by Enhancing Existing Synonyms**

Use the Modeling View to enhance an existing synonym by adding a segment.

1. From the Data Servers area, open a synonym by double-clicking a Master File.

    The Metadata canvas opens.

2. Click the *Modeling View* tab.

    The Modeling View tab opens in the workspace.

3. Right-click a segment in the workspace and select *Insert*.

4. Insert tables (segments) through one of the methods listed:

    To insert a segment from an existing synonym as child or root:

    a. Select *Insert*, then *Reference to Existing Synonym as Child* or *Reference to Existing Synonym as Root*.

        The Insert Reference to Existing Synonym dialog box opens.

    b. Select a synonym to be inserted and click *Select*.

        Note: Use this method if you are creating a cluster join with an existing table or synonym.

        Tip: Click *Save As* from the Modeling View File menu if you do not want to modify the original synonym.

    To insert a segment via Metadata Import:

    a. Select *Insert*, then *Segment via Metadata Import*. This 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.

    b. When this option is selected, you are first presented with the Adapter dialog box where you can select a configured adapter connection to continue or configure a new adapter, if necessary.

    To insert a segment manually:

    Select *Insert*, then *Segment Manually*. This enables you to assign values to segment attribute fields.

    Note: Use this method if you are coding a new Master File, as you would for a Db2 Web Query data source.

    The segment is added in the Modeling View.

IBM

5. Right-click the join line between the segments.

   Note: This option is only available when using relational tables.

6. Select a *Join Type* from the shortcut menu.

The Join Condition field automatically creates a Join if identical fields exist in both segments.

Defining a Business View of a Master File

By defining a Business View of a Master File, you are creating an alternative view of the Master File and can limit the fields available or create a subset of fields from the original Master File. Fields can be grouped into meaningful folders. Field names, titles, and descriptions can be customized for each Business View.

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 for DEFINE, COMPUTE, 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.

When opening a Master File in the Synonym Editor and clicking the Business View button on the toolbar, a Business View pane opens inside the Synonym Editor where Business View attributes can be added. Adding Business View attributes at this point would update the Master File by inserting the Business View attributes at the end of the file, and once the file is saved, the Business View becomes the active view when the file is used for reporting.

If you do not wish to turn your Master File into a Business View, it is recommended that you save the Business View using a different name or preferably start by creating a new Master File by selecting *New*, then *Synonym via Metadata Canvas*. Then, insert a reference to an existing synonym and continue to build the Business View. In this case, the Business View points to the cross-referenced Master File, and all of the actual fields and security information comes from the referenced file when the Business View is used in Db2 Web Query tools (such as Joins and Defines) and in reports.

You may create a Business View for an existing Master File by using the Synonym Editor.

Procedure: How to Create a Business View Using the Metadata Canvas

1. From the Content group, click *Data* and then click *Synonym via Metadata Canvas*. Select a Master File and click *OK*.

2. Right-click a segment, point to *Insert*, and then click *Reference to Existing Synonym*.

3. From the Insert Reference to Existing Synonym dialog box, select the synonym for which you want to create a Business View.

The referenced file is added to the new synonym and its fields are visible in the left frame.

4. Select *Business View* from the Tools group to open the Business View panel.

   Note: A Business View Master File may contain only one root folder.

   Tip: Right-click a node in Business View and select *Properties* to open the Properties panel and view additional information for items selected in the Business View pane. Use the Properties pane to change titles, descriptions, or field names. Items that cannot be edited are grayed out.

5. In the Business View panel, right-click the file name and select *Create Default Business View* or *New Folder*.

   The *Create Default Business View* option duplicates the segments and fields that are available in the Master File. You can reorganize the view as necessary. The *New Folder* option creates a root folder to which you can add fields from the Master File on the left pane by selecting them and then dragging them inside the folder. You can create additional folders to create the structure you want.

6. To add additional folders for the Business View, right-click the Business View root folder that was created in the Business View pane and select *New Folder*.

   Note: Multiple subfolders can be created and folders may be empty for organizational purposes.

7. Select fields from the Master File on the left and drag them to the appropriate folder in the Business View pane. Press the Shift or Ctrl key while selecting multiple fields.

   Note: If needed, fields may be duplicated by placing them in multiple folders, but any given folder may contain a field only once.

   The selected fields appear in the Business View pane.

Reference: Usage Notes for Business Views

❏ When creating a Business View using a referenced Master File:

   ❏ The detailed information about fields, such as USAGE and ACTUAL formats or indexes remain in the referenced Master File.

   ❏ All information about Cluster Master Files remain in the referenced Master File.

   ❏ DBA attributes specified in the referenced Master File are respected by the Business Views

❏ When a Master File 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 purposes. 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-Db2 Web Query data sources support retrieval requests against a Business View.

❑ The referenced Master File or the Master File currently being used to create the Business View, may contain Defines, Computes, Filters, and other fields for use in the Business View.

Reference: Using a Business View Master File

When you use the Business View Master File with 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.

The Table List shows all available Master Files, including the Business Views, that are available. The Remarks column shows a description from the original Master File unless the Business View file contains its own remarks.

Applying Database Administrator (DBA) Security

You can secure Master Files on a file-by-file basis. For each data source, security can be maintained at two different levels.

❑ **Database Administrator Level.** You specify the Database Administrator (DBA) password for the data source. The DBA has unlimited access to the Master File and data source and can set up or change security restrictions for individual users. Only the Database Administrator can encrypt or decrypt a data source.

❑ **User Level.** You specify user passwords for the data source. A user password identifies a user who has access to that data source. When you specify a user password, you must also set at least the type of file access: read, write, read/write, or update. Security for each user can be further limited by restricting access to segments, fields, or field values. For more information, see *Limiting Data Source Access: The RESTRICT Attribute* on page 166. Once a user password has been established, you can apply the same restrictions to multiple users. For more information, see *Applying Security Restrictions for Multiple Users* on page 161.

Note: You cannot specify a Database Administrator (DBA) password during the Create Synonym process. You must use the Metadata canvas.

When security is specified, the Database Administrator or user must enter a password to get access to the data source. When the user no longer needs access to the data source, you can delete their security.

Before adding any type of security to a data source, the Database Administrator must be aware of certain DBA guidelines. See *DBA Guidelines* on page 159.

Procedure: **How to Set Up Security for the Database Administrator**

1. On the Metadata tab, from the Tools group, click the *DBA* button.

    The DBA pane opens, as shown in the following image.
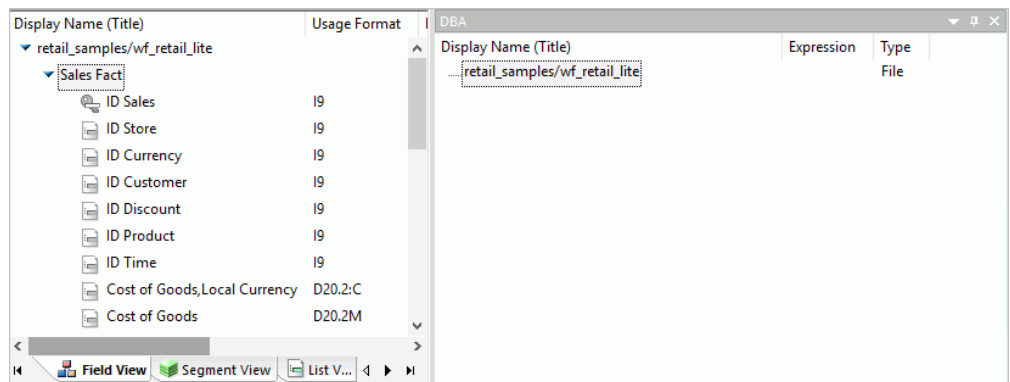
    Note: The DBA pane is available from the Field View, Segment View, List View, and Modeling View tabs.

2. Right-click the file name in the DBA window, point to *Insert*, then click *DBA*.

A default DBA password will be created for the Master File. You can change this value, delete it, add users to specify file restrictions, or add file names to specify data source-specific restrictions to the current data source. You can also specify a separate DBA file that contains DBA security restrictions.

Note: When the password is created and the cursor is in that field, you can right-click and use the edit options to undo, select all, cut, copy, paste, or delete the password.

Procedure: **How to Set Up Security for a User**

1. In the DBA pane, right-click the DBA icon to insert user restrictions or specify a DBA file.

2. Once you add a user, you can continue to insert file access restrictions by right-clicking the user field and selecting *Insert*.

3. Select the type of access: Read, Write, Read/Write, or Update.

4. Specify the type of restriction for each option: Restriction to Field, Value, Segment, Noprint, or Same Restriction.

   Note: The Same Restriction option is activated when there are multiple users.

5. Click *OK* to save the Master File with the user password and restrictions.

Reference: **DBA Guidelines**

You can ensure that the security restrictions you place on Master Files are correct by adhering to the following guidelines:

❏ Every file with access limits must have a DBA password.

❏ No segment, field, or field value restrictions may be specified at the Database Administrator level. The Database Administrator must have unlimited access to the data source and all cross-referenced data sources.

❏ Once security restrictions have been applied, the Database Administrator should conduct thorough testing of every restriction before the data source is used. It is particularly important to check field values to make sure they do not contain errors. If they are in error, user access to the field data will be unnecessarily restricted.

❏ All groups of cross-referenced data sources should have the same security restrictions.

❏ You must have a DBA password to encrypt and decrypt or restrict existing data sources.

❏ The Database Administrator can change any type of security restriction.

❏ Access levels affect the fields users can access. The Database Administrator must consider what commands each user will need. If a user does not have access rights, that user will receive a message.

Reference: **DBA Pane**

The following options are available from the DBA pane when the DBA password is selected.

DBA password

By default, the DBA password is the same as the user ID used to connect to the Reporting Server. Using the Rename option from the DBA password shortcut menu, you may enter a different password of up to 64 characters. This is the password of the DBA who will be creating and maintaining the current data source. The DBA has full access to the data source and the corresponding Master File, controls the access rights of other users, and has encryption privileges. For more information, see *Encrypting and Decrypting a Master File* on page 162.

DBAFILE

Select the name of the Master File that contains your DBA security restrictions. Other Master Files can use the DBA security restrictions in this Master File.

Insert Filename

Enter the name of the Master File to which user security will be applied. This option is used to add data source-specific restrictions to the current data source. It includes a FILENAME attribute for the selected Master File. The FILENAME attribute in the referenced Master File must be the same as the FILENAME attribute in the DBA section of the current data source.

Insert Users

Enter the passwords (up to 64 characters) of users whose access rights will be granted for the current data source.

File Access

For user access, select one of the following options:

❏ Select *Read Access* for full viewing rights.

❏ Select *Write Access* to permit additions or changes to the data source.

❏ Select *Read/Write Access* for both of the above.

❏ Select *Update Access* to permit changes to field values.

Restrictions: Segment, Field, Value, Noprint, Same

When the file access is selected, continue to select the type of restriction you wish to apply.

❏ Select *Segment* to grant access to individual segments.

❏ Select *Field* to grant access to individual fields.

❏ Select *Value* to limit access to values that meet a test condition.

❏ Select *Noprint* to specify fields you do not want to display in a report.

❏ Select *Same* to apply the same restrictions as other users that are already set up.

Applying Security Restrictions for Multiple Users

You can specify restrictions for one user and apply the same restrictions to other users. This helps when you want to set the same restrictions for a group of users.

Procedure: **How to Apply Previously Defined Restrictions to Another User**

1. In the DBA pane, right-click the DBA password, point to *Insert*, and then click *User*.

2. Right-click the newly added user and select *Insert* to specify the desired type of access restriction you would like to apply.

   Available access types are Read Access, Write Access, Read/Write Access, and Update Access.

3. Right-click an access type and select *Insert*, then *Same Restriction*.

   Note: The Same Restriction option is only available when there are multiple users. A drop-down combo box is activated in the Properties panel with a NAME attribute.

4. Click the arrow on the drop-down combo box next to the NAME attribute in the Properties panel, and then select the user with the security restrictions that would apply to the new user.

   Security restrictions from the user selected in the drop-down combo box are applied to the new user. You can apply the security restrictions to other users by repeating steps 1 to 4.

   Note: You must have created at least one user security restriction to apply security restrictions to multiple users.

Deleting a DBA or User Password

The DBA can delete a DBA password or security for a user when it is no longer needed.

Procedure: **How to Delete a User Password**

1. On the DBA pane, select the user password you wish to delete.

2. Right-click and select *Delete* or press *Delete* on the keyboard.

If you delete the user based upon whom you have assigned security restrictions for other users, you must reset security restrictions for all users attached to the user you deleted.

Procedure: **How to Delete a DBA Password**

Deleting a DBA password will delete all user security for that data source.

On the DBA pane select the DBA password, then right-click and select *Delete* or press the Delete key.

All security information is removed.

Encrypting and Decrypting a Master File

The DBA may use the Encrypt and Decrypt attributes from the Metadata canvas to scramble and unscramble some or all of the contents of a data source. When you encrypt Master Files, they are secure from unauthorized examination.

Encryption at the data source level scrambles the entire contents of that Master File so it is unreadable. When you encrypt a Master File, you can decrypt it. Decrypting unscrambles the contents to its readable state.

Before you can encrypt or decrypt any Master File, you must specify the DBA password. If you do not specify a DBA password, you will not be able to encrypt or decrypt the file.

Procedure: **How to Encrypt a Master File**

1. On the Metadata tab, from the Tools group, click the *DBA* button.

    The DBA pane opens.

2. Create and save the Master File with the DBA password.

3. From the Metadata canvas Field View tab, select a segment from the Master File hierarchy (left pane).

    The values for the selected segment appear in the Properties panel on the right.

4. Select the *ENCRYPT* check box.

5. Click *Save* from the File menu to encrypt the Master File.

Procedure: **How to Decrypt a Master File**

1. At the encrypted segment level in the Master File hierarchy, clear the *ENCRYPT* attribute.

2. Click *Save* from the File menu to decrypt the Master File.

Implementing DBA Security Using Db2 Web Query Language

The declarations (called security declarations) follow the END command in a Master File and tell Db2 Web Query that security is needed for the data source and what type of security is needed. The word END on a line by itself in the Master File terminates the segment and field attributes and indicates that the access limits follow. If you place the word END in a Master File, it must be followed by at least a DBA attribute. Each security declaration consists of one or several of the following attributes:

❑ The DBA attribute gives the name or password of the Database Administrator for the data source. The Database Administrator has unlimited access to the data source and its Master File.

❑ The USER attribute identifies a user as a legitimate user of the data source. Only users whose name or password is specified in the Master File of a Db2 Web Query data source with security placed on it have access to that data source.

❑ The ACCESS attribute defines the type of access a given user has. The four types of access available are:

RW, which allows a user to both read and write to a data source.

R, which allows a user only to read data in a data source.

W, which allows a user to only write new segment instances to a data source.

U, which allows a user only to update records in a data source.

❑ The RESTRICT attribute specifies certain segments or fields to which the user is not granted access. It can also be used to restrict the data values a user can see or perform transactions on.

❑ The NAME and VALUE attributes are part of the RESTRICT declaration.

Describe your data source security by specifying values for these attributes in a comma-delimited format, just as you specify any other attribute in the Master File.

Example: **Implementing DBA Security in a Master File**

The following is a Master File that uses security features:

```
FILENAME = PERS, SUFFIX = FOC,$
SEGMENT = IDSEG, SEGTYPE = S1,$
 FIELD = SSN            ,ALIAS = SSN    ,FORMAT = A9   ,$
 FIELD = FULLNAME       ,ALIAS = FNAME  ,FORMAT = A40  ,$
 FIELD = DIVISION       ,ALIAS = DIV    ,FORMAT = A8   ,$
SEGMENT=COMPSEG, PARENT=IDSEG, SEGTYPE=S1,$
 FIELD = SALARY         ,ALIAS = SAL    ,FORMAT = D8   ,$
 FIELD = DATE           ,ALIAS = DATE   ,FORMAT = YMD  ,$
 FIELD = INCREASE       ,ALIAS = INC    ,FORMAT = D6   ,$
END
DBA=JONES76,$
USER=TOM    ,ACCESS=RW, $
USER=BILL   ,ACCESS=R  ,RESTRICT=SEGMENT  ,NAME=COMPSEG    ,$
USER=JOHN   ,ACCESS=R  ,RESTRICT=FIELD    ,NAME=SALARY     ,$
                                           NAME=INCREASE   ,$
USER=LARRY  ,ACCESS=U  ,RESTRICT=FIELD    ,NAME=SALARY     ,$
USER=TONY   ,ACCESS=R  ,RESTRICT=VALUE    ,NAME=IDSEG,
    VALUE=DIVISION EQ 'WEST' ,$
USER=MARY   ,ACCESS=W  ,RESTRICT=VALUE    ,NAME=SALTEST,
    VALUE=INCREASE+SALARY GE SALARY,$
                                           NAME=HISTTEST,
    VALUE=DIV NE ' ' AND DATE GT 0,$
```

Identifying the DBA: The DBA Attribute

The first security attribute should be a password that identifies the Database Administrator. This password can be up to 64 characters long and, by default, is not case-sensitive. It can include special characters. If the DBA password contains blanks, it must be enclosed in single quotation marks ('). Since nothing else is needed, this line is terminated by the usual delimiter (,$).

Example: **Identifying the DBA Using the DBA Attribute**

```
DBA=JONES76,$
```

Identifying Users With Access Rights: The USER Attribute

The USER attribute is a password that identifies the users who have access to the data source. A USER attribute cannot be specified alone. It must be followed by at least one ACCESS restriction (discussed in *Specifying an Access Type: The ACCESS Attribute* on page 165) to specify what sort of ACCESS the user is granted.

Before using a secured data source, a user must enter the password using the SET PASS or SET USER command. If that password is not included in the Master File, the user is denied access to the data source. When the user does not have a password, or has one that is inadequate for the type of access requested, a message appears.

Syntax: **How to Set the USER Attribute**

Any user whose name or password is not declared in the Master File is denied access to that data source. The syntax of the USER attribute is

```
USER = name
```

where:
name

> Is a password of up to 64 characters for the user. The password can include special characters and is not case-sensitive. If the password contains blanks, it must be enclosed in single quotation marks (').

You can specify a blank password (default value if not previously changed). Such a password does not require the user to issue a SET PASS= command. A blank password may still have access limits and is convenient when a number of users have the same access rights.

Example: **Setting the USER Attribute**

```
USER=TOM,...
```

An example of setting a user password to blank, and access to read only follows:

```
USER= , ACCESS=R,$
```

Establishing User Identity

A user must enter his or her password before using any Db2 Web Query data source that has security specified for it. A single user may have different passwords in different files. For example, in file ONE, the rights of password BILL apply, but in file TWO, the rights of password LARRY apply. Use the SET PASS command to establish the passwords.

Specifying an Access Type: The ACCESS Attribute

The ACCESS attribute specifies what sort of access a user is granted. Every security declaration, except the DBA declaration, must have a USER attribute and an ACCESS attribute.

The following is a complete security declaration, consisting of a USER attribute and an ACCESS attribute.

```
USER=TOM, ACCESS=RW,$
```

This declaration gives Tom read and write (for adding new segment instances) access to the data source.

Access levels affect what kind of commands a user can issue. Before you decide what access levels to assign to a user, consider what commands that user will need. If a user does not have sufficient access rights to use a given command, a message appears.

ACCESS levels determine what a user can do to the data source. Use the RESTRICT attribute (discussed in *Limiting Data Source Access: The RESTRICT Attribute* on page 166) to limit the fields, values, or segments to which a user has access. Every USER attribute must be assigned an ACCESS attribute. The RESTRICT attribute is optional. Without it, the user has unlimited access to fields and segments within the data source.

Limiting Data Source Access: The RESTRICT Attribute

The ACCESS attribute determines what a user can do with a data source.

The optional RESTRICT attribute further restricts a user access to certain fields, values, or segments.

The RESTRICT=VALUE attribute supports those criteria that are supported by the IF phrase. The RESTRICT=VALUE_WHERE attribute supports all criteria supported in a WHERE phrase, including comparison between fields and use of functions. The WHERE expression will be passed to a configured adapter when possible.

Syntax: How to Limit Data Source Access

```
...RESTRICT=level, NAME={name|SYSTEM} [,VALUE=test],$
```

or

```
...RESTRICT=VALUE_WHERE, NAME=name,  VALUE=expression; ,$
```

where:

level

Can be one of the following:

❏ **FIELD,** which specifies that the user cannot access the fields named with the NAME parameter.

❏ **SEGMENT,** which specifies that the user cannot access the segments named with the NAME parameter.

❑ **SAME,** which specifies that the user has the same restrictions as the user named in the NAME parameter. No more than four nested SAME users are valid.

❑ **NOPRINT,** which specifies that the field named in the NAME or SEGMENT parameter can be mentioned in a request statement, but will not display. You can use a VALUE test to limit the restriction to values that satisfy an expression. For example, consider the following RESTRICT=NOPRINT declaration. User MARY can only display the IDs of those employees whose salaries are less than 10000.

```
USER=MARY    ,ACCESS=R  ,RESTRICT=NOPRINT  ,NAME=EMP_ID ,
    VALUE=CURR_SAL LT 10000;,$
```

name

Is the name of the field or segment to restrict. When used after NOPRINT, this can only be a field name. NAME=SYSTEM, which can only be used with value tests, restricts every segment in the data source, including descendant segments. Multiple fields or segments can be specified by issuing the RESTRICT attribute several times for one user.

Note: With value restrictions, NAME=segment restricts the named segment and any segment lower in the hierarchy, whether or not an alternate file view changes the retrieval view. This means that if a parent segment has a value restriction, and a join or alternate file view makes a child segment the new root, the value restriction on the original parent will still apply to the new root.

VALUE

Specifies that the user can have access to only those values that meet the test described in the *test* parameter.

test

Is the value test that the data must meet before the user can have access to it. The test is an expression supported in an IF phrase.

VALUE_WHERE

Specifies that the user can have access to only those values that meet the test described in the *expression* parameter.

expression;

Is the value test that the data must meet before the user can have access to it. The test is an expression supported in a WHERE phrase.

Note: The semicolon (;) is required.

Example: **Limiting Data Source Access**

```
USER=BILL ,ACCESS=R ,RESTRICT=SEGMENT ,NAME=COMPSEG,$
```

Placing Security Information in a Central Master File

The DBAFILE attribute enables you to place all passwords and restrictions for multiple Master Files in one central file. Each individual Master File points to this central control file. Groups of Master Files with the same DBA password may share a common DBAFILE which itself has the same DBA password.

There are several benefits to this technique, including:

❏ Passwords only have to be stored once when they are applicable to a group of data sources, simplifying password administration.

❏ Data sources with different user passwords can be JOINed. In addition, individual DBA information remains in effect for each data source in a JOIN.

The central DBAFILE is a standard Master File. Other Master Files can use the password and security restrictions listed in the central file by specifying its file name with the DBAFILE attribute.

Syntax: **How to Place Security Attributes in a Central Master File**

```
END
DBA=dbaname, DBAFILE=filename ,$
```

where:

dbaname

Is the same as the dbaname in the central file.

filename

Is the name of the central file.

You can specify passwords and restrictions in a DBAFILE that apply to every Master File that points to that DBAFILE. You can also include passwords and restrictions for specific Master Files by including FILENAME attributes in the DBAFILE.

Hiding Restriction Rules: The ENCRYPT Command

Since the restriction information for a Db2 Web Query data source is stored in its Master File, encrypt the Master File in order to prevent users from examining the restriction rules. Only the Database Administrator can encrypt a description. You must set PASS=DBA*name* before you issue the ENCRYPT command.

Syntax: ## How to Hide Restriction Rules: ENCRYPT Command

```
ENCRYPT FILE filename
```

where:

filename

Is the name of the file to be encrypted.

Example: ## Encrypting and Decrypting a Master File

The following is an example of the complete procedure:

```
SET PASS=JONES76
ENCRYPT FILE PERS
```

The process can be reversed in order to change the restrictions. The command to restore the description to a readable form is DECRYPT.

The DBA password must be issued with the SET command before the file can be decrypted. For example:

```
SET PASS=JONES76
DECRYPT FILE PERS
```

Encrypting Data

You may also use the ENCRYPT parameter within the Master File to encrypt some or all of its segments.

Encryption takes place on the segment level. That is, the entire segment is encrypted. The request for encryption is made in the Master File by setting the attribute ENCRYPT to ON.

Example: ## Encrypting Data

```
SEGMENT=COMPSEG, PARENT=IDSEG, SEGTYPE=S1, ENCRYPT=ON,$
```

You must specify the ENCRYPT parameter before entering any data in the data source. The message NEW FILE... must appear when the encryption is first requested. Encryption cannot be requested later by a change to the Master File and cannot be removed after it has been requested or any data has been entered in the data source.

Analyzing Procedures Using Impact Analysis

You can use Impact Analysis to generate a report that identifies the procedures that access a specific Master File or field within a Master File. Impact Analysis helps you analyze the potential impact of modifying or deleting Master Files or fields and 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 Db2 Web Query procedures. This enables you to see if changes in the original Master File will impact fields used in the Business View.

Impact Analysis is accessible from the Repository and Data Servers development areas of the Configured Environments tree.

Db2 Web Query customers should launch the Impact Analysis from the Repository node on the Explorer tree. The search will be performed on the repository folders where Web Query procedures reside. DataMigrator customers should launch the Impact Analysis from the Data Servers node or Metadata canvas. The search will be performed on the Reporting Server application path where DataMigrator for i flows reside.

Procedure: How to Use Impact Analysis From the Configured Environments Tree

1. You can use Impact Analysis from the Configured Environments tree in the following ways:

   ❑ From the Repository area, right-click a folder and then click *Impact Analysis* to limit the search to the selected folder and its subfolders.

   ❑ Right-click the *Repository* node and then click *Impact Analysis* to search all folders within that node.

   ❑ From the Data Servers area, expand an application, right-click a Master File, and then click *Impact Analysis* from the shortcut menu. This method is for DataMigrator customers.

   The Impact Analysis canvas 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:

   ❏ **Selected Master File.** The Master File you selected when opening the canvas will be searched, by default. Proceed to step 3.

   ❏ **Different Master File.** Click the ellipsis button (...) to the right of the File Name search field and select a different Master File in the Select a Master File dialog box that opens.

   ❏ **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 shortcut menu.

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 Select 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 Result pane.

   The following image shows the New Report tab of the Impact Analysis canvas populated with File Name search criteria, multiple Search Paths, and a report displayed in the Impact Analysis Result pane.
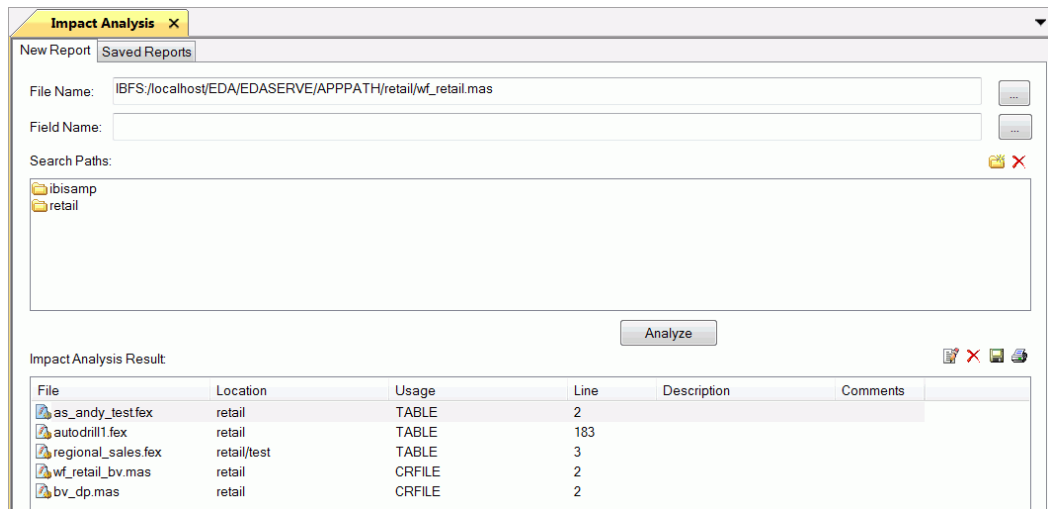
You have options to edit procedures, delete a report item, export a report, and print the report.

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\DevWorkbench82\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:

`drive:\ibi\DevWorkbench82\bin\IARepository.xml`

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.

Note: You can select an Impact Analysis report and click the Report Description icon to display details of the report.

Creating Reports

Reports are the foundation of business intelligence and business analytics, enabling you to gain valuable insights and make improved business decisions. In Developer Workbench, you can create and edit reports using the Report canvas. The Report canvas is driven by the same InfoAssist+ tool that is available in the Db2 Web Query browser interface. When you create a new report, you use the Report Wizard. In report mode, you can create and style simple or complex reports, add data to the Report canvas, and style that data creating a graphical representation of the report page. This allows you to view how the report displays at run time.

This topic provides an overview of reporting, the types of reports you can create with Developer Workbench, how to launch the Report Wizard to create reports, and a sample GUI procedure, with its associated Db2 Web Query syntax, that you can use to create each type of report.

For information on the Report Wizard, see *Launching the Report Wizard* on page 173.

In this chapter:

❏   Launching the Report Wizard

Launching the Report Wizard

The Report Wizard allows you to create a new report procedure or SQL report procedure, and open an existing report.

To open the Report Wizard:

❏   On the *Home* tab, in the *Content* group, click *Report*.

❏   In the Environments Tree panel, right-click an application folder, point to *New*, and click *Report*.

   Note: If you choose to create a new report using the shortcut menu in the Environments Tree panel, you will skip the first screen of the Report Wizard. You will not have to specify whether you are creating a report or SQL report. You will instead be brought to the Select Data Source window, with the location for your report already selected.

The Report Wizard opens, as shown in the following image.
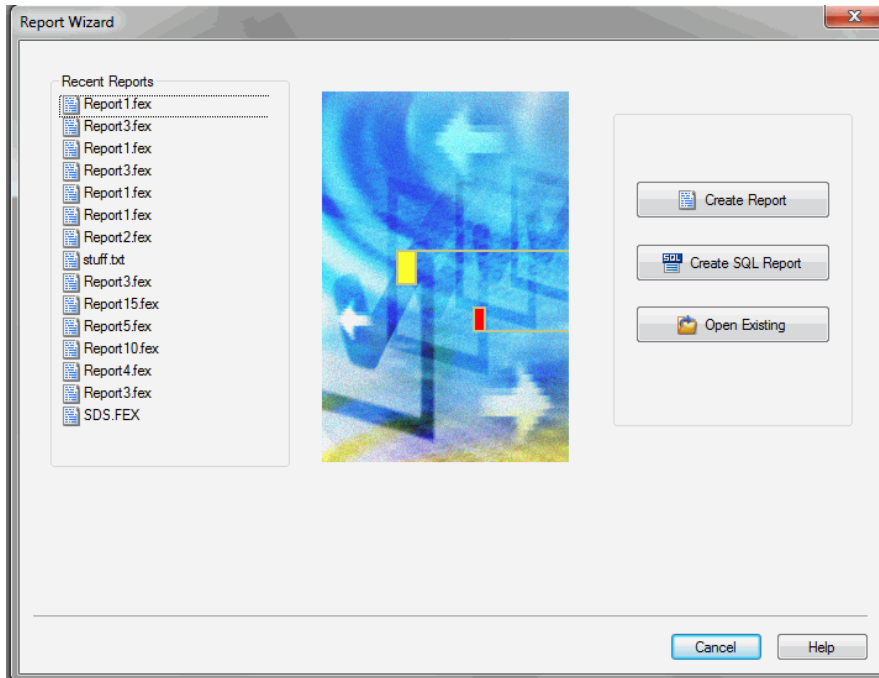
From the Report Wizard, you can create a new report or SQL report procedure or open a recent procedure.

Procedure: How to Create a Report Using the Report Wizard

After you launch the Report Wizard, you can begin to create your report procedure.

Note: If you choose to create a new report using the shortcut menu in the Environments Tree panel, you will skip the first screen of the Report Wizard. You will not have to specify whether you are creating a report or SQL report. You will instead be brought to the Select Data Source window, with the location for your report already selected.

1. Click *Create Report*.

    The Report Wizard - Select Procedure Location window opens.

2. Navigate to where you want to create the new report and click *Next*.

    The Select Data Source window opens.

3. Choose an application folder in the application folder tree, and select a Master File in the Master files list. You can select the *Use Qualified Path* check box if you only want to display the data sources in the application folder and generate a qualified file name reference.

4. Click *Finish*.

   The Report canvas opens.

5. Double-click fields in the Data pane, or drag them on to the canvas to add them to the report.

6. Format your report by using the options on the Report, Format, Data, Layout, View, and Field tabs.

7. To save your report, click *Save* on the Quick Access Toolbar, or click *Save* or *Save As* from the Application menu.

8. To run your report, click *Run* on the Quick Access Toolbar, or click *Run* from the Application menu.

Procedure: **How to Create an SQL Report From an External .sql File**

After you launch the Report Wizard, you can begin to create your SQL report procedure. This enables you to browse and select external procedures that exist in the domain.

Note: If you choose to create a new SQL report using the shortcut menu in the Environments Tree panel, you will skip the first screen of the Report Wizard. You will not have to specify whether you are creating a report or SQL report. You will instead be brought to the SQL Report Wizard - Welcome window.

1. Click *Create SQL Report*.

   The Report Wizard - Choose location for the new SQL Report window opens.

2. Select a location for your SQL report and click *Next*.

   The SQL Report Wizard - Welcome window opens.

3. Click the *Included from an external '.sql' file* option.

4. Click *Next*.

   The SQL Report Wizard - Data access information window opens.

5. In the Select the SQL database engine area, select a database from the drop-down list. The drop-down list changes depending on your server configuration.

6. In the Select connection area, choose a connection from the drop-down list generated from the engine you selected.

7. Click *Next*.

   The SQL Report Wizard - Include external SQL file window opens.

8. Click *Browse* to select an external SQL file name or type the external SQL file name in the field.

   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. The default limit is 100.

9. Click *Next*. The SQL Report Wizard - Summary of SQL options window opens. Review your options and click *Next*, which opens the Report Wizard - SQL Data Source window.

10. Navigate to where you want to create the new procedure.

11. In the Enter a Procedure Name field, type a name for the SQL report.

   Note: Typing a procedure name is not required to create a new report. If you do not type a procedure name in the Enter a Procedure Name field, a default procedure name will be provided until you save the procedure.

12. Click *Finish*.

   The Report canvas opens. You can now add data to your report and format the data using the tabs on the ribbon.

Procedure: **How to Pass SQL Commands to the RDBMS Using SQL Passthru**

After you launch the Report Wizard, you can begin to create your SQL report procedure by typing SQL commands that will be passed on to the RDBMS with the SQL Passthru feature.

Note: If you choose to create a new SQL report using the shortcut menu in the Environments Tree panel, you will skip the first screen of the Report Wizard. You will not have to specify whether you are creating a report or SQL report. You will instead be brought to the *Report Wizard - Choose location for the new SQL Report* window, with the location for your report already selected.

1. Click *Create SQL Report*.

   The Report Wizard - Choose location for the new SQL Report window opens.

2. Select a location for your SQL report and click *Next*.

   The SQL Report Wizard - Welcome window opens.

3. Click *Type SQL statements in the report request*.

4. Click *Next*.

   The SQL Report Wizard - Data access information window opens.

5. In the Select the SQL database engine area, select a database from the drop-down list. The drop-down list changes depending on your server configuration.

6. In the Select the connection area, choose a connection from the drop-down list generated from the engine you selected.

7. Click *Next*.

   The SQL Report Wizard - Enter SQL statements window opens.

8. In the field box, type the SQL statements you want to pass to the RDBMS.

   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. The default limit is 100.

9. Click *Next*.

   The SQL Report Wizard - Summary of SQL options window opens.

10. Review your options and click *Next*.

    The Report Wizard - SQL Data Source window opens.

11. Navigate to where you want to create the new procedure.

12. In the Enter a Procedure Name field, type a name for the SQL report.

    Note: Typing a procedure name is not required to create a new report. If you do not type a procedure name in the Enter a Procedure Name field, a default procedure name will be provided until you save the procedure.

13. Click *Finish*.

    The Report canvas opens. You can now add data to your report and format the data using the tabs on the ribbon.

Procedure: **How to Import SQL Commands From an Existing .sql File**

You can import commands from existing .sql files. This enables you to modify SQL code after importing it from an external file to the procedure being built. You can modify the request using bits of the code.

Note: If you choose to create a new SQL report using the shortcut menu in the Environments Tree panel, you will skip the first screen of the Report Wizard. You will not have to specify whether you are creating a report or SQL report. You will instead be brought to the *Report Wizard - Choose location for the new SQL Report* window, with the location for your report already selected.

1. Click *Create SQL Report*.

   The Report Wizard - Choose location for the new SQL Report window opens.

2. Select a location for your SQL report and click *Next*.

    The SQL Report Wizard - Welcome window opens.

3. Click *Import from an existing '.sql' file* and click *Next*.

    The SQL Report - Data access information window opens.

4. In the Select SQL database engine area, select a database from the drop-down list. The drop-down list changes depending on your server configuration.

5. In the Select the connection area, choose a connection from the drop-down list generated from the engine you selected.

6. Click *Next*.

    The SQL Report Wizard - Import external SQL file window opens.

7. Type the SQL file name that you want to import or click *Browse* to select it.

    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. The default limit is 100. You can edit these options here or in the next step.

8. Click *Next*.

    The SQL Report Wizard - Enter SQL statements window opens.

9. In the field box, type the SQL statements you want to pass to the RDBMS.

10. Click *Next*.

    The SQL Report Wizard - Summary of SQL options window opens.

11. Review your options and click *Next*.

    The Report Wizard - SQL Data Source window opens.

12. Navigate to where you want to create the new procedure.

13. In the Enter a Procedure Name field, type a name for the SQL report.

    Note: Typing a procedure name is not required to create a new report. If you do not type a procedure name in the Enter a Procedure Name field, a default procedure name will be provided until you save the procedure.

14. Click *Finish*.

    The Report canvas opens. You can now add data to your report and format the data using the tabs on the ribbon.

Chapter **5**

Creating Charts and Visualizations

A chart often conveys meaning more clearly and effectively than data displayed in tabular form. A chart enables you to visually communicate quantitative information. On a chart, you can give data a shape and form, and reveal patterns and relationships among many data values. A chart can highlight anomalies that require further investigation.

