

WebFOCUS

Online Help PDF

WebFOCUS Business User Edition Release 8.2 Version 01

Active Technologies, EDA, EDA/SQL, FIDEL, FOCUS, Information Builders, the Information Builders logo, iWay, iWay Software, Parlay, PC/FOCUS, RStat, Table Talk, Web390, WebFOCUS, WebFOCUS Active Technologies, and WebFOCUS Magnify are registered trademarks, and DataMigrator and Hyperstage are trademarks of Information Builders, Inc.

Adobe, the Adobe logo, Acrobat, Adobe Reader, Flash, Adobe Flash Builder, Flex, and PostScript are either registered trademarks or trademarks of Adobe Systems Incorporated in the United States and/or other countries.

Due to the nature of this material, this document refers to numerous hardware and software products by their trademarks. In most, if not all cases, these designations are claimed as trademarks or registered trademarks by their respective companies. It is not this publisher's intent to use any of these names generically. The reader is therefore cautioned to investigate all claimed trademark rights before using any of these names other than to refer to the product described.

Copyright © 2016, by Information Builders, Inc. and iWay Software. All rights reserved. Patent Pending. This manual, or parts thereof, may not be reproduced in any form without the written permission of Information Builders, Inc.

Contents

1. WebFOCUS Business User Edition	21
Introducing the WebFOCUS Business User Edition	21
WebFOCUS BUE Components.	21
WebFOCUS Installation Overview.	22
WebFOCUS and Your Network.	22
Installation	22
Installing WebFOCUS Business User Edition on a Windows Environment.	22
WebFOCUS Business User Edition System Requirements for Windows.	22
Hardware Requirements.	22
Software Requirements.	23
Installing WebFOCUS Business User Edition.	23
Running WebFOCUS Business User Edition Services Manually.	31
Uninstalling WebFOCUS Business User Edition.	33
Troubleshooting WebFOCUS Business User Edition.	35
Port Assignment.	35
WebFOCUS Business User Edition Log Files.	35
Troubleshooting the Uninstall Process.	36
Installing the WebFOCUS Business User Edition on a Linux Environment.	38
WebFOCUS Business User Edition System Requirements on a Linux Environment. .	39
Installation Prerequisites.	39
System Resource Limits.	40
Installing WebFOCUS Business User Edition Using Console Mode.	40
Configuring Java Memory Allocation for the Application Server.	42
Launching WebFOCUS Business User Edition.	42
Running WebFOCUS Business User Edition Servers Manually.	43
Updating WebFOCUS Business User Edition.	44
Uninstalling WebFOCUS Business User Edition.	45
Configuring for Email Distribution.	45
Navigating the Environment	48
Navigating the WebFOCUS Business User Edition Portal.	48
Getting Started Page.	48
Home Page.	49

Menu Bar	49
Resources Tree	51
Node Context Menu Options	51
Quick Links	52
Understanding Domain Content	53
Selecting a Domain	53
Identifying Types of Content in the Resources Tree	53
Identifying Content in the Resources Tree	54
Options for Folders and Items	55
Viewing Folder and Item Properties	59
Navigating the Administration Console	63
Navigating the Configuration Tab	64
Navigating the Security Tab	65
Navigating the ReportCaster Tab	66
Navigating the Diagnostics Tab	66
Reviewing Version Information	67
Reviewing Client Verification	68
HTTP Request Headers and JVM Property Information	68
Navigating the InfoAssist+ Interface	69
InfoAssist+ Application Window	69
Application Main Menu	70
Quick Access Toolbar	73
Ribbon	74
Working With the Ribbon	75
Home Tab	76
Insert Tab	76
Format Tab	77
Data Tab	78
Slicers Tab	78
Layout Tab	79
View Tab	79
Field Tab	80
Series Tab	81