It is important that you choose a chart that is appropriate for your data. Developer Workbench provides a complete chart library of both basic and advanced charts. You can choose from a wide variety of charts to best represent the data that you want to display.

In Developer Workbench, you can create and edit charts using the Chart canvas. The Chart canvas is driven by the same InfoAssist+ tool that is available in the Db2 Web Query browser interface. When you create a new chart, you use the Chart Wizard.

This topic provides an overview of the types of charts you can create with Db2 Web Query Developer Workbench and how to launch the Chart Wizard to create charts.

For more information on creating visualizations, click the Help icon , located in the upper-right corner of the canvas.

In this chapter:

❏ Launching the Chart Wizard

❏ Creating Visualizations

Launching the Chart Wizard

To open the Chart Wizard:

❏ On the *Home* tab, in the *Content* group, click *Chart*.

❏ In the Environments Tree panel, right-click an application folder, point to *New*, and click *Chart*.

Note: If you choose to create a new chart using the shortcut menu in the Environments Tree panel, you will skip the first screen of the Chart Wizard. You will not have to specify whether you are creating a chart or SQL chart. You will instead be brought to the Select Data Source window, with the location for your chart already selected.

The Chart Wizard opens, as shown in the following image.
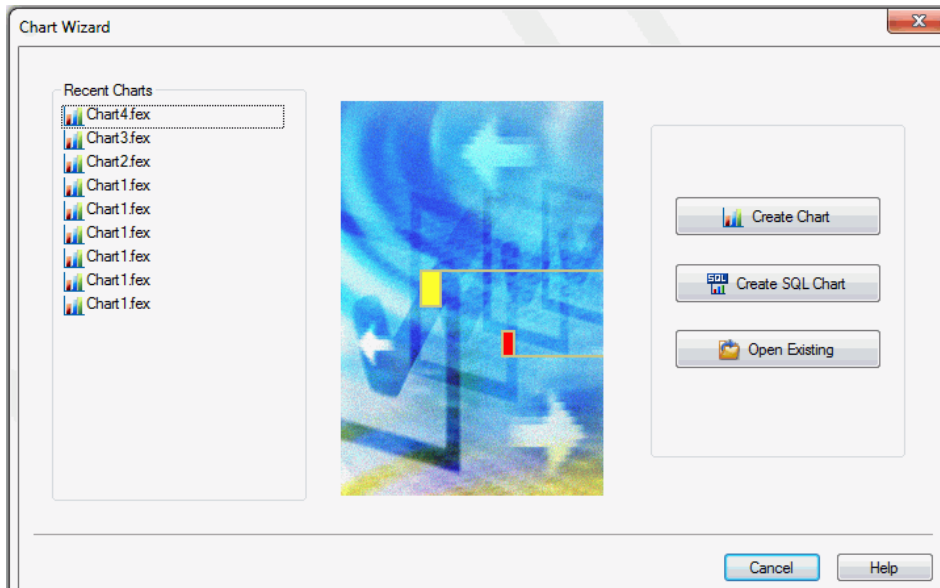
From the Chart Wizard, you can create a new chart or SQL chart procedure or open a recent procedure.

Procedure: **How to Create a Chart Using the Chart Wizard**

After you launch the Chart Wizard, you can begin to create your chart procedure.

Note: If you choose to create a new chart using the shortcut menu in the Environments Tree panel, you will skip the first screen of the Chart Wizard. You will not have to specify whether you are creating a chart or SQL chart. You will instead be brought to the Select Data Source window, with the location for your chart already selected.

1. Click *Create Chart*.

   The Chart Wizard - Choose location for the new chart window opens.

2. Navigate to where you want to create the new chart and click *Next*.

   The Select Data Source window opens.

3. Choose an application folder in the application folder tree, and select a Master File. You can select the *Use Qualified Path* check box if you want to only display the data sources in the application folder.

4. Click *Finish*.

   The Chart canvas opens.

5. Double-click fields in the Data Panel to add them to the chart.

   You can use the features and functions on the ribbon that is located in the Chart canvas to format the chart.

6. To save your chart, click the *Save* button on the Developer Workbench Quick Access Toolbar, or select *Save* from the Application menu.

7. To run your chart, click the *Run* button on the Developer Workbench Quick Access Toolbar.

Procedure: **How to Create an SQL Chart From an External .sql File**

After you launch the Chart Wizard, you can begin to create your SQL chart procedure.

Note: If you choose to create a new SQL chart using the shortcut menu in the Environments Tree panel, you will skip the first screen of the Chart Wizard. You will not have to specify whether you are creating a chart or SQL chart. You will instead be brought to the SQL Chart Wizard - Welcome window, with the location for your chart already selected.

1. Select *Create SQL Chart*.

   The Chart Wizard - Choose location for the new chart window opens

2. Navigate to where you want to create the new chart and click *Next*.

   The SQL Chart Wizard - Welcome window opens.

3. Click the *Included from an external '.sql' file* option. This enables you to browse and select external procedures that exist in the domain of the project.

4. Click *Next*.

   The SQL Chart Wizard - Data access information window opens.

5. In the Select the SQL database engine area, select a database from the drop-down list. The drop-down list changes depending on your Reporting Server configuration.

6. In the Select the connection area, choose a connection from the drop-down list generated from the engine you selected.

7. Click *Next*.

   The SQL Chart Wizard - Include external SQL file window opens.

8. Click *Browse* to select an external SQL file name or type the external SQL file name in the field.

   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 *Next*.

   The SQL Chart Wizard - Summary of SQL options window opens.

10. Review your options and click *Finish*.

    The Chart Wizard - SQL Data Source window opens.

11. Navigate to where you want to create the new procedure.

12. In the Enter a Procedure Name field, type a name for the SQL chart.

    Note: Typing a procedure name is not required to create a new chart. If you do not type a procedure name in the Enter a Procedure Name field, a default procedure name will be provided until you save the procedure.

13. Click *Finish*.

    The Chart canvas and the Procedure View panel open. You can now add data to your chart and format the data using the tabs in the ribbon.

Procedure: How to Pass SQL Commands to the RDBMS Using SQL Passthru

After you launch the Chart Wizard, you can begin to create your SQL chart procedure.

Note: If you choose to create a new SQL chart using the shortcut menu in the Environments Tree panel, you will skip the first screen of the Chart Wizard. You will not have to specify whether you are creating a chart or SQL chart. You will instead be brought to the *Chart Wizard - Choose location for the new SQL Chart* window, with the location for your chart already selected.

1. Click *Create SQL Chart*.

   The Chart Wizard - Choose location for the new SQL Chart window opens.

2. Navigate to where you want to create the SQL chart and click *Next*.

   The SQL Chart Wizard - Welcome window opens.

3. Click *Type SQL statements in the chart request*. This enables you to type SQL commands that will be passed on to the RDBMS with the SQL Passthru feature.

4. Click *Next*.

   The SQL Chart Wizard - Data access information window opens.

5. In the Select the SQL database engine area, select a database from the drop-down list. The drop-down list changes depending on your Reporting Server configuration.

6. In the Select the connection area, choose a connection from the drop-down list generated from the engine you selected.

7. Click *Next*.

   The SQL Chart Wizard - Enter the SQL statements window opens.

8. In the field box, type the SQL statements you want to pass to the RDBMS.

   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. The default limit is 100.

9. Click *Next*.

   The SQL Chart Wizard - Summary of SQL options window opens.

10. Review your options and click *Finish*.

    The Chart Wizard - SQL Data Source window opens.

11. Navigate to where you want to create the new procedure.

12. In the Enter a Procedure Name field, type a name for the SQL chart.

    Note: Typing a procedure name is not required to create a new chart. If you do not type a procedure name in the Enter a Procedure Name field, a default procedure name will be provided until you save the procedure.

13. Click *Finish*.

    The Chart canvas and the Procedure View panel open. You can now add data to your chart and format the data using the tabs in the ribbon.

Procedure: **How to Import SQL Commands From an Existing .sql File**

You can import commands from existing .sql files.

Note: If you choose to create a new SQL chart using the shortcut menu in the Environments Tree panel, you will skip the first screen of the Chart Wizard. You will not have to specify whether you are creating a chart or SQL chart. You will instead be brought to the *Chart Wizard - Choose location for the new chart* window, with the location for your chart already selected.

1. Open the Chart Wizard and click *Create SQL Chart*.

   The Chart Wizard - Choose location for the new chart window opens.

2. Navigate to where you want to create the new chart and click *Next*.

   The SQL Chart - Welcome window opens.

3. Click *Import from an existing '.sql' file* and click *Next*. This enables you to modify SQL code after importing it from an external file to the procedure being built. You can modify the request using bits of the code.

   The SQL Chart - Data access information window opens.

4. In the Select the SQL database engine area, select a database from the drop-down list. The drop-down list changes depending on your Reporting Server configuration.

5. In the Select the connection area, choose a connection from the drop-down list generated from the engine you selected.

6. Click *Next*.

    The SQL Chart Wizard - Import external SQL file window opens.

7. Type the SQL file name that you want to import or click *Browse* to select it.

    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. The default limit is 100. You can edit these options here or in the next step.

8. Click *Next*.

    The SQL Chart Wizard - Enter SQL statements window opens.

9. In the field box, type the SQL statements you want to pass to the RDBMS.

10. Click *Next*.

    The SQL Chart Wizard - Summary of SQL options window opens.

11. Review your options and click *Next*.

    The Chart Wizard - SQL Data Source window opens.

12. Navigate to where you want to create the new procedure.

13. In the Enter a Procedure Name field, type a name for the SQL chart.

    Note: Typing a procedure name is not required to create a new chart. If you do not type a procedure name in the Enter a Procedure Name field, a default procedure name will be provided until you save the procedure.

14. Click *Finish*.

    The Chart canvas and the Procedure View panel open. You can now add data to your chart and format the data using the tabs in the ribbon.

Creating Visualizations

In Developer Workbench, you can create and build visualizations. Visualizations centralize information by providing different views of data that are pertinent to a particular objective (for example, reviewing trends or fluctuations in data over a period of time or within a region).

A visualization provides you with a quick glance of information on a single screen. Visualizations support the use of different types of charts, maps, and grids. For example, you may want to use a bar, pie, and line chart to show different views of the same data. Alternatively, you may want to offset a particular visual by showing other types of related data that employ a different type of visual. You can also add a text cell to your visualization to provide explanatory text or information that other users can reference. Visualizations allow you to monitor changes in data. They also serve to provide information in real-time, based on changes in underlying data or other components. A visualization can be updated, changed, or revised at any time to account for shifts in data needs.

Using InfoAssist+

You can create visualizations from the Repository node in the Environments Tree panel or Environments Detail panel.

To access the online Help for visualizations, click the *Help* icon , located in the upper-right corner of the canvas.

Procedure: How to Create a New Visualization

1. From the Repository node in the Environments Tree panel or Environments Detail panel, right-click a folder, point to *New*, and click *Visualization*.

2. Select a Master File from the list and click *Open*.

The Db2 Web Query InfoAssist+ interface opens, as shown in the following image, where you can create charts, maps, and grids to visually represent your data.
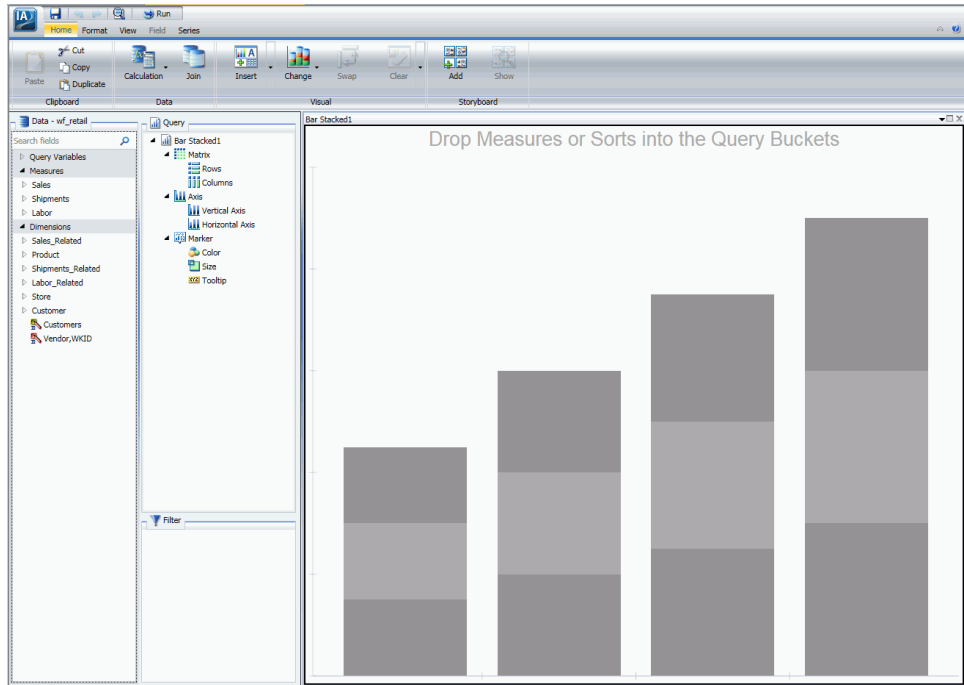
The following image shows a visualization that contains multiple visuals, including a bubble, gauge, ring pie, and treemap.
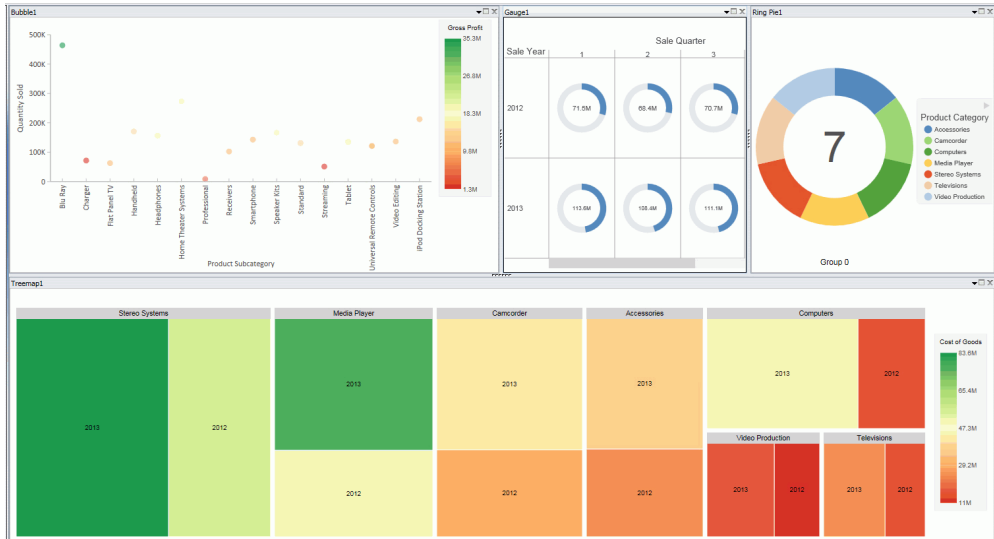
Note: For more information on creating visualizations, click the Help icon 🔵, located in the upper-right corner of the canvas.

Creating HTML Pages

In Developer Workbench, you can create and edit HTML pages using the HTML canvas. When you create a new HTML page, you use the HTML/Document Wizard.

This topic describes how to launch the HTML/Document Wizard to create HTML pages. It also identifies and explains the tabs and panels that are available when you are developing HTML pages in the HTML canvas.

In this chapter:

❑ Launching the HTML/Document Wizard

❑ Accessing HTML Page Components

❑ Using Parameters

❑ Viewing Object Attributes

❑ Modifying Object Population Settings

❑ Using Tasks & Animations

❑ Working with Requests & Data Sources

❑ Specifying Browser Defaults

❑ Chaining in the HTML Canvas

❑ Using JavaScript Code with HTML Canvas Pages

❑ Creating Responsive Web Pages

❑ Designing Content for Smartphones

❑ Cascading Style Sheet Class Mapping List

Launching the HTML/Document Wizard

The HTML/Document Wizard allows you to create a new HTML Page, or open a recent procedure.

To open the HTML/Document Wizard:

❑ On the *Home* tab, in the *Content* group, click *HTML/Document*.

❑ In the Environments Tree panel, right-click an application folder, point to *New*, and click *HTML File*.

Note: If you choose to create a new HTML page using the shortcut menu in the Environments Tree panel, your screen will not include a Navigator panel since the location for your HTML page is already selected.

The HTML/Document Wizard opens, as shown in the following image.
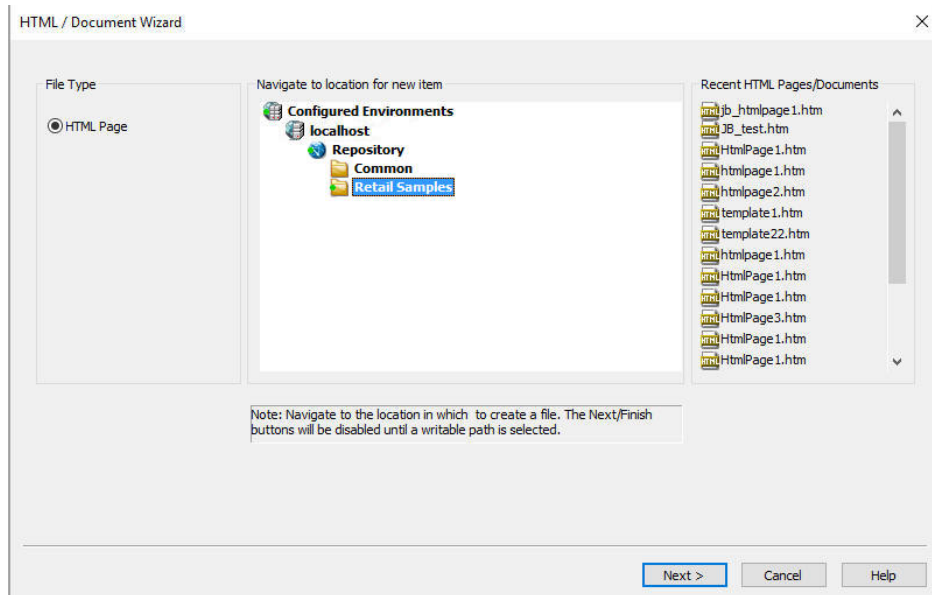
From the HTML/Document Wizard, you can open a recent HTML Page/Document or create a new one.

Procedure: How to Create an HTML Page Using the HTML/Document Wizard

After you have launched the HTML/Document Wizard, you can begin to create your HTML Page.

1. Navigate to where you want to create the new procedure or select a recent procedure and click *Next*.

   Note:

   ❑ The Next button is available after you select a valid path.

   ❑ If you used the shortcut menu in the Environments Tree panel to create your HTML page, a location is already selected.

2. Optionally, you can add themes and settings to your HTML page.

3. Click *Finish*.

   The HTML canvas, File/Folder Properties panel, Properties panel, Settings panel, Requests & Data sources panel, and Tasks & Animations panel open. You can now add data to your HTML page and format the data using the tabs in the ribbon.

4. To save your HTML page, click the *Save* button on the Quick Access Toolbar or select *Save* from the Application menu.

5. To run your HTML page, click *Run* on the Quick Access Toolbar or click *Run* from the Application menu.

Accessing HTML Page Components

The following topics describe the tabs and panels that are available when you are developing HTML pages in the HTML canvas.

Inserting Components in an HTML Page Using the Components Tab

You can add a variety of components to an HTML page found in the command groups in the Components tab. For example, suppose you want to add a button to your page that a user can click to refresh the data. The button is a component. You select it from the Generic Elements command group in the Components tab, then add it to your page as desired.

The Components tab contains the Reports, Generic Elements, Containers, and Objects groups, as shown in the following image.
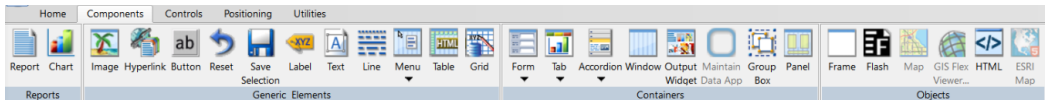
When you click a command from a ribbon, your mouse pointer turns into a cross-hair to let you draw the object in the HTML canvas. If you click a command in error, press the Escape key to return your mouse to a pointer.

Inserting a Report or Chart in an HTML Page Using the Reports Group

From the Reports group, you can add a report or chart to your HTML page. The Reports group is shown in the following image.
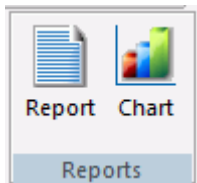
The commands in the Reports group are:

Report

Inserts a report object. You can add reports to the HTML canvas that will display when you run the layout. Reports are added by referencing existing reports in the repository.

You can also include parameters in a report whose values can be assigned with controls that are added with the HTML canvas.

Chart

Inserts a chart object. You can add charts to the HTML canvas that will display when you run the layout. Charts are added by referencing existing charts in the repository.

Procedure: How to Add a New Report or Chart to an HTML Page

1. Insert a report or chart object by doing one of the following:

   ❑ On the *Components* tab, in the *Reports* group, click the *Report* or *Chart*.

   ❑ Right-click in the layout and select *New Report* or *New Chart* from the shortcut menu.

   The pointer changes into a crosshair.

2. Drag the crosshair to create a report or chart object and adjust it to the size that you want.

   A report or chart object is created in the layout and assigned the name report*(n)* or chart*(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 chart associated with it. Once a report or chart is associated with the object, the object displays the contents of the report or chart if live or simulated data is active (live data is the default) or a colored placeholder if preview is off in the HTML Page tab, located in the Developer Workbench Options dialog box.

3. Create a report or chart by doing one of the following:

   ❑ Double-click the placeholder.

      or

   ❑ Right-click the placeholder and select *New Report* for a report, or *New Chart* for a chart.

   The Open File dialog box appears.

4. Select the Master File you want to use and click *OK*.

   The Report canvas opens for reports and the Chart canvas opens for charts.

5. Optionally, after creating the report or chart, you can change its properties by adjusting the properties displayed in the Properties panel.

Procedure: How to Add an Existing Report or Chart to a Layout

1. Insert a report or chart object by doing one of the following:

   ❑ On the *Components* tab, in the *Reports* group, click *Report* or *Chart*.

❏ Right-click in the layout and select *New Report* or *New Chart* from the shortcut menu.

The pointer changes into a crosshair. Drag the crosshair to create a report or chart object and adjust it to the size you want.

A report or chart object is created in the layout. The object will appear in gray and white to indicate that the placeholder does not have a report or chart associated with it. Once a report or chart is associated with the object, the object displays the contents of the report or chart if live or simulated data is active (live data is the default) or a colored placeholder if preview is off in the HTML Page tab, located in the Developer Workbench Options dialog box.

2. Right-click the report or chart.

❏ Select *Reference existing procedure*.

The Open File dialog box appears.

3. Select the procedure you want to add to the layout.

4. Click *OK*.

The report or chart object appears in the Design view of the HTML canvas.

Adding Basic Elements to an HTML Page Using the Generic Elements Group

You can add a variety of basic elements. The Generic Elements group is shown in the following image.
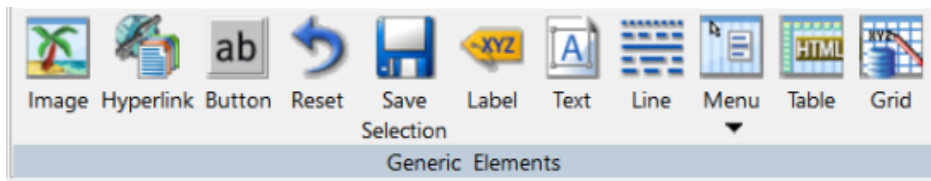
The commands in the Generic Elements group are:

Image

Inserts an image. 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. After you run your report and click the image, you can launch a URL or run a report the same way by clicking a hyperlink or push button.

Note: When inserting images, images must be referenced from a specific directory location.

Hyperlink

Inserts a hyperlink.

Button

Inserts a button. A push button enables you to execute a report or link to a URL or HTML file. This behavior is similar to a hyperlink.

Note: You can either double click to change the text or can use the Text property in the Properties panel.

Reset

Inserts a reset button. A reset button enables you to revert the entire page back to its initial settings.

Save Selection

Inserts a Save Selection button. At run time, the user can select given parameters and save them in a snapshot by clicking the Save Selection button. This creates a static HTML file in the same folder as the source file.

The static Save Selection files are not editable in the HTML canvas or in the text editor. To verify that a file is a Save Selection file:

1. Right-click the file in the Configured Environments tree and select *Properties* from the shortcut menu.

    The File/Folder Properties panel opens.

2. The value for the Properties attribute is *tool=saveparam*. This indicates that the file is not editable in the HTML canvas or in the text editor.

Label

Inserts a label. The label component lets you create and name a label, and link it to a control. To link a label to a control, you can select a value from the HtmlFor property drop-down list in the Properties panel, or you can select the *Bind label to* option from the shortcut menu, when the label and control are both selected.

Text

Inserts a text box. You can add text to the layout. This is useful for including headings for your webpage, or adding directions or explanations for your report or chart.

Line

Inserts a line. You can add a horizontal or vertical line to the layout. This is useful for distinguishing between sections of your launch or display page.

Menu

Inserts a menu. You can add a horizontal or vertical menu to the layout.

Table

Inserts a table.

Grid

Inserts a grid.

Procedure: **How to Format Text in an HTML Page**

You may apply various formatting and style options to words and individual text characters within the text object.

Note: Any formatting and styling that you have applied to individual text strings within the text object will remain unchanged. Changes made to the entire text object are only applied to part of the text string that has not been formatted.

1. On the *Components* tab, in the *Generic Elements* group, click *Text*.

   The pointer changes to a crosshair.

2. Drag your pointer across the canvas to create a text object.

   A text object with the default text, *Enter text*, is created.

3. Select the text that you wish to format:

   ❏ To format the entire text object, single-click the text object in the layout.

   ❏ To format an individual word or text character, highlight part of the text within the text object.

4. Right-click the text, point to *Style*, and then click *Font*.

   The Style Composer dialog box opens with Font selected automatically.

   Note: You can also access font formatting options in the Properties panel.

5. Select from the formatting options available. You can change the type, style, color, size, and effect of the font.

   Note: When you enter a *Specific* font size, the unit defaults to px (pixels).

6. Click *OK* to close the Style Composer dialog box.

The formatting options that you selected are applied to the text.

Procedure: **How to Insert a Bulleted List or Numbered List Into a Text Element**

To insert a bulleted list or numbered list into a text object:

1. On the *Components* tab, in the *Generic Elements* group, click *Text*.

   The pointer changes to a crosshair.

2. Drag your pointer across the canvas to create a text object.

   A text object with the default text, *Enter text*, is created.

3. Add multiple lines of text to the text object.

4. Highlight and right-click the text that you want to include in the list.

5. In the shortcut menu, select one of the following list options:

   ❏ **Bullets**

      ❏ Disc

      ❏ Circle

      ❏ Square

      ❏ None

   ❏ **Numbering**

      ❏ Numbers

      ❏ Lowercase Letters

      ❏ Uppercase Letters

      ❏ Small Roman numerals

      ❏ Large Roman numerals

      ❏ None

Note:

❏ Alternatively, you can select a bullet type before typing text to begin the list. Pressing enter will begin the next item in the list on a separate line.

❏ To change the bullet or number list type of an existing list, place your pointer on the list level you want to change and reselect a bullet or number list type. Selecting *None* will remove the bullets or numbers for that level and move any nested lists up one level. In order to switch between bullets and numbers, you must first remove the current list option by selecting *None* and then applying the list option you want.

Procedure: **How to Insert Nested Lists Into a Text Element**

To insert a nested list into the text object:

1. On the *Components* tab, in the *Generic Elements* group, click *Text*.

   The pointer changes to a crosshair.

2. Drag your pointer across the canvas to create a text object.

   A text object with the default text, *Enter text*, is created.

3. Add a list to the text object. For more information, see *How to Insert a Bulleted List or Numbered List Into a Text Element* on page 196.

4. Place your pointer after a list item.

5. Right-click, point to *Nested List* and then select a bulleted or numbered list option.

   A list is started within the current list, allowing you to enter text on that list level.

   Note: Pressing Tab while your pointer is on the same line as a list item will move that item one level down, resulting in a nested list. The bullet or number type selected is the next list type in the right-click shortcut menu. For example, if you have a bulleted list that uses the disc bullet type, pressing Tab to move an item down one level will cause that nested list to have the circle bullet type.

   You can continue to nest lists within other lists by using the same steps shown above.

   Note: You cannot skip a list level. For example, in order to insert a nested bulleted list or nested numbered list on a lower level, there must be a list one level up from it.

Adding Containers to an HTML Page Using the Containers Group

You can add specific containers that group objects together on an HTML page. The Containers group is shown in the following image.

The commands in the Containers group are:

Form

Inserts either a multi-layer form or single-layer form.

Tab

Inserts a tab control. Tab controls enable you to create multiple pages in one HTML form and present a better display for viewing secondary information. You can select *Top*, *Bottom*, *Left*, or *Right*. This means you can choose to add a tab control that displays tabs at the top, bottom, left, or right on the control.

You can drag and resize this object, at run time, if the Enable dragging and Enable resizing properties, in the Properties panel, are set to *Yes*.

To change the header text, you can either double click and change the header text or enter a value in the Selected Page: Title property.

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 and graph objects, images, and lines.

The Tab control can be displayed as a full screen or part of an HTML page.

Accordion

Inserts an Accordion styled box. You can drag and resize this object, at run time, if the Enable dragging and Enable resizing properties, in the Properties panel, are set to *Yes*.

To change the header text, you can either double click and change the header text or enter a value in the Selected Page: Title property.

Window

Inserts a window. You can drag and resize this object, at run time, if the Enable dragging and Enable resizing properties, in the Properties panel, are set to *Yes*.

To change the header text, you can either double click and change the header text or enter a value in the Selected Page: Title property.

Output Widget

Inserts an output widget. The output widget container includes buttons in the widget title bar that let you display or hide filter controls and that let you maximize to display full-screen, and minimize to return it to its original size.

Maintain Data App

Inserts a Maintain Data app window. This component is not applicable to Db2 Web Query, as it requires the Maintain feature.

Group Box

Inserts a group box. A group box can be used to create a border around a group of objects, for example, forms or reports and charts.

Panel

Inserts a panel to group objects together. The panel is invisible at run time.

Procedure: **How to Use a Form Object to Create a Pop-Up Dialog Box**

You can use a form object to create a pop-up dialog box.

1. Create a form object.
2. Create a button.
3. Select the form.
4. In the Properties panel, change the *Display container as* property to *As popup dialog*.
5. Create a new task in the Tasks section of the Tasks & Animations panel.
6. Use the button you create as the trigger.
7. Make the button object toggle the visibility of the form you created.
8. Run the page and click the button to display the form as a dialog box.

Procedure: **How to Create a Pop-Up Dialog Box or Pop-Up Window From a Form Control**

1. Add a button object to your HTML page.

2. Add a form control to your HTML page.

3. Delete the run and back buttons created with the form control.

4. While the form control is selected, in the Properties panel, change the Display container as property to either *As popup dialog* or *As popup window*.

   As popup dialog causes the form to display as a pop-up dialog box. When the dialog box is displayed, you must click *Close* to return to your HTML page.

   As popup window causes the form to display as a pop-up window. When the window is displayed, you can resize and move the window around your screen.

5. Create a new task in the Tasks section of the Tasks and Animations panel that uses the button object to toggle the visibility of the form you created.

6. Run your HTML page.
   When you click the button you added to the HTML page, the form will be displayed as either a pop-up dialog box or a pop-up window, depending on what option you selected.

Reordering Tabs, Accordions, and Windows

You can reorder tabs, accordions, and windows by dragging the page that you want to appear first onto the page that you want it to display before. For example, if you have 3 tabs, Tab1, Tab2, and Tab3, and you drag Tab3 onto Tab1, Tab3 will now display ahead of Tab1. The order of the tabs will then be Tab3, Tab1, Tab2. You can reorder windows when they are in tile view.

Adding Objects to an HTML Page Using the Objects Group

You can add objects other than reports, charts, elements, or containers to customize your HTML page. The Objects group is shown in the following image.
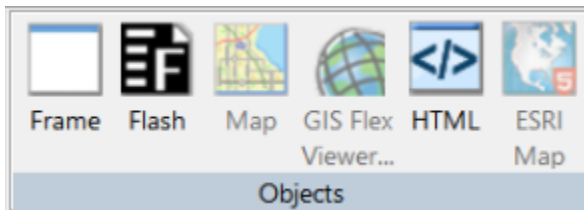
The commands in the Objects group are:

Frame

Inserts a frame object. 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.

Flash

Inserts a Flash component. You can add SWF files that are Adobe® Flash Player compatible to accompany reports or graphs on an HTML page.

Note: When inserting Flash animations, only files that are 1 MB or smaller can be run using the HTML canvas.

Map

Inserts a map object. You can add a Google™ or ESRI map to your HTML page. Maps are services offering powerful, user-friendly mapping technology that can be customized to show points on a map with drill-down capabilities. You can customize the map properties and bind them to a report or chart. This feature is not applicable to Db2 Web Query. Inserting a map can be done by inserting an existing Chart that has been formatted as an ESRI map.

GIS Flex Viewer

Inserts a GIS Flex Viewer object. The GIS Flex Viewer contains numerous controls, a report, and a map object. This command is not applicable to Db2 Web Query, as it requires application development features on the Data Server.

HTML

Inserts an HTML object. Developers can use the Settings panel to type a valid snippet of HTML code for the HTML object, such as HTML code to browse for a file.

ESRI Map

Inserts an ESRI map object. You can use the Settings panel to configure the properties and components. This feature is not applicable to Db2 Web Query. Inserting a map can be done by inserting an existing Chart that has been formatted as an ESRI map.

Changing Load Order With HTML Object Manipulation

In an HTML page, objects load in the order in which they were added to the canvas. A developer using raw HTML has the option of editing the source code to change the load order. Such edits leverage the Document Object Model (DOM) for HTML and XML documents.

In Developer Workbench, however, the source code cannot be edited. Therefore, the HTML object manipulation menu is provided to allow you to change the load order, if necessary.

To change the load order, select multiple objects or components in the HTML canvas, then right-click and select HTML object manipulation. The following menu options are available:

❏ **Append selected.** Available when the last multi-selected object is a container. Use this option to move one or more multi-selected objects into the container.

❏ **Swap selected.** To swap positions for two multiselected objects.

❏ **Delete contains only.** To delete containers only.

❏ **Merge containers.** To merge containers.

Accessibility Support for Displaying Objects in the Order of the Document Object Model

You can add elements/objects independently of screen reading order and then change the order later on during the development process.

Using the Accessible property, available on the Properties panel for a document, you can indicate that the Document Object Model (DOM) should be rewritten in the order of objects on the page, left to right, top to bottom. In addition, all tab index values should be set to the value -1.

Procedure: How to Display Objects in the Order of the DOM

1. Create a new HTML page with multiple controls on the page.

2. On the Properties panel for the document, select *Yes* from the Accessible drop-down menu, as shown in the following image.
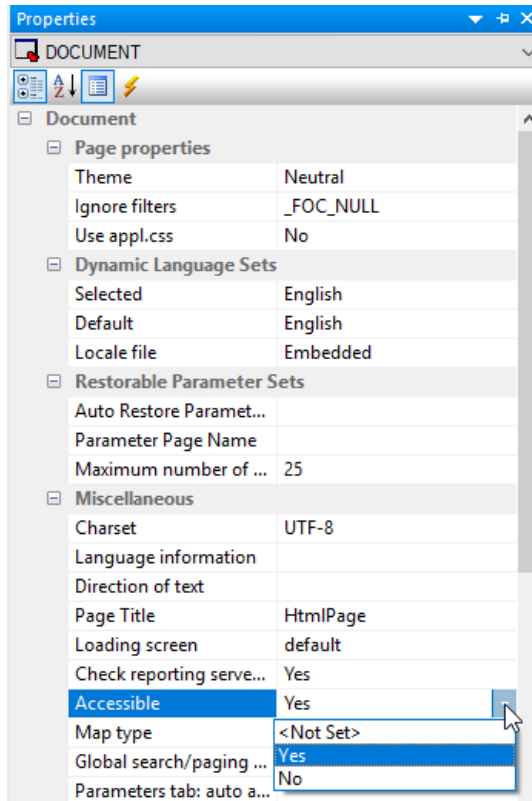
The following tab order warning message appears, as shown in the following image.
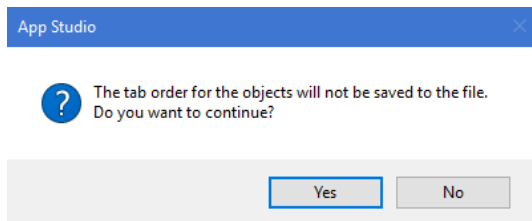
Note: The Warning message displays only once, when the Accessibility property is set to *Yes*.

Selecting *Yes* will write the tab order of objects on the page, left to right, top to bottom. Selecting *No* will keep the tab order in the page.

Inserting Controls in an HTML Page Using the Controls Tab

You can use the Controls tab and group to insert controls in an HTML page. Controls enable you to prompt users for a parameter value. When you create a parameter as part of a report or graph, the HTML canvas automatically adds a control, Submit button, and Reset button for the parameter to your layout, and the parameter appears on the Parameters tab. You can also add an input control and bind it to a parameter.

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. The 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 control, as well as the parameters associated with each control, can be controlled with the Properties tab of the Properties window, and with the Parameters tab. The Controls tab is shown in the following image.
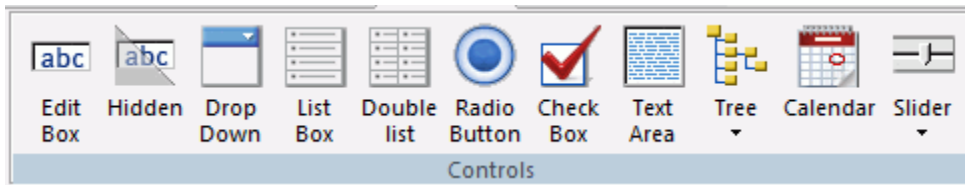
The controls are:

Edit Box

Inserts a text box. A text box enables you to enter a value in an entry field. You can specify a list of static or dynamic accepted values in Settings panel and, at run time, when you type the first letter of a value into the text box, that value will be listed as an autocomplete suggestion.

Hidden

Inserts a hidden control. A hidden input control allows parameter values to be used in a control without the user seeing them. When a hidden control is used, the current input control assigned to the parameter will not be visible. The value of the parameter can be entered in the Properties panel of the Parameters tab, or supplied through chaining.

Drop Down

Inserts a drop-down list. A drop-down list enables you to select single or multiple values from a list of supplied values. In order to provide multiple values, the procedure must be set up to accept multiple values.

❏ A single-select drop-down list enables you to select only one value for each time a request is run.

❏ A multiselect drop-down list enables you to select multiple values using the check boxes adjacent to the values.

You can use a dynamic or static list of values for the drop-down list.

List Box

Inserts a list. This 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 multiselect 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.

Double list

Inserts a double list. Displays multiselect 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.

Radio Button

Inserts a radio button. Radio buttons enable you to select a single value from a list of supplied values. Radio button values can be static or dynamic.

Use the Type property to control the appearance of the radio buttons. From the drop-down list box in the Type field, select *Standard* to display standard radio buttons.

From the drop-down list box in the Type field, select *Push button* to display push buttons.

Check Box

Inserts a check box. Check boxes enable you to select a single value from a list of supplied values. Note that if there are multiple check box input controls that are grouped together, you may select the Multiple properties for each control. Multiple ensures that you can select a single value from each check box control. Check box list values can be dynamic or static.

Use the Type property to control the appearance of the check boxes. From the drop-down list box in the Type field, select *Standard* to display standard check boxes.

From the drop-down list box in the Type field, select *Push button* to display push buttons.

Text area

Inserts a text area. A text area is a single-select 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 Properties panel on the Parameters tab.

If you want to display text in the control:

❏ When hard wrap is enabled, at run time, a line break is shown at the point where it was specified.

❏ For soft wrap, the line should wrap automatically depending on the size of the text area control.

If you want to submit values in the control:

❏ For single-select text area controls, only the first value up to the end of the line will be submitted.

❏ For multiselect text area controls, all of the values separated by OR or AND will be submitted.

Tree

Inserts either a single-source tree control or multi-source tree control. By using a tree structure in an HTML report, you can show hierarchical data from a multi-dimensional data source (for example, SAP BW), that uses the parent/child model. You may also use a tree control for non-hierarchical data sources. 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 bound to a tree control, the tree control uses option buttons for each node in the hierarchy. If the parameter is defined as Multiselect OR or Multiselect AND, and that parameter is bound to a tree control, then the tree control uses check boxes for each node in the hierarchy, enabling you to select multiple nodes.

Calendar

Inserts a 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 or parameter that is added to or referenced in the HTML canvas, and contains date parameters, will have a Calendar control type available in the New Parameters dialog box. When you select the *Calendar* control type, a text box with a calendar icon displays in the Design view of the layout. The text box is the only control available for the calendar, and the icon always displays 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 Db2 Web Query date format that specifies the complete date from the list of supported data types in Calendar Properties.

Slider

Inserts a horizontal or vertical slider control that has a numeric range of values to be used with a report or chart. This enables you to use a slider bar to select from a range of values.

Adding Paging/Search Capabilities to a Control

With a List Box, Drop Down List, or the first control of a Double List, you can add additional paging and search capabilities. Designed for controls that have a large number of valid values, this functionality enables you to:

❏ Page through a large list of values before making a selection.

❏ Search for a value within a control.

These options give you a quick way of narrowing your search when you know the value, or partial value, that you want to use to filter your page.

The *Add 'Paging/Search' control* function is available on the HTML canvas through the right-click context menu for the List Box, Drop Down List, and the first control of a Double List.

Use this function to enhance the capabilities of an individual control. If you want to create a single Paging/Search option that applies to multiple controls on a page, see *Adding a Paging/ Search Control to an HTML Page* on page 209.

Procedure: **How to Add Paging/Search Capability to a Control**

To add paging and search capabilities to a control that has a large number of valid values:

1. On the HTML canvas, select a List Box, Drop Down List, or the first control of a Double List, then right-click the control and select *Add 'Paging/Search' control* from the context menu.

   The paging/search control is added to the canvas below the input control. The following image shows the paging/search control added below a Drop Down List for Product Subcategory:
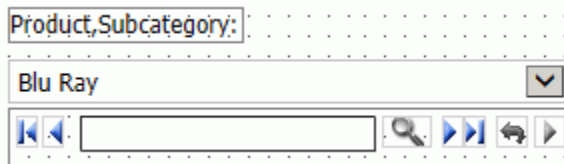
   Note, the blue left and right arrow buttons are for paging. The gray magnifying glass and arrow buttons are for searching.

   At run time, the paging/search control shows a number or page range of values. The default range is 1-50, meaning the input control starts at the first value and displays a range of 50 values per page.

   To change the default range, click the paging/search control and view the *Number or Range* attribute on the Properties tab. Change the value here as needed.

2. Run the HTML page.

   The paging/search control shows the specified range. For example, 1-50.

3. Use the arrow buttons to page through the values to be displayed for the input control. You may select First, Previous, Next, or Last.

   For searching, enter a search term in the paging/search control and press the Enter key or click the Search (magnifying glass) button.

   To display search criteria, click the search arrow button on the far right. The search criteria list is displayed. You can select any of the following options:

   ❏ **Match Prefix.** Search value prefix must match the prefix of the value in the input control.

❑ **Match Case.** Search value must match the value in the input control exactly.

❑ **Match Whole Word Only.** Search value word must exist in the string.

❑ **Instant Search.** The value is automatically searched as you type in the input control.

Toggle the search arrow button to hide the search criteria list.

4. Click the Run button to refresh the report, showing the value selected in the input control.

Adding a Paging/Search Control to an HTML Page

The *Global search/paging control* property lets you add additional capabilities to an HTML page that contains multiple instances of the following controls: List Box, Drop Down List, and the first control of a Double List. Designed for controls that have a large number of valid values, this functionality enables you to:

❑ Page through a large list of values before making a selection.

❑ Search for a value within multiple controls.

The *Global search/paging control* is a DOCUMENT level property.

Use this option to enhance the functionality of an HTML page. If you want to enhance the functionality of an individual control, see *Adding Paging/Search Capabilities to a Control* on page 207.

Procedure: How to Add a Paging/Search Control to an HTML Page

Use this procedure to add global paging and search capabilities to an HTML page with multiple controls.

The Global control option overrides individual control options. If *Keep selected values* is selected for Global searching/paging, that will override whether the double-list has it, regardless of whether it is set for the control itself.

1. Click the *Global search/paging control* property in the Properties tab for the DOCUMENT object and select *Yes* from the drop-down menu.

2. Run the HTML page.

   The Search/Paging control appears in the upper right corner of the window (Home position).

3. Click an input control.

   The Search/Paging control appears below the selected input control.

The label for the input control is displayed showing that it is linked to that input control. If no label is associated with the input control, the window will show the unique identifier for the input control, for example, combobox1.

If you want multiple pages, type a range in the Search/Paging field in the format *1-2 of 4* and press the Enter key.

4. Click the *Show settings* button on the Global Search and Paging window to expand the window and show the search and paging options.

The Search/Paging options are:

❏ **Match Prefix.** Search value prefix must match the prefix of the value in the input control.

❏ **Match Case.** Search value must match the value in the input control exactly.

❏ **Match Whole Word Only.** Search value word must exist in the string.

❏ **Instant Search.** The value is automatically searched as you type in the input control.

❏ **Auto-link to selected control.** Link the Search/Paging control to the selected control (default).

❏ **Keep selected values.** This option only affects a double-list. When selected, this option accumulates the values you searched for, in the right-side pane. When you search for another value, the previously searched values will not be deleted.

Toggle the *Up* arrow button to display and collapse the Search/Paging options. Click the *Home* icon on the Search/Paging window to move the window back to the Home position.

Procedure: How to Format Selected and Unselected Push Buttons for Radio Button and Checkbox Controls

This procedure shows how you create a push button form from a radio button or checkbox control. It also shows how you can apply background color to push buttons to distinguish between selected and unselected values.

1. In the HTML canvas, create a radio button or check box control by clicking *Radio Button* or *Check Box* on the *Controls* tab of the ribbon. Populate the control with multiple values.

2. Select the control and open the *Properties* panel.

3. In the Properties panel, from the Type drop-down menu, select *Push button*.

Two additional properties appear: *Selected pushbutton background color* and *Unselected pushbutton background color*.

4. From the Selected pushbutton background color drop-down menu, select a background color.

5. From the Unselected pushbutton background color drop-down menu, select a different background color.

6. Save and run the HTML file.

   The specified background colors appear for the selected and unselected values.

Procedure: **How to Set Calendar Properties**

1. Add a calendar to the HTML page by clicking the *Calendar* command on the Controls tab.

   Position the cursor over the HTML canvas. The pointer changes to a crosshair.

2. Drag the crosshair to create a calendar and adjust it to the size you want.

   A calendar placeholder is created in the layout and assigned the name calendar*(n)*, where *(n)* is a number.

3. Optionally, change the calendar properties by adjusting the properties displayed in the Properties panel.

4. Bind an existing parameter to the calendar.

   Binding a parameter to a calendar control creates an incoming parameter. The incoming parameter value must be in a Db2 Web Query date format that specifies the complete date from the list of supported data types in Calendar Properties. The incoming parameter value will populate the calendar with date values.

   ❏ Click the *Parameters* view tab.

   ❏ Select the center of the parameter name object, drag the parameter to the center of the calendar object, and release the mouse to complete the binding.

   ❏ To unbind the parameter from the calendar, select the arrow head on the line, so that the line is bold. Right-click and click *Break binding*.

5. Bind the calendar to a parameter.

   Binding the calendar to a parameter will populate the parameter with a date value.

   ❏ Click the *Parameters* view tab.

   The Settings panel opens, showing the calendar setup options. The calendar setup options enable you to set the range of dates available to the user at run time. Available dates are represented as an active hyperlink (blue and underlined). Unavailable dates are static (black without underlines).

Note: The Settings panel for a calendar contains different options depending on the selected data type.

❑ Create the values for the calendar. You can create Static or Dynamic values.

When the Current/Start date option is checked, the current date will be used in the calendar control at run time.

The Date Range options for setting up the calendar include:

❑ **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 icon, or by clicking the month, day, or year from the controls.

The pop-up calendar icon appears in the From and To sections when the *Static* Date Range is selected. If you click the pop-up calendar icon, a pop-up calendar appears and shows the current date selected and circled in red by default. 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 you 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). This is selected as the default Date Range.

❑ **Dynamic.** This option allows you to point to a request that returns a range of dates. Clicking the *Select custom request* drop-down list box allows you to choose a preexisting request. The request 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/YYYY.

❑ Select the center of the calendar, drag the calendar to the center of the parameter object, and release the mouse to complete the binding.

Working With a Double-Head Slider Control

A double-head slider control is a new control you can access from the HTML canvas. A double-head slider control is a type of slider control in which you can select a range of values for displaying data in a report or chart. Rather than having one slider head, you have two. In order to use a double-head slider control, you must have two variables in the report or chart that will be controlled by the double-head slider control. One variable will be controlled by one head of the double-head slider control and the other variable will be controlled by the other head of the double-head slider control. The variables need to be specified in the procedure as FROM and TO.

Note: You can use the Report canvas to create a parameter that accepts a range of values using the Expression Builder dialog box. The Chart canvas requires you to manually create a parameter that accepts a range of values.

Procedure: **How to Create a Double-head Slider Control From the Ribbon**

1. While in the HTML canvas, click either *Horizontal* or *Vertical*, from the *Slider* command, in the *Controls* group, on the *Controls* tab.

    The pointer changes to a crosshair.

2. Drag the crosshair to create a single-head slider control.

3. While the single-head slider control is selected, in the Properties panel, change the Range property from *No* to *Yes*.

    Changing the Range property to Yes changes the single-head slider control to a double-head slider control. You can now use this control to select a range of values to display.

Procedure: **How to Create a Double-head Slider Control Using the New Parameters Dialog Box**

1. Create a report object or chart object on the HTML canvas.

    The pointer changes to a crosshair.

2. Drag the crosshair to create a report or chart

3. Reference a report or chart with two variables used in a FROM... TO... relationship.

    The New Parameters dialog box opens.

4. In the Control Type column of the New Parameters dialog box, right-click the first parameter and then click *Slider*, and select either *Double-head Horizontal* or *Double-head Vertical*.

    Note that the Control Type for the second parameter changes to *Append to Above*. This means that the second parameter has been recognized as being part of a range and will be controlled by the double-head slider control specified in the first parameter.

Procedure: How to Create an HTML Page That Uses a Double-head Slider Control

1. Create a new HTML page.

2. Add a report object to the HTML canvas.

3. Reference a report that contains a variable that accepts a range of values.

   The New Parameters dialog box opens.

4. In the Control Type column of the New Parameters dialog box, right-click the first parameter, click *Slider*, and then select either *Double-head Horizontal* or *Double-head Vertical*.

   Note that the Control Type for the second parameter changes to *Append to Above*. This means that the second parameter has been recognized as being part of a range and will be controlled by the double-head slider control specified in the first parameter.

5. Click *OK* to close the New Parameters dialog box and add the double-head slider control to the HTML page.

6. Click the *Parameters* view tab at the bottom of the HTML canvas.

7. Drag the unbound parameter out of the Unbound Parameters box.

8. Drag the second half of the slider control to the unbound parameter.

9. Click the slider control and click the Settings panel.

10. In the Input Control Population section, change the Data type to *Dynamic*.

11. Select the data source you used in the Data Source drop-down list.

12. In the Value Field, click the ellipsis (…) and select the field being controlled by the double-head slider control.

13. Save and run your HTML page.

    The double-head slider control controls what you see in your report or chart by allowing you to select a range of values rather than one specific value or all values. The first head updates the FROM value and the second head updates the TO value.

Procedure: How to 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 Edit Box object to view the associated properties in the Properties panel.

2. 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.

Using Multi-Select Lists

Two types of multi-select lists can be used in a report: a regular list box or a drop-down list. When using either type of multi-select input control to supply parameter values, the Multiple property value indicates whether multiple values can be selected from a list of supplied values at run time.

Procedure: **How to Create a List Box with Multiple Values**

A multi-select list box enables you to select multiple values by using the Ctrl key while selecting values. In order to select multiple values in the list box, the procedure must be set up to accept multiple values. Ensure that the Variable Type for the parameter value is Multiselect OR or Multiselect AND in the procedure.

1. From the HTML canvas, insert a report with parameters that accept multiple values.

   For example, create a report with *Multiselect OR* as the variable type for the parameter, accepting a dynamic list of values from a Master File.

2. When the New Parameters dialog box is displayed, accept the default control type of *List box* and click *OK*.

   A list box is created in the layout and assigned the name listbox*(n)*, where *(n)* is a number.

3. Select *Multiple* from the Multiple drop-down list in the Properties panel.

   This indicates that multiple items can be selected from the list box.

4. Save and run the HTML page.

5. Select multiple values by using the Ctrl key while selecting values from the list box.

6. Click the *Run* button to run the report with the selected value parameters.

Procedure: **How to Create a Drop-Down Check Box List with Multiple Values**

A multiselect drop-down list enables you to select multiple values using the check boxes adjacent to the values. In order to select multiple values in the drop-down list, the procedure must be set up to accept multiple values. Ensure that the Variable Type for the parameter value is Multiselect OR or Multiselect AND in the procedure.

1. From the HTML canvas, insert a report with parameters that accept multiple values.

   For example, create a report with *Multiselect OR* as the variable type for the parameter, accepting a dynamic list of values from a Master File.

2. When the New Parameters dialog box appears, accept the default control type of *List box* and click *OK*.

A list box is created in the layout and assigned the name listbox*(n)*, where *(n)* is a number.

3. Right-click the control, select *Set Control Type,* and select *Drop down list* to change the control type.

   Note: In the Properties panel, the Multiple property for the control is automatically set to *Multiple*.

4. Save and run the HTML page.

5. Select multiple values using the check boxes displayed when you click the down arrow.

6. Click outside of the control to close the drop-down list.

7. Click the *Run* button to run the report with the selected value parameters.

Saving Control Selections in a Browser Session

You can link the controls on HTML pages so that they automatically default to the same selected value when you switch between multiple pages in the same browser session. To do this, you need to assign a common value to the *Global name* property for each control that you want to link.

For example, you may have two HTML pages that contain controls with information that relates to regional sales. You can assign a value to the *Global name* property, such as *Region*, to each of the controls that you want to link. When you run those pages in the same browser session, and choose a value from one of the linked controls, such as *Southeast*, the controls on the other pages will refresh and display the information for the Southeast, by default.

Note: A selected value is retained as the default only during a single browser session. The value is not retained after you close the browser.

Procedure: **How to Save Control Selections in a Browser Session**

This procedure describes how to assign a value to the *Global name* property, which enables you to save control selections within a browser session.

Note: A selected value is retained as the default only during a single browser session. The value is not retained after you close the browser.

1. Create or open an HTML page that contains a control.

2. Click the control to highlight it.

3. In the Properties panel, under Miscellaneous, type a value in the *Global name* field. This value can be any alphanumeric string, such as *GlobalRegion1*.

4. Save the HTML page.

5. Repeat steps 1-4 for any controls that you want to link. You must assign the same *Global name* property value for each control that you want to link.

Using Tree Controls

You can insert a single-source tree control or a multi-source tree control in an HTML page.

Procedure: ### How to 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 page. 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. From the Controls tab, insert a tree control.

   Tip: You may select Single source Tree control or Multi source Tree control. If no type is selected, Single source Tree control is the default. Single source Tree controls must be populated from a multi-dimensional data source such as SAP BW, SSAS, or Essbase.

The pointer changes to a crosshair.

2. Drag the crosshair to create a tree control, and adjust it to the size you want.

   A tree control is created in the layout and assigned the name treecontrol*(n)*, where *(n)* is a number.

3. Optionally, you may click the *Expanded* property from the Properties panel to show the tree control expanded at run time.

4. Optionally, you may click the *Hyperlink* property from the Properties panel to show the tree nodes as hyperlinks, instead of radio buttons at run time.

5. From the Settings panel, click *Dynamic* as the Data type.

6. Select *Explicit (Requests panel)*, select a procedure from the Request drop-down list, and then click *OK*.

   The procedure name is added as the explicit procedure.

   Note: In prior releases, an explicit procedure was known as an "external" procedure.

7. Click the browse (...) button adjacent to *Value from field* to select a field from the hierarchy or type the field manually.

   The Value from field is the data source field from which the values will be retrieved.

8. Click the browse (...) button adjacent to *Display from field* to select a field from the hierarchy or type the field manually.

   The Display field is the text that represents the parameter value in the tree control.

9. Save and run the page to view the multi-dimensional data source in the tree control.

Procedure: How to Populate a Multi Source Tree Control

You can show a tree structure for a non-hierarchical data source by using a tree control. By identifying the number of levels for the tree control, you are able to populate each level of the tree control with its own procedure. Setting the number of levels creates a tree structure by which each level is its own subcontrol, chained together with no conditions.

This procedure describes how to add parameters for a tree control, where the number of levels property is set.

1. In the HTML canvas, insert a Multi source tree control.

   ❑ Click *Multi source Tree control* from the Tree control drop-down list, located on the Controls tab.

   The pointer changes to a crosshair.

2. Drag the crosshair to create a tree control, and adjust it to the size you want.

   A tree control is created in the layout and assigned the name treecontrol*(n)*, where *(n)* is a number.

3. From the Properties panel, type in the Number of levels for the tree control, and press the Enter key.

   This enables you to specify the number of levels to populate.

4. Optionally, you may click the *Expanded* property from the Properties panel to show the tree control expanded at run time.

5. With the tree control selected, click the *Parameters* tab.

   The tree control object shows the set number of levels.

6. Select each level of the tree control and create the settings for its data population.

   When creating a Multi source Tree control, the static data type is not available. If creating static values for the tree control, you must create a single source tree control.

7. Optionally, to add an additional level for the tree control, right-click the tree control object on the Parameters tab and select *Add level*.

   Note: Add level only appears for a Multi source tree control object.

8. Click the added level to view the Settings panel for that level.

9. Switch to the *Design* tab of the HTML canvas to preview the populated tree control.

   Note that the Properties window drop-down list for the tree control shows each level of input values.

10. Save and run the page.

    Note: If a user selects a lower level node in one level and a higher level node in another level, when the procedure is executed, only the lowest level selections will take effect. For example, you have 3 levels: COUNTRY, CAR, and MODEL. Under ENGLAND, TRIUMPH you select TR7. Under FRANCE, you select PEUGEOT. At run time you will only receive the records for TR7 because you did not select a MODEL under the FRANCE node.

    The tree control populates each level with values.

    Note: A value must be selected for each level before you can click the *Save Selection* button.

    If a selected value is specified for a field that is not in level1, then corresponding selected values must also be specified for the preceding levels.

Only the first and second levels load at run time. If a selected value is specified for a field in level1, only values from the first two levels will be selected.

Procedure: **How to Add a Tree Control to an HTML Page Using Static Values**

This procedure describes how to add static data type parameters for a tree control, where the Number of levels property for the tree control is not set. This enables you to add a static list of values.

1. In the HTML canvas, insert a tree control.

2. From the Settings panel, select *Static* as the Data type.

   Static is selected, by default. You may select an item, delete it, or add a subitem.

3. Create the parameter values for the control:

   ❏ Click the *New* button to add a list of values. The values are added in a sequential hierarchical structure. The last value added appears in the Value and Display Value fields.

   ❏ To edit the value, manually type the desired value in the Value and Display Value fields.

   ❏ Click *Append child item* from the Static values drop-down list to append a value at the level currently selected, and create a new value as the child of the selected value.

   ❏ Click *Insert before* from the Static values drop-down list to insert a value before the selected value. Note the number of the value.

   ❏ Click *Insert after* from the Static values drop-down list to insert a value after the selected value. Note the number of the value.

   Repeat these steps until the list contains all of the values you want to include.

   ❏ Optionally, click the *Delete* button to eliminate any values.

4. Click the *Selected* check box to show the entry in the Value field as the default value.

5. Select the *Send display value* check box to send the display value, rather than the actual data, to the parameter.

6. Save and run the page to populate the tree control with static values.

Procedure: **How to Create a New Tree Control From the New Parameters Dialog Box**

When a report contains one or more new amper variable parameters, the New Parameters dialog box opens when you save the report and return to the HTML canvas. You can assign a new Single source or Multi source Tree control from the HTML page to the parameter from the New Parameters dialog box.

For each parameter, you will find Name and Control Type fields, a Create control check box, and options to set the Control Type to a Single source or Multi source Tree control.

1. Reference a report that contains parameters.

    When referencing a report with parameters, the New Parameters dialog box opens, prompting you to create the control type.

2. Select the new tree control from the New Parameters dialog box.

    The Control Type column refreshes, showing the selected control.

3. Click *OK* to close the New Parameters dialog box.

    The report is added and the associated parameters are bound to the tree control.

Reference: **Usage Notes for Chaining Tree Controls**

The following usage notes apply when chaining tree controls. You may chain controls from the New Parameters dialog box and from the Parameters tab.

❏ When the Multi source Tree control is a link in the chain, the New Parameters dialog box enables you to share parameters with the same multi source control.

❏ You can chain a Multi source Tree control to a Single source Tree control.

❏ You can chain a Single source Tree control to another Single source Tree control.

❏ You can chain a Multi source Tree control to another Multi source Tree control.

❏ You can chain a tree control to another non-tree control, such as Drop down list or List box.

❏ Chaining cannot be done with only field names.

❏ Dynamic population of controls with field names need to use SYSCOLUMN calls.

Positioning Objects in an HTML Page Using the Positioning Tab

Positioning options enable you to multi-select objects and position them relative to one another. The Positioning tab contains the Positioning, Relationships, Text Alignment, and Responsive Design groups, as shown in the following image.
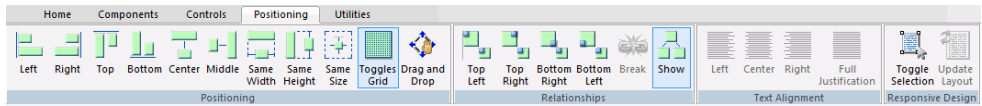
Aligning Objects in an HTML Page Using the Positioning Group

The Positioning group contains the Left, Right, Top, Bottom, Center, Middle, Same Width, Same Height, Same Size, Toggles Grid, and Drag and Drop commands.

Left

Aligns the objects to the left. This works in relationship positioning.

Relates an object to the left of another object.

Right

Aligns the objects to the right. This works in relationship positioning.

Relates an object to the right of another object.

Top

Aligns the objects to the top of the highest selected object. This works in relationship positioning.

Bottom

Aligns the objects by the bottom of the lowest selected object. This works in relationship positioning.

Center

Aligns the objects to the center of the page. This works in relationship positioning.

Aligns objects at the horizontal center point of the object.

Middle

Aligns the objects to the middle of the page. This works in relationship positioning.

Aligns objects at the vertical center point of the object.

Same Width

Makes two or more objects the same width.

Same Height

Makes two or more objects the same height.

Same Size

Makes two or more objects the same size.

Toggles Grid

Turns the grid on and off.

Drag and Drop

Turns parent/child Drag and Drop on and off.

When this command is off (default), drag and drop is used only to reposition objects on the HTML canvas.

When this command is on, drag and drop is used to designate a parent/child relationship between two objects.

Procedure: **How to Designate a Parent/Child Relationship using Drag and Drop**

There are instances where you want an object in your HTML canvas to be a child to another object. For example, you may have a report (child) that you want to place within a tab container (parent).

1. Open an HTML file in which the two objects have been created.
2. Click the *Positioning* tab.
3. Click the *Drag and Drop* command to toggle it on.

   Alternatively, you can select *Toggle Drag and Drop* from the HTML canvas shortcut menu.

   The Drag and Drop command on the Positioning tab is now highlighted.
4. Click the designated child object and drag it toward the designated parent object.

   A small rectangle appears below your cursor, but the child object itself does not move.
5. Move your cursor to the designated parent object and drop.

   The parent/child relationship is now created.
6. To restore the default drag and drop, click the *Drag and Drop* command to toggle it off.

   Note: When the Drag and Drop command is on, you are unable to reorder tabs or page containers, when using the Tab, Accordion, or Window components.

Relating Objects in an HTML Page Using the Relationships Group

The Relationships group contains the Top Left, Top Right, Bottom Right, Bottom Left, Break, and Show commands.

Top Left

Relates two or more objects on a page. Two objects must be selected. Relates an object to the top left of another object.

Top Right

Relates two or more objects on a page. Two objects must be selected. Relates an object to the top right of another object.

Bottom Right

Relates two or more objects on a page. Two objects must be selected. Relates an object to the bottom right of another object.

Bottom Left

Relates two or more objects on a page. Two objects must be selected. Relates an object to the bottom left of another object.

Note: The above four commands have been deprecated and will be removed from a future release of Developer Workbench.

Break

Breaks the relationship between the objects selected.

Show

Shows or hides the relationship between objects.

Aligning Text in an HTML Page Using the Text Alignment Group

The Text Alignment group contains the Left, Center, Right, and Full Justification commands. These commands align text within the HTML page.

Left

Aligns selected text to the left.

Center

Aligns selected text to the center.

Right

Aligns selected text to the right.

Full Justification

Fully justifies the selected text, meaning that the text fills the width of the text box.

Enabling Responsive Design for an HTML Page Using the Responsive Design Group

The Responsive Design group contains the Toggle Editing, Toggle Selection, and Update Layout commands. These commands are enabled when the Responsive design property is set to *Yes*.

Toggle Selection

Toggle on to allow containers that include content to be dragged to a new position. Toggle off to prevent those containers from being repositioned.

Update Layout

Click to refresh the HTML canvas after you reposition containers, to show how the containers will stack when folded. This command is activated when the Autosize Enable property is set to Yes for both the document and the object.

Changing HTML Page Properties Using the Utilities Tab

You can change HTML page properties using the Utilities tab. For example, suppose you want to test your HTML page in a specific browser. The default browser specification is considered a property of your HTML page. To run your page in a different browser, open the Utilities tab and select the desired browser from the Preview command group.

The Utilities tab is shown in the following image.
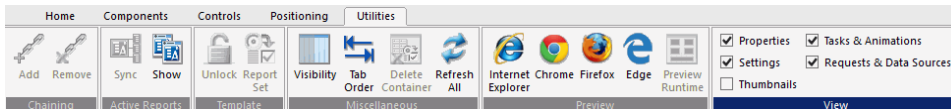
Chaining Objects in an HTML Page Using the Chaining Group

You can add or remove chaining options using the Chaining group. The Chaining group is shown in the following image.
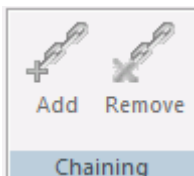
The commands are:

Add

Chains two or more objects that are selected in the Parameters tab.

Remove

Unchains two or more selected, chained objects in the Parameters tab.

Synchronizing an Active Technologies Report in an HTML Page Using the Active Reports Group

You can synchronize an active report. The active reports group is shown in the following image.
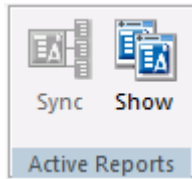
The commands are:

Sync

Syncs an active report.

Show

Shows the synchronization of the canvas.

Unlocking a Template for an HTML Page Using the Template Group

You can unlock a template to add controls, text, and buttons. The Template group is shown in the following image.
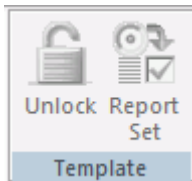
The commands are:

Unlock

Unlocks the template that you are using and allows for full control of the page.

Report Set

In template mode, this adds a title bar and a form with controls, text, and buttons as a set.

Working With the Miscellaneous Group in the HTML Canvas

The Miscellaneous group contains the Visibility, Tab order, Delete Container, and Refresh All commands. The Miscellaneous group is shown in the following image.

The commands are:

Visibility

Toggles the visibility of hidden objects. Hidden objects are objects that have the *Visibility* property, in the Properties panel, set to *hidden*.

For more information on using the Visibility command, see *How to Toggle the Visibility of a Hidden Object* on page 227.

Tab order

Shows the order of the tabs.

Delete Container

Deletes the selected container, without deleting the container content.

Refresh All

Refreshes the HTML page.

Procedure: **How to Toggle the Visibility of a Hidden Object**

1. Insert an object into the HTML canvas.

2. Select the object you created, if it is not already selected.

3. In the Properties panel, change the *Visibility* property to *hidden*.

    The object on the canvas is hidden from view.

4. On the *Utilities* tab, in the *Miscellaneous* group, click the *Visibility* command.

The hidden object is now displayed on the canvas. You can click the *Visibility* command again to hide the object again. At run time, this object will be hidden.

Previewing HTML Output Using the Preview Group

The Preview group, as shown in the following image, allows you to preview HTML output in Internet Explorer, Chrome, Firefox, and Edge browsers to be sure that your content displays correctly.

The Preview Runtime option is available for Responsive HTML pages only. It uses your default browser to preview your HTML run time layout with simulated data. This may be faster than running the page if it contains large amounts of data.

Note: Alternatively, you can right-click in the HTML canvas, select *Preview*, and select a browser or Preview Runtime.

The browser preview options are:

Internet Explorer

Preview your HTML output in Internet Explorer.

Chrome

Preview your HTML output in Chrome.

Firefox

Preview your HTML output in Firefox.

Edge

Preview your HTML output in Edge.

Preview Runtime

Available for Responsive HTML pages only. It uses your default browser to preview your HTML run time layout with simulated data.

Displaying Panels in an HTML Page Using the View Group

The View group allows you to display or hide the panels associated with an HTML Page. This group is accessible in the Utilities tab of the HTML canvas and in the Text Editor tab when accessing the Embedded JavaScript and Embedded CSS view tabs. The View group is shown in the following image.
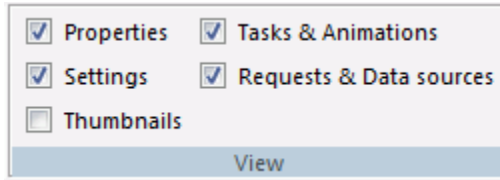
The commands are:

Properties

Toggles the visibility of the Properties panel.

Settings

Toggles the visibility of the Settings panel.

Thumbnails

Toggles the visibility of the Thumbnails panel.

Tasks & Animations

Toggles the visibility of the Tasks & Animations panel.

Requests & Data sources

Toggles the visibility of the Requests & Data sources panel.

Using Parameters

Parameters, or "variables," are a good way to make a single HTML page valuable to a wide range of end users. For example, if you have a parameter for region, the end users can select their own region at run time. This improves performance, saves users from scrolling through data that is not pertinent, and saves you from having to build multiple pages.

Parameter values and input controls can be created with a dynamic or static list of values:

❏ A dynamic list retrieves values from a specified data source when the request is run. These values are always current to your data source.

❏ A static list consists of a list of values you supply. These values do not change unless you change them.

❑ An active control lists active report values that mimic active report menu items.

Note: The active 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 TOC control list gives you the ability to integrate a report with a Table of Contents and On Demand Paging in the HTML canvas.

Note: TOC controls cannot be associated to any parameters in the layout. This type of control can only be associated with certain input controls in the layout.

Working With the Parameters Tab

The Parameters tab enables you to create and modify parameter values, input controls, and customize parameter conditions. You can also bind parameters to controls and chain controls to one another. The Parameters tab consists of the following components:

❑ Input control objects.

You may select the input control object to view and edit the settings of the control.

❑ Creating an input control from the Design view prompts you to create a bound parameter on the Parameters tab.

❑ Editing an input control, which is inserted when setting input controls for new parameters.

❑ Add new parameters.

Right-click anywhere on the Parameters tab to add a new parameter.

Note: Manually adding a parameter creates an unbound parameter.

❑ Rearrange the order of objects.

When you move a parameter on the Parameters tab, a property value is automatically set. On the Properties panel, under Document properties, the *Parameters tab: auto arrange* property is automatically set to *No* when you move a parameter. This means the parameter will remain where you moved it after you navigate away, then navigate back to the Parameters tab.

You can rearrange the order of parameters and controls on the Parameters tab, which will automatically set the *Parameters tab: auto arrange* property value to *No*. This means that if you navigate away from the Parameters tab, and then return to it, the objects remain the in the position you placed them. Additionally this means that the Auto Arrange Objects option, which is selected in the HTML tab of the Developer Workbench Options dialog box by default, will be overridden.

You can change the *Parameters tab: auto arrange* property value to *Yes*, to undo any moves that you made previously, and to ensure that the objects on this tab use the auto arrange functionality.

To inherit the Auto Arrange Objects option value, you can set the *Parameters tab: auto arrange* property value to *<Not Set>*.

The *Parameters tab: auto arrange* property is available in the Properties panel. To access this property, you must select DOCUMENT, from the list at the top of the panel.

When you select an object on the Parameters tab, the properties for that object will automatically display on the Properties panel. When you select a parameters object from the Properties panel drop-down, the object is automatically selected in the Parameters tab.

❏ Refresh unresolved parameters.

All parameters on the Parameters tab are parsed every two minutes to check if any are unresolved. If there are, their surrounding polygon is colored red. If you want to check for unresolved parameters on demand, right-click and select *Refresh unresolved*.

❏ Binding controls and parameters.

Input controls and parameters can be bound and unbound from the Parameters tab.

You may bind a parameter to an input control, or bind an input control to a parameter.

❏ Binding a parameter to a control makes it an incoming parameter that will populate the control. Drag a parameter object to a control object on the Parameters tab.

❏ Binding a control to a parameter will populate the parameter. Drag a control object to a parameter object on the Parameters tab.

❏ Chain one control to another.

Chaining will populate controls based on the selected value from the prior control in the chain. You can chain static and dynamic controls, link or unlink parts of a chain, and create conditions on links in a chain. Chains are represented by lines connecting control objects on the Parameters tab. By clicking the arrow head in a link of a chain, the Settings panel enables you to modify and set properties and conditions of the chain.

Note: Chaining is applicable only for controls, not parameters.

Adding a New Unbound Parameter

An unbound parameter is useful when passing a parameter value used on another page. You may also bind the new parameter to a control to create an incoming parameter, or bind a control to the parameter.

Procedure: **How to Add a New Unbound Parameter**

The following steps describe how to add a new parameter:

1. Right-click anywhere on the Parameters tab and select *Add parameter*.

    Enter the parameter value information. Options are *Single select*, *Multiselect OR*, and *Multiselect AND*.

2. If using single value, select *Single select*.

    Note: Single select is the default option when adding a new parameter.

    a. Enter the Selected Value to be assigned to a single variable.

    b. Enter the name for the parameter in the Name field, or keep the default name.

    c. Optionally, you may use the Format field to define the format of the parameter, such as A20, or D12.2.

        If this field is left blank, it automatically applies the Alphanumeric format to the value field.

3. If using a multiselect value, select *Multiselect OR* or *Multiselect AND*.

    The Value, Display, and Selected columns appear.

    a. Click the *New* button to enter a list of static values.

    b. In the Value column, enter the value to be passed to the selected parameter.

    c. In the Display column, enter the text that represents the parameter value in the control the user views.

    d. In the Selected column, check the box for the value you want to be selected by default. More than one value can be selected.

    Repeat these steps until the list contains all of the values you want to include.

4. Optionally, you may select values and click the *Delete* button to eliminate any values, and use the up and down arrows to rearrange the order of the values.

5. To modify the parameter value, right-click the parameter on the Parameters tab and select *Settings* to make your edits.

    Tip: You may also use the Undo and Redo buttons located on the Standard toolbar. Note that undo/redo treats the entire Settings panel as one action.

6. Optionally, bind a control to a parameter to populate the parameter. Select the center of the parameter name object and drag the parameter to the center of the control object.

7. Optionally, bind the new parameter to a control to create an incoming parameter. Select the center of the control object and drag the control to the center of the parameter object.