Understanding the Canvas.....	81
Using the Query Pane and Filter Pane on the Canvas.....	82
Understanding Output Options.....	83
Using the Navigation Taskbar.....	85
Using the Status Bar.....	86
Understanding the Resources Panel.....	86
Using the Data Pane.....	87
Using the Data Pane to Add Fields to a Report.....	88
Using the Query Pane and Filter Pane.....	90
Using Field Containers	91
Field Containers for Reports.....	91
Field Containers for Charts and Visualizations.....	91
Using Shortcut Menu Options in the Query Pane.....	95
Dynamic Grouping.....	98
2. Setting Up Your Environment	103
Using the Administration Console	103
Opening the Administration Console.....	103
Using the Administration Console Menu Bar.....	105
Using the Licenses Menu.....	105
Reviewing Client License Information.....	105
Reviewing User Audit Information.....	106
Reviewing Third Party Licenses.....	109
Clearing the Cache.....	110
Closing the Administration Console.....	110
Opening Administration Console Help.....	110
Configuring Security.....	110
Using Internal Security.....	111
Using External Security.....	113
Understanding Custom Settings.....	113
Understanding NLS Settings.....	115
Customizing the Dynamic Language Switch.....	117
Understanding Redirection Settings.....	118

Changing Application Settings	120
Changing Client Settings	124
InfoAssist+ Properties.....	124
Understanding InfoAssist+ Auto Drill Properties.....	131
Reporting Server Configuration Settings.....	133
Using the ReportCaster Console	137
Using the ReportCaster Console.....	138
Server Status.....	139
Viewing the Distribution Server Status.....	140
Separate Job Queues for Each Data Server.....	143
Viewing, Stopping, Suspending, and Restarting the Distribution Server.....	143
Job Status.....	144
Job Log.....	145
ReportCaster Configuration.....	146
Configuration Icons.....	146
Configuration Tab Folders.....	147
Changing Configuration Settings.....	148
Changing Default User IDs.....	149
Distribution Server Settings.....	149
General Preferences.....	152
Specifying Schedule Format Settings.....	153
Specifying Schedule Distribution Method Settings.....	156
Email Distribution.....	158
Notification.....	163
Zip Settings.....	166
Other Schedule Defaults.....	168
Log Purge.....	169
LDAP Setting.....	170
Data Servers Settings.....	173
Configuring Reporting Servers With ReportCaster.....	178
Using Blackout Periods.....	180
Configuring Weekly Blackout Periods.....	186
Weekly Blackout Period Settings.....	187

Configuring Monthly Blackout Periods.	189
Monthly Blackout Period Settings.	190
Configuring Single Day Blackout Periods.	192
Single Day Blackout Period Settings.	192
Configuring Every Day Blackout Periods.	193
Every Day Blackout Period Settings.	194
Importing Blackout Periods.	196
Viewing Blackout Period Import File Format.	198
Single Date Blackout Period Import File Entry Layout.	198
Weekly Blackout Period Import File Entry Layout.	198
Monthly Blackout Period Import File Entry Layout.	199
Every Day Blackout Period Import File Entry Layout.	199
Extracting Blackout Period Profiles.	205
Extracting File Formats.	205
Global Updates.	207
Purge Logs.	210
Working With Domains.	211
Managing Domain Users.	212
Granting Access to a Domain.	212
Managing Domain Private Resources.	212
Deleting Domains.	212
Managing Domain Users After Deleting a Domain.	213
Working With Folders.	213
Managing Users.	217
Understanding Users.	217
Understanding User Name Requirements.	218
Managing Groups.	222
Understanding Groups.	222
Creating Groups From Domains.	222
Managing Private Resources.	225
Authenticating Users to Your Active Directory.	225
Configuring Active Directory/LDAP Authentication.	225
Creating BUE User Accounts When Configured for Active Directory Authentication.	229