Procedure: How to Link an Unbound Parameter to a Procedure
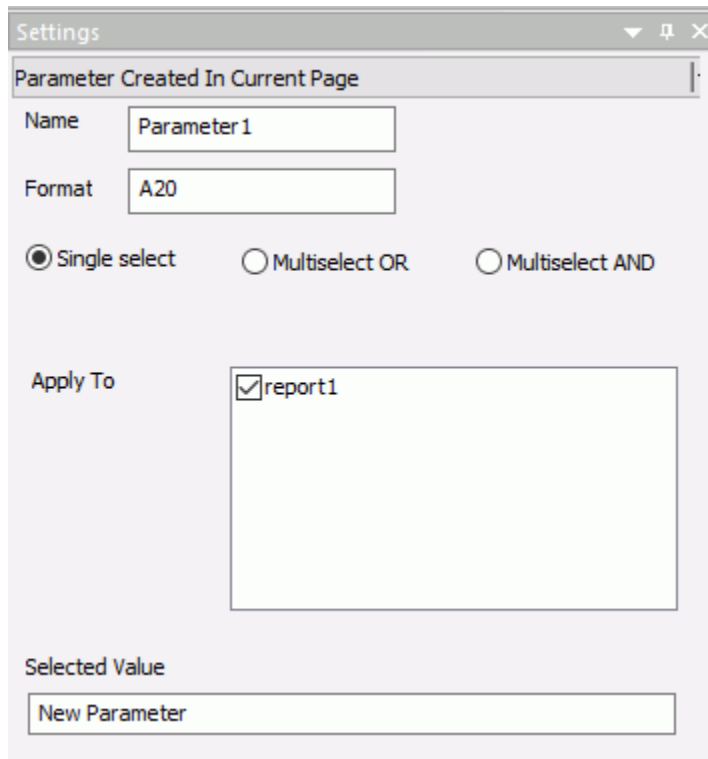
The following steps describe how to link an unbound parameter to a procedure. As an example, a heading is added to a report in the steps below.

1. Create a new HTML page.

2. From the Requests & Data Sources panel, click the *New* drop-down arrow and select *External Request - Db2 Web Query Procedure for i Using InfoAssist+*.

3. Select any simple or parameterized procedure.

   Note: In this example, we are using a report.

4. Insert a frame in the canvas.

5. In the Tasks & Animations panel, select the *load* task.

6. Click the arrow in the Requests/Animations section, select *Run Request* and select your procedure.

7. Select the *Target type* as *Frame*.

8. Select the frame from the *Target/Template Name* drop-down list.

9. In the Requests & Data Sources panel, right-click *Parameters Created in Current Page* and select *New Parameter*.

   Note: You can also create a new unbound parameter as described in *How to Add a New Unbound Parameter*.

10. Navigate to the Parameters tab and drag *Parameter1* outside the Unbound Parameters box.

11. Select *Parameter1* and in the Settings panel, enter *A20* in the *Format* field.

12. Select the checkbox in the *Apply To* field next to the procedure you want to link, as shown in the following image.

13. Enter an alphanumeric value in the *Selected Value* field.

14. Open the procedure.

15. From the *Report* group on the *Report* tab, use the Header & Footer option to add a Report Header.

16. Enter *&Parameter1* in the Report Heading text box.

17. Run the HTML page.

   Notice that the value entered in the Settings panel for Parameter1 is displayed in the report heading.

Creating a Static List of Values

When creating a list of static values, you can select from the following options:

❏ Add ignore value

❏ Add everything value

❏ Add 'No Selection' value

❏ Use values from procedure

❏ Use values from external file

❏ Use values from Library

❏ Use values from Library (include versions)

When the options are added to the Value list, the display text can be customized, but the value cannot be changed.

You may create an unbound static parameter, an incoming static parameter (a parameter that is bound to a control), or a control that is bound to a static parameter.

Reference: **Settings Panel (Incoming Static Parameter and Unbound Control)**

The Settings panel appears when creating or editing a static value on the Parameters tab.

The options available depend on the type of static value.

The Settings panel contains the following fields and options when *Static* is selected as the Data type.

Data type

Determines whether values are obtained from a static or dynamic list, an active report, or table of contents.

Static. Uses a static list of values you supply. A list of static values can also be created in the Report canvas.

Static values

Is a list of supplied values for a static list.

Value. The value to be passed to the selected parameter.

Display. The text that represents the value in the control the user views. Press the Ctrl + Shift keys to add a value to the Display field.

Selected. The value to act as the default value. If the control is multiselect, more than one value can be selected.

New. Creates a new value.

Delete. Deletes a supplied value from the list.

Move Up. Moves the selected value up in the list.

Move Down. Moves the selected value down in the list.

Send display value

Select this option to send the display value, rather than the actual data, to the parameter.

Values are procedures names

Select this option to have a control populated with procedure names, so that when a value is selected, that procedure executes. The Value column is the procedure name itself and cannot be edited. The Display column is editable.

Reference: Settings Panel (Unbound Parameter)

The Settings panel appears when creating or editing a static value on the Parameters tab.

The options available depend on the type of static value.

The Settings panel contains the following fields and options when adding an unbound parameter with Single select. Single select is the default option when adding a new parameter.

Name

The default name assigned to the parameter. Optionally, you may enter a new name for the parameter.

Format

The Format field defines the format of the parameter, such as A20, or D12.2.

This field is optional. If this field is left blank, it automatically applies the Alphanumeric format to the value field.

Selected Value

Enter the selected value to be assigned to the parameter.

Static values

Is a list of supplied values for a static list.

Value. The value to be passed to the selected parameter.

Display. The text that represents the value in the control the user views. Press the Ctrl + Shift keys to add a value to the Display field.

Selected. The value to act as the default value. If the control is multiselect, more than one value can be selected.

New. Creates a new value.

Delete. Deletes a supplied value from the list.

Move Up. Moves the selected value up in the list.

Move Down. Moves the selected value down in the list.

Reference: Settings Panel (Bound Parameter)

The Settings panel appears when selecting a bound parameter on the Parameters tab. The Settings panel for a parameter is read-only, and displays the values for the bound control.

The Settings panel contains the following read-only values:

Value

Shows the selected value for the static parameter data.

Display

Shows the static parameter display value.

Procedure: How to Add a New Static Value

The steps below describe how to manually add a new static value:

You may create an unbound static parameter, an incoming static parameter (a parameter that is bound to a control), or a control that is bound to a static parameter.

1. Create a new parameter.

    a. Right-click anywhere on the *Parameters* tab and select *Add parameter*.

    b. Enter the parameter value information in the Settings panel. Options are *Single select*, *Multiselect OR*, and *Multiselect AND*.

    c. If using single value, select *Single select*.

        Single select is the default option when adding a new parameter.

        ❏ Enter the Selected Value to be assigned to a single variable.

        ❏ Enter the name for the parameter in the Name field, or keep the default name.

        ❏ Optionally, you may use the Format field to define the format of the parameter, such as A20, or D12.2.

        If this field is left blank, it automatically applies the Alphanumeric format to the value field.

    d. If using a multiselect value, select *Multiselect OR* or *Multiselect AND*.

The Value, Display, and Selected columns appear.

❏ Click the *New* button to enter a list of static values.

❏ In the Value column, enter the value to be passed to the selected parameter.

❏ In the Display column, enter the text that represents the parameter value in the control the user views.

❏ In the Selected column, check the box for the value you want to be selected by default. More than one value can be selected.

Repeat these steps until the list contains all of the values you want to include.

❏ Optionally, you may select values and click the *Delete* button to eliminate any values, and use the up and down arrows to rearrange the order of the values.

An unbound static parameter is useful when passing a parameter value used on another page. You may also bind the new parameter to a control to create an incoming parameter, or bind a control to the parameter.

2. Bind the new parameter to a control.

Binding a new parameter to a control creates an incoming parameter. An incoming parameter is a static parameter that is bound to a control. The parameter value will populate the control.

a. Select the *Design* tab and create an input control. For example, insert a list box or a drop-down list.

b. Click the *Parameters* tab.

The Settings panel appears for the control.

c. Select the center of the parameter name object and drag the parameter to the center of the control object.

d. To unbind the parameter, select the arrow head on the line, so that the line is bold, right-click, and select *Break binding*.

3. Create a control that is bound to a parameter.

Create a control with static values and bind the control to a parameter to populate the parameter with the control values.

a. From the Design view of the HTML canvas, select a control.

The pointer changes into a crosshair.

b. Drag the crosshair to create the control and adjust it to the size you want.

c. Click the *Parameters* tab.

The Settings panel appears for the control.

d. Select *Static* as the Data type.

Static is selected, by default.

e. Create the parameter values for the control:

❏ In the Value column, enter the value to be passed to the control.

❏ In the Display column, enter the text that represents the static parameter value in the control the user views.

❏ In the Selected column, check the box for the value you want to be selected by default. More than one value can be selected.

Repeat these steps until the list contains all of the values you want to include.

❏ Optionally, you may select values and click the *Delete* button to eliminate any values, and use the up and down arrows to rearrange the order of the values.

f. Bind the new control to a parameter: Select the center of the control object and drag the control to the center of the parameter object.

g. To unbind the control, select the arrow head on the line, so that the line is bold, right-click and select *Break binding*.

h. To change the default type of control, right-click the control object on the Parameters tab or the Design tab and select *Set Control Type*.

The options are Calendar, Check box, Drop down list, Hidden, List box, Radio button, Text Area, Edit box, Single source Tree control, and Multi source Tree control.

4. To modify the static value, right-click the control, or parameter, on the Parameters tab and select *Settings* to make your edits.

Procedure: **How to Add an Ignore Value**

The add ignore value option sends _FOC_NULL to the server at run time and is intended for use with complex applications. The add ignore value option is available for Multiselect OR and Multiselect AND static parameters.

1. From the HTML canvas, use controls to supply parameter values for a report.

A report with parameters requires that you to select values (at run time) in order to generate the output.

2. Click the *Parameters* tab.

   The parameters associated with each control can be controlled with the Parameters tab. The properties of a control can be controlled with the Properties tab.

3. Select a multiselect control object from the Parameters tab.

   The Settings panel opens.

4. Select *Add ignore value* from the Static values drop-down list.

5. Optionally, select *Send display value* to send the display value, rather than the actual data, for the parameter values in the report.

6. Close the Settings panel.

7. Run the HTML page and select the *Ignore All* value to ignore the parameter values.

Note: Sending _FOC_NULL to a procedure will result in any clause of that procedure that uses that variable to be ignored.

Procedure: How to 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. The add everything value option is available for Multiselect OR and Multiselect AND static parameters.

The add everything value option is not available for a Double List Control.

1. From the HTML canvas, use controls to supply parameter values for a report.

2. Click the *Parameters* tab.

3. Select a multiselect control object from the Parameters tab.

   The Settings panel opens.

4. Select *Add everything value* from the Static values drop-down list.

5. Optionally, select *Send display value* to send the display value, rather than the actual data, for the parameter values in the report.

6. Run the HTML page and click the *Select All* parameter value to view all the parameter values.

Procedure: **How to Use Values From a Procedure**

This is the default option which populates the static list with field names predefined in the procedure. The use values from procedure option is available for Multiselect OR and Multiselect AND static parameters, and when adding static field values from the Report canvas.

1.   From the HTML canvas, use controls to supply parameter values for a report.

2.   Click the *Parameters* tab.

3.   Select a multiselect control object from the Parameters tab.

     The Settings panel opens.

4.   Select *Use values from procedure* from the Static values drop-down list.

     The field names from the procedure appear in the Settings panel.

5.   Optionally, select *Send display value* to send the display value, rather than the actual data, for the parameter values in the report.

6.   Run the HTML page and select the parameter values from the procedure.

Procedure: **How to Import Values From an External File**

This option enables you to use a local external file to provide values for the parameter. The import values from an external file option is available for Multiselect OR and Multiselect AND static parameters.

1.   From the HTML canvas, use controls to supply parameter values for a report.

2.   Click the *Parameters* tab.

3.   Select a multiselect control object from the Parameters tab.

     The Settings panel opens.

4.   Select *Use values from external file* from the Static values drop-down list.

     The Open File dialog box appears.

5.   Select a text file from your local machine and click *Open*.

     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 data value and *Boston* is the display value.

     The imported values are loaded into the Static values area of the Parameters tab.

If there is only one value on the line in the text file, the value will populate both the data value and the display values.

6. Run the HTML page to see the imported values for the selected parameter.

Creating a Dynamic List of Values

Dynamic values are available by default if a parameter used in the procedure is associated with the selected control. A dynamic list retrieves values from a specified data source when the request is run.

Reference: **Settings Panel (Dynamic Values)**

The Settings panel appears when creating or editing a dynamic parameter on the Parameters tab.

The Settings panel contains the following fields and options when Dynamic is selected as the Data type:

Data type

Determines whether values are obtained from a static or dynamic list, an active report, or table of contents.

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.

Default

Is the data source from which the values will be retrieved. This is the default value when *Dynamic* is selected as the Data Type.

Explicit (Requests panel)

Is the existing procedure that will be called.

You may modify the explicit procedure directly from the Settings panel on the Parameters tab. If you modify the request, you can save the explicit procedure and overwrite the original request.

Data Source

Is the data source from which the values will be retrieved.

Value from field

Is the data source field from which the values will be retrieved.

Display from field

Is the text that represents the parameter value in the control the user views.

There should be a relationship between the Value from field and the Display from field. The Display from field is user-friendly text corresponding to the Value from field.

Sort

Clicking *Sort* enables you to set the sort order for displaying values in dynamic list controls. This option is useful when you want to sort each control independently of the others.

By default, the request retrieves dynamic display values from the BY sort field in the request. The results display values based on the value field.

Sort by

When Sort is enabled, you may sort the display value by the Value field or the Display field selected from the Settings panel. The default is Value field.

Sort order

When Sort is enabled, you may select the sort order as Ascending or Descending. The default sort order is Ascending.

Check for duplicate values

When creating a dynamic list of values for a report, you may remove duplicate values from input controls.

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.

Add 'No selection' option

Optimizes performance by populating a chain one control at a time, instead of all the controls when the page initially loads. Selecting the Add 'No selection' option enables you to populate controls when necessary.

Cache run time data

When adding dynamic parameters to the HTML page, input controls retrieve data through procedures. Select this option to cache the run-time data for the selected input control. This setting is off by default.

This setting overrides the Default caching option from the HTML Page tab, which is located in the Developer Workbench Options dialog box.

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.

Use last modified filter only

Send display value

Select this option to send the display value, rather than the actual data, to the parameter.

Selected Value

Enter the value to be selected as the default value whenever the procedure is run.

Reference: Settings Panel (Bound Parameter)

The Settings panel appears when selecting a bound parameter on the Parameters tab. The Settings panel for a parameter is read-only and displays the values for the bound control.

The Settings panel contains the following read-only values:

Data Source

Shows the selected Master File for the parameter data source.

Value from

Shows the value field for the dynamic parameter data.

Display from

Shows the dynamic parameter display field.

Multiselect

Shows OR or AND, if there is a Multiselect OR or Multiselect AND dynamic parameter.

Selected Value

Shows the selected value, if there is one assigned to the variable.

Procedure: How to Create a Dynamic Value

The steps below describe how to create a dynamic list of values.

You may need to create a dynamic control that is bound to a parameter. Creating a control with dynamic values and binding the control to a parameter will populate the parameter with the control values.

1. From the Design view of the HTML canvas, select a control from the Controls submenu of the Insert menu. For example, insert a list box or drop-down list.

   The pointer changes into a crosshair.

2. Drag the crosshair to create the control and adjust it to the size you want.

3. Click the *Parameters* tab.

   The Settings panel opens.

4. Select *Dynamic* as the Data type.

   The dynamic value options appear.

5. Create the dynamic values for the control.

   a. If you are using a default procedure to supply dynamic values, follow the steps below.

      ❏ Click the *browse (...)* button adjacent to the *Data Source* drop-down list.

         The Open File dialog box appears.

      ❏ Select the Master File name and click *OK*.

      ❏ Click the *browse (...)* button adjacent to the *Value from* field.

         The Object Inspector opens with the field names from the selected Master File.

      ❏ Double-click a field name to add it to the Value from field.

         Tip: You may also use the Object Inspector icons to select a field and close the Object Inspector. The green icon is OK, the red icon is Cancel, double-clicking a value will select the value and close the dialog box without using any button, and pressing the Esc key will cancel the dialog box without using any button.

      ❏ Optionally, you may click the *browse (...)* button adjacent to the *Display from* field to select a different field name for the Display field. (You may also use the Object Inspector icons to select a field and close the Object Inspector).

   b. If you are using an explicit procedure to supply dynamic values, follow the steps below.

      ❏ Select *Explicit (Requests Panel)*.

      ❏ Select a procedure from the *Requests* drop-down list.

         The parameter names from the procedure are automatically added to the *Value from* field and *Display from* field.

      ❏ Optionally, you may click the *browse (...)* button adjacent to the *Value from* field and *Display from* field to select different field names.

> **Tip:** You may also use the Object Inspector icons to select a field and close the Object Inspector. The green icon is OK, the red icon is Cancel, double-clicking a value will select the value and close the dialog box without using any button, and pressing the Esc key will cancel the dialog box without using any button.
>
> Only the parameter names from the explicit procedure will be available for selection.

6. Optionally, select *Add "ALL" option* to add the select ALL data source values to the control.

7. Optionally, select *Add 'No selection' option* to optimize performance by populating a chain one control at a time instead of all the controls when the page initially loads.

8. Optionally, select *Cache run time data* to cache the run time data for the selected input control.

9. 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.

10. Optionally, click the *Sort* option to enable and select the sort order options for displaying values in dynamic list control.

11. Optionally, select *Check for duplicate values* to remove any duplicate value entries from the input control at run time.

12. Bind the new control to a parameter. Select the center of the control object and drag the control to the center of the parameter object.

13. To unbind the control, select the arrow head on the line, so that the line is bold, right-click and select *Break binding*.

14. To change the default type of control, right-click the control object on the Parameters tab or the Design tab and select *Set Control Type*.

    The options are Calendar, Check box, Drop down list, Hidden, List box, Radio button, Text Area, Edit box, Single source Tree control, and Multi source Tree control.

15. To modify the dynamic value, right-click the control and select *Settings* to make your edits.

Procedure: How to Create Dynamic Parameters by Adding a Filter

You may create new dynamic parameters by adding a filter to a report or graph component in the layout.

1. To create dynamic parameters for your report or graph in the Design view, right-click the report or graph object and select *Add a filter* from the shortcut menu.

   The Filter options dialog box opens.

2. 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 Settings panel on the Parameters tab.

If the New Parameters dialog box does not appear, ensure that *Show New Parameters dialog* is selected from the HTML Page tab. To access the HTML Page tab, select *Options* from the Window menu to open the Developer Workbench Options dialog box. From the Developer Workbench Options dialog box, select the HTML Page tab.

3. Click *OK* to close the New Parameters dialog box.

4. The filter appears above the report or graph object.

    Repeat this procedure for each additional parameter for the report or graph.

Procedure: **How to Sort the Dynamic List of Values**

This option is useful when you want to sort each control independently of the others.

Note: If sort options are not selected, the request retrieves dynamic display values from the BY sort field in the request, and the results display values based on the value field. Sort options are not selected by default.

1. Select a dynamic control from the Parameters tab.

    The Settings panel opens.

2. Select *Sort* to enable the sort options.

    You may select the Sort by and Sort order options for the control.

3. Select the Sort by options:

    ❏ Sort by Value sorts the value by the field name from the Value field. This is the default Sort by selection.

    ❏ Sort by Display sorts the value by the field name from the Display field.

4. Select the Sort order options:

    ❏ Sort order Ascending sorts the value from lowest to highest. This is the default Sort order selection.

    ❏ Sort order Descending sorts the value from highest to lowest.

5. Run the HTML page to see the sort results.

Procedure: How to Check for Duplicate Values

When creating a dynamic list of values for a report, you may remove duplicate values from input controls. This is useful if you are using your own procedure that does not use a structured data source.

The Check for duplicate values option is turned off by default.

1. From the HTML canvas, create an input control with a dynamic list of values.

2. Select the input control and click the *Parameters* tab.

   The Settings panel opens for the input control.

3. Select the *Check for duplicate values* check box.

   The Check for duplicate values option is only available when creating a dynamic list of values for an input control.

4. Save and run the HTML page.

   The input control removes duplicate value entries.

   The following example shows a list box with a list of city values. The first list box shows the list of values with duplicate entries. The second list box shows the list with duplicate values removed.

Automatically Populating Fields With Dynamic Values

When the name of a dynamic parameter matches a corresponding field name in a data source, the HTML canvas automatically populates the field name values 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 fields in the Settings panel of the Parameters tab (when Dynamic is selected as the Data type). This generates a layout report that is ready to run as long as the parameter names match the field names.

Example: Automatically Populating Fields With Dynamic Parameter Values

When the following report request is called from the HTML canvas with a push button control, the Settings panel for the PRODUCT parameter on the Parameters tab is automatically populated to dynamically retrieve the values of the PRODUCT field.

```
TABLE FILE GGSALES
SUM UNITS
BY PRODUCT WHERE ( PRODUCT EQ '&PRODUCT.Product:.' );
END
```

IBM

Creating a Static or Dynamic Parameter Value List

When creating a static or dynamic list of values, you may add an ALL value to the list of values and/or send the display value in a parameter.

The ALL feature allows developers to automatically add an ALL value to a list of 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 control list to the server at run time. The Add everything value option is only available with a Multiselect OR variable type.

When using a Dynamic or active report Data type, you may enter the value(s) to be selected as the default value whenever the procedure is run.

Note: You may also enter the selected value when adding a new unbound parameter on the Parameters tab.

Procedure: How to Send the Display Value for Static and Dynamic Controls

From the HTML canvas, you may send the display value, rather than the actual data, to the parameter. The display value can also be used for headings and footings in the report output.

1. From the HTML canvas, import or create 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 is added to the HTML canvas.

2. To send the display value for the parameter selection, select the control object (for example, select listbox*n*) in the Design view and click the *Parameters* tab.

3. Navigate to the Settings panel.

Note: The options available in the Settings panel vary, depending on the type of values (static or dynamic) you are creating.

4. For a static list of values, the Value, Display, and Selected columns appear on the Settings panel.

   ❑ In the Value column, enter the value to be passed to the control.

   ❑ In the Display column, enter the text that represents the static parameter value in the control the user views.

   ❑ In the Selected column, check the box for the value you want to be selected by default. More than one value can be selected.

   Repeat these steps until the list contains all of the values you want to include.

   ❑ Select *Send display value*.

   Tip: You may also update the display values from the Variable Editor dialog box in the Report canvas.

   ❑ Optionally, you may select values and click the *Delete* button to eliminate any values, and use the up and down arrows to rearrange the order of the values.

   ❑ Click the *Design* tab to view the display values in the control object of the HTML canvas.

5. For a dynamic list of values, the Value from field and the Display from field appear on the Settings panel.

   ❑ Add the request to the Requests & Data sources panel.

   ❑ Select *Explicit (Requests Panel)*.

   ❑ Select the procedure from the Requests drop-down list.

   ❑ Click the ellipsis button from the Value field.

   The Object Inspector opens with the field names from the selected Master File.

   ❑ Double-click a field name to add it to the Value field.

   The selected field is automatically added to the Display field.

   ❑ Optionally, you may click the Display field ellipsis button to select a different field name for the Display field.

   ❑ Optionally, select *Add "ALL" option* to automatically add an ALL value to a list of parameter values.

❑ Optionally, select *Add 'No selection' option* to optimize performance by populating a chain one control at a time instead of all the controls when the page initially loads.

❑ Select *Send display value*.

When Send display value is selected, the &Variable gets the display value, instead of the actual value.

6. Add *&Variable_TEXT* to the report heading or footing.

Note: If the report procedure uses it, *&Variable_TEXT* will always be passed, regardless of if Send display value is selected.

The heading as Display Value: &STCD_TEXT, where Store Code (STCD) is the variable name.

Tip: This is different from adding the actual value from the data source, where clicking the variable name would add <STCD to the report heading.

7. Save and close the report to return the HTML canvas.

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 canvas, as no control type is needed.

8. Run the HTML page.

9. Select the parameter for the report and run the report.

The display value is shown in the report heading.

Note: The HTML canvas passes &Variable_TEXT if the report procedure uses it, regardless of whether the send display value check box is checked or unchecked.

Procedure: **How to Use Procedure Names as Values**

The *Values are procedures* names option lets you populate a control with procedure names or HTML file names. When that procedure name is clicked, the procedure executes.

1. Create an HTML page that contains a listbox, a push button, and a report.

Note: In this procedure a listbox is used, however, the following controls are also able to use the Values are procedures names option: double list, drop-down, radio button, and check box.

2. Select the listbox to bring up the Settings panel.

Note: If the Settings panel does not open, select *View* and click *Settings*.

3. Select *Static* as the Data type.

4. At the bottom of the Settings panel, select *Values are procedures/html files*.

5. Click the *New* button and select procedures from your directory.

    Note: You can add multiple procedure names to the Settings panel by multi-selecting procedures while in the Open File dialog box.

6. Once the procedures have been added to the Settings panel, edit the display names of the procedures by double-clicking the display contents if they are not already highlighted.

7. Right-click the button you created and click *Create Hyperlink*.

    The Hyperlink Properties dialog box opens.

8. Create a hyperlink that opens a selected procedure from a control in the report frame created earlier.

    a. For the Action, select *Procedures from control* from the drop-down list.

        This option coincides with the Values are procedures names option found in the Settings panel. This option will point to an entire procedure for the hyperlink, rather than a simple value. This option is only available when a control on the HTML page is using the Values are procedures names option.

    b. Select *listbox1* as the Source.

        The source can be different if you use a different control. For example, combobox1, customselect1, radio1, or checkbox1.

    c. Select *Frame* as the Target Type.

        You could also select *New Window* as the target if you wanted the procedure to open in a new window.

    d. Select *report1* as the Target/Template Name.

9. Run the page.

10. Select the procedure from the listbox and click the button.

    The report is run.

Procedure: How to Use Selected Values as the Default Value

When using a Dynamic or active report Data type, you may enter the values 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 in the data source, the values are selected by default. If the selected value is not available in the data source, then the value(s) that you entered are ignored and the first value retrieved from the data source is selected.

1. Create or import a report that contains a parameter value.

   The report and control is added to the HTML canvas.

2. To enter the selected value to be used as the default value, select the control object (for example, select listbox*n*) in the Design view and click the *Parameters* view tab.

   The Settings panel opens.

   Note: The options available in the Settings panel vary, depending on the type of values (static or dynamic) you are creating.

3. For a dynamic list of values, the Value from field and the Display from field appear on the Settings panel.

   a. If using a default procedure to supply dynamic values, follow the steps below.

      ❏ Select *Default*.

      ❏ Click the ellipsis button adjacent to the Data Source field.

         The Open File dialog box appears.

      ❏ Select the Master File name and click *OK*.

      ❏ Click the ellipsis button adjacent to the Value from field.

         The Object Inspector opens with the field names from the selected Master File.

      ❏ Double-click a field name to add it to the Value from field.

         The selected field is automatically added to the Display from field and the source code for the default procedure appears.

      ❏ Optionally, you may click the Display from field ellipsis button to select a different field name for the Display from field.

      ❏ Optionally, select *Add "ALL" option* to automatically add an ALL value to a list of parameter values.

      ❏ Optionally, select *Add 'No selection' option* to optimize performance by populating a chain one control at a time instead of all the controls when the page initially loads.

❏ Enter the exact parameter value in the Selected Value input field, as it appears in the data source.

Parameter values are case sensitive.

❏ Optionally, you may enter more than one value by using a semicolon (;) between the values. For example, *CA;GA*.

You may enter selected values with a semicolon or a comma. Additionally, you may also have embedded commas or semicolons in the data values. If you have these embedded characters, you must enclose all of the values in the input area in double quotation marks ("). For example:

ENGLAND;ITALY

ENGLAND,ITALY

"ENGLAND","IT,ALY"

Note that even though one value in the last set has the embedded comma, both values need to be enclosed in double quotation marks (").

b. If using an explicit procedure to supply dynamic values, follow the steps below.

❏ Select *Explicit (Requests Panel)*.

❏ Select the procedure from the *Requests* drop-down list.

The parameter names from the procedure are automatically added to the Value from field and Display from field, and the source code for the explicit procedure appears.

❏ Optionally, you may click the Value from field and Display from field ellipsis button to select a different field name or to type the field name manually.

Note: Only the parameter names from the explicit procedure will be available for selection.

❏ Optionally, select the *Add "ALL" option* to automatically add an ALL value to a list of parameter values.

❏ Enter the exact parameter value in the Selected Value input field, as it appears in the data source.

Parameter values are case-sensitive.

❏ Optionally, you may enter more than one value by using a semicolon (;) between the values. For example, *CA;GA*.

You may enter selected values with a semicolon (;) or a comma (,). Additionally, you may also have embedded commas or semicolons in the data values. If you have these embedded characters, you must enclose all of the values in the input area in double quotation marks ("). For example:

ENGLAND;ITALY

ENGLAND,ITALY

"ENGLAND","IT,ALY"

Note that even though one value in the last set has the embedded comma, both values need to be enclosed in double quotation marks (").

4. For an active report list of values, the Available active reports, Menu Option Types, and Common Columns appear on the Settings panel.

❏ Select one or more active reports from the list of Available active reports. The selected report will be bound to the active report control in the layout.

When an active report is selected, Refresh for active reports is enabled, by default.

❏ Select the Menu Options Types for the active report control to sort, filter, list or select columns, and/or change presentation styles of the bound active report and the associated report and graph objects synchronized to the active report.

❏ Optionally, select the *Add "ALL" option* to automatically add an ALL value to a list of parameter values.

❏ Enter the exact parameter value in the Selected Value input field, as it appears in the data source.

Parameter values are case-sensitive.

❏ Optionally, you may enter more than one value by using a semicolon between the values. For example, *CA;GA*.

You may enter selected values with a semicolon or a comma. Additionally, you may also have embedded commas (,) or semicolons (;) in the data values. If you have these embedded characters, you must enclose all of the values in the input area in double quotation marks ("). For example:

ENGLAND;ITALY

ENGLAND,ITALY

"ENGLAND","IT,ALY"

Note that even though one value in the last set has the embedded comma, both values need to be enclosed in double quotation marks (").

5. For a single-select unbound parameter, the Selected Value input field appears on the Settings panel.

    ❑ Enter the exact parameter value in the Selected Value input field, as it appears in the data source.

    Parameter values are case sensitive.

    ❑ Optionally, you may enter more than one value by using a semicolon between the values. For example, *CA;GA*.

    You may enter selected values with a semicolon or a comma. Additionally, you may also have embedded commas or semicolons in the data values. If you have these embedded characters, you must enclose all of the values in the input area in double quotation marks. For example:

    ENGLAND;ITALY

    ENGLAND,ITALY

    "ENGLAND","IT,ALY"

    Note that even though one value in the last set has the embedded comma, both values need to be enclosed in double quotation marks (").

    Even though it is a Single select parameter, multiple values are specified and selected.

6. Click the *Run* button to run the report with the selected value parameters.

    The report output appears.

    The selected value, if available from the data source, is automatically selected (highlighted) in the parameter list.

    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.

Viewing Object Attributes

The Properties panel shows the attributes of the object or objects selected on the canvas. For example, if you select a hyperlink object, the Properties panel shows the different attributes associated with a hyperlink, such as Enable Dragging. General attributes for the entire HTML page are shown when no object is selected. The Properties panel is available when you are using the HTML canvas. The attributes can be sorted by category or alphabetically.

You can also select Events. When Events is selected, a list of all available JavaScript events that can be used in conjunction with an object appears. The events that are available change depending on what type of object is selected. For example, a report object has different events available than a button object. When no object is selected, events for the HTML page are displayed. Double-clicking on an event will create a function block for the selected object, using that event. You can view the created functions in the Embedded JavaScript and Embedded CSS tabs, where you can type the JavaScript code to execute when the selected event occurs.

For more information on customizing the Properties panel, see *Customizing the Properties Panel for HTML Objects Using the Properties Toolbar* on page 258.

Color Selection

For properties that affect color, such as Background-color, Border-color, and Color, you can enter the hexadecimal value of the color you want to use directly into the property, rather than opening the Color Picker dialog box. If you use the Color Picker dialog box to select a color, the hexadecimal value of that selected color will be entered into the selected property. For more information on the Color Picker dialog box, see *Color Picker Dialog Box* on page 263.

Sizing

Size property fields for a selected object can be set using the Left, Top, Width, and Height fields. The default setting is in pixels (px), and the settings are also displayed in the Locations field. You can also select *<Not set>*, *auto*, or *inherit* in the Left, Top, Width, and Height fields but only pixels are displayed in the Location field.

The Position field also contributes to the location of an object in the HTML canvas. The default value is absolute, but you can change it to one of the values in the list below:

❏ **<Not Set>.** The element is positioned relative to its first positioned (not static) ancestor element.

❏ **inherit.** Inherits this property from its parent element

❏ **absolute.** The element is positioned relative to its first positioned (not static) ancestor element. This is the default value.

❏ **fixed.** The element is positioned relative to the browser window.

❏ **relative.** The element is positioned relative to its normal position, so setting the Left field to 20px adds 20 pixels to the elements left position.

❏ **static.** Elements render in order, as they appear in the document flow.

Controls

The Name or Unique Identifier attribute settings for controls appear automatically in the Parameters tab of the HTML canvas.

❑ When you change the Name attribute for a control, the new name automatically appears in the Parameters tab.

❑ Some controls do not have a Name attribute. For these controls, the Unique Identifier setting automatically appears in the Properties panel instead.

Customizing the Properties Panel for HTML Objects Using the Properties Toolbar

The Properties toolbar contains the buttons and menus that you need to navigate and customize the contents of the Properties panel.

Categorized

Groups the attributes by category.

Alphabetical

Sorts the attributes alphabetically, without categories.

Properties

Shows the property names and values. This is the default setting. Properties and Events cannot be selected at the same time.

Events

Shows the JavaScript events that can be associated with the selected object on the canvas. Properties and Events cannot be selected at the same time.

Resizing HTML Components Using the Autosize Children Option

The Autosize Children option can resize all child components on an HTML page to automatically fit the maximum available screen space on any monitor or device where it might be viewed. For example, a single component will take up one hundred percent of the available space, while two components will take up fifty percent each.

This option is used so that when your HTML page is run on different devices, components and controls do not appear off the screen and have to be scrolled to in order to use them. To accommodate this, the orientation of components may be different on different devices. For example, a report and chart that appear side-by-side when run on a desktop monitor, may appear stacked one above the other when run on a tablet. Similarly, this difference could occur when run on a 24-inch desktop monitor and a 26-inch desktop monitor.

There are autosize Enable properties that work at both the document and component levels:

❏ **Autosize Children Enable.** This property is set at the document level. When set to *Yes*, it enables some or all of the child components in the document to resize automatically.

The Autosize Children Enable property is also available at the component level for the form, group box, and panel container objects.

❏ **Autosize settings Enable.** This property is set for each component in the document. The setting for this option may be:

   ❏ *Yes.* Enables the component to resize automatically.

   You can prevent the component from becoming too small using the Min-width and Min-height options. Type the minimum number of pixels for the width, height, or both, in the appropriate fields. When a component reaches the minimum width or height that you specified, a scroll bar appears and the size of the component does not decrease further.

   ❏ *No.* Prevents the individual component from resizing automatically, even when the document and other components resize. Components tagged in this way are rendered first, when the HTML page runs.

The following image shows a report component and a chart component on the HTML canvas where Autosize Children is enabled. Both components share the available space equally.
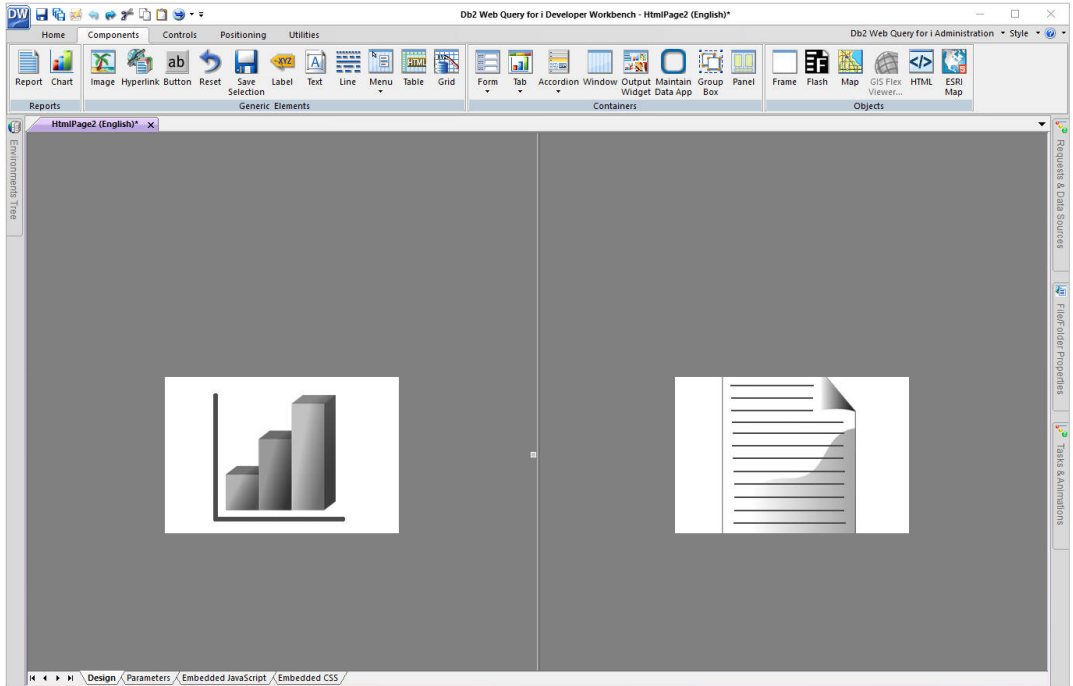
You can enable Autosize Children for an existing HTML file or create a new file with the option enabled.

Procedure: **How to Enable Autosize Children for an Existing HTML File**

1. Open the HTML file.

    The HTML canvas opens.

2. In the Properties panel for the document, set the Autosize Children Enable property to *Yes*.

3. For each component on the page that you want to resize automatically, set the Autosize settings Enable property to *Yes*.

4. Right-click the HTML canvas and click *Update Layout*.

    The components are resized to occupy all of the space on the canvas.

Procedure: **How to Create a New HTML File with Autosize Children Enabled**

1. On the *Home* tab, in the *Content* group, click *HTML/Document*.

You can also create a new HTML file from the Application menu or by using the shortcut menu in the Environments Tree panel, for a folder that supports content creation.

The HTML/Document Wizard opens.

2. Navigate to where you want to create your HTML page and click *Next*.

The Themes and Settings window of the HTML/Document Wizard opens.

3. In the Other settings area, select the *Autosize reports/charts* check box.

Selecting this option sets the Autosize Children Enable property for the document to *Yes*.

4. Click *Finish*.

The HTML canvas opens.

5. For each component on the page that you want to resize automatically, set the Autosize settings Enable property to *Yes*.

Working With Autosize Children

Working with Autosize Children entails some practices and conditions beyond those found in the native HTML canvas. This is because, when Autosize Children is enabled, components automatically move and resize themselves on the canvas.

Adding Components to the HTML Canvas

When you add a component to the canvas with the Autosize Children option enabled, the existing components will temporarily move out of the way so that you can draw the new one. The components will then reposition and resize automatically.

If the screen does not refresh, you can right-click the canvas and click *Update Layout*. There is a small amount of canvas space shown to the bottom and right of the screen, which allows you to access the shortcut menu. This extra canvas space is not shown at run time.

Note: If Update Layout does not resize the component automatically, check the properties. The Autosize Children Enabled property for the document must be set to *Yes*, and the Autosize settings Enable property for the component must be set to *Yes*. Components with an Autosize settings Enable property set to *No* will not resize automatically on the canvas or at run time.

To change the order of the components, drag a component to the front or to the left of another component.

The following items apply to form, group box, and panel containers:

❑ Forms, group boxes, and panels that contains components and controls must have the Autosize Children Enable property set to *Yes*, and the Autosize settings Enable property set to *Yes*.

❑ The components that are children of a form, group box, and panel have their Autosize settings Enable property automatically set to *Yes*. For controls that are children of a form, group box, or panel, you must manually set the Autosize settings Enable property to *Yes*.

❑ The form, group box, or panel should have the Number of Columns property set to the value you want. For example, if you want three reports or charts to be next to each other, set the value for the Number of Columns property to *3*.

If an object has the Autosize settings Enable property set to *No*, and you want that object to occupy the entire width of the available space, you must set the Width property to 100%.

Note: Reports and charts do not automatically re-execute once the page loads and the screen size changes. You must re-execute reports and charts to ensure that the content refreshes. If the output format of a chart is HTML5, the chart content refreshes automatically and you will not have to re-execute.

Procedure: How to Add Content Inside a Container

You can easily add content inside a container, such as a panel, in the native HTML canvas. However, if the Autosize Children property is enabled, the container moves when you try to add content to it. To prevent this movement, use the following steps:

1. Click to select and freeze the container to which you want to add content.

2. On the *Components* tab, in the *Generic Elements* group, choose an object, such as a text box, that you want to add to the container.

3. Click in the container again to add the element. For example, draw the text box and type the text.

4. Click the element to select it.

   In the Properties panel, set the Autosize settings Enable property to *Yes*, if you want the element to resize automatically.

   Click *No*, if you want the element to remain fixed in the container.

5. Right-click the canvas and click *Update Layout*.

   The content now appears inside the container.

Working With Containers

If you place an object that has Autosize settings Enable property set to *No* inside a container that has Autosize Children Enable property set to *Yes*, the object will move to the upper-left corner of the container.

When a container has Autosize Children Enable property set to *Yes*, and a report or chart is added to that container, the Autosize Children settings property for the report or chart is automatically set to *Yes*.

When a Tab, Accordion, or Window container has Autosize Children Enable property set to *Yes*, and multiple reports or charts are added to it, use the Number of columns property. Select the number of columns you want from the drop-down list and arrange the contents in the container, as desired. This ensures the contents will be arranged the same for all browsers. If you do not use the Number of columns property, different browsers may arrange the contents in different ways.

Reference: **Color Picker Dialog Box**

The Color Picker dialog box contains the Web Palette tab, Named Colors tab, System Colors tab, and Custom Color tab. From these tabs, you can pick different colors.

❏ **Web Palette.** You can choose a color from common web colors.

❏ **Named Colors.** You can choose a color from common named colors.

❏ **System Colors.** You can choose a color based on the colors of Developer Workbench.

❏ **Custom Color.** You can drag red, green, and blue sliders to create custom colors.

The hexadecimal value for any color you select or create is displayed at the bottom of the dialog box. If you want to enter a specific hexadecimal value, you must enter using the Properties panel or the Style Composer dialog box, in the appropriate area. For example, you can enter a hexadecimal value on the Background-color property line.

Modifying Object Population Settings

In Developer Workbench, you use the Settings panel to modify the population settings of components, controls, and parameters. The Settings panel has context-sensitive sections that are displayed or hidden, depending on the object selected on the canvas. When you select an object, the settings for that object are displayed in the Settings panel.

Settings are grouped under the following sections, and the sections and settings change, depending on which component, control, or parameter is selected.

❑ **Manage CSS and Scripts Section**

When no object is selected on the canvas, you can manage your cascading style sheet files and JavaScript files using the Manage CSS and Scripts section. The Settings panel displays this section by default.

Using the CSS button and JavaScript button , you can search for cascading style sheet files and JavaScript files, located in your Db2 Web Query environment. You can reference web-accessible Cascading Style Sheet files and JavaScript files by typing the URL, of those files, in the URL/Find File area.

❑ **Parameter Created In Current Page Section**

To create a parameter in the current page, right click in the Parameters tab and click *Add Parameter*. The Settings panel displays the Parameter Created in Current Page section. Type the parameter name and format. Click the radio button to specify Single select, Multiselect OR, or Multiselect AND. Type a value in the Selected Value field.

❑ **The Dynamic Parameter Data Section**

When a dynamic parameter is selected on the Parameters tab, the Settings panel displays the data source file, Value field, and Display field. If the value is multiselect, you can click the drop-down list in the Multiselect field and choose OR, AND, BY, ACROSS, or NONE. Next, type the Selected Value and the Preview Value.

❑ **Input Control Population Section**

When a control is selected on the canvas and the Data type is set to Dynamic, you can drag fields and parameters from the Requests & Data sources panel to the Settings panel.

Working With the Selection To Area in the Settings Panel

The Selection to area in the Settings panel, assists in directing where control and grid selections are sent. For example, you can choose to send any selections made from a drop-down control to a specific parameter.

You can click the *Use 'Requests_Data sources' panel* button to open the Requests & Data sources panel next to the Selection to area. This allows you to drag any parameter from the Requests & Data sources panel, to the Selection to area.

Using Tasks & Animations

Using the Tasks & Animations panel, you can submit requests for parsing, execute tasks when specific criteria are met, and add jQuery animations to your HTML page. The Tasks & Animations panel is shown in the following image.
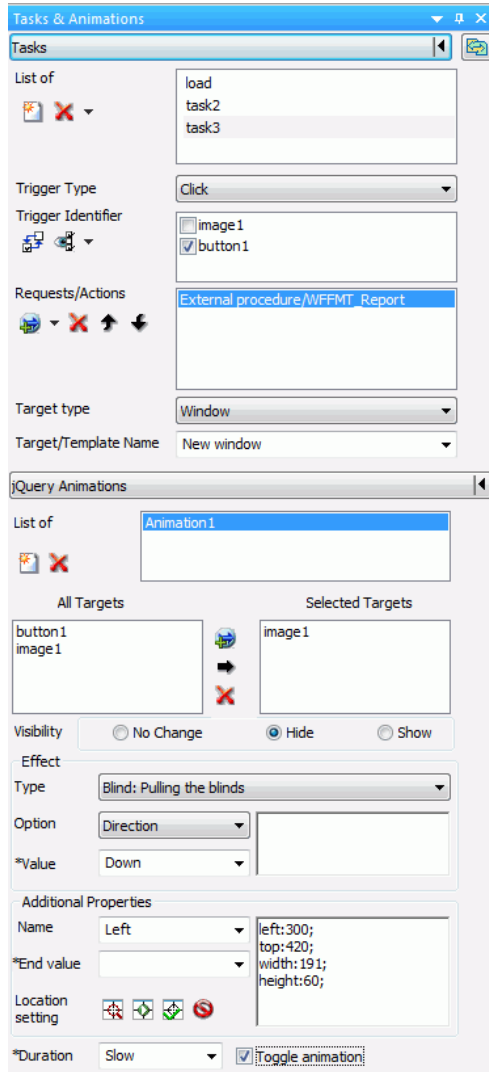
For more information on customizing the Tasks & Animations panel, see *Executing jQuery Animations Using the jQuery Animations Section* on page 270.

Executing a Task Using the Tasks Section

The Tasks section of the Tasks & Animations panel allows you to execute a request when specific criteria are met. For example, you can create a task that runs an embedded report when a button is clicked.

For more information, see *How to Automatically Execute a Request When a Page is Loaded* on page 278.

Tasks

Toggle details

Opens the Tasks dialog box which displays the Tasks section of the Tasks & Animations panel horizontally rather than vertically. Tasks are displayed in a list. This list shows the Task Name, Trigger Type, Trigger Identifier, Requests/Actions, Target Type, and Target Name, of each task, next to each other. When the Tasks dialog box is opened, the Tasks section of the Tasks & Animations panel is collapsed.

List of

A list of tasks defined in the HTML page.

New task

Creates a new task. A task is an actionable item.

Delete

Deletes a selected task. You can use the drop-down list to delete a selected task or to delete all untriggered tasks.

Trigger Type

Determines when a task should start. You can choose:

❏ Click

❏ Click link

❏ Double click

❏ Mouse down

❏ Mouse enter

❏ Mouse leave

❏ Mouse move

❏ Mouse over

❏ Mouse out

❏ Mouse up

❏ Blur

❏ Selection Changed

❏ Checkbox: Check

❏ Checkbox: Uncheck

❏ TBD

❏ Maps: Selected marker drill down

❏ Maps: anywhere click

❏ Grids: Cell Modified

❏ Grids: Cell Selection Changed

❏ Grids: Cell Edit Start

❏ Grids: Cell Edit Finish

❏ Grids: Column Sized

❏ Grids: Column Selection Changed

Note: The TBD trigger type can be used when a task is executed by an API call.

Trigger Identifier list

A list of all possible triggers based on what option you have selected from the Trigger filters list.

Clear Triggers Selection

Deselects any object you have selected in the Trigger Identifier list.

Buttons, Images, Hyperlinks/Selected Only

You can select whether to display buttons, images, hyperlinks, or other objects as possible Trigger Identifiers. The options are:

❏ Buttons, Images, Hyperlinks

❏ Buttons

❏ Images

❏ Hyperlinks

❏ Menu Items

❏ All

❏ Selected Only

When *All* is selected, objects that are not buttons, images, or hyperlinks become available for use as Trigger Identifiers. For example, with All selected, a text object can be a Trigger Identifier.

Selected only will make objects you have selected on the canvas appear in the Trigger Identifier list.

Requests/Actions list

A list that displays the order in which the requests, for the selected tasks, will execute any Wait for completion requests and any refresh requests. You can use Wait for completion requests to have certain requests execute before others. For example, if request1 is above the Wait for completion action and request2 is below that, then request1 will run first and when that is completed, request2 will then run.

Requests selections

Allows you to select which request(s) to add to the Requests/Actions list. Each request selected populates the Target type field with the available targets in which that request can be executed.

❏ **Run Request.** Displays the available reports, created in the Request & Data sources panel.

❏ **Schedule Request.** Displays the available reports, created in the Request & Data sources panel.

❏ **Run Animation.** Displays the available jQuery animations.

❏ **Execute Task.** Displays the available tasks, to enable you to execute one task from within another.

❏ **Refresh.** Creates an action to refresh the selected target.

❏ **Wait for completion.** Allows you to specify that one request musts complete before the next one begins.

❏ **JavaScript call.** Creates a JavaScript call action. When using the JavaScript call, different results occur depending on what is returned. If the JavaScript call returns 0, then all actions after it run. If the JavaScript call returns 1, then one action after the call is skipped, and all actions after that are then run. If the JavaScript call returns 2, all actions after it are skipped.

Delete

Deletes a request from the Requests/Actions list.

Move Up

Moves an item on the Requests/Actions list up one place.

Move Down

Moves an item on the Requests/Actions list down one place.

Target type

The Target type option contains a list of targets in which the request can be executed. These targets can be controls, frames, windows, or distribution methods when the Request is set to Schedule. These options can be different for specific actions.

The Window Target type will execute the request in a window.

The InfoWindow Target type will execute the request in the Db2 Web Query generated InfoWindow.

The Ajax call Target type will execute the request as an asynchronous call in JavaScript.

The Frame Target type will execute the request in a selected frame.

The Input Control Target Type is only available when you have a refresh request selected in the Requests/Actions list. This Target Type specifies that an input control will be refreshed. You specify which control should be refreshed by selecting one from the Target/Template Name list.

The active report Target Type is only available when you have a refresh request selected in the Requests/Actions list. This Target Type specifies that an active report will be refreshed. You specify the active report that will be refreshed by selecting one from the Target/Template Name list.

The Refresh BI Portal Target Type is only available when you have a refresh request selected in the Requests/Actions list. This Target Type specifies that a Db2 Web Query BI Portal page will be refreshed. If you select *All pages of the portal*, all BI Portal pages will be refreshed. If you select *Current page only*, only the current BI Portal page will be refreshed.

The JavaScript function Target Type is only available when you have selected *JavaScript call* in the Requests/Actions field. This Target Type specifies that the selected task will call a JavaScript function that you type in the Function name field.

Target/Template Name

Contains options that are associated with the Target type option.

Note: When you type a frame name in the Target/Template Name field, if the frame does not exist, a dialog box displays, asking if you want to create a new frame.

This field title changes to Function name under the following conditions:

❏ The Trigger Type *TBD* was selected.

❏ The Requests/Actions *JavaScript call* was selected.

❏ The Target type *JavaScript function* was selected.

In the Function name field, you can type the function that you want to call. You can also include any parameters that are required. Parameters must be included in parentheses and multiple parameters separated by commas. If you had previously typed a function in this field, you can select it from the drop-down menu.

Size (Width / Height)

Sets the Width and Height of an InfoWindow. Only available for the InfoWindow Target Type.

Current pages only

Refresh the current Db2 Web Query BI Portal page. Only available when there is a request to refresh the Db2 Web Query BI portal.

All pages of portal

Refresh all Db2 Web Query BI Portal pages. Only available when there is a request to refresh the Db2 Web Query BI portal.

Executing jQuery Animations Using the jQuery Animations Section

The jQuery Animations section, in the Tasks & Animations panel, allows you to execute jQuery animations on your HTML page. The animation effects in the jQuery Animations section can be used individually or in conjunction with each other to create more complex animations.

For more information, see *How to Animate an Object Using the Tasks & Animations Panel* on page 273.

List of

A list of all animations present in the HTML page. This list will change when animations are added or deleted.

New

Adds a new animation to the HTML page. The newly added animation appears in the List of area.

Delete

Removes an animation from the HTML page. The deleted animation is removed from the List of area.

All Targets

A list of all possible targets that can be used in the current animation being defined.

Selected Targets

A list of all targets that will be used in the current animation.

Add canvas selection

Adds the currently selected object on the canvas to the Selected Targets list. Multiple objects can be selected and added at the same time.

Add from list

Adds a selected object from the All Targets list to the Selected Targets list. Multiple objects can be selected and added at the same time.

Remove

Removes an object from the Select Targets list when that object is selected. Multiple objects can be selected and removed at the same time.

Visibility

You can select whether or not to animate the target so that it is hidden or displayed. You can also select not to animate the target for visibility. The options are No Change, Hide, and Show.

Effect

The Effect group contains animation effects and the options you can use to customize those animations. Animation effects can make the target bounce, shake, pulsate, and much more. You can also customize the options associated with that animation effect. The options correlate to the effect used. Such options can be how many times the target bounces after you specify that you want the bounce effect.

Type

A drop-down list where you can select the animation effect you want to use.

Option

You can select which option value you would like to edit, if available. For example, selecting the Bounce animation type allows you to select Distance and Times in the Option drop-down list. When you select Distance or Times, you can then edit the value of either of those options in the Value area. If you select Times and then enter 3 into the Value area, your object will bounce 3 times. The options available are different depending on the effect type you choose.

Value

You can enter a number that pertains to the currently selected option and that affects the animation. For example, if you select the *Slide* type and the *Distance* option, the number you enter in the Value area will be the distance the object will slide when it is animated. The value options available are different depending on the effect type you choose.

Effect text area

As the options for the effect are set, the syntax for that effect are displayed here. You can manually adjust this syntax to affect the animation.

Additional Properties

The Additional Properties group contains animations that change the location and size of the target and the components of that target (text size, text width, border size).

Name

A drop-down list where you can select different properties that can be animated. Such properties include height, opacity, font size, and others.

End value

The value for the property you choose from the Name field. For example, if you choose Left and enter 10, the object you are animating would move 10 pixels to the left. The reason you do not need to specify a start value is because the current location of the object is the start value.

IBM

Location setting

You can animate the target so that the location and size of that target is changed once the animation is activated.

Use location of selected target

This will use the location of the currently selected target. Coordinates, height, and width of the object are entered automatically into the Location setting box. This is used when the developer plans to move the selected target object and the animation will move it back.

Start location setting

Inserts an adjustable placeholder object into the canvas, at the location of the selected target. You can move and resize this placeholder object. This object represents where the animation will move and what size it will be when the animation is complete.

Set location settings

Sets the adjustable object location and size. This will be where the animated object moves and what size it will be. It is the ending location of the selected target.

Cancel setting

Cancels the setting of the location for the animation.

Additional Properties text area

Once the location has been determined using the Location setting commands, the syntax for that location is displayed here. You can manually adjust this syntax to affect the animation.

Duration

How quickly the animation will execute. You can choose either Slow or Fast.

Toggle animation

Allows you to revert to the pre-animation state. For example, if the target moves from right to left, the next invocation of the trigger will move the target back to the original position at the right.

Procedure: **How to Animate an Object Using the Tasks & Animations Panel**

The following are examples for how to create and animate an HTML page with jQuery animations.

Note: Any of the jQuery animations in the jQuery Animations section can be used individually or together. Using different jQuery animations together will create a more complex animation.

1. Create an HTML page.

2. Insert an image object and a button object onto the HTML page.

3. On the Tasks & Animations panel, in the jQuery Animations section, click *New*.

    A new animation is added to the animation list.

4. To make the image object, image1, the target, do one of the following:

    ❏ Select the image object on the canvas and click the *Add canvas selection* command.

    ❏ Select image1 from the All Targets area and click the *Add from list* command.

    ❏ Double-click on image1 in the All Targets area to move it to the Selected Targets area.

    The image object, image1, is added to the Selected Targets area.

5. Select Hide for the Visibility option.

    This animation option will cause the image to be hidden when the button is clicked.

6. On the Tasks & Animations panel, in the Tasks section, click *New*.

    A new task is added to the tasks list.

7. Select *Click* from the Trigger Type drop-down list.

8. From the Trigger Identifier list, select the button object, *button1*.

    The button will be used to activate the animation.

9. Select *Animation1* from the Requests/Actions option.

When the animation is activated using the button, the image will hide.

Procedure: **How to Select an Effect and Customize the Options for an Animation**

The following is an example for how to select an effect for an animation and customize the options for the effect.

Note: Any of the jQuery animations in the jQuery Animations section can be used individually or together. Using different jQuery animations together will create a more complex animation.

1. Create an HTML page.

2. Insert an image object and a button object onto the HTML page.

3. On the Tasks & Animations panel, in the jQuery Animations section, click *New*.

    A new animation is added to the animation list.

4. To make the image object, image1, the target, do one of the following:

   ❏ Select the image object on the canvas and click the *Add canvas selection* command.

   ❏ Select the image1 from the All Targets area and click the *Add from list* command.

   ❏ Double-click on image1 in the All Targets area to move it to the Selected Targets area.

   The image object, image1, is added to the Selected Targets area.

5. Select an Effect and customize the options for that effect.

   a. Select *Blind* from the Type drop-down list.

      This will cause the image to appear as if window blinds have been pulled over it.

   b. Select *Direction* from the Option drop-down list.

      Note: Each Effect has different options available to customize. Blind has the Direction option to customize where Bounce has the Distance and Times options.

   c. Select Horizontal from the Value field.

      This will cause the image to be animated with a horizontal blind effect.

      Note: The Value field will either be a drop-down list or a text entry box depending on what you selected for the Option drop-down list. For example, Direction will use this as a drop-down list while Times will use this as a text entry box.

6. On the Tasks & Animations panel, in the Tasks section, click *New*.

   A new task is added to the tasks list.

7. Select *Click* from the Trigger Type drop-down list.

8. From the Trigger Identifier list, select the button object, *button1*.

   The button will be used to activate the animation.

9. Select *Animation1* from the Requests/Actions option.

When the animation is activated using the button, the image will animate with a blind effect.

Procedure: **How to Create a Move or Resize Animation**

The following is an example for how to create a move or resize animation.

Note: Any of the jQuery animations in the jQuery Animations section can be used individually or together. Using different jQuery animations together will create a more complex animation.

1. Create an HTML page.

2. Insert an image object and a button object onto the HTML page.

3. On the Tasks & Animations panel, in the jQuery Animations section, click *New*.

    A new animation is added to the animation list.

4. To make the image object, image1, the target, do one of the following:

    ❑ Select the image object on the canvas and click the *Add canvas selection* command.

    ❑ Select image1 from the All Targets area and click the *Add from list* command.

    ❑ Double-click on image1 in the All Targets area to move it to the Selected Targets area.

    The image object, image1, is added to the Selected Targets area.

5. Use the Location setting options, found in the Additional Properties area, to animate the object so that it moves when activated.

    a. Click the *Start location setting* command.

        A placeholder object is shown overlapping your image.

    b. Move the placeholder object to a different location on the HTML page.

    c. Click the *Set location setting* command.

        This removes the placeholder object and adds syntax to the Additional properties text area. When your animation is activated, it will move to a different location that you specified using the placeholder object.

6. Select *Fast* from the Duration drop-down list.

    This will make the animation run at a faster speed.

7. Check the *Toggle Animation* check box.

    Checking the Toggle Animation check box will allow you to click the button after the animation has run and restore the image to its original state (in its original location and unhidden).

8. On the Tasks & Animations panel, in the Tasks section, click *New*.

    A new task is added to the tasks list.

9. Select *Click* from the Trigger Type drop-down list.

10. From the Trigger Identifier list, select the button object, *button1*.

    The button will be used to activate the animation.

11. Select *Animation1* from the Requests/Actions option.

When the animation is activated using the button, the image will move to the location that you set. If you click the button a second time, it will move back to the original location.

Procedure: **How to Create a Hyperlink in Developer Workbench to Drill to a Db2 Web Query Procedure**

You use the Hyperlink component and the Tasks & Animations panel to create hyperlinks. The following is an example of creating a hyperlink to drill to a Db2 Web Query procedure.

1. In the *Components* tab, in the *Generic Elements* group, click *Hyperlink*.

   A task is added to the Tasks section of the Tasks & Animations panel with a trigger type and trigger identifier already chosen.

2. In the Requests & Data sources panel, create a new external request for a Db2 Web Query Procedure.

3. Drag the request on to the canvas to create a frame.

4. In the Tasks section of the Tasks & Animations panel, in the Requests/Actions list, click the Requests selection command and select the request you created in step 2.

5. In the Trigger Type drop-down list, select *Frame*.

6. In the Target/Template Name drop-down, select the frame you created in step 3.

   When the HTML page is run and you click on the hyperlink, the Db2 Web Query Procedure you chose will run in the frame.

Procedure: **How to Create a Hyperlink in Developer Workbench to Execute a URL**

1. In the *Components* tab, in the *Generic Elements* group, click *Hyperlink*.

   A task is added to the Tasks section of the Tasks & Animations panel, with a trigger type and trigger identifier already chosen.

2. In the Requests & Data Sources panel, click the *New* drop-down arrow and select *Url Request*.

3. Type the URL in the Enter Url dialog box.

4. Navigate to the *Tasks & Animations* panel, select the task for the hyperlink, and verify that the trigger identifier displays the unique ID of the hyperlink object.

5. Click the arrow in the Requests/Actions section, select *Run Request*, and then select the URL request.

6. Select the Target type from the drop-down list (for example, *Window* or *Frame*).

7. Select the appropriate Target/Template name from the drop-down list.

Procedure: **How to Automatically Execute a Request When a Page is Loaded**

You can use the Requests and Tasks sections to create a request that executes when your HTML page is loaded. To do this, you must use the load task, in the Task Section. The load task is automatically added to every page and runs all selected requests when the page initially loads.

1. Select the *load* task.

2. Select the *Load* trigger type in the Trigger Type drop-down list. This option is selected by default for a load task.

3. Select a request from the Requests/Actions list.

4. Select the destination of the procedure by using the Target type drop-down list. For example, you can have the procedure run in a new window or in a report frame on the HTML page.

5. Select the frame in which the procedure will run. This option is only needed if you want to run the procedure, in a frame, on the HTML page, and not a new window.

Working with Requests & Data Sources

A request is a definition of something that can be executed. For example, you can execute a default procedure, explicit procedure, HTML page, URL, and more. If a request has parameters, those parameters will be indented and listed under the request name. By default, all parameters are sent to the Reporting Server. However, you can choose to not send a parameter in the request section by using the shortcut menu of the parameter, and clicking *Don't Send*. If you choose to not send a parameter, you must verify that your procedure can work without that value. For example, it contains a -DEFAULT value for the parameter.

When creating an external request that uses an existing file, the request inherits the name of that external file. For example, if you create an external request that uses an existing Db2 Web Query procedure called Parameter_Report, then your external request will inherit the name Parameter_Report.

Each request displays two folders beneath it, the Columns and Parameters folders. The Columns folder contains the fields used in the request. The Parameters folder contains the parameters used in the request. The Parameters folder is expanded by default, while the Columns folder is collapsed by default.

The Data Sources folder contains all data sources, used for default procedures and default control population requests, in the HTML page. Referenced data sources are not displayed in the Requests & Data sources panel. You can add additional data sources to the HTML page by embedding additional requests. You can also add additional data sources by right-clicking on the Data Sources folder, and using the shortcut menu to click *Add Data Source*.

You can refresh all requests, except empty requests, using the shortcut menu. Right-click a request and click *Refresh*.

You can also duplicate a request by using the shortcut menu, on the request you want to duplicate, and clicking *Clone*. When you clone an embedded request, you create a copy of the default request with the name, *name_1*. Name is the name of the original default request. For example, if you clone a request named SalesReport, the cloned request would be called SalesReport_1. Cloning an explicit request does not make a second copy of the referenced procedure.

You can also make edits to a selected procedure using the shortcut menu and clicking *Open in text editor*, as shown in the rightmost image below. For a default procedure, the new procedure tab opens with an extension of .vrt to show it is virtual. With a referenced procedure, the procedure tab opens with an extension of .fex.

In the Requests & Data sources panel, each icon next to an item indicates the type of request or parameter. For example, a parameter that was created in the current HTML page has the parameter symbol and an addition sign on it. Another example is how an external URL request has the word *URL* on it.

Two sample Requests & Data sources panels are shown in the following images.
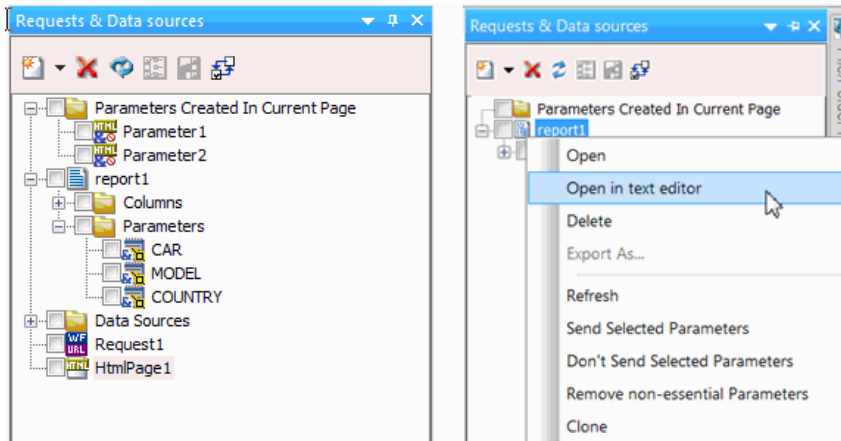
You can drag parameters and fields to the Settings panel to configure the selected component or control. You can drag requests from the Tasks & Animations panel to the HTML canvas, to create objects and controls for the request.

New request

Creates a new request. A request is an item that can be executed.

Delete

Deletes a selected request.

Refresh parameters

Refreshes the parameters used in the request.

Create Input Controls

Opens the New Parameter dialog box where you can select which input controls to create for parameters that are checked off in the Parameters list box, and if you want those controls chained to the parameter.

Save Selection

Saves the parameters you currently have checked off in the Parameters list box. You can choose to send more or less parameters than the procedure in the request requires. If you send less parameters, then you need to make sure the remaining parameters are handled by a -DEFAULT, -SET, or some other construct in the request.

Clear Selection

Clears all selected items in the Requests & Data sources panel.

Procedure: **How to Create a Default Procedure Request in Developer Workbench**

1. Add an embedded procedure to the Requests & Data source panel. Click the *New* drop-down arrow, point to *Embedded Request*, and then click *Import Existing* to embed an existing procedure file.

    The Open File dialog box appears, prompting you to select a procedure file.

    Note: The New Report, New Chart, or New Document options under *Embedded Request* are not supported.

2. Select a procedure file and click *OK*.

    Once you are done selecting the procedure, save and close the canvas.

3. In the Requests & Data sources panel, you may drag the request that you embedded on to the canvas.

    A procedure object is created on the canvas. If the procedure contains parameters, a form control is also added to the canvas. A task is created in the Tasks section of the Tasks & Animations panel.

Specifying Browser Defaults

You can use the Style Composer to control default settings for font, background properties, position mode, flow control, margins, list styles, and visual effects.

To access the Style Composer, right-click an object on your HTML page and select *Style* from the shortcut menu.

Reference: **Specifying Font Styles Using the Style Composer**

To specify the font styles that will be used in the browser for your HTML page, 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 are displayed in a browser.

The options include: *None, Underline, Strikethrough, Overline, Capitalization*.

Reference: **Specifying Background Properties Using the Style Composer**

To specify the background styles that will be used in the browser for your HTML page, 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 background color of the HTML page.

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*.

Note: When using a background image with scrolling enabled, you must specify the horizontal and vertical positions. If you do not specify these positions, your background image will not show. The horizontal and vertical positions are relative to the window and not the individual element.

Reference: **Specifying Text Styles Using the Style Composer**

To specify the text styles that will be used in the browser for your HTML page, 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 text.

You can specify: *Horizontal, Vertical, Justification*.

Spacing between

Determines the spacing between text.

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*.

Reference: **Specifying Position Mode Using the Style Composer**

To specify the position mode that will be used in the browser for your HTML page, 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.

Reference: **Specifying Layout Styles Using the Style Composer**

To specify the layout styles that will be used in the browser for your HTML page, 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*.

Reference: **Specifying Edge Styles Using the Style Composer**

To specify the margins, padding, and border styles that will be used in the browser for your HTML page, make your selections in the Edges window of the Style Composer.

The Edges window of the Style Composer is comprised of the following elements:

Margins

From which you can specify: *Top, Bottom, Left, Right*.

Padding

From which you can specify: *Top, Bottom, Left, Right*.

Reference: **Specifying List Styles Using the Style Composer**

To specify the list styles that will be used in the browser for your HTML page, 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*.

Reference: **Specifying User Interface Effects Using the Style Composer**

To specify the interface styles and visual effects that will be used in the browser for your HTML page, 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*.

Reference: **Specifying Border Settings Using the Style Composer**

To specify the Border settings that will be used in the browser for your HTML page, make your selections in the Border window of the Style Composer.

The Borders window of the Style Composer is comprised of the following elements:

Borders

Determines which border should be styled, what type of style should be applied, the width of the border, and the color of the border.

Rounded Border

Determines the whether your border corners will be rounded. You can style each individual corner to be rounded. You can also specify by how much they are rounded and whether all corners should be equally rounded.

Chaining in the HTML Canvas

You may chain controls to one another on the Parameters tab and apply conditions to links in the chain. Chaining will populate controls based on the selected value from the prior control in the chain. You can chain static and dynamic controls, link or unlink parts of a chain, and create conditions on links in a chain. Chains are represented by lines connecting control objects on the Design or Parameters tab. Note that chaining is applicable only for controls, not parameters.

Note:

❏ Although you may chain controls from the Design tab, you may only create conditions to links in the chain through the Parameters tab.

By clicking the arrow head in a link of a chain, the Settings panel enables you to modify and set the properties and conditions of the chain.

❏ If using static controls, you must apply conditions for each link in the chain. Conditions need to be created for each value of the control chained from, and those values must be mapped to the correct value(s) that will be displayed in the control that it is being chained to.

Automatically Chaining Parameters From the New Parameters Dialog Box

The auto chain option enables you to automatically chain selected controls from the New Parameters dialog box. Chaining populates controls based on the selected value from the prior control in the chain. The auto chain option is useful since it creates the chain, or links of a chain, automatically.

Note: Automatic chaining creates a basic chain with default functionality that does not include any conditions. You may create conditions for a chain through the Parameters tab.

When importing or referencing a report with parameters to an HTML page, the controls are not chained by default. You may choose to include or exclude individual controls in a chain with the Chain control column from the New Parameters dialog box.

Additionally, when the auto chain option is selected, a separator is added to the parameters list on the New Parameters dialog box. A separator is used to separate controls into multiple chains and can be moved up or down in the chain sequence.

Procedure: How to Auto Chain Controls From the New Parameters Dialog Box

The auto chain option creates the chain, or links of a chain, automatically. When the auto chain option is selected, a separator is added to the parameters list. A separator is used to separate controls into multiple chains and can be moved up or down in the chain sequence.

1. In the HTML canvas, import or reference a report with parameters.

   The New Parameters dialog box opens.

2. Select *Auto chain controls in above specified order*.

   Tip: You may use the up or down arrows to change the order of the selected control before selecting this option.

The Chain control option is selected for all controls and a separator is added as the last object to the list of parameters.

3. To create multiple chains, click the separator row and use the up or down arrows to change the location of the separator in the chain.

   Note: If the default separator is moved up, another separator is added to the end of the list.

4. Click *OK* to close the New Parameters dialog box and add the control to the HTML page.

   On the Design tab, when creating multiple chains from the New Parameters dialog box, each set of chained parameters appears on a new line, regardless of the grouping option selected from the New Parameters dialog box. This behavior is set through the *Start each chain on a new line* option, located on the Form Settings dialog box in the HTML Page tab of the Developer Workbench Options dialog box. This enables you to see the relationship of the chains within the form. Start each chain on a new line is selected by default.

   On the Parameters tab, chains are represented by lines connecting control objects.

5. You can remove a chain by clicking *Break binding* from the shortcut menu when a chain link is selected.

Procedure: How to Chain Controls From the New Parameters Dialog Box

The Chain control column enables you to include or exclude individual controls in a chain, from the New Parameters dialog box.

1. In the HTML canvas, import or reference a report with parameters.

   The New Parameters dialog box opens.

2. Select the *Chain control* check box for the controls to be included in the chain.

   The controls are chained in the order that they appear on the New Parameters dialog box. You may use the up or down arrows to change the order of the selected control before chaining controls.

   Note: If a control is excluded from a chain, the chain automatically links only the selected controls.

3. Click *OK* to close the New Parameters dialog box and add the control to the HTML page.

On the Design tab, when creating multiple chains from the New Parameters dialog box, each set of chained parameters appears on a new line, regardless of the grouping option selected from the New Parameters dialog box. This behavior is set through the *Start each chain on a new line* option, located on the Form Settings dialog box in the HTML Page tab of the Developer Workbench Options dialog box. This enables you to see the relationship of the chains within the form. Start each chain on a new line is selected by default.

On the Parameters tab, chains are represented by lines connecting control objects.

4. You can remove a chain by clicking *Break binding* from the shortcut menu when a chain link is selected.

Creating Pop-Up Controls

For a DOCUMENT object, you can use pop-up controls to:

❏ Create modern looking controls that will run seamlessly on any device.

❏ Create controls that look the same, but offer different functionality based on single or multiple value selection.

To create pop-up controls, you can:

❏ Set the Create popup controls property on the Properties panel for the DOCUMENT object.

❏ Use the Popup style right-click shortcut option, which allows a control to be an original or a pop-up style control. (Disabled for Edit Box, Text Area, Calendar, and Slider).

❏ Use the New Parameters dialog box. This is available as:

  ❏ An item in the Parameters grid as a check box (Popup).

  ❏ A check box to select all parameters in the grid to be a Popup Control (Create popup controls).

The Create popup controls property on the Properties panel is shown in the following image. Possible values are Yes and No. No is the default value.