Configuring WebFOCUS BUE for SSL	230
Establishing the Self-Signed Certificate as a Trusted Certificate.....	232
Importing the Trusted Certificate into the Keystore.....	232
Updating the Connector Protocols in the Tomcat Server.xml File.....	233
Creating a Change Management Package	233
Changing InfoAssist+ User Preferences	238
View.....	239
Layout.....	240
Format.....	240
Environment and Styling.....	240
Changing Global Preferences.....	240
Configuring Hyperstage	241
Hyperstage Overview.....	241
Hyperstage and PostgreSQL.....	241
Configuring the Hyperstage Database.....	242
Configuring Hyperstage.....	242
Configuration Tips and Examples.....	243
Using the Hyperstage Database Beyond WebFOCUS.....	244
Starting and Stopping the Hyperstage Server	244
Quick Copy For Hyperstage Using Extended Bulk Load Utility.....	246
Managing Hyperstage Tables.....	247
About the Hyperstage Database Files.....	247
About Supported Data Types.....	248
Creating and Dropping Tables.....	250
Character Set Support.....	251
Supported Character Sets.....	251
Collations and Comparisons.....	251
Padding.....	252
Hyperstage Backup and Recovery.....	252
Backup Procedure.....	253
Restore Procedure.....	253
Functions and Operators.....	253
Hyperstage Optimizer Supported Functions and Operators.....	253

Comparison Functions and Operators.....	253
Control Flow Functions.....	253
String Functions.....	254
Numeric Functions.....	254
Date and Time Functions.....	256
Text Search and Other Functions.....	256
Group By Aggregate Functions.....	256
Hyperstage Data Tools.....	257
Hyperstage Consistency Manager.....	257
Hyperstage Consistency Manager Tests.....	258
About Rebuilding or Repairing Knowledge Nodes.....	261
About Cleanup Procedures.....	261
Distributing the Quick Data Add-In File	261
WebFOCUS Quick Data Options.....	267
Configuring a Default WebFOCUS BUE Environment.....	269
3. Working With Data	273
Working With Data Overview	273
Understanding Your Data Structures.....	274
Preparing Data for Upload	274
Naming Conventions and Excel Sheet Names.....	275
Removing Introductory Information.....	276
Placing Column Titles in the First Row.....	276
Removing Aggregated Information.....	277
Using Excel Named Ranges.....	277
Preparing Hierarchical Data Columns.....	278
Removing Date Formulas.....	279
Uploading and Appending Spreadsheets	279
Uploading Images and Documents	282
Connecting to Data and Editing Data	284
Creating a Synonym With the Connect to Data Wizard	284
Step One: Connect to SQL.....	285
Step Two: Import Metadata.....	287

Step Three: Edit Metadata.....	288
Editing And Deleting Metadata.....	289
Using Data Preparation Options	290
Categorize Fields Into Measures, Dimensions and Hierarchies Screen Overview	291
Ribbon.....	291
Diagnostics.....	292
Undo/Redo.....	292
Preparation.....	293
Templates.....	295
Metadata.....	295
Sample Data.....	297
Options.....	297
Load.....	302
Load and Next.....	303
Windows.....	304
Table/Column Window.....	304
Measures/Dimensions Window.....	305
Using the Properties Dialog Box.....	312
Filters/Groups Window.....	317
Output Window.....	320
Creating Clusters.....	321
Pivoting Repeating Columns Into Rows.....	322
Creating Hierarchies.....	325
Joining and Blending Data	326
Joins.....	326
Blending Data.....	327
Creating Virtual Fields	329
Selecting a Temporary Field.....	329
Detail (DEFINE).....	330
Summary (COMPUTE).....	330
Creating Temporary Fields Independent of a Master File.....	331
Enabling the Display of Missing Values for a DEFINE or COMPUTE.....	332

4. Creating Content	333
Generating Sample Content	333
Generating Sample Content Overview.....	333
Analyzing Sample Content.....	337
Creating Reports	343
Choosing a Report Output.....	344
Creating Charts	345
Chart Outputs.....	348
Creating Your Own Chart Types	349
Introducing Chart Extensions.....	349
Creating a Chart Extension.....	350
Using the Chart Extension API.....	351
Rendering Charts.....	351
Configuring Your Chart Extension.....	355
Creating a config Object for Chart Engine Configuration.....	355
Configuring the Chart Interface.....	357
Accessing Data for Your Extension.....	359
Defining and Using Buckets in an Extension.....	360
Handling Partial and Null Data in an Extension.....	363
Installing a Chart Extension.....	363
Using Your Extension in a WebFOCUS Request.....	365
Building a Document	367
Inserting Reports From Multiple Data Sources.....	367
Inserting a New Report.....	368
Inserting a New Chart.....	369
Inserting an Existing Report.....	370
Creating a Document From a Single Report.....	370
Inserting Text and Images.....	370
Editing Components in a Document.....	371
Creating Multi-Page Documents	378
Creating Multi-page Documents.....	379
Creating a Multi-page Active Technologies Dashboard.....	379