Properties	▼ 🔲 ✕
🔲 DOCUMENT	⌄

⊟ **Document**	
⊟ **Page properties**	
Theme	Neutral
Ignore filters	_FOC_NULL
Use appl.css	No
⊟ **Restorable Parameter Sets**	
Auto Restore Parameter Values	
Parameter Page Name	
Maximum number of Saved sets	25
⊟ **Miscellaneous**	
Charset	UTF-8
Page Title	HtmlPage
Loading screen	
Check reporting servers access	Yes
Html alias	Custom
Accessible	
Map type	Google map
Global search/paging control	
Autoplay control	
Parameters tab: auto arrange	
Dynamic Styling	
Mobile	No
Responsive design	No
Global responsive margin	
New container margin	4
Create popup controls	No
	Yes
	No
⊟ **Body**	
⊟ **Core attributes**	
Unique Identifier	
Class identifier (CSS)	IBI_PageBg
Title	HtmlPage

The Popup style right-click shortcut option for a drop-down control is shown in the following image. (Disabled for Edit Box, Text Area, Calendar, and Slider).
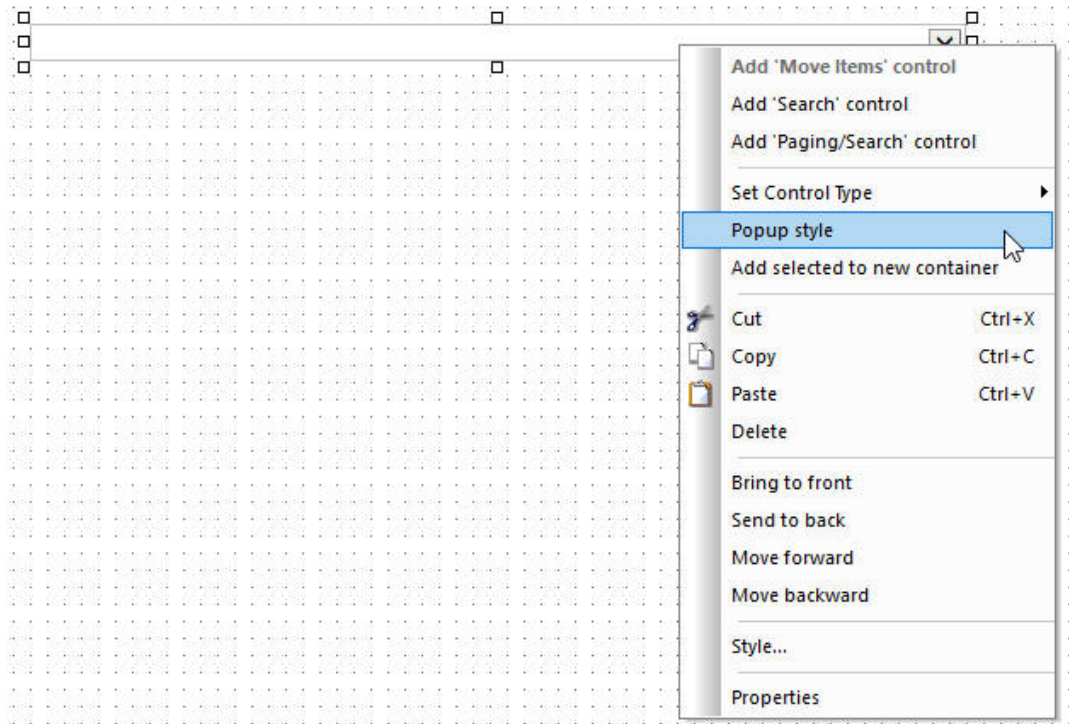
The New Parameters dialog box is shown in the following image. This options is available as a item in the Parameters grid as a check box (Popup) and also as a check box to select all parameters in the grid to be a Popup Control (Create popup controls).
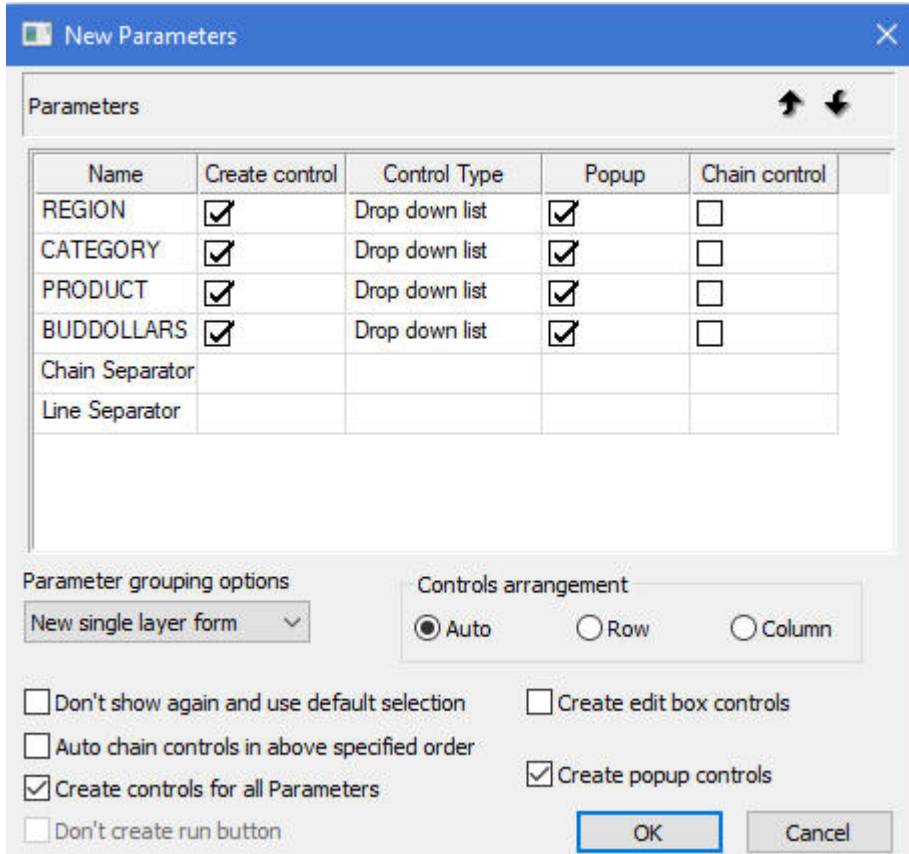
The following image shows an example of a pop-up control.
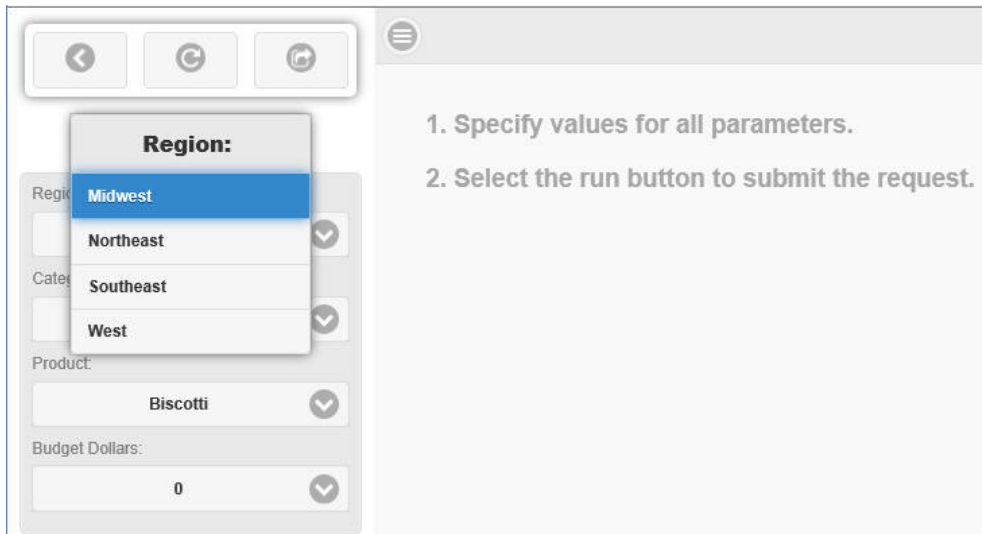
Using the Chain Separator and Line Separator

The chain separator and line separator options allow for chains and parameters to be grouped or split depending on which settings are turned on. When used, the two separators do the following:

Chain separator. When *Start each chain on a new line* is on, this separator creates new chain groups on new lines. When *Start each chainon a new line* isoff, this separator starts a new chain wherever it is placed in the New Parameters dialog box. The controls are positioned in one row and wrap at the end of the form.

Note: *Start each chain on a new line* is turned on, by default. This option can be found in the Form Settings dialog box, in the HTML Page section of the Developer Workbench Options dialog box, as shown in the following image.
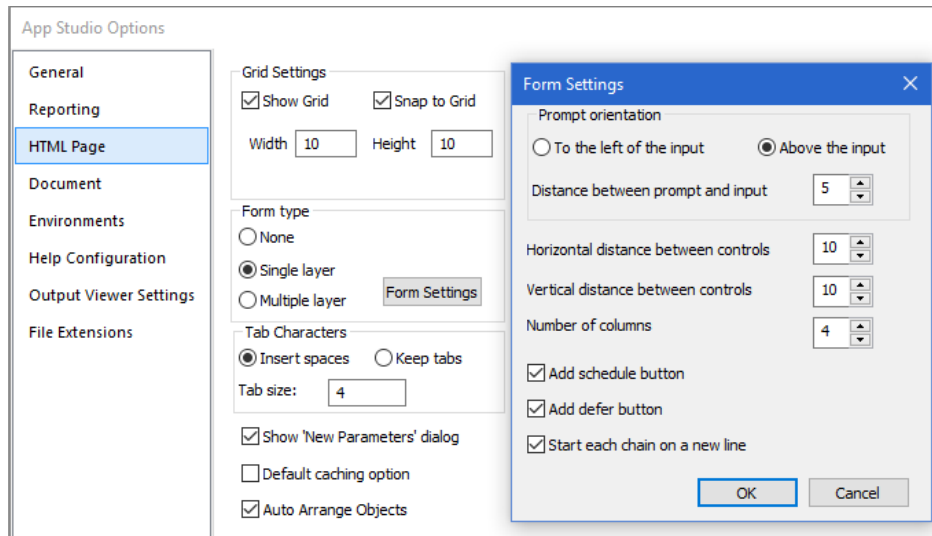
Line separator. This separator creates a line break wherever it is placed.

Chaining Controls on the Parameters Tab

Chaining enables you to associate two or more related values. When you chain controls together, chained 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 value will be available instead of all the plants in the data source. Each time a selection is made, all chained controls that come after will be dynamically updated. Chaining also enables you to add, remove, and reverse the order of controls in the chain.

Values 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.

If a parameter has two or more incoming bindings, the value selected last will be displayed as the parameter.

Note: When a parameter is populated by two or more controls, the value of the last control used will be assigned to the parameter.

Procedure: **How to Chain Controls on the Parameters Tab**

1. Create an HTML page using input controls to supply parameter values.

2. Click the *Parameters* tab.

3. Select the center of the control object and drag the control to the center of the next control object in the chain.

    Repeat this step for each link in the chain.

4. Optionally, apply condition settings to the chain to determine how parameters are populated.

    If using static controls, you must apply conditions for each link in the chain. Conditions need to be created for each value of the control chained from, and those values must be mapped to the correct value(s) that will be displayed in the control that it is being changed to.

Procedure: **How to Remove a Link in the Chain**

1. While on the Parameters tab, select the arrow head on the line so that the line is bold.

2. Right-click and select *Break binding*.

3. Repeat this step for each link in the chain that you want to break.

Procedure: **How to Reverse the Order of Chained Controls**

1. While on the Parameters tab, select the arrow head on the line so that the line is bold, right-click and select *Break binding*.

2. Select the center of the control object and drag the control to the center of the next control object in the chain.

    Notice the direction of the arrow between the control objects. You may reverse the direction of the link in the chain or reverse the order of the chain by changing the direction of each link.

    ❏ If reversing the direction of a link in the chain, drag the control object in the desired order.

    ❏ If reversing the order of a chain, drag the control objects in the desired order.

Tip: In some scenarios, when reversing the order of chained values, you may want to move the controls from the default location on the Parameters tab so that you can better see the direction of the chain. Moving objects on the Parameters tab will not affect the Design view of your layout.

❏ Press the Shift key and select the control object and bound parameter to move the objects as a set.

❏ Chain the control objects together.

3. Optionally, apply condition settings to the chain to determine how parameters are populated.

Arranging Controls

You can choose the arrangement of controls in the New Parameters dialog box, without any additional steps, after the parameter controls are generated. This gives you the flexibility to design your HTML page, without the need for moving the controls after creation.

The Controls arrangement option, as shown in the following image, indicates the placement of controls.
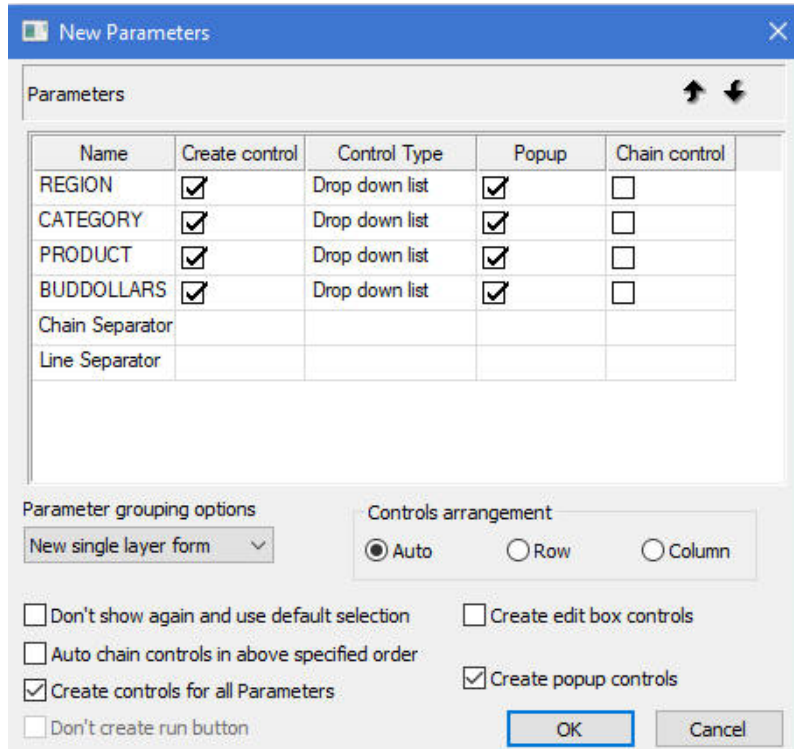
The following are possible values for the Controls arrangement option:

❏ **Auto.** This value depends on the value in the Number of columns property in the Form settings dialog box. This dialog box is in the HTML Page section in the Developer Workbench Options dialog box. Based on this value, the controls will fold to the next line. The valid values for Number of columns are 1 to 99. Auto is the default value.

❏ **Row.** Places the controls horizontally within the form.

❏ **Column.** Places the controls vertically within the form.

Applying Conditions to a Chain

A chain contains conditions for each link in the chain. The conditions are linked to the values being selected in the control object. You may apply multiple conditions to one link. The settings for the condition describe how the link should behave. The following options are available:

❏ Apply *Actions* for the links on the chain.

❏ Apply the *Values compare operator* for the condition.

❏ Apply *Selected values* with a *Multiselect operator* for the condition.

❏ Apply the *Resolves parameter* values for the condition.

❏ Apply the *Compare operator* for the condition.

If using static controls, you must apply conditions for each link in the chain. Conditions need to be created for each value of the control chained from, and those values must be mapped to the correct values that will be displayed in the control that it is being changed to.

Reference: ## Settings Panel (Conditions)

The Settings panel appears when creating a condition for a chain link on the Parameters tab. A chain contains conditions for each link in the chain.

The conditions are linked to the values being selected in the control object. The settings for the condition inherit the values of the prior bound control and provide additional condition settings. This section describes the additional condition settings.

The Settings panel contains the following fields and options when creating a condition:

Conditions

The conditions list enables you to create multiple conditions for the link. *Default* is the only initial condition.

❏ To create a new condition, click the *New* icon. Condition*(n)* is created, where *(n)* is the number, and added to the Conditions drop-down list. You may type in a unique condition name, choose Selected values, and set the condition settings.

❏ Click the *Delete* icon to remove the selected condition from the list. Note that the *Default* condition name cannot be deleted.

Actions

Select an action for the chain link to control. The options offer variations to populate, show, hide, execute, and select the values. The list of available options are:

❑ **Populate, show.** Populates the control and displays it at run time. This is the default action for all conditions.

❑ **Populate, hide.** Populates the control and does not display it at run time.

❑ **Populate with alternate, show.** Populates the control with alternate values derived from a procedure, or value list, that is not the default and displays the control at run time.

❑ **Populate with alternate, hide.** Populates the control with alternate values derived from a procedure, or value list, that is not the default and does not display the control at run time.

❑ **Show.** Shows the control, but does not populate it.

❑ **Hide.** Hides the control, but does not populate it.

❑ **Execute.** Executes the bound object. For example, if you bind a control to a Submit button and change the value in the control at run time, the report/chart automatically executes when you change the value, without having to click the Submit button.

❑ **Select.** Selects the bound object. For example, if you bind a control to a tab item and select a value in the control at run time, the bound object (the tab item) is automatically selected as the active tab on the page.

❑ **Use As Target.** This action is used in conjunction with a control whose values are procedure names or HTML file names. Selecting this option, binds the control to a frame for the output.

Values compare operator

Values compare operator provides chaining logic scenarios to include such options as Equal, Not Equal, Greater Than, Less Than, and so on. This option sets the condition for how to populate the control being linked to.

Equal is the default Values compare operator.

Selected values

Selected values enable you to provide the values used in the condition. When creating a new condition, the Selected values section is activated. You may type selected values in the input box or click the ellipsis button to select values from the list.

The list of values that appears is based on the values of the prior bound control in the chain.

When selected values are entered, the Multiselect operator field is activated.

Multiselect operator

The Multiselect operator options are activated when selected values are entered for the condition. Options are *One of* or *All of*. One of is based on one of the values shown in the Selected values, being selected in the prior control, in the chain. *All of* is based on the value of all of the Selected values, being selected in the prior control, in the chain.

One of is the default multiselect operator.

Resolves parameter ("To:" field is required)

From. The From field specifies where to get the value used in the To field, if the control being chained from is an activeX control.

To. The To field is used to dynamically generate the selection list used to populate the control being chained to. This field displays the parameter whose value will drive the condition evaluation. The parameter name linked to the prior control in the chain is displayed by default. The ellipsis button provides a pop-up dialog of the other parameter values (from the report) to be resolved.

Compare operator

The compare operator provides parameter chaining logic scenarios to include such options as Equal, Not Equal, Greater Than, Less Than, and so on. This sets the compare operator to populate the control.

Equal is the default Compare operator.

Procedure: **How to Create a New Condition**

1. Insert a report with parameters in the HTML canvas.

2. Drag control objects on the Parameters tab to create a chain.

   Chains are represented by lines connecting control objects on the Parameters tab.

   Chaining controls will populate parameters with values at run time, based on values selected in prior controls on the chain.

3. Click a link in the chain.

   The Settings panel shows the bound control values and the *Default* condition settings for the link in the chain.

4. Click the *New* icon to create a multiple condition for the chain.

Condition*(n)* is created, where *(n)* is the number, and added to the Conditions drop-down list, and the Selected values section is activated. You may type in a unique condition name.

5. You may type in a unique condition name, choose Selected values, and set the condition settings for the new condition.

6. Optionally, you may click the *Delete* button to remove the selected condition from the list.

   Note: *Default*, the initial condition, cannot be deleted.

Procedure: **How to Select the Action for a Condition**

1. Insert a report with parameters in the HTML canvas.

2. Drag control objects on the Parameters tab to create a chain.

   Chains are represented by lines connecting control objects on the Parameters tab.

   Chaining controls will populate parameters with values at run time, based on values selected in prior controls on the chain.

3. Click a link in the chain.

4. In the Setting panel, select the action for the condition from the *Actions* drop-down list. For example, to hide the control being chained to, select *Hide*.

   Populate, show is the default option.

When running the HTML page, the action for the chained control is applied.

Procedure: **How to Select the Values Compare Operator for a Condition**

1. Insert a report with parameters in the HTML canvas.

2. Drag control objects on the Parameters tab to create a chain.

   Chains are represented by lines connecting control objects on the Parameters tab.

   Chaining controls will populate parameters with values at run time, based on values selected in prior controls on the chain.

3. Click a link in the chain.

4. In the Setting panel, select the chaining logic for the parameter being chained to, in the condition, from the Values compare operator drop-down list.

   Equal is the default option.

The compare operator is applied to the value selected.

Procedure: How to Apply Selected Values With a Multiselect Operator to a Condition

1. Insert a report with parameters in the HTML canvas.

2. Drag control objects on the Parameters tab to create a chain.

    Chains are represented by lines connecting control objects on the Parameters tab.

    Chaining controls will populate parameters with values at run time, based on values selected in prior controls of the chain.

3. Click a link in the chain.

4. In the Settings panel, click the *New* icon to create a multiple condition for the chain.

    Condition*(n)* is created, where *(n)* is the number, and added to the Conditions drop-down list, and the Selected values section is activated. You may type in a unique condition name.

5. You may type selected values in the input box or click the ellipsis button to select values from the list.

    The list of values that appears is based on the values of the prior bound control in the chain.

    Tip: You may also use the pop-up icons to select a field and close the pop-up dialog box. The green icon is OK, the red icon is Cancel, double-clicking a value will select the value and close the dialog box without using any button, and pressing the Esc key will cancel the dialog box without using any button.

    When selected values are entered, the Multiselect operator field is activated.

6. Select the chaining logic for the selected values from the *Multiselect operator* drop-down list.

    ❏ *One of* is based on one of the values shown in the Selected values, being selected in the prior control, in the chain. This is the default selection.

    ❏ *All of* is based on the value of all of the Selected values, being selected in the prior control, of the chain.

The selected values and multiselect operator are applied to the condition.

Procedure: **How to Resolve Parameters for a Condition**

It is recommended that you populate the controls first, before chaining. When you populate first, certain information is obtained that allows the determination of the best choice for *Resolves parameter*. If you chain first and then populate, the information cannot be obtained because the chaining is already established. If you chain first, you must manually set *Resolves parameter*.

1. Insert a report with parameters in the HTML canvas.

2. Drag control objects on the Parameters tab to create a chain.

   Chains are represented by lines connecting control objects on the Parameters tab.

   Chaining controls will populate parameters with values at run time, based on values selected in prior controls on the chain.

3. Click a link in the chain.

4. In the Setting panel, click the *Resolves parameter* ellipsis button to select a parameter name to resolve. If a custom procedure that has a filter (or filters) populates the control, the values list shows the parameters from the filters. If a data source populates the control, the values list shows all of the fields from the data source shown in the Object Inspector.

   The value in the Resolves parameter field should be either:

   ❏ The field name that limits the values for the next control in the chain, if a data source populates the control.

   ❏ The parameter name from the procedure, if a procedure populates the control.

   In most cases, this value will be populated by default and will not need to be changed.

   Tip: You may also use the pop-up icons to select a field and close the pop-up dialog box. The green icon is OK, the red icon is Cancel, double-clicking a value will select the value and close the dialog box without using any button, and pressing the Esc key will cancel the dialog box without using any button.

The parameter value is resolved in the chain if no filter is specified.

Procedure: How to Select the Compare Operator for the Parameter

An example of when to apply chaining logic is when 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 parameter compare operator, 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.

1. Insert a report with parameters in the HTML canvas.

2. Drag control objects on the Parameters tab to create a chain.

    Chains are represented by lines connecting control objects on the Parameters tab.

    Chaining controls will populate parameters with values at run time, based on values selected in prior controls on the chain.

3. Click a link in the chain.

4. In the Setting panel, select the chaining logic option from the Compare operator drop-down list. This sets the compare operator to populate the control.

    Equal is the default option.

The compare operator is applied to the parameter selected.

Procedure: How to Enable Cache Processing for Chained Values

You may enable cache processing for chained values in two ways:

❏ Enable the caching option for the HTML page and all objects on the page.

❏ Enable cache run time data for a dynamic control or a condition.

   A chain contains conditions for each link in the chain. The conditions are linked to the values being bound to the control object. If you change the options for the condition, it will also be applied to the control, and vice versa.

Caching options are turned off by default.

1. To enable caching options for all objects on the HTML page, in the Developer Workbench Options dialog box:

    ❏ Select the *HTML Page* tab.

    ❏ Select *Default caching option*.

    ❏ Click *OK* to close the Developer Workbench Options dialog box.

2. To enable caching options for a dynamic control or condition:

A chain contains conditions for each link in the chain. The conditions are linked to the values being chained to the control object. If you change the options for the condition, it will also be applied to the control, and vice versa.

For a dynamic control:

1. Create a dynamic input control to supply parameter values.

2. Select the dynamic control object from the Parameters tab.

   The Settings panel opens, showing the dynamic control options.

3. Select *Cache run time data* to cache the run time data for the selected input control.

   Note: This setting overrides the *Default caching option* from the HTML Page tab, in the Developer Workbench Options dialog box.

4. Select the center of the control object and drag the control to the center of the next control object.

   When binding controls, the conditions inherit the values set in the dynamic control settings.

For a dynamic condition:

1. Click a link on the chain to open the Settings panel for the condition.

2. Select *Cache run time data* to cache the run time data for the selected input control. This option is only available for dynamic controls.

   This setting overrides the *Default caching option* from the HTML Page tab, in the Developer Workbench Options dialog box.

When running the HTML page, data for the chained value is cached to improve performance.

Using JavaScript Code with HTML Canvas Pages

Although the HTML canvas is fully integrated with JavaScript, it is suggested that you do not create custom JavaScript that manipulates the HTML canvas generated controls, as Db2 Web Query cannot support such custom JavaScript code. Additionally, there is no guarantee that the JavaScript code will work correctly in future releases.

Note:

❏ The HTML canvas run time is a set of JavaScript files. You should not be calling the functions within them directly. Should you call these functions directly, your code may not function in future releases and IBM cannot be held responsible.

❑ If you want JavaScript to run after a page loads completely, but before any reports are executed, you need to create a function called onInitialUpdate (this function was called onInitialUpdate() in previous releases). If you have another function, in that function, you can call IbComposer_onInitialUpdate(). Your code should be added inline in the HTML file after the line:

```
//End function window_onload
```

The HTML canvas run time will call onInitialUpdate() if it exists.

Function: IbComposer_removeSelectOption

IbComposer_removeSelectOption removes values from a listbox, drop down, or the 'from' listbox of a double list control.

Syntax: How to Remove Listbox or Drop Down Values

```
IbComposer_removeSelectOption(controlID,arr[]);
```

controlID

Alphanumeric

Is the unique identifier of the control from which values are obtained.

arr[]

Alphanumeric Array

Is the array that contains single or multiple values.

Note: When the IbComposer_removeSelectOption method is called, the values specified by the second parameter (arr[]) will be removed.

Example: Removing Values From a Listbox

```
function button1_onclick(event) {
   var eventObject = event ? event : window.event;
   var ctrl = eventObject.target ? eventObject.target :
   eventObject.srcElement;
   // TODO: Add your event handler code here
   var readVals = [];
   readVals = IbComposer_getCurrentSelection('listbox1');
   IbComposer_removeSelectOption('combobox1',readVals);
}
```

Function: IbComposer_runAnimation

IbComposer_runAnimation runs the animation defined by the user in the Tasks and Animations panel.

Syntax: **How to Run an Animation**

```
IbComposer_runAnimation(name);
```

```
name
```

Alphanumeric

Is the name of the animation specified by the user in the Tasks and Animations panel.

Example: **Running an Animation**

```
function submit1_onclick(event) {
    IbComposer_runAnimation('animation1');
}
```

Function: IbComposer_triggerExecution

IbComposer_triggerExecution allows the user to execute a specific task defined in the Tasks section of the Tasks and Animations panel.

Syntax: **How to Execute a Specific Task**

```
IbComposer_triggerExecution(taskName,index);
```

where:

```
taskName
```

Alphanumeric

Is the name of the task specified by the user in the Tasks section of the Tasks and Animations panel in the HTML canvas.

```
index
```

Numeric

Is the task number starting from the number 1.

Example: **Executing a Task**

```
function submit1_onclick(event) {
    IbComposer_triggerExecution('task1',1);
}
```

Function: IbComposer_getRequestRefProcedure

IbComposer_getRequestRefProcedure returns the name of the procedure, given the unique identifier of the request referencing the procedure. This was formerly available in the myXmlRoot document.

Syntax: **How to Return the Procedure Name**

```
IbComposer_getRequestRefProcedure(Drilldown);
```

where:

Drilldown

Alphanumeric

Is the name of the request provided.

Example: **Returning a Procedure Name**

```
function button1_onclick(event) {
    IbComposer_getRequestRefProcedure('Drilldown');
}
```

Function: IbComposer_getRequestTarget

IbComposer_getRequestTarget returns the name of all the target frames used by the request. This was formerly available in the myXmlRoot document.

Syntax: **How to Return the Target Frame Names**

```
IbComposer_getRequestTarget(Drilldown,bShow);
```

where:

Drilldown

Alphanumeric

Is the name of the request provided.

bShow

Boolean

Is an operator that can be set to true to show the names, or set to false to hide them.

Example: **Returning Target Frame Names**

```
function button1_onclick(event) {
    IbComposer_getRequestTarget('Drilldown','true');
}
```

Function: IbComposer_populateDynamicCtrl

IbComposer_populateDynamicCtrl allows you to determine from which control data is drawn from to populate a destination control.

Syntax: **How to Populate a Control Dynamically**

```
IbComposer_populateDynamicCtrl(controlID,fromControlId);
```

where:

controlID

Alphanumeric

Is the unique identifier of the control from which values are obtained.

fromControlId

Alphanumeric

Is the unique identifier of the control from which values are obtained, when two or more controls are chained to a destination control. For example, if listbox1 and listbox2 are both chained to listbox3, by default, the values of listbox3 will be determined by listbox1. You can use IbComposer_populateDynamicCtrl('listbox3','listbox2') to make listbox2 determine the values of listbox3. This identifier is optional.

Example: **Populating a Control Dynamically**

```
function button3_onclick(event) {
    var acc = IbComposer_populateDynamicCtrl('listbox3','listbox2');
    acc.selectNextPage();
}
```

Function: IbComposer_getComponentById

IbComposer_getComponentById obtains a component by using its ID.

Syntax: **How to Obtain a Component by Using Its ID**

```
IbComposer_getComponentById(controlID);
```

where:

controlID

>   Alphanumeric

>   Is the unique identifier of the control from which values are obtained.

Example: **Obtaining the Accordion Report By Using Its ID**

```
function button3_onclick(event) {
    var acc = IbComposer_getComponentById('accordion1');
    acc.selectNextPage();
}
```

Function: IbComposer_getCurrentSelection

IbComposer_getCurrentSelection retrieves the current selected values from a control.

Syntax: **How to Retrieve the Current Selected Value From a Control**

```
IbComposer_getCurrentSelection(controlID,[layer],[bDateObj]);
```

where:

controlID

>   Alphanumeric

>   Is the unique identifier of the control from which values are obtained.

layer

>   Integer

>   Is an optional parameter used to specify the layer number in a Multi source Tree control if a Multi source Tree control is being used. The layer number starts with 1 for the first layer.

bDateObj

>   Boolean

>   Is an optional operator that can be set to true to return a date object or to false to return a string. If you do not specify a value for this operator, it returns a string.

Example: **Retrieving the Current Selected Value for a Drop Down List**

```
function button1_onclick(event) {
   var values = IbComposer_getCurrentSelection('combobox1');
   for(var I = 0; i < values.length; i++)
   alert(values[i]);
}
```

Function: IbComposer_getCurrentSelectionEx

IbComposer_getCurrentSelectionEx retrieves the current selected actual or display values from a control. The function can also be used to get the index values for List Boxes, Drop Down Lists, and Double Lists.

Syntax: **How to Retrieve the Current Selected Value, Actual Value, or Display Value From a Control**

```
IbComposer_getCurrentSelectionEx(controlID,[layer]);
```

where:

controlID

> Alphanumeric
>
> Is the unique identifier of the control from which values are obtained.

layer

> Integer
>
> Is an optional parameter used to specify the layer number in a Multi source Tree control if a Multi source Tree control is being used. The layer number starts with 0 for the first layer.

Example: **Retrieving the Current Selected Index Value, Actual Value, and Display Value for a Multi-Select List**

```
function button1_onclick(event) {
   var values = IbComposer_getCurrentSelectionEx('combobox1');
   for(var i = 0; i < values.length; i++)
   {
   alert("Index Value: " + values[i].getIndex() + "\n" +
   "Actual Value: " + values[i].getValue() + "\n" +
   "Display Value: " + values[i].getDisplayValue());
}
```

Function: IbComposer_getClickedRow

IbComposer_getClickedRow retrieves the current clicked row of an HTML table.

Syntax: **How to Retrieve the Current Clicked Row of an HTML Table**

```
IbComposer_getClickedRow(clickEventTarget);
```

where:

clickEventTarget

    Alphanumeric

    Is the name of the event for the clicked row.

Example: **Retrieving the Current Clicked Row of an HTML Table**

```
function IbComposer_getClickedRow(clickEventTarget) {
    return getCurrentClickedRow(clickEventTarget);
}
```

Function: IbComposer_getClickedColumn

IbComposer_getClickedColumn retrieves the current clicked column of an HTML table.

Syntax: **How to Retrieve the Current Clicked Column of an HTML Table**

```
IbComposer_getClickedColumn(clickEventTarget);
```

where:

clickEventTarget

    Alphanumeric

    Is the name of the event for the clicked column.

Example: **Retrieving the Current Clicked Column of an HTML Table**

```
function IbComposer_getClickedColumn(clickEventTarget) {
    return getCurrentClickedColumn(clickEventTarget);
}
```

Function: IbComposer_getClickedCellValue

IbComposer_getClickedCellValue retrieves the current clicked cell value of an HTML table.

Syntax: **How to Retrieve the Current Clicked Cell Value of an HTML Table**

```
IbComposer_getClickedCellValue(clickEventTarget);
```

where:

clickEventTarget

    Alphanumeric

    Is the name of the event for the clicked cell.

Example: **Retrieving the Current Clicked Cell Value of an HTML Table**

```
function IbComposer_getClickedCellValue(clickEventTarget) {
    return getClickedCellValue(clickEventTarget);
}
```

Function: IbComposer_setCurrentSelection

IbComposer_setCurrentSelection sets the current selected values for control parameters.

Syntax: **How to Set the Current Selected Value for a Control**

```
IbComposer_setCurrentSelection(controlID,arrValues,bUpdateDependencies);
```

where:

controlID

    Alphanumeric

    Is the unique identifier of the control for which to set the values.

arrValues

    Array

    Is the array of values to be set.

bUpdateDependencies

    Boolean

    Is an operator that can be set to true to update chained controls and triggered events. The default is false.

Example: **Setting the Current Selected Value for a List Box**

```
function button2_onclick(event) {
    var arr = [];
    arr.push('ITALY');
    arr.push('JAPAN');
    IbComposer_setCurrentSelection('listbox1',arr,'false');
}
```

Function: IbComposer_execute

IbComposer_execute executes a report or chart.

Syntax: **How to Execute a Report or Chart**

```
IbComposer_execute(reportID,[outputTarget]);
```

where:

reportID

Alphanumeric

Is the unique identifier of the report or chart to execute.

outputTarget

Alphanumeric

Is the optional parameter to set the target of the output. Is one of the following:

❑ The name of a frame.

❑ *'_blank'*.

❑ *'_target'*.

❑ The name of a new window.

Example: **Executing a Report in a New Window**

```
function button3_onclick(event) {
    IbComposer_execute('report1','newwin');
}
```

Function: IbComposer_isSelected

IbComposer_isSelected determines if a control or value is selected.

Syntax: **How to Determine If a Control or Value Is Selected**

```
IbComposer_isSelected(controlID,[testValue]);
```

where:

controlID

Alphanumeric

Is the unique identifier of the control being tested.

IBM

testValue

    Alphanumeric

Is the optional parameter the control is being checked against.

Example: **Determining If a Check Box Is Selected**

```
function checkbox1_onclick(event) {
    var curValue = IbComposer_isSelected('checkbox1');
    IbComposer_showHtmlElement('form1',curValue);
}
```

Function: IbComposer_showHtmlElement

IbComposer_showHtmlElement shows or hides an HTML element.

Syntax: **How to Show or Hide an HTML Element**

```
IbComposer_showHtmlElement(elementID,bShow);
```

where:

elementID

    Alphanumeric

Is the unique identifier of the element which is shown or hidden.

bShow

    Boolean

Is an operator that can be set to true to show the element and false to hide it.

Example: **Showing or Hiding a Check Box**

```
function checkbox1_onclick(event) {
    var curValue = IbComposer_isSelected('checkbox1');
    IbComposer_showHtmlElement('form1',curValue);
}
```

Function: IbComposer_enableHtmlElement

IbComposer_enableHtmlElement enables or disables an HTML element.

Syntax: **How to Enable or Disable an HTML Element**

```
IbComposer_enableHtmlElement(elementID,bEnable);
```

where:

elementID

> Alphanumeric

> Is the unique identifier of the element which is enabled or disabled.

bEnable

> Boolean

> Is an operator that can be set to true to enable the element and false to disable it.

Example: **Enabling or Disabling Elements**

```
function checkbox2_onclick(event) {
    IbComposer_enableHtmlElement('listbox1',
IbComposer_isSelected('checkbox2','country'));
    IbComposer_enableHtmlElement('combobox1',
IbComposer_isSelected('checkbox2','car'));
    IbComposer_enableHtmlElement('listbox2',
IbComposer_isSelected('checkbox2','model'));
    IbComposer_enableHtmlElement('combobox2',
IbComposer_isSelected('checkbox2','dcost'));
}
```

Function: IbComposer_ResetDownChainControls

IbComposer_ResetDownChainControls resets the controls down the chain from the current control to have correct corresponding values.

Syntax: **How to Reset Chain Controls**

```
IbComposer_ResetDownChainControls(ctrl);
```

where:

ctrl

> Alphanumeric

> Is the unique identifier of the first control.

Example: **Resetting the Chain Started by a List Box**

```
function button4_onclick(event) {
    var arr = [];arr.push('ENGLAND');
    IbComposer_setCurrentSelection('listbox1',arr,'false');
    IbComposer_ResetDownChainControls('listbox1');
}
```

Function: IbComposer_selectTab

IbComposer_selectTab selects the tab specified by the tabNumberToSelect and makes it the active tab.

Syntax: **How to Select a Tab and Make It Active**

```
IbComposer_selectTab(tabControlID,tabNumberToSelect);
```

where:

tabControlID

> Alphanumeric

> Is the unique identifier of the tab control being made active.

tabNumberToSelect

> Integer

> Is the number of the tab to make active.

Example: **Making a Tab Active**

```
<FORM id=form1 onsubmit="OnExecute(this);
    IbComposer_selectTab('tab1',1) name="form1">
```

Function: IbComposer_selectTemplateTab

IbComposer_selectTemplateTab selects a tab on a template page and makes it the active tab.

Syntax: **How to Select a Template Tab and Make It Active**

```
IbComposer_selectTemplateTab(tabId);
```

where:

tabId

> Alphanumeric

> Is the unique identifier of the tab control being made active.

Example: **Making a Template Tab Active**

```
function submit1_onclick(event) {
    IbComposer_selectTemplateTab('tab5');
}
```

Function: IbComposer_goToPageOfMultiPageControl

IbComposer_goToPageOfMultiPageControl selects a page in a multipage control, such as a tab, accordion, or window.

Syntax: How to Select a Page in a Multipage Control

IbComposer_goToPageOfMultiPageControl(*controlID*, *page*)

where:

controlID

    Alphanumeric

    Is the unique identifier of the control from which values are obtained.

page

    Alphanumeric

    Specifies a page number (for example 1, 2, and so on), or true for the previous page and false for the next page.