Navigating the Page Menu.....	380
Using the Active Cache Option.....	381
Enabling Active Cache Through InfoAssist+.....	381
Building Visualizations	381
Creating a Visual.....	382
Changing the Visual Type.....	383
Selecting a Visual.....	384
Grids.....	386
Bar Charts.....	388
Line Charts.....	390
Area Charts.....	392
Pie Charts.....	393
Ring Pie Charts.....	395
Scatter Charts.....	397
Bubble Charts.....	398
Matrix Marker.....	400
Treemaps.....	401
Gauges.....	402
Heatmaps.....	403
Interacting With Visualizations.....	404
Minimizing or Maximizing a Visual.....	410
Renaming a Visual.....	411
Creating Matrix Charts	412
Using Active Technologies	416
Active Technologies Report Overview.....	417
Security Features.....	419
Handling a Large Amount of Data.....	419
Usage Notes for Active Technologies.....	420
Creating an Active Technologies Report.....	420
Active Technologies Report Menu Options.....	420
Active Technologies Cell Menu Options.....	424
Configuring Active Technologies Report Options.....	425
General Tab.....	426

Menu Options Tab.....	426
Colors Tab.....	428
Advanced Tab.....	429
Creating an Active Technologies Chart.....	430
Active Technologies Options for Charts.....	430
Creating an Active Technologies Dashboard.....	432
Active Technologies Dashboard Prompts.....	433
Target Reports.....	433
Using Multiple Reports as Targets and Sources.....	434
Using Navigation Options for Reports	443
Creating Maps to Illustrate Trends	444
InfoAssist+ and Esri Integration	446
Creating and Customizing Maps in InfoAssist+.....	449
Adding a Custom Geographic Role.....	468
Building InfoMini Applications	475
Understanding InfoMini Applications.....	475
Using the InfoMini Button.....	476
Understanding the InfoMini Layout.....	477
Creating an InfoMini Application.....	478
Viewing Data Behind Visuals	479
Creating Multi-Y Axis Comparative Visuals.....	481
Creating HOLD Files	483
Valuable Applications of HOLD Files.....	483
Storing HOLD Files.....	483
Output Formats for Reports.....	483
Creating Hold Files.....	484
FOCUS Format Index Fields.....	487
Creating a Subquery Filter Using a HOLD File.....	487
Creating Shortcuts and URLs	488
Creating Shortcuts.....	488
Creating URLs.....	489
Working With the WebFOCUS BUE Portal	490
Accessing Page Shortcut Menu Options.....	493

Creating Report Queries With InfoAssist+	493
Ribbon Command Reference	497
Ribbon Commands for Reports.....	497
Home Tab.....	497
Format Tab.....	499
Data Tab.....	501
Slicers Tab.....	502
Layout Tab.....	503
View Tab.....	504
Field Tab.....	506
Ribbon Commands for Charts.....	511
Home Tab.....	511
Format Tab.....	513
Data Tab.....	518
Slicers Tab.....	519
Layout Tab.....	520
View Tab.....	521
Field Tab.....	523
Series Tab.....	526
Ribbon Commands for Documents.....	528
Home Tab.....	528
Insert Tab.....	531
Format Tab.....	532
Data Tab.....	534
Slicers Tab.....	535
Layout Tab.....	536
View Tab.....	537
Field Tab.....	539
Series Tab.....	544
Ribbon Commands for Visualizations.....	545
Home Tab.....	545
Format Tab.....	547
View Tab.....	549