Example: Selecting a Page for a Multipage Control

The following example shows how to select a page in a window multipage control. In this example, the control will start on page 3.

```
function button2_onclick(event) {
var eventObject = event ? event : window.event;
var ctrl = eventObject.target ? eventObject.target : eventObject.srcElement;
// TODO: Add your event handler code here
IbComposer_goToPageOfMultiPageControl('windowPanel1', '3');
}
```

The following example shows how to select the previous page in a window multipage control.

```
function button2_onclick(event) {
var eventObject = event ? event : window.event;
var ctrl = eventObject.target ? eventObject.target : eventObject.srcElement;
// TODO: Add your event handler code here
IbComposer_goToPageOfMultiPageControl('windowPanel1', 'true');
}
```

The following example shows how to select the next page in a window multipage control.

```
function button2_onclick(event) {
var eventObject = event ? event : window.event;
var ctrl = eventObject.target ? eventObject.target : eventObject.srcElement;
// TODO: Add your event handler code here
IbComposer_goToPageOfMultiPageControl('windowPanel1', 'false');
}
```

Function: IbComposer_getAllAmpersValues

IbComposer_getAllAmpersValues is used to retrieve the current selected values from all the controls on your page layout. It then takes those values and assembles them as a string that can be added to the end of a URL call. An example of this would be having a REGION control and multiselecting MidEast, NorthEast, and NorthWest. It will assemble these selections as shown below:

```
&REGION=%27MidEast%27%20OR%20%27NorthEast%27%20OR%20%27NorthWest%27
```

This function can be used in conjunction with the Business Intelligence Portal, where the generated string is appended to all Business Intelligence Portal calls that run reports or charts. This allows the parameter values to affect all portal components, even if new ones are added or existing ones are removed at run time.

Syntax: **How to Retrieve all Parameter Values**

```
IbComposer_getAllAmpersValues([verifySelection]);
```

where:

verifySelection

Boolean

Is an optional parameter. When true and when the Selection required property for the control is set to *Yes*, this returns an empty string for the parameter controls that do not have a selection made.

Note: All controls have the Selection required property. This property is set to *Yes* by default. If a control has no valid selection made at run time, a red box appears around it and the following status bar message displays:

```
Please make required selections
```

Example: **Retrieving a List of All Parameters Selected in a Report**

```
function button1_onclick(event) {
   var val = IbComposer_getAllAmpersValues();
   alert(val);
   OnExecute(event);
}
```

Function: IbComposer_showLayer

IbComposer_showLayer shows or hides the specified layer.

Syntax: **How to Show or Hide a Layer**

```
IbComposer_showLayer(layerName,bShow);
```

where:

layerName

> Alphanumeric

> Is the name of the layer, which will be shown or hidden.

bShow

> Is an operator that can be set to true to show the layer or false to hide it.

Example: **Showing or Hiding a Layer**

```
function button1_onclick(event) {
   var eventObject = event ? event : window.event;
   var ctrl = eventObject.target ? eventObject.target :
   eventObject.srcElement;
   // TODO: Add your event handler code here
   IbComposer_showLayer('Customers','true');
}
```

Function: IbComposer_preventModifyIFrameDocument

IbComposer_preventModifyIFrameDocument can prevent CSS modification of any iframe within a document, by calling it from onInitialUpdate.

Syntax: **How to Prevent Modification of a Document within an IFrame**

```
function onInitialUpdate() {
   IbComposer_preventIFrameUpdates();}
```

Function: IbComposer_setCalendarDatesRange

IbComposer_setCalendarDatesRange sets the range for the target calendar, given the date on the source calendar and a range in days.

Syntax: How to Set a Date Range for a Target Calendar

```
IbComposer_setCalendarDatesRange(elementId, fromDate, toDate)
```

where:

elementID

Alphanumeric

Is the unique identifier of the target calendar control for which you want to set the date range.

fromDate

Date object

Is the value of the date from the source calendar.

Note: This can be obtained by using the IbComposer_getCurrentSelection function. This function should have the third parameter set to True to return a Date object.

toDate

Date object

This value must be calculated using fromDate and the number of days needed to show the range.

Example: **Setting the Date Range for a Target Calendar**

```
function button1_onclick(event) {
var eventObject = event ? event : window.event;
var ctrl = eventObject.target ? eventObject.target : eventObject.srcElement;
// TODO: Add your event handler code here
    var curDate = IbComposer_getCurrentSelection('calendar1', 0, true);
      var fromDate= new Date(curDate);
      var toDate = new Date();
      var nOfDays= IbComposer_getCurrentSelection('slider1');
      if (isNaN (nOfDays-0)&& nOfDays != null) {
          alert("Please enter a numeric value!!");
      }
      else {
          var time = fromDate.getTime()+(nOfDays*3600000 * 24);
          toDate.setTime(time);
          IbComposer_setCalendarDatesRange("calendar2", fromDate,
          toDate);
      }
}
```

Creating Responsive Web Pages

Responsive web design is an approach to webpage creation where content automatically realigns to fit a variety of different window sizes and screen sizes. This allows the content from one HTML file to automatically adapt to multiple footprints such as a desktop, laptop, tablet, or smartphone.

Unlike a generic HTML page, the containers in a responsive page reorder automatically as the screen size changes. Where a generic page shows partial content with a horizontal scroll bar, the responsive page shows full content, re-ordered, with no horizontal scroll bar.

How Responsive Containers Fold

The behavior that makes a page responsive is called "folding." This section details the way responsive containers fold, or re-arrange, as screen size changes. The order in which responsive containers fold is illustrated in the following image.

Figure A illustrates a responsive layout with a row-based 4-2 format, meaning there are four containers on row one and two containers on row two.

Figure A also illustrates the filter row, containing four controls. At run time, the filter row can either be hidden or displayed.

Figure B illustrates the way containers fold responsively when the screen width decreases. The content in each container resizes accordingly, as the screen width changes.
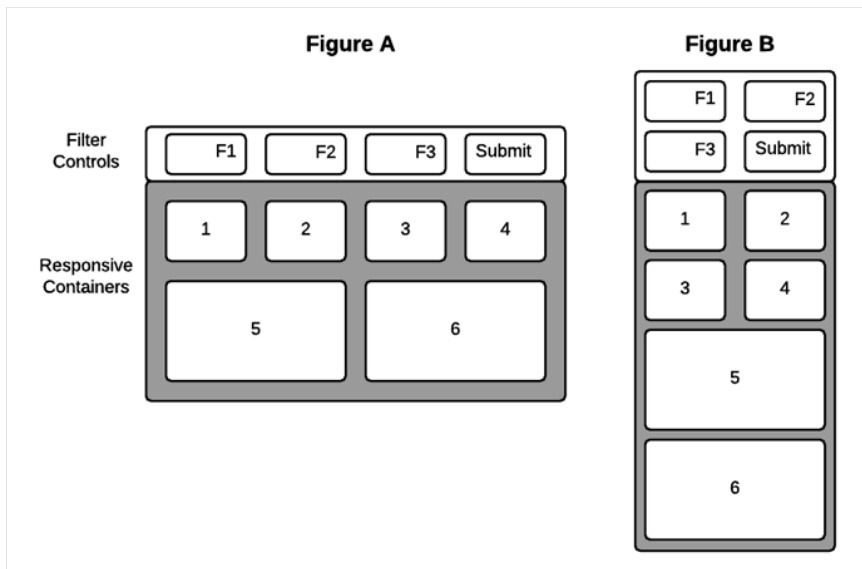
Note: Objects within a responsive container are positioned absolutely. That is, they do not fold. Responsive containers fold in relation to one another, but the objects inside the containers remain fixed.

Objects inside responsive containers can scale to defined limits, as detailed in *How to Set Minimum and Maximum Panel Sizes* on page 333. However, the objects within a container will not fold.

Procedure: **How to Control Folding with the Flow Direction Property**

You can control the column/row orientation by which responsive containers re-arrange themselves.

1. Select a container in a responsive HTML page.

2. Click the drop-down arrow in the *Flow Direction* field of the Properties panel and select one of the following:

   ❏ **Not Set.** - Allows you to fold objects based on the available space (default).

   ❏ **Row.** - Allows you to fold objects horizontally by row.

   ❏ **Column.** - Allows you to fold objects vertically by column.

   Note: The Flow Direction property does not display for a responsive container that contains an element that is not responsive. For example, a button contained in a responsive panel will disable (hide) the Flow Direction property for the panel. To prevent this, right-click the button and click *Add selected to new container* from the context menu. Then, when you click the original panel, the Flow Direction property is available again.

3. Run the page. Resize the browser to observe the folding.

Selecting a Responsive Method

There are two ways to create responsive pages in Developer Workbench: using a Responsive Template, and using the HTML Canvas in Responsive Mode. The following diagram shows the high-level decision tree that lets you quickly find the best way to create your responsive page:
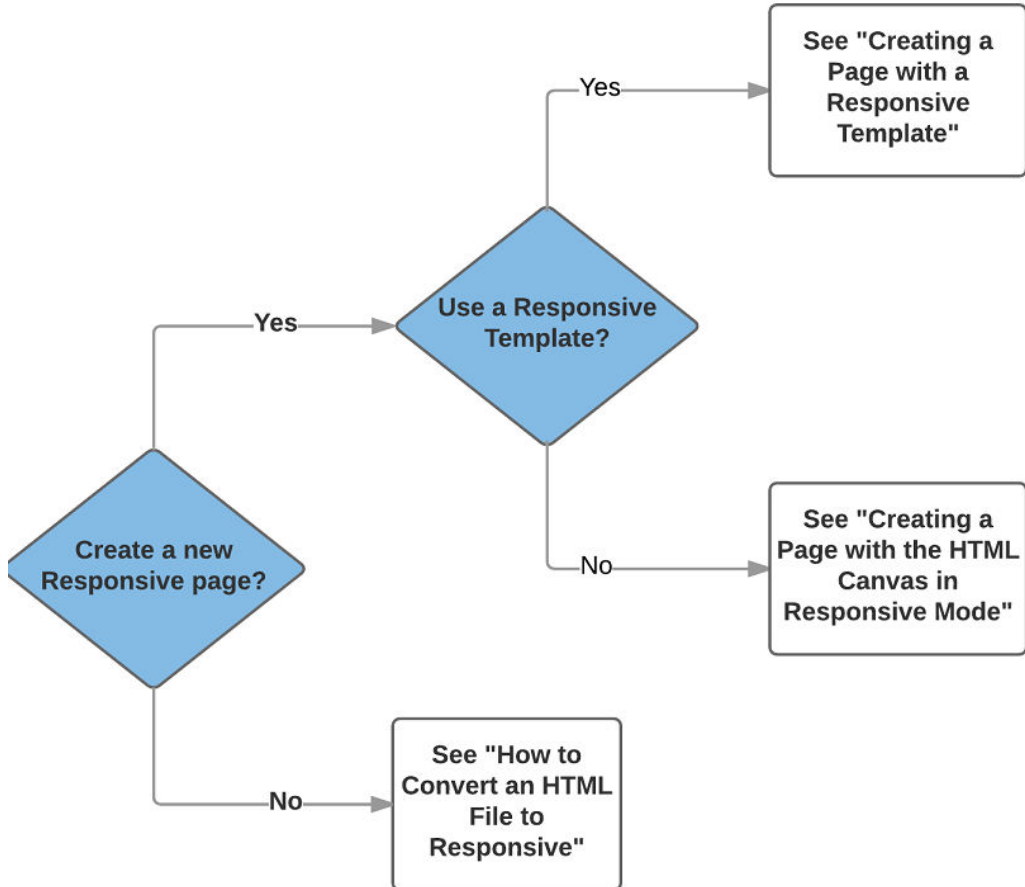
1. **Using a Responsive Template.** This method lets you create a page by populating responsive Output Widget containers in a pre-defined template or a layout that you create at design time. You use the HTML/Document wizard to specify a layout that is either row-based or column-based.

Key Characteristics

The Output Widget container, shown in the following image, is the key characteristic of a Responsive Template.

The Output Widget container has the following features:

❏ **Title Bar.** Including an editable text title.

❏ **Image.** Such as a company logo, that can be inserted on the left side of the title bar.

❏ **Arrow Toggle.** On the right side of the title bar, to display or hide widget-level controls.

❏ **Expand Toggle.** On the right side of the title bar, to make the widget contents display full screen or original size.

If you want to create a responsive page with one or more Output Widget containers, the Responsive Template method is the most efficient approach. This method is used only for creating new responsive HTML files. For more information, see *Creating a Page With a Responsive Template* on page 324.

2. **Using the HTML Canvas Responsive Mode.** This "free form" method lets you create a responsive page with no pre-defined layout. This manual approach is perhaps not as fast as using a template, but may be preferable if you want to create a responsive page with containers of many different types.

   This method can also be used to convert an existing HTML file to responsive. For more information, see *Creating a Page With the HTML Canvas in Responsive Mode* on page 336.

Creating a Page With a Responsive Template

There are six pre-defined responsive templates available through the HTML/Document Wizard. These represent the most commonly used page layouts. You can use the layout from a template as-is, or modify the layout further after initiating it with a template.

As an alternative to using a pre-defined template, the HTML/Document Wizard allows you to create a custom page layout at design time. You can specify the number of rows and columns, and whether the page has a row- or column-based orientation. These and other page options are detailed in the following sections.

Understanding Responsive Templates

When you select a responsive template from the HTML/Document Wizard, it generates a responsive layout with the specified number of rows (or columns) and widget containers. The parts of the responsive layout are highlighted in the following image, which was generated using the 2-2 responsive template.
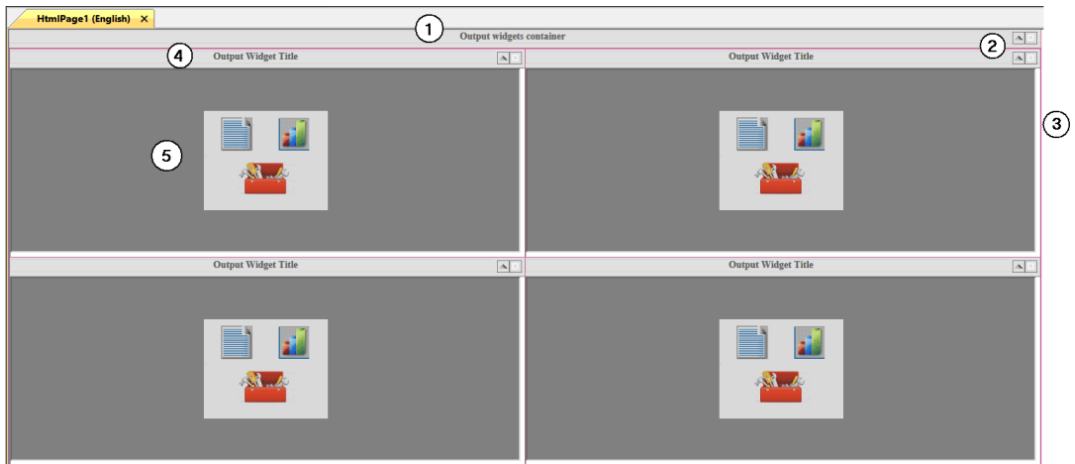
Document. Although not enumerated in the previous image, a responsive template generates an HTML document that includes the following responsive properties, editable in the Properties panel:

❏ **Load in iFrame object.** To load the entire page as a widget (a single DIV item) for use in a BI Portal. The default value for this property is *Yes*.

❏ **Responsive: min width.** Sets the minimum width (in pixels or percentage) beyond which the page size cannot be decreased. The default minimum width is 320 pixels.

❏ **Responsive: max width.** Sets the maximum width (in pixels or percentage) beyond which the page will not rescale larger. This also defines the width of the canvas at design time. This limit is indicated by a black vertical line on the right side of the canvas, as shown in item 3 in the previous image. The default maximum width is Full screen.

At run time, the layout populates the full browser.

1. Main Widget. Defined as windowPanel1 <DIV> in the Properties panel, the default title is *Output widgets container*.

❏ **Grid margin.** Defines the margin around all four edges of a cell expressed in pixels.

❏ **Template orientation.** Determines if the content folds by row or by column. The default orientation is *Row-based*. To change this, you can select *Column-based*.

❏ **Widget Title.** You can delete the default and type the desired title for the widget that encompasses the entire page contents.

❏ **Display image.** Enables you to display an image in the upper left corner of the widget. The default setting is *Yes*.

❏ **Image source.** Used to select the image to be displayed. Once selected, the path to the image is provided.

❏ **Auto-hide inputs panel.** The row of controls used to filter the content for the page is hidden by default. To always display the row of controls, select *No*.

❏ **Select animation.** Enables you to select any animations that have been set for this window panel.

2. Widget display buttons. The page, and each widget within it, has two buttons in the upper-right corner: an arrow and a box. Toggle the arrow to display or hide the row of controls. Toggle the box to expand a widget to full screen or return it to original size.

3. Vertical design boundary. This is a visual indicator on the far right side of the canvas that shows the responsive maximum width set in the Properties panel for the document. The default is 1500 pixels.

4. Widget Title. Defined as windowPanel2 <DIV> in the Properties panel (for the first widget), the default widget title is *Output Widget Title*. You can delete the default and type the desired widget title in the Properties panel. At design time, a red box appears around the widget title and contents to indicate a responsive container.

5. Widget contents. Defined as iframe1 <IFRAME> in the Properties panel (for the first widget), you can right-click to add content to the widget through the following shortcut menu:

❏ **Reference existing procedure.** To reference an existing procedure in the widget.

❏ **Use as Toolbox.** A general option to use the widget as needed. For example, you may decide to create a button to execute an action.

The content in pages derived from a responsive template loads in the following order at run time: left to right - top to bottom.

To add a header to your responsive page, right-click in the title bar and select *Add page header* from the shortcut menu. Alternatively, you could use the Add page header option in the HTML/Document Wizard, as detailed in the following procedure.

Procedure: **How to Select a Responsive Template**

Select a responsive template from the HTML/Document Wizard as follows:

1. Open the HTML/Document Wizard using one of the following methods:

   a. On the Home tab, in the Content group, click *HTML/Document*; or

   b. In the Environments Tree panel, right-click an application folder, click *New*, and click *HTML/Document*.

   The HTML/Document Wizard opens. If you used option **a.** above, the HTML/Document Wizard contains a navigation window that you can use to specify the folder where you want to save the new file.

2. Click *Next*.

   The Templates, Settings, and Themes page opens, containing the Responsive Templates section, as shown in the following image.
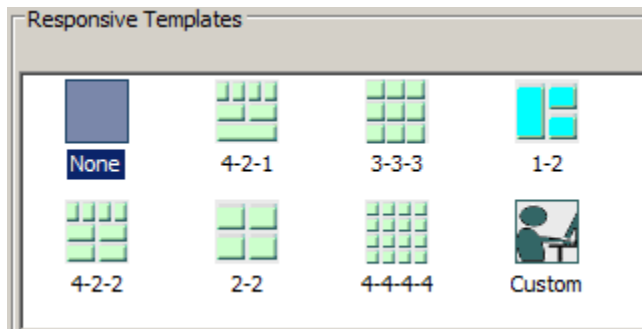
3. Click on a desired responsive template.

   ❏ Selecting *None* results in a generic HTML canvas that is not responsive.

   ❏ Selecting *Custom* lets you design a layout that differs from those offered by the responsive templates. For more information, see *How to Create a Custom Layout* on page 335.

   For purposes of this procedure, do not select *None* or *Custom*.

4. Use the Input Controls section at the bottom of the page as follows:

   ❏ Select *Auto-hidden* to hide the controls input panel. This is the default. At run time, users display the panel by clicking the down arrow in the title bar.

❏ Select *Always show* to display the controls input panel. At run time, users hide the panel by clicking the up arrow in the title bar.

❏ Clear the check box for *Use a single input control panel* if you want to use widget-level controls instead of, or in addition to, page-level.

5. Select *Run requests on load* to load the contents of all the widgets when the page first opens.

6. Select *Add page header* if you want to add space above the title bar, within the responsive panel1 <DIV>. You could use this area to add a graphic and/or company logo, for example.

7. Click *Finish* to close the HTML/Document Wizard.

The HTML canvas opens with the automatically generated responsive layout corresponding to the selected template.

Note: It is a best practice to expand Developer Workbench to full screen when working with a responsive layout. This lets you access the greatest number of widgets without having to scroll.

Reference: Populating Widgets in a Responsive Template

A layout generated by a responsive template contains blank widgets in responsive containers. This section contains guidelines for editing and adding objects to widgets.

❏ To add content to a header, click the desired element from the Components tab and drag and drop it on the header area to define it.

To increase the height of a page header, click the *Title Bar* and drag it down to expose more room above it. Click the header area to highlight the responsive red box. Drag the lower border of the red box down to increase the height of the header.

❏ To change a widget title, click the default title, *Output Widget Title*, and open the Properties panel. In the Widget Title field, type the new widget title.

❏ To display an image in the top left corner of a widget title bar, set the Display image value to *Yes*. Use the Image source field to lookup the desired image file.

❏ To add objects to a widget, right-click in the widget to display the shortcut menu. Click *reference existing procedure*.

Note: When using a layout generated by a responsive template, you cannot add objects to the canvas outside the layout.

❑ To add multiple objects to a widget, drag the additional object from the ribbon to the widget. For example, on the *Home* tab, in the *Content* group, click the *Report* button and drag it to a widget that already contains an object, and drop it. A box is automatically drawn within the widget to contain the new report object.

Note: All of the objects in a responsive container must be responsive. You cannot include responsive and non-responsive objects in the same responsive container.

❑ To set an equal margin around all four sides of a widget, enter a number of pixels in the Margin property for the widget <IFRAME>.

❑ To move contents from one widget to another, click the Positioning tab and, in the Responsive Design group, click *Toggle Selection*. Drag contents between widgets as needed and click *Preview Runtime*, on the Utilities tab.

Procedure: **How to Add or Remove a Widget**

You can change the layout of your responsive page at design time by adding or removing a widget, as detailed below.

1. Open a responsive HTML page.

2. To add a widget, use one of the following methods:

    a. Select a widget next to the location where you want to add a new one. Right-click on the title bar of that widget and select *Insert new widget before* or *Insert new widget after* from the shortcut menu.

        The new widget is added.

    b. Alternatively, you can right-click on the title bar of the HTML page and select *Template configuration...* from the shortcut menu. The Template Configuration dialog box shows the row (or column) numeric configuration. For example, 3-2.

        Change the numeric configuration to include the new widget. For example, to add a new widget to the second row, change the value to 3-3.

        The new widget is added.

3. To remove a widget, use one of the following methods:

    a. Select a widget that you want to remove. Right-click on the title bar of that widget and select *Delete* from the shortcut menu.

        The widget is removed.

    b. Alternatively, you can right-click on the title bar of the HTML page and select *Template configuration...* from the shortcut menu. The Template Configuration dialog box shows the row (or column) numeric configuration. For example, 3-2.

Change the numeric configuration to reduce the number of widgets. For example, to remove a widget from the first row, change the value to 2-2.

The widget is removed.

Procedure: **How to Add an Additional Object to a Widget**

You can put more than one object (such as a report or chart) in an output widget, as detailed in this procedure.

1. Click an object that has already been added to a widget.

   ❏ If the widget contains a report or chart, you can add another report or chart, but not any other type of object.

   ❏ If the widget contains a toolbox, you can add an additional report or chart, but it will not have the Autosize option set by default. You must set this option manually.

   Open the Properties panel to verify that the IFRAME is selected.

2. On the *Components* tab, click the object that you want to add. For example, *Report* or *Chart*.

   The widget can be divided vertically (with the two objects, side by side) or horizontally (with the two objects one above the other). Move your cursor to the right or left side, or the top or bottom of the widget.

   The existing content automatically moves to the opposite side of your cursor.

3. Click on the side where you want to insert the new object.

   The widget is automatically divided into two equal columns or rows. The side you selected contains the report or chart icon in an iframe.

   Note: If you change your mind after you add the new iframe, you may right-click and delete it. However, the original iframe will not automatically expand to fill the original space in the widget. At run time, the content of the original iframe does fill the original space. But at design time it remains partial. You can click the original iframe and manually drag its border to fill the widget space, if you want the appearance of the page at design time to more closely reflect the appearance at run time.

4. Right-click the icon and select the desired option from the shortcut menu to create the new object. For example, Reference existing procedure.

5. Click the widget title bar and open the Properties panel. The Number of columns property defaults to Auto. Use the drop-down menu to select the number of columns you want in the widget. Select *2* if you have two objects side by side. Select *1* if you have two objects, one above the other.

Run the page to preview widgets that have multiple objects to be sure the content remains legible.

Procedure: How to Apply Page-Level Controls in a Responsive Page

You may often have a responsive page that contains multiple reports that use the same controls. For example, if users can select region and month, they may want to show the same region and month selection for multiple reports in the page. In this case, you can apply common controls at the page level.

Your page can also include one or more reports that use controls that differ from the common ones. In this case you can apply additional controls at the widget level, as defined in *How to Apply Widget-Level Controls in a Responsive Page* on page 332.

This procedure details how you apply controls at the page level.

1. Create multiple reports or charts that use the same controls and add them to widgets in a responsive layout as detailed in *How to Select a Responsive Template* on page 327.

   When you add a report with controls to a widget, the New Parameters dialog box opens. You can use this dialog to chain controls together. Chaining lets you filter control values based on the selected value from the prior control in the chain. For example, if State and City controls are chained, the City values will be filtered to show only cities in the selected state.

   For more information see *Chaining in the HTML Canvas* on page 284.

2. Run the page. Note that:

   ❏ The control names and values are appended to the page title in the title bar.

   ❏ The controls input panel does not display. This is the default.

3. Click the down arrow in the right corner of the title bar to display the controls input panel.

   Close the HTML output and return to the HTML canvas.

4. If you want the controls input panel to display when the page opens:

   a. Click the title bar and open the Properties panel.

      The properties for windowPanel1 <DIV> are displayed.

   b. Change the Auto-hide inputs panel value to *No*.

   c. Run the page. Note that the controls input panel now does display.

5. To display the controls on the left side of the input panel:

   a. Click the controls input panel and open the Properties panel.

b. Change the Control panel value to *Yes*.

c. Run the page and expand it to full screen. Note that the controls in the input panel display together on the left side of the panel, where the user expects them to be.

If you do not set the Control panel value to *Yes*, the controls will automatically expand to the right to use the full width as the page size increases.

The selections that the user makes in the controls input panel are automatically applied to all the reports on the page that share the controls.

Procedure: ## How to Apply Widget-Level Controls in a Responsive Page

You may often have a responsive page that contains multiple reports that use the same controls. For example, if users can select region and month, they may want to show the same region and month selection for multiple reports in the page. In this case you can apply common controls at the page level, as defined in *How to Apply Page-Level Controls in a Responsive Page* on page 331.

Your page can also include one or more reports that use controls that differ from the common ones. In this case you can apply additional controls at the widget level.

This procedure details how you apply controls at the widget level when different controls are used at the page level.

1. Open a responsive HTML file that uses page-level controls, such as the one created in *How to Apply Page-Level Controls in a Responsive Page* on page 331.

   Add a new widget to a row as detailed in *How to Add or Remove a Widget* on page 329.

2. Create a report or chart that uses controls and add it to the new widget. For purposes of this procedure, use different controls than the ones used in the other reports in this page.

   When you add a report with controls to a widget, the New Parameters dialog box opens. By default, this dialog is set to create a control. Click *OK*.

3. Run the page. Note that the controls input panel displays in the new widget by default.

4. To hide the controls input panel in a new widget:

   a. Add a new widget to a row as detailed in *How to Add or Remove a Widget* on page 329. Add the report you created in Step 2 to the new widget.

      The New Parameters dialog box opens.

   b. In the Parameter grouping options field select *New single layer form* and click *OK*.

      The New Parameters dialog box closes. The input panel is hidden.

The selections that the user makes in the controls input panel at the page level are automatically applied to all the widgets on the page that share controls. Additionally, the user can make a selection in the independent widget that does not share controls.

Procedure: **How to Animate the Controls Input Panel**

Creating a page with a responsive template enables you to add animation that appears when a user opens or closes a controls input panel at run time. This consists of creating and naming an animation, then applying it to the window panel that corresponds with the input controls, as detailed below.

1. Open an HTML page that was a) created with a responsive template; and b) contains input controls.

2. Open the *Tasks & Animations* panel. In the JQuery Animations section, click the *New* button.

   The system gives the new animation a default name. For example, *Animation1*. If you wish to rename the animation, click the default name and type the new name.

   Note: It is not necessary to specify a selected target for the animation. Leave this field blank.

3. In the Effect section, use the drop-down menu in the Type field to select an animation. If you wish, you may select an Option and Value to apply to the selected animation.

4. At the bottom of the panel, click *Toggle animation*.

   The animation is ready to be applied to the window panel.

5. In the responsive layout, click the title bar under which the control input panel appears.

6. Open the Properties panel. In the Output Widget section, click the *Select animation* field. Use the drop down menu to highlight the desired animation name. For example, *Animation1*.

7. Run the HTML page. Use the up/down arrow button on the title bar to see the animation when you hide/reveal the controls input panel.

Procedure: **How to Set Minimum and Maximum Panel Sizes**

Responsive panels automatically resize as the viewing window changes, but panels do not have a default minimum or maximum size. You can set minimum or maximum size, in pixels or percentages, to best suit the panel contents, as follows.

Note: Panels in a responsive page resize within the context of the minimum and maximum page width. To view or edit these values, open the Properties panel and select *DOCUMENT* from the drop-down menu at the top of the panel. For Responsive: min width, the default is 320px. For Responsive: max width, the default is Full screen.

1. Click a responsive panel that contains controls, or click the title bar of a widget, and open the *Properties* panel.

   The Properties panel for the panel <DIV> includes the following properties: Min-width, Max-width, Min-height, and Max-height. These values are blank by default.

2. You can manually type a number of pixels or a percentage. For example, *50px* or *50%*.

   Alternatively, you can use the shortcut menu as follows:

3. Right-click a responsive panel that contains controls, or right-click the title bar of a widget. From the shortcut menu, select *Update Sizes*.

   The Update Sizes shortcut menu displays.

   Select Update minimum size to populate the Min-width and Min-height properties with the number of pixels or percentages used by the selected container. In other words, the container will not scale smaller than the size shown in the responsive layout.

   Note: Update minimum size does not change the object size in design time, but it does change the size in run time.

   Select *Use Current* to use sizes from the selected container as follows:

   ❏ **Size as min size.** To populate the Min-width and Min-height properties with the number of pixels used by the selected container.

   ❏ **Size as max size.** To populate the Max-width and Max-height properties with the number of pixels used by the selected container.

   ❏ **Height as min height.** To populate the Min-height property with the number of pixels used by the selected container.

   ❏ **Width as min width.** To populate the Min-width property with the number of pixels used by the selected container.

   ❏ **Height as max height.** To populate the Max-height property with the number of pixels used by the selected container.

   ❏ **Width as max width.** To populate the Max-width property with the number of pixels used by the selected container.

Procedure: **How to Load a Widget at Run Time**

In some cases, the content you add to a widget might not load at run time, leaving the widget blank. This could occur if you did not select the *Run requests on load* option from the HTML/Document wizard, or if you added a new widget to a layout.

To cause widget content to load at run time:

1. Open the *Tasks & Animations* panel, select the *Load Task*, and then select *Load Trigger Type*.

2. In the Requests/Actions section, click the down arrow next to the Add Request button.

    The shortcut menu is displayed.

3. Select *Run Request* and then select an iframe, for example *iframe2*.

    The iframe is added to the Requests/Actions field and is highlighted.

4. In the Target Type field, select *Frame*. In the Target/Template field, select the iframe that corresponds to the one that is highlighted, for example *iframe2*. Run the page.

    The content loads at run time.

5. Repeat these steps for all the iframes in your page that do not load at run time.

Procedure: **How to Create a Custom Layout**

In the HTML/Documents Wizard, the Templates, Settings, and Themes page includes six responsive templates that represent the most commonly used layouts. You can use these layouts as-is, or use them as a point of departure for a modified layout that contains more or fewer widgets. For more information, see *How to Add or Remove a Widget* on page 329.

Instead of using a template, you can create a custom layout, as detailed in this procedure.

1. Open the HTML/Document Wizard using one of the following methods:

    a. On the *Home* tab, in the *Content* group, click *HTML/Document*.

        or

    b. In the Environments Tree panel, right-click an application folder, click *New*, and click *HTML/Document*.

    The HTML/Document Wizard opens. If you used option **a.** above, the HTML/Document Wizard contains a navigation window that you can use to specify the folder where you want to save the new file.

2. Click *Next*.

    The Templates, Settings, and Themes page opens, containing the Responsive Templates section.

3. In the Responsive Templates section, click *Custom*. Click Finish.

   The layout opens and the Template configuration dialog is activated.

4. In the Orientation field, select *Row-based* or *Column-based*.

5. Use the Select row number and Number of containers in a row field as follows:

   a. With number 1 showing in the Select row number field, use the drop down list in the Number of widgets in a row field to specify the number of widgets you want in row 1.

      Note: Use the drop down list. Do not type in the field.

   b. Select *2* in the drop down list in the Select row number field.

   c. Use the drop down list in the Number of widgets in a row field to specify the number of widgets you want in row 2.

   d. Repeat this sequence as needed to define your custom layout.

      Close the Template configuration dialog.

6. Define the page-level behavior of input controls as follows:

   ❏ In the Properties panel for windowPanel1 <DIV>, set Auto-hide inputs panel to *Yes*. At run time, users reveal the panel by clicking the down arrow in the title bar. This controls page-level behavior.

   ❏ In the Properties panel for windowPanel2 <DIV>, set Auto-hide inputs panel to *Yes*. At run time, users reveal the panel by clicking the down arrow in the title bar. This controls widget-level behavior. Repeat this for each widget for which you want to use widget-level controls.

   ❏ Select *No* to display the controls input panel. At run time, users hide the panel by clicking the up arrow in the title bar.

7. To add a page header, right-click in the title bar of the page and select *Add page header* from the shortcut menu. You could use this area to add a graphic and/or company logo, for example.

Creating a Page With the HTML Canvas in Responsive Mode

This "free form" method lets you create a responsive page with no pre-defined layout. This manual approach is perhaps not as fast as using a template, but may be preferable if you want to create a responsive page with containers of many different types. You can also use this method to convert an existing HTML file to responsive.

The following containers are responsive, by default:

❏ Report

❏ Chart

❏ Form (Single Layer and Multiple Layer)

❏ Tab (Top, Bottom, Left, and Right)

❏ Accordion (Vertical and Horizontal)

❏ Window

❏ Output Widget

❏ Group Box

❏ Panel

❏ Frame

Procedure: **How to Create New Content in Responsive Mode**

You can create new responsive content using the HTML canvas in responsive mode, as defined in this procedure.

1. Open the HTML/Document Wizard using one of the following methods:

    a. On the *Home* tab, in the *Content* group, click *HTML/Document*; or

    b. In the Environments Tree panel, right-click an application folder, click *New*, and click *HTML/Document*.

    The HTML/Document Wizard opens.

2. Click *Next*.

    The Templates, Settings, and Themes page opens, containing the Page Settings section, as shown in the following image.
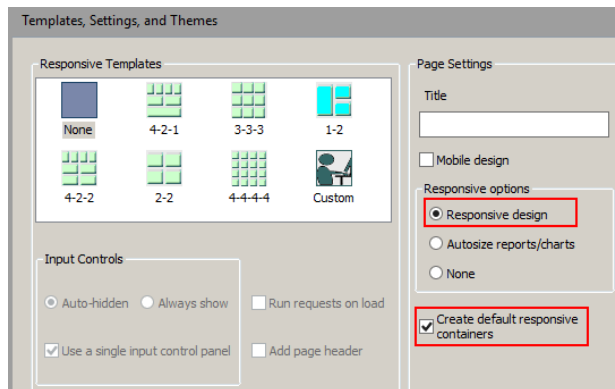
3. In the Page Settings section, select *Responsive design*.

   Note:

   ❑ The Create default responsive containers option is selected, by default. When you create a control using the New Parameters dialog box, the forms and the controls will be created with responsive containers.

   ❑ If you clear the Create default responsive containers check box, no responsive containers will be drawn for the form object.

   ❑ Controls are not responsive, by default. If you draw a control, right-click the control and select *Add Selected to a new container* to draw a responsive container around the object.

   ❑ You can make a non-responsive container responsive by selecting the container and, in the Properties panel, selecting *Yes* for the Enable Responsive property.

4. Click *Finish*.

   The HTML canvas opens. In the Properties panel, in the DOCUMENT properties, the Responsive design value is set to *Yes*.

   As an alternative to using the HTML/Document Wizard, you can open a file in the HTML canvas and manually change the Responsive design property value to *Yes*.

   You can add content to the HTML canvas before turning Responsive Design on. However, there are advantages to turning Responsive Design on before adding content. Design time benefits include:

   ❑ Having the responsive minimum and maximum widths established

   ❑ Having the maximum width visually indicated by the horizontal line on the right side of the canvas

   ❑ Having the Preview Runtime capability to preview the run-time appearance

## *Procedure:* How to Convert an HTML File to Responsive

1. To enable responsive behavior in an existing HTML file, double-click the file in the Environments Tree panel, or right-click it and click *Open*.

   The HTML file is open in the HTML canvas.

2. In the Properties panel, select *DOCUMENT* from the drop-down menu.

   The Document properties are displayed.

3. In the Responsive design property, select *Yes* from the drop-down menu.

   A message appears asking of you want to group control elements into responsive containers.

4. Indicate your preference for responsive containers as follows:

   a. Click *Yes* to add multiple containers in the control area as shown in the following image.
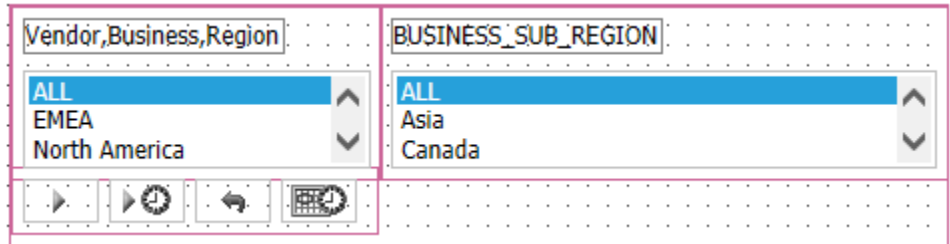
   Note there are three containers within the fourth, larger container. The containers are red, indicating that they are responsive.

   b. Click *No* to add no containers. This may be preferable if your page contains many nested objects, otherwise each object will get its own container.

   You can add additional containers at a later time:

   1. Multi-click the control objects you want to group in a container.

   2. Right-click one of the highlighted objects and select HTML object manipulation from the shortcut menu.

   3. Select *Merge containers*.

   4. To make a frame responsive, select it and open the Properties panel. Set the Enable responsive property value to *Yes*.

The HTML canvas is now in responsive mode. The frames and controls on your HTML canvas are each enclosed in a responsive container, indicated by a red box.

Note: if the HTML file had the Autosize Children option enabled, it will be superseded by the new responsive behavior, which automatically controls sizing.

Two additional responsive properties appear in the Properties panel, whose values you may edit:

❏ **Responsive: min width.** Elements will not be repositioned or resized if the width of the container is less than this minimum, specified in pixels or percentage.