Field Tab.....	550
Series Tab.....	551
5. Customizing Content	553
Customizing Reports	553
Styling Reports.....	556
Report Style Dialog Box.....	566
Color Dialog Box.....	567
Report Group.....	568
Report Features.....	570
Customizing Charts and Visualizations	571
Chart Features.....	571
Labeling Charts.....	572
Accessing Chart Formatting Tools.....	573
Using Live Preview.....	573
Formatting a Series.....	573
Associated Dialog Boxes.....	574
Format Series Dialog Box.....	574
Edit Title Dialog Box.....	575
Traffic Light Condition Dialog Box.....	575
Series Elements Shortcut Menu.....	576
Using Series Properties.....	578
Enhancing Series Using the Series Tab.....	579
Formatting Charts Using the Series Tab.....	580
Formatting Data Labels.....	587
Associated Dialog Boxes.....	587
Format Labels Dialog Box.....	588
Style Dialog Box.....	591
Line Style Dialog Box.....	592
Data Labels Elements Shortcut Menu.....	592
Using Data Labels Properties.....	592
Formatting a Legend.....	594
Format Legend Dialog Box.....	594

Legend Elements Shortcut Menu.	595
Using Legend Properties.	596
Formatting Gridlines.	598
Format Grid Lines Dialog Box.	598
Gridline Elements Shortcut Menu.	601
Using Gridline Properties.	601
Formatting Axis Labels.	608
Format Axis Dialog Box.	609
Secondary Axes Options.	613
Axis Elements Shortcut Menu.	613
Using Axis Properties.	614
Formatting a Frame and a Background.	617
Frame & Background Dialog Box.	618
Frame and Background Shortcut Menu.	624
Using Frame and Background Properties.	625
Formatting a Gauge Chart.	626
Format Gauge Dialog Box.	626
Gauge Elements Shortcut Menu.	630
Using Gauge Properties.	630
Formatting Page Headings and Page Footings.	635
Using Additional Formatting Features.	638
Collation in Charts.	642
Using Storyboards	642
Animating Visualizations	644
Using Filters to Customize the Display of Data	645
Including or Excluding a Filter.	647
Using the Prompt Functionality to Select Field Information at Run Time.	647
Using Auto Drill	648
Using the Auto Linking Feature to Link Content	650
Using Multi Drill	656
Adding Parameters for Data Selection at Run Time	667
Working With Slicers	670
Creating Slicers.	670

Filtering With Slicers.....	672
How Slicers Cascade Together.....	672
Changing a Field Format	673
Dynamic Grouping	675
6. Using Content	679
Creating Distribution Lists	679
Creating a Distribution List.....	679
Maintaining Distribution Lists	684
Editing and Deleting a Distribution List.....	684
Accessing Distribution Lists.....	685
Bursting a Report.....	686
Bursting Guidelines and Limitations.....	688
Specifying Multiple Email Addresses.....	690
Creating Schedules	692
About the Basic Scheduling Tool.....	692
Basic Scheduling Tool Quick Access Toolbar.....	693
Basic Scheduling Tool Ribbon.....	693
Creating a Schedule in the Basic Scheduling Tool.....	695
About Tasks in the Basic Scheduling Tool.....	697
Task Options in the Basic Scheduling Tool.....	697
Specifying Parameter Values.....	698
Deleting a Parameter.....	709
Creating a New Parameter.....	709
Advanced Task Settings.....	711
Distribution Options in the Basic Scheduling Tool.....	712
Using the Email Distribution Option in the Basic Scheduling Tool.....	712
Using the Printer Distribution Option in the Basic Scheduling Tool.....	717
Notification Options in the Basic Scheduling Tool.....	718
Setting On Error and Always Notification in the Basic Scheduling Tool.....	719
About Properties in the Basic Scheduling Tool.....	720
About Recurrence in the Basic Scheduling Tool.....	721
The Run Once Interval.....	722

The Minutes Interval.....	723
The Hourly Interval.....	724
The Daily Interval.....	725
The Weekly Interval.....	726
The Monthly Interval.....	727
The Yearly Interval.....	729
The Custom Interval.....	729
Advanced Settings.....	731
Maintaining Schedules	733
About Maintaining a Schedule in the Basic Scheduling Tool.....	733
Editing a Schedule in the Basic Scheduling Tool.....	735
Copying a Schedule in the Basic Scheduling Tool.....	736
Deleting a Schedule in the Basic Scheduling Tool.....	736
Sharing Schedules.....	736
Publishing Schedules.....	738
Tracking Schedules	738
Log Reports.....	738
Tracking Schedules in the Console.....	738
Using Schedule Logs.....	739
Checking the Job Status.....	739
Using Favorites	744
Publishing Content	745
Using Visualizations at Run-Time	746
Working With Filters at Run Time.....	747
Viewing Data at Run Time.....	750
Using the Run-Time Toolbar Options.....	750
Using the Deferred Report Status Interface	751
Deferred Report Status Interface Features.....	752
Sort Controls for the Deferred Report Status Interface.....	752
Deferred Report Status.....	752
Deferred Report Expiration Setting.....	756
Saved Deferred Output Subject to Temporary Expiration.....	756
Setting the Automatic Refresh Interval.....	756

Viewing Deferred Reports.....	757
Reviewing Deferred Report Parameters.....	757
Saving Deferred Reports.....	758
Deferred Status Delete Confirmation Messages.....	758
7. Troubleshooting	761
Troubleshooting WebFOCUS Business User Edition	761
Starting the Reporting Server Service.....	761
Starting WebFOCUS BUE.....	761
Port Assignment.....	762
WebFOCUS Business User Edition Log Files.....	762
Troubleshooting the Uninstall Process.....	762
Monitoring WebFOCUS Sessions	766
Exporting the Session Monitor Log	768
Viewing WebFOCUS Sessions	769
Reviewing the Session Viewer Main Page.....	769
Reviewing the Session Details Page.....	772
Reviewing Trace Entries.....	775
Reviewing Expanded URL Details.....	776
Reviewing Reporting Server Request Details.....	778
Reviewing Reporting Server Response Details.....	779
Working With Log Files	779
Understanding Warning Messages in InfoAssist+	780
InfoAssist+ Warning Messages.....	781
Unsupported Syntax and Objects.....	782

WebFOCUS Business User Edition

With the WebFOCUS Business User Edition (BUE), you can perform reporting, analysis, and discovery tasks using a single tool.

In this chapter:

- ❑ [Introducing the WebFOCUS Business User Edition](#)
 - ❑ [Installation](#)
 - ❑ [Navigating the Environment](#)
-

Introducing the WebFOCUS Business User Edition

This topic provides an overview the WebFOCUS Business User Edition (BUE).

WebFOCUS BUE Components

WebFOCUS BUE is a web-based environment, consisting of the following components:

InfoAssist+

Perform reporting and analysis as well as data discovery, from a single user interface. These categories of self-service analytics include the following capabilities:

- ❑ **Reporting and Analysis.** Create ad hoc queries; develop reports, charts, documents and other analytic views for personal usage or sharing and distribution in static or interactive formats. You can also create more advanced information assets, such as multi-page analytic documents and multi-source dashboards.
- ❑ **Data Discovery.** Explore your data to find trends, patterns, and gain new insight, by creating visualizations. Running against a high-speed data sandbox and other data sources, you can create interactive visualizations that can be saved, shared, published, and delivered as PowerPoint presentations to help you build business cases through storytelling.

Business User Edition Portal

Provides access to a single web-based environment, where you can create, access, or manage content and groups. You can develop new content or interact with content that has been created by other users, as well as create personal portal pages.

ReportCaster

Provides scheduling and distribution capabilities for reports and procedures, as well as independent files and URLs. Using ReportCaster, reports and files are automatically distributed on a monthly, weekly, daily, or specific basis.

WebFOCUS Installation Overview

This section briefly explains the different WebFOCUS installation components.

WebFOCUS and Your Network

WebFOCUS seamlessly integrates into your existing network by connecting web servers and application servers to your data. Users then access WebFOCUS through a web browser.

The main requirements for installing WebFOCUS are:

- ❑ **Web Browser.** To access WebFOCUS applications, you need a web browser and a TCP/IP connection to a web server or application server.
- ❑ **Web Server and Application Server.** WebFOCUS runs in part through a web server or application server. WebFOCUS BUE installs and configures a required application server.
- ❑ **Data.** WebFOCUS BUE accesses data through a High-Speed Analytics Sandbox (Hyperstage), a hybrid in-memory data store that stores, in a very compressed form, large amounts of data for retrieval, aggregation, and calculation.