❏ **Responsive: max width.** Elements will not be repositioned or resized if the width of the container is greater than this maximum, specified in pixels or percentage.

A black vertical line appears on the right side of the HTML canvas in responsive mode, as a visual indicator of the responsive maximum width. You may need to scroll right to see this line.

5. To move contents from one container to another, click the *Positioning* tab and, in the Responsive Design group, click *Toggle Selection*. Drag contents between containers as needed and click *Preview Runtime*, on the Utilities tab.

6. Click the *Run* button on the Application menu and view the HTML output full screen. Gradually decrease the width to observe the responsive folding.

Procedure: **How to Restrict the First Row to Controls Only**

When you create a page with a responsive template, the first row of the page is automatically restricted to input controls (when applicable). Content such as reports or charts will not fold up to populate the first row, regardless of how wide the browser may stretch at run time.

When you create a page using the HTML canvas in responsive mode, this restriction is not automatic. Therefore a report or chart could fold up to populate the first row, if the browser width is increased. This behavior may be considered undesirable.

Ensure that the first row contains controls only, as follows:

1. Open an HTML page that was:

   ❏ Created with the HTML canvas in Responsive mode

      and

   ❏ Contains input controls.

2. Verify that the input controls are all grouped within a single container. If not:

   a. Hold down the Ctrl key and multi-select all the controls.

   b. Right-click on one of the controls and select HTML object manipulation.

   c. From the shortcut menu, select *Add selected to new container*.

      Note that a box appears that contains all the selected controls. The box is red, indicating it is responsive.

3. To display the controls on the left side of the input panel:

   a. Click the controls input panel and open the Properties panel.

   b. Change the Control panel value to *Yes*.

   c. Run the page and expand it to full screen. Note that the first row contains controls only. Also note that the controls in the input panel display together on the left side of the panel, where the user expects them to be.

If you do not set the Control panel value to *Yes*, the controls will automatically expand to the right to use the full width as the page size increases.

Procedure: **How to Create a Responsive Page for Use in a BI Portal**

The Db2 Web Query BI Portal gives users access to BI content, including dashboards, visualizations, reports, charts, graphs, maps, and interactive InfoApps. The BI Portal allows users to analyze vital information quickly, link content and reports together, and easily tailor information to their needs.

This procedure is comprised of two parts: creating a report for use in the responsive portal and creating a responsive HTML page using the HTML canvas.

1. Select or create a report with the following criteria:

    ❏   Has dynamic parameters specified in a WHERE clause.

    ❏   Has _FOC_NULL as the default.

    ❏   Has a parameter that is a Multiselect OR.

    ❏   Has a value of *No* in the File/Folder Properties panel - Prompt for Parameters property.

2. Create a responsive HTML page as follows:

    a. Open the *HTML/Document Wizard* from the *Home* tab, in the *Content* group, or from the shortcut menu in the Environments Tree panel. Click *Next*.

        The Templates, Settings, and Themes page opens.

    b. In the Page Settings area, click *Responsive design* and click *Finish*.

        The HTML canvas opens. In the Properties panel, in the DOCUMENT properties, verify that the Responsive design object property is set to *Yes* and the Load in iFrame object property is set to *No*.

        The Load in iFrame object property is used to enable objects in a BI Portal to overlap the edges of a container. For example, when you have a pop-up calendar that populates a date field, the calendar can overlap the control container to allow the user to select a date.

        Note: The Load in iFrame object property creates a <DIV> for the page instead of an <iFrame> tag. A BI Portal is limited to only one HTML Canvas page that is enclosed in a <DIV> tag.

        Note that the Responsive: min width is set to 320px and the Responsive: max width is set to Full screen. These default values can be edited in pixels or percentages.

     c.   In the Requests and Data sources panel, click the drop down arrow next to the New icon, and select the option Requests-Parameters-Filter Panel.

         The Open File dialog box opens.

     d.   Select the procedure you designated in Step 1 and click *OK*.

         The New Parameters dialog opens.

     e.   Click *Autochain controls in above specified order*, and click *OK*.

         The New Parameters dialog closes and the HTML canvas is opened, showing the controls in containers. The red boxes indicate that responsive design is activated.

     f.   Open the Tasks & Animations panel and verify that the Submit button (task 2) Requests/Actions value is Refresh1 and the Target Type is Refresh BI Portal.

     g.   Right-click the canvas and select *Preview Runtime* or click the *Preview Runtime* button on the Utilities tab. Note that the content expands to the maximum width. The black vertical line on the right side of the canvas is a visual indicator of the specified Full screen max width.

     h.   Save the HTML page.

Reference: **Tips for Using Responsive Design**

The following are some best practices in working with Responsive Design to create responsive HTML pages:

❑ When creating a new responsive HTML page:

    ❑ Use a Responsive template to generate a layout with widgets. The HTML/Document Wizard lets you control the number of rows and the number of widgets per row, lets you determine how input controls will appear, lets you add a page header, and lets you run requests automatically when the page loads at run time.

    ❑ Use Responsive Design in the HTML canvas to generate a layout without widgets. This method lets you design a page for a BI Portal.

❑ On the Positioning tab, the Positioning group contains the Same Width, Same Height, and Same Size commands that behave as follows in responsive design mode:

    ❑ **Same Width.** Sets the minimum and maximum width to that of the control object (the previously selected object).

    ❑ **Same Height.** Sets the minimum and maximum height to that of the control object (the previously selected object).

    ❑ **Same Size.** Sets the minimum and maximum width and height to that of the control object (the previously selected object).

To use these commands:

1. Click the object whose size you want to duplicate.

2. User the Control key and click the secondary objects that you want to resize.

3. Click the desired command button from the Positioning group.

❏ On the Positioning tab, the Responsive Design group contains the Toggle Selection and Update Layout commands. These commands are enabled in responsive design mode.

   ❏ **Toggle Selection.** Toggle on to allow responsive containers that include content to be dragged to a new position. Toggle off to prevent those containers from being repositioned.

   ❏ **Update Layout.** This command is activated in a responsive page when the Autosize children property is set to Yes. Click to refresh the HTML canvas after you reposition containers, to show how the containers will stack when folded.

❏ You can control the run-time alignment of contents within a responsive container using the *Content alignment* property. This property appears for a parent responsive container that contains multiple responsive objects.

   Click the down arrow to display the following property values:

   ❏ **Blank <Not Set>.** The objects are aligned toward the left side of the container by default.

   ❏ **Start.** The objects are aligned on the left side of the container.

   ❏ **End.** The objects are aligned on the right side of the container.

   ❏ **Center.** The objects are centered within the container.

   ❏ **Space between.** The objects are evenly distributed by width. The first object is close to the left side, the last object is close to the right side, and the objects in between are spaced equally.

   ❏ **Space around.** The objects are evenly distributed by width and height.

   The alignment in the HTML canvas does not change after you specify a content alignment property value. Click *Run* on the Quick Access Toolbar to see the specified alignment in the browser.

❏ A black vertical line appears on the right side of the HTML canvas in responsive mode, as a visual indicator of the breakpoint width for desktop display. In order to control the behavior of your responsive containers, position them to the left of this line on the HTML canvas. Content appearing to the right of this line will fold arbitrarily.

❏ If you have both Mobile Design and Responsive Design enabled, Mobile Design takes precedence when you view content on the smartphone. Mobile Design uses jQuery mobile controls. Responsive Design uses browser-generated controls.

Designing Content for Smartphones

You can design content specifically to be used on smartphones. Content designed in this way is automatically sized, and takes full advantage of the page-swipe navigation and touch-screen capabilities of smartphones.

Note: Tablets can display native Developer Workbench HTML content without modification. This section pertains to designing HTML content for smartphones only.

The following steps outline how you can design HTML content for smartphones:

1. Create and save your content in an HTML file that is built in the native Developer Workbench HTML canvas.

2. Enable Mobile Layout for the HTML file by using the Enable mobile property.

   For more information, see *Enabling Mobile Layout in an HTML File* on page 349.

3. Select either the Basic or Advanced Mobile Layout.

   For more information, see *Working With the Basic and Advanced Mobile Layouts* on page 350.

4. Populate the smartphone pages with your HTML content.

5. View the results on your smartphone.

   For more information, see *Viewing Mobile Output on a Smartphone* on page 356.

Designing Output for Smartphones With the Mobile Layout Functionality

Before you use the Mobile Layout functionality, you must create and save your content on the native HTML canvas. The example shown in the following image contains four objects: a text box containing a title, a drop-down control, a frame containing a report, and a frame containing a chart.

Frames

Frames in your HTML content automatically appear in the Mobile Layout. In the Basic Mobile Layout, individual frames appear separately on individual pages. In the Advanced Mobile Layout, all frames appear together on a single page. You can change either layout to display frames on pages as you like.

The page order, when viewed on a smartphone, is determined by the frame order in the Tasks area of the Tasks & Animations panel. You can change the page order in the Mobile Layout, as needed.

Animations that you have targeted to a frame included in the Mobile Layout will run on your smartphone.

Controls

To maximize the capabilities of a touch screen, single-select and multi-select controls both become listbox jQuery mobile controls, with one allowing single-select and the other multiselect.

Note: jQuery mobile controls do not display when you design your HTML page using the Mobile Layout, but will display when the page is viewed on a smartphone.

Other Objects

Some objects that you add to the HTML canvas are not automatically promoted to pages in the Mobile Layout. This is true for text boxes and images.

Objects that are not automatically promoted to pages appear below a horizontal line that underlines the page preview. You can drag objects above the line onto pages, as needed. Objects that remain below the line, and any animations targeted to them, will not appear when you view the pages on your smartphone.

Note: The size of an object that you drag into a page is accurately previewed in the page, so you can determine if the object needs to be resized. If so, you must resize it in the HTML canvas.

When you drag an object from below the horizontal line onto a page, the system recognizes this as a manual modification to the Mobile Layout. This triggers a message: Manual modification of mobile layout will stop default generation of the pages. It will be your responsibility to maintain it.

Accepting this means that changes to the underlying data will not flow automatically to the mobile pages. You must open the HTML file, right-click the canvas, and select *Update Layout*.

To take advantage of the default generation of the mobile pages, it is recommended that you avoid making manual modifications.

Note: You can undo manual modification of the Mobile Layout by right-clicking the canvas and clicking *Reset mobile to default*. This restores automatic update of the HTML pages, and removes any manual modifications.

A page in Mobile Layout appears as shown in the following image. In this example, the page content displays in a frame, which is scrollable when you view it on a smartphone. Below the frame is the navigation area.

Styling for the components of the Mobile Layout is controlled by the cascading style sheet. For more information, see *Cascading Style Sheet Class Mapping List for Mobile Components* on page 356.

The Navigation Area

The following image displays the navigation area that appears below the page content.
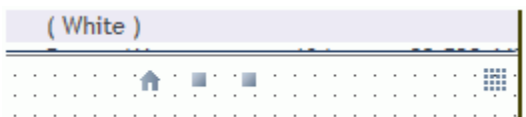
The icons in our example indicate that there is a home page, which is represented by the house icon, and two additional pages, which are represented by two square icons. In addition, there is a page combination tool, which is represented by a tile icon.

The default home page appears first in the page order. You can change the home page as needed.

You can use the square icons and the house icon to navigate between the pages.

You can use the tile icon to vertically combine objects in a page, in cases where there is more than one object on a page. Toggle the tile icon to change vertical combinations.

Adding/Removing Pages

The following list describes the shortcut menu options on the Mobile Layout canvas, which you can use to add and remove pages.

❏ **Show all pages.** Displays all pages on the Mobile Layout canvas. Both the Basic and Advanced Mobile Layouts show one page, by default.

❏ **Add empty page.** Adds an empty page to the Mobile Layout canvas. You can then drag content into the page.

❏ **Remove empty page.** Deletes any empty pages on the Mobile Layout canvas.

❏ **Remove selected page.** Deletes a page that has content. The system recognizes this as a manual modification to the Mobile Layout. This triggers a message: Manual modification of mobile layout will stop default generation of the pages. It will be your responsibility to maintain it.

Accepting this means that changes to the underlying data will not flow automatically to the mobile pages. You must open the HTML file, right-click the HTML canvas, and select *Update Layout*.

❏ **Reset mobile to default.** Restores automatic update of the HTML pages, and removes any manual modifications.

Note: When you delete content from the HTML canvas, you must also manually delete it from the Mobile Layout. Deleting content from the Mobile Layout that was previously deleted from the HTML canvas is not interpreted as a manual modification, and will preserve the default generation of the pages.

Moving Content Between Pages

You can change the order of the pages or display more than one object on a page.

To display more than one object on a page horizontally, click the object in one screen and drag it to another.

To display more than one object on a page vertically, use the tile icon . Toggle the tile icon to remove vertical combinations.

To move a control or an object that appears below the horizontal line, drag the object to the desired page. The system recognizes this as a manual modification to the Mobile Layout. This triggers a message: Manual modification of mobile layout will stop default generation of the pages. It will be your responsibility to maintain it.

Accepting this means that changes to the underlying data will not flow automatically to the mobile pages. You must open the HTML file, right-click the HTML canvas, and select *Update Layout*.

Note: You can undo manual modification of the Mobile Layout by right-clicking the canvas and clicking *Reset mobile to default*. This restores automatic update of the HTML pages, and removes any manual modifications.

To take advantage of the default generation of the pages, it is recommended that you avoid making manual modifications.

Enabling Mobile Layout in an HTML File

An HTML file is designated for mobile output when the Enable mobile property is set to *Yes*. This designation enables a Mobile Layout option, where you can edit and preview your mobile output.

Note: An HTML file that has Mobile Layout enabled can additionally be viewed on a desktop or tablet. In these cases, page design is derived from the HTML canvas and the page design on the Mobile Layout is ignored. This eliminates the need to maintain different copies of the same pages for different output.

You can enable mobile output for an existing HTML file, or create a new file with mobile output enabled.

Procedure: **How to Enable Mobile Output for an Existing HTML File**

1. Open the HTML file.

   The HTML canvas opens.

2. Open the Properties panel.

3. Change the *Mobile* property from *No* to *Yes*.

The HTML canvas shortcut menu now includes mobile output options.

Procedure: **How to Create a New File With Mobile Output Enabled**

1. On the Home tab, in the Content group, click *HTML/Document*. You can also create a new HTML file from the Application menu or by using the shortcut menu in the Environments tree panel, for a folder that supports content creation.

   The HTML/Document Wizard opens.

2. Navigate to where you want to create your HTML page and click *Next*.

   The Themes and Settings window of the HTML/Document Wizard opens.

3. Select the *Mobile design* check box.

   Selecting this option automatically sets the *Mobile* property to *Yes* in the Properties panel.

4. Click *Finish*.

   The HTML canvas opens. The canvas shortcut menu now includes mobile output options.

Working With the Basic and Advanced Mobile Layouts

There are two different layouts for designing HTML output for smartphones, Basic and Advanced.

❏ **Basic Mobile Layout.** Enables you to build simple smartphone pages that you can navigate using graphical icons and page-swipe. The Basic Mobile Layout displays individual frames from your HTML canvas on individual mobile pages, by default. You can change this as needed.

❏ **Advanced Mobile Layout.** Offers the same features as the Basic, but also gives you the additional ability to add a table of contents with or without a logo, and add a logo on each page.

In the Advanced Mobile Layout, all frames from your HTML canvas appear together on a single mobile page, by default. You can change this as needed.

Having selected one layout, you may choose to switch to the other, if needed. For more information, see *How to Change Between Basic and Advanced Mobile Layouts* on page 352.

Procedure: **How to Use the Basic Mobile Layout**

You can work with the Basic Mobile Layout when you have saved content in the HTML canvas and when you have set the Enable mobile property to *Yes*.

1. Right-click the HTML canvas and click *Show Mobile Layout (basic)*.

   The Basic Mobile Layout is displayed.

Styling for the components of the Mobile Layout is controlled by the cascading style sheet. For more information, see *Cascading Style Sheet Class Mapping List for Mobile Components* on page 356.

2. Edit the mobile content in the following ways:

❑ To preview all pages, right-click the Mobile Layout canvas and click *Show all pages*. This displays all pages side-by-side.

❑ A horizontal line appears below the navigation area. Any objects that appear below the line come from the HTML canvas, but were not automatically added to the Mobile Layout.

To add objects to the Mobile Layout, drag them from the area below the line to the desired place in the layout.

The system recognizes this as a manual modification to the Mobile Layout. This triggers a message: Manual modification of mobile layout will stop default generation of the pages. It will be your responsibility to maintain it.

Accepting this means that changes to the underlying data will not flow automatically to the mobile pages. You must open the HTML file, right-click the HTML canvas, and select *Update Layout*.

Note: You can undo manual modification of the Mobile Layout by right-clicking the canvas and clicking *Reset mobile to default*. This restores automatic update of the HTML pages, and removes any manual modifications.

To take advantage of the default generation of the pages, it is recommended that you avoid making manual modifications.

❑ To combine pages horizontally, drag one page to another.

❑ To change the home page, select the desired page, right-click the Mobile Layout canvas, and click *Set current page as home page*.

Additionally, you can drag a page to the home page screen. If necessary, drag the old page to a different screen.

Note: When you change the page order by manually dragging in the Mobile Layout, it overrides the page order specified in the Tasks & Animations panel.

❑ If there are any empty pages in your preview, right-click the canvas and click *Remove empty pages*.

3. Save the HTML file.

You can now view the HTML file on a smartphone. The layout is derived from the Mobile Layout. For more information, see *Viewing Mobile Output on a Smartphone* on page 356.

Additionally, you can view the HTML file on a desktop or tablet. The layout is derived from the HTML canvas.

Procedure: How to Change Between Basic and Advanced Mobile Layouts

To change between Basic and Advanced Mobile Layouts:

1. Right-click the existing *Mobile Layout* canvas and click *Reset mobile to default*.

    This restores the Mobile Layout to its original state.

2. Right-click the existing *Mobile Layout* canvas and click *Back to desktop*.

    The HTML canvas opens.

3. Right-click the HTML canvas and click the new layout, *Show Mobile Layout (basic)* or *Show Mobile Layout (advanced)*.
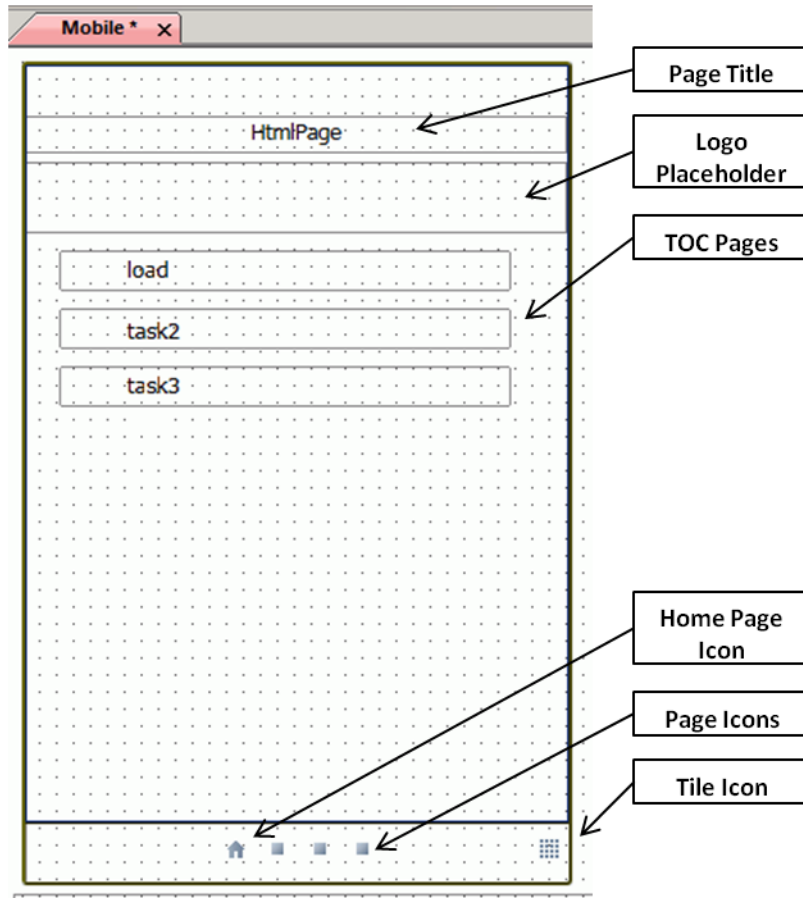
Procedure: How to Create a Table of Contents for Mobile Output

With the Advanced Mobile Layout, you can include a table of contents page to help users navigate quickly. This may be useful when your mobile output contains a large number of pages.

To design output for smartphones that includes a table of contents:

1. Create an HTML page.

2. Right-click the *HTML* canvas and click *Show mobile layout (advanced)*.

    The Mobile Layout opens and displays the table of contents preview, as shown in the following image.

Mobile * ✕

HtmlPage

load

task2

task3

Page Title

Logo Placeholder

TOC Pages

Home Page Icon

Page Icons

Tile Icon

The Mobile Layout default display shows a single page, which previews the appearance of the table of contents in a smartphone format.

In our example, this rectangle contains the title *HtmlPage*.

Styling for the components of the Mobile Layout is controlled by the cascading style sheet. For more information, see *Cascading Style Sheet Class Mapping List for Mobile Components* on page 356.

3. Edit your layout in the following ways:

   ❑ To change the title, use the drop-down at the top of the Properties panel and select *table_of_contents*. In the *Title* field, type the new title.

   The new title now appears in the table of contents preview.

❏ You can choose to show or to hide the table of contents on a smartphone. The default is to show it.

To hide the table of contents on a smartphone, select *No* from the *Show at run time* property field.

In the preview, the rectangles for each page represent hyperlinks in the table of contents, which open the selected page. In our example, there are hyperlinks for the *task2* and *load* page titles.

❏ To change page titles, select the *Title* property field for a specified page and type the desired title.

The new titles appear at the top of the page and on the table of contents.
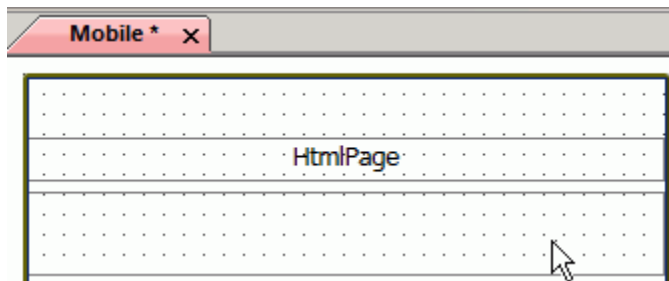
4. Save the HTML file.

    You can now view the HTML file with a table of contents on a smartphone. For more information, see *Viewing Mobile Output on a Smartphone* on page 356.

Procedure: **How to Add a Logo to a Mobile Table of Contents**

You can add a logo to the table of contents page of your mobile layout.

1. Open the Advanced Mobile Layout of an HTML file that has a table of contents.

    The logo placeholder is indicated by the cursor in the following image.

2. In the drop-down at the top of the Properties panel, click *table_of_contents*.

    The table of contents properties display.

3. In the *Logo* field, click the ellipsis button.

    The Open File dialog box appears.

4. Select the desired logo file and click *OK*.

    The logo now appears in the table of contents preview.

Note: The logo placeholder box provides an accurate preview of the way the logo will appear on a smartphone. It will indicate if the graphic file you selected needs to be resized.
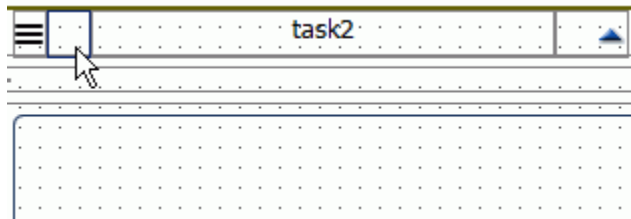
5. Save the HTML file.

   You can now view the HTML file with a table of contents on a smartphone. For more information, see *Viewing Mobile Output on a Smartphone* on page 356.

Procedure: **How to Add a Logo to Each Page of the Contents**

You can add a logo to each page of the table of the contents in your mobile layout.

1. Open the Advanced Mobile Layout of an HTML file that has a table of contents.

2. Open a page, other than the table of contents page.

   The small square near the top left of the page, indicated by the cursor in the following image, is a placeholder for a logo.

3. In the drop-down at the top of the Properties panel, click *table_of_contents*.

   The table of contents properties fields display.

4. In the All swipes logo field, click the ellipsis button.

   The Open File dialog box appears.

5. Select the desired logo file and click *OK*.

   The logo now appears on each page of the preview.

   Note: The logo placeholder for the individual pages is a small box. It provides an accurate preview of the way the logo will appear on a smartphone, and indicates whether the graphic file that you selected needs to be resized.

6. Save the HTML file.

   You can now view the HTML file with a table of contents on a smartphone. For more information, see *Viewing Mobile Output on a Smartphone* on page 356.

Viewing Mobile Output on a Smartphone

There are multiple ways that mobile output from Developer Workbench can be viewed on a smartphone, depending upon the configuration of your environment. These include:

❑ Mobile Favorites

❑ The Mobile Faves App

These ways are outlined in the following section.

Note: Check with your system administrator for your best practice in viewing mobile output.

Mobile Favorites

Mobile Favorites are HTML files, reports, graphs, hyperlinks, and other item types residing in the Domains area, that you can add to the Mobile Favorites list in the BI Portal. Adding items to the Mobile Favorites list enables you to display them on mobile devices.

Note: Content residing on your local folder is not accessible through Mobile Favorites. It must reside in the Domains area of Developer Workbench.

Mobile Faves App

The Mobile Faves app is automatically set up to obtain content from a server on the Internet, where you can view sample business content. You can add your own Db2 Web Query servers to the list, including those that contain the HTML files that are enabled for mobile output. You will see a list of available folders saved to the selected server connection. These folders contain all of the content previously selected as Mobile Favorites from your Db2 Web Query portal.

iPhone or Android users can download the free Mobile Faves app from the App Store or the Google Play Store. For more information, refer to the *Mobile Faves for Android User's Manual* or the *Mobile Faves for iOS User's Manual*.

Cascading Style Sheet Class Mapping List for Mobile Components

This section contains a list of IBI classes that are used to style various components and controls for Mobile design.

Components/Controls	Classes Used
Smartphone container border color	```.IBI_SmartPhoneContainer { border-color:#4C5E76; }```

Components/Controls	Classes Used
Control panel border color	```\n.IBI_MobileControlPanel {\n border-top-color: #4C5E76;\n}\n```
Control panel font	```\n.IBI_MobileControlPanel {\n font-size:small;\n color: green;\n}\n```
Page content font	```\n.IBI_MobilepageContent {\n font-size: large;\n color: Red;\n}\n```
TOC content font	```\n.IBI_MobileTocContent {\n font-size: large;\n color: Blue;\n}\n```
Label font	```\n.IBI_MobileLabel {\n font-size: large;\n color: Azure;\n}\n```
HTML title	```\n.IBI_MobileTocTitle {\n font-size: x-large;\n color: BlueViolet;\n}\n```
Task name in TOC	```\n.IBI_MobileTocItem {\n font-size: x-large;\n color: ForestGreen;\n}\n```
Control panel text	```\n.IBI_MobileSwipeHeaderText {\n font-size: small;\n color: Crimson;\n}\n```

Cascading Style Sheet Class Mapping List

This section contains a list of IBI classes that are used to style various components and controls.

Components/Controls	Classes Used
Body	IBI_PageBg
Report	IBI_Report-iFrame; IBI_rounded_m
Chart	IBI_Report-iFrame; IBI_rounded_m
Hyperlink	IBI_LinkItem
Button	IBI_button
Reset	IBI_button
Label	IBI_ReportControlLabel
Form	IBI_ReportControlPanel; IBI_rounded_m
Submit Button	IBI_button; IBI_btn-run
Reset Button	IBI_button; IBI_btn-reset
Save Selection Button	IBI_btn-saveselection
Defer Button	IBI_btn-defer
Schedule Button	IBI_btn-schedule
Panel	IBI_Panel; IBI_rounded_m
Frame	IBI_rounded_m
Edit box	IBI_ReportControlTarget; IBI_rounded_s
Drop down	IBI_ReportControlTarget; IBI_rounded_s
List Box	IBI_ReportControlTarget; IBI_rounded_s
Double List	IBI_ReportControlTarget, IBI_btn-up, IBI_btn-down, IBI_btn-left, IBI_btn-right
Radio Button	IBI_ReportControlTarget; IBI_Radio
Check Box	IBI_ReportControlTarget; IBI_CheckBox
Text Area	IBI_rounded_m

Components/Controls	Classes Used
Single Source Tree	IBI_ReportControlTarget
Multi Source Tree	IBI_ReportControlTarget
Horizontal Slider	IBI_ReportControlTarget; IBI_rounded_s; IBI_SliderInput; IBI_buttonSliderNav; IBI_btn-left; IBI_btn-right; IBI_SliderTrack; IBI_SliderHandle
Vertical Slider	IBI_ReportControlTarget; IBI_rounded_s; IBI_SliderInput; IBI_buttonSliderNav; IBI_buttonSliderNavVert; IBI_btn-up; IBI_btn-down; IBI_SliderTrack; IBI_SliderTrackVert; IBI_SliderHandle; IBI_SliderHandleVert
Info Window	IBI_InfoWnd; IBI_InfoWndTitleBar; IBI_InfoWndFrame
Tab	IBI_MultiContentPlugin
Tab Header	IBI_pageHeader
TabHeader - Top Tab	IBI_pageHeaderTab_TopBottom
TabHeader - Bottom Tab	IBI_pageHeaderTab_Bottom
TabHeader - Left Tab	IBI_pageHeaderTab_Left
TabHeader - Right Tab	IBI_pageHeaderTab_Right
Tab Selected	IBI_pageHeader_Selected
Vertical Tab/Accordion/Window header	IBI_pageHeader_Vertical
Vertical Tab/Accordion/Window Selected	IBI_pageHeaderVertical_Selected
Tab Heading Text	IBI_pageHeaderText
Tab Heading Text - Left Tab	IBI_textVerticalTopToBottom
Tab Heading Text - Right Tab	IBI_textVerticalBottomToTop

Components/Controls	Classes Used
Tab/Window/Accordion Content	IBI_pageContent
Tab/Window/Accordion Content - Top	IBI_pageContentTab_Top
Tab/Window/Accordion Content - Bottom	IBI_pageContentTab_Bottom
Tab/Window/Accordion Content - Left	IBI_pageContentTab_Left
Tab/Window/Accordion Content - Right	IBI_pageContentTab_Right
Tab/Accordion/Window Min/Max box	IBI_windowNavBox
Tab/Accordion/Window images	IBI_windowMaxRestore; IBI_windowMaximize; IBI_windowMinimize; IBI_windowTiles; IBI_windowFlipRight; IBI_windowFlipLeft
Mobile components	
Smartphone container border color	IBI_SmartPhoneContainer
Control panel	IBI_MobileControlPanel
Page content	IBI_MobilepageContent
TOC content	IBI_MobileTocContent
Label	IBI_MobileLabel
TOC title	IBI_MobileTocTitle
TOC task name	IBI_MobileTocItem
Control panel text	IBI_MobileSwipeHeaderText

Editing Source Syntax

In Developer Workbench, you can edit the source syntax for procedures, procedure components, and other types of files required by your projects.

This topic describes the Text Editor that is available when you are editing the source syntax for procedures, procedure components, and other types of files.

Note: Text editing is not the recommended way for developing new content. The preferred way is to use the different canvases. Text editing should be reserved for troubleshooting or developing code that cannot be accomplished using the canvases.

In this chapter:

❑ Accessing the Text Editor

Accessing the Text Editor

Developer Workbench provides a fully integrated Text Editor that you can use to edit the source syntax for procedures, procedure components, and other types of files required by your applications.

You can access the Text Editor tab by right-clicking a procedure in the Environments Tree panel, and selecting *Open in Text Editor*. You can also access the Text Editor tab by selecting the Embedded JavaScript or Embedded CSS view tabs in the HTML canvas. Some commands are unavailable while in the Text Editor tab accessed through the Embedded JavaScript or Embedded CSS view tabs.

The Text Editor tab is shown in the following image.
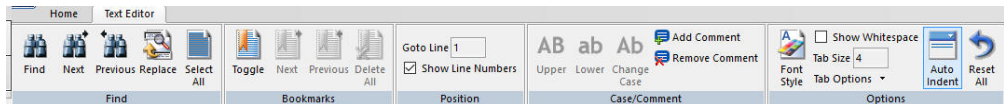
In addition, you can:

❑ Take advantage of color-coded syntax designed to make writing, editing, and debugging procedures easier.

❑ Bookmark lines of a file for quick editing and easy reference.

❑ Find and replace text.

❑ Run procedures or procedure components directly from the Editor.

Finding or Replacing Text Using the Find Group

The Find group contains options that enable you to find or replace text. The Find group is shown in the following image.
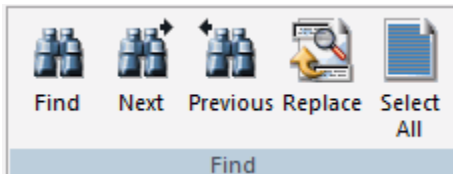
The commands in this group are:

Find

Finds the specified text. You can also press Ctrl+F to activate the Find dialog box.

Next

Finds the next instance of the specified text.

Previous

Finds the previous instance of the specified text.

Replace

Replaces specific text with different text.

Select All

Selects all of the text in the procedure.

Reference: **Find Dialog Box**

You can use the options on the Find dialog box to indicate how to search for information.

The options on the Find dialog box are:

Find what

Provides a text box where you can specify the text that you want to find. Optionally, you can click the arrow to select a search term or phrase that was previously specified.

Match whole word only

Select this option to match the whole word only.

Match case

Select this option to match the uppercase or lowercase value, as specified in the Find what field.

Direction

Select up or down to indicate the direction in which the Text Editor should search.

Find Next

Allows you to find the next instance of your search term.

Mark All

Marks all instances of your search term in the Text Editor.

Cancel

Cancels the current search and closes the Find dialog box.

Reference: Replace Dialog Box

You use the options on the Replace dialog box to indicate how to use the find feature to find and replace information.

The options on the Replace dialog box are:

Find what

Provides a text box where you can specify the text that you want to find. Optionally, you can click the arrow to select a search term or phrase that was previously specified.

Replace with

Provides a text box where you can specify the text that is going to replace the text for which you are searching. Optionally, you can click the arrow to select a replacement term or phrase that was previously specified.

Match whole word only

Select this option to match the whole word only.

Match case

Select this option to match the uppercase or lowercase value, as specified in the Find what and Replace with fields.

Replace In

Enables you to specify where you would like the replacement to occur. Options include Selection or Whole file.

Find Next

Allows you to find the next instance of your search term.

Replace

Replaces the search information that you specified in the Find next field with the text or other information that you indicated in the Replace with field.

Replace All

Allows you to automatically replace all instances of the search term you indicated in the Find next field with the information you specified in the Replace with field.

Cancel

Cancels the current search and replaces and closes the Replace dialog box.

Working With Bookmarks Using the Bookmarks Group

The Bookmarks group enables you to turn bookmarks on or off, find the next bookmark, find the previous bookmark, and delete all bookmarks. The Bookmarks group is unavailable when you access the Text Editor tab through the Embedded JavaScript or Embedded CSS view tabs, in the HTML canvas. The Bookmarks group is shown in the following image.
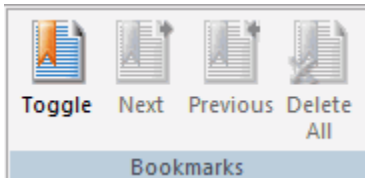
The commands are:

Toggle

Turns bookmarks on or off.

Next
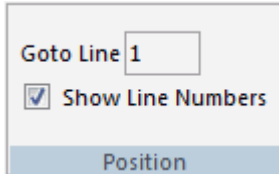
Goes to the next bookmark.

Previous

Goes to the previous bookmark.

Delete All

Deletes all bookmarks.

Placing the Cursor Using the Position Group

The Position group allows you to place your cursor at the desired line and allows you to turn off line numbers. The Position group is shown in the following image.
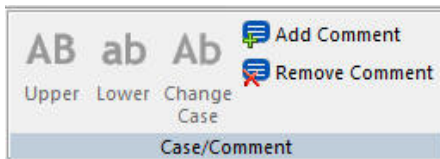

The commands are:

Goto Line

Displays the current line your cursor is on. You can type a different line number into the text box to place your cursor on that line.

Show Line Numbers

When selected, displays line numbers. This option is selected, by default.

Changing the Case of Text Using the Case/Comment Group

The Case/Comment group contains options for changing selected text to uppercase or lowercase. The Case/Comment group is unavailable when you access the Text Editor tab through the Embedded JavaScript or Embedded CSS view tabs, in the HTML canvas. The Case/Comment group is shown in the following image.

The commands are:

Upper

Changes the selected text to uppercase (all capital letters).

Lower

Changes the selected text to lowercase (all lowercase letters).

Change Case

Changes the case of the selected text to lowercase or uppercase depending on how the selected text appeared first. You can also change the case of selected text by using the shortcut menu on the canvas, while text is selected.

Add Comment

Inserts a new comment line if your pointer appears on a new line or changes an active line of text to a comment. You can also use the shortcut menu on the canvas to add a comment.

Remove Comment

Deletes the previous comment change. This option is similar to the Undo command on the Quick Access Toolbar. You can also use the shortcut menu on the canvas to remove a comment.

Formatting Text Using the Options Group

The Options group contains commands that you can use to format the text in a procedure. The Options group is unavailable when you access the Text Editor tab through the Embedded JavaScript or Embedded CSS view tabs, in the HTML canvas. The Options group is shown in the following image.
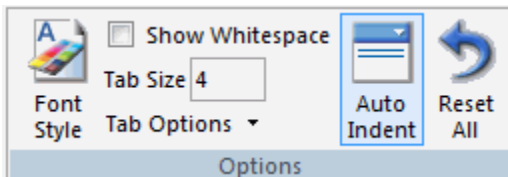
Note: When you access the Text Editor through the Embedded JavaScript or Embedded CSS view tabs, the only option in the Options group is Font Style.

The commands are:

Font Style

Opens the Font and Color Settings dialog box from which you can modify the appearance of the syntax in a procedure.

Show Whitespace

Replaces spaces between characters with a visual indicator.

Tab Size

Controls the number of spaces a tab character uses.

Tab Options

Provides the options to control tab characters. You can substitute spaces for tab characters or keep tab characters in the syntax.

Auto Indent

Automatically indents the syntax within a procedure.

Reset All

Restores all options back to the default settings.