Installation

Installing WebFOCUS Business User Edition on a Windows Environment

This topic provides the step-by-step instructions that you need to install WebFOCUS Business User Edition (BUE).

You can begin using WebFOCUS BUE immediately after you install the product. No additional configuration steps are required.

WebFOCUS Business User Edition System Requirements for Windows

To install WebFOCUS BUE, you must have administrator privileges. Additionally, your machine must meet the system requirements that are defined in the Hardware and Software Requirements topics.

Hardware Requirements

- ❑ Quad-core processor, 2.5 GHz minimum per core is recommended.

- ❑ 16 GB of RAM or higher is recommended for optimal performance. 8 GB of RAM minimum.
- ❑ 8 GB of free disk space.
- ❑ Ensure that the Windows page file is properly configured and that is set to *Automatically manage paging file size*. For Windows 64-bit, the recommended paging file size is three times the size of the RAM.

Software Requirements

- ❑ Windows 10, 8.1, 7, Windows 2012, 2012 R2, or 2008 R2 Service Pack 1 Server editions.

Note: The Windows 10 Anniversary upgrade Version 1607 (OS Build 14398.82) is not supported. This version causes Hyperstage to become unstable and disconnect.

- ❑ 64-bit Version of Windows.

- ❑ Supported browsers: Internet Explorer[®] 11, Microsoft Edge[®], Google Chrome[™] (latest version), Mozilla Firefox[®] (latest version).

Note: Accessibility Compliance is not supported at this time.

Installing WebFOCUS Business User Edition

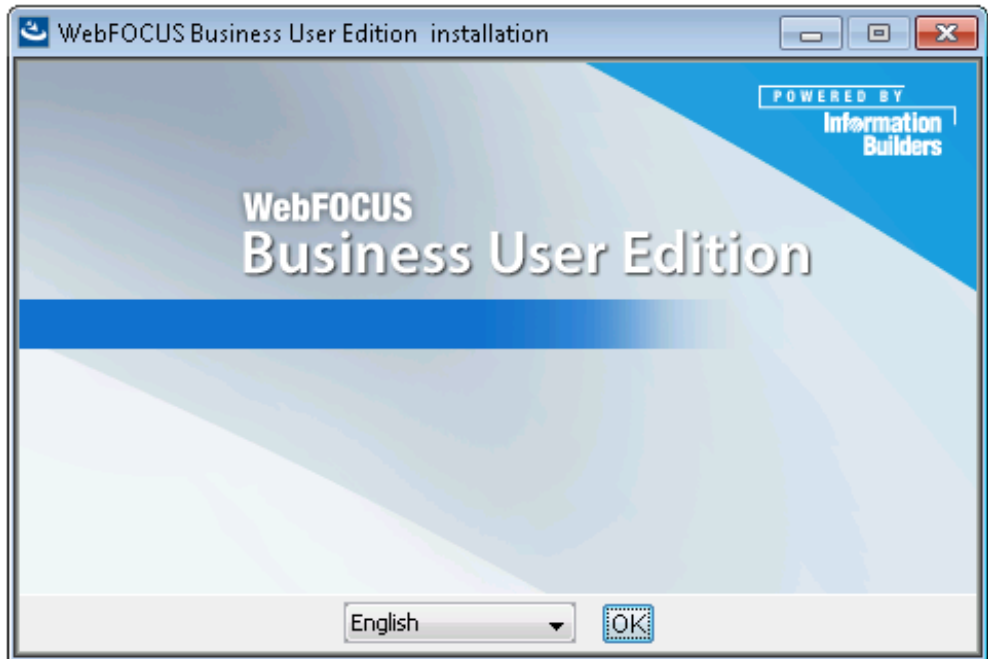
This section describes how to install WebFOCUS BUE.

Procedure: How to Install WebFOCUS Business User Edition

1. Download the installation files.
2. Double-click *installWebFOCUS_BUE8201.exe*.

Note: Java Version 8 Update 102 (8u102) and Tomcat 8.0.36 are included with the WebFOCUS BUE product installation.

The InstallAnywhere installation program opens, as shown in the following image.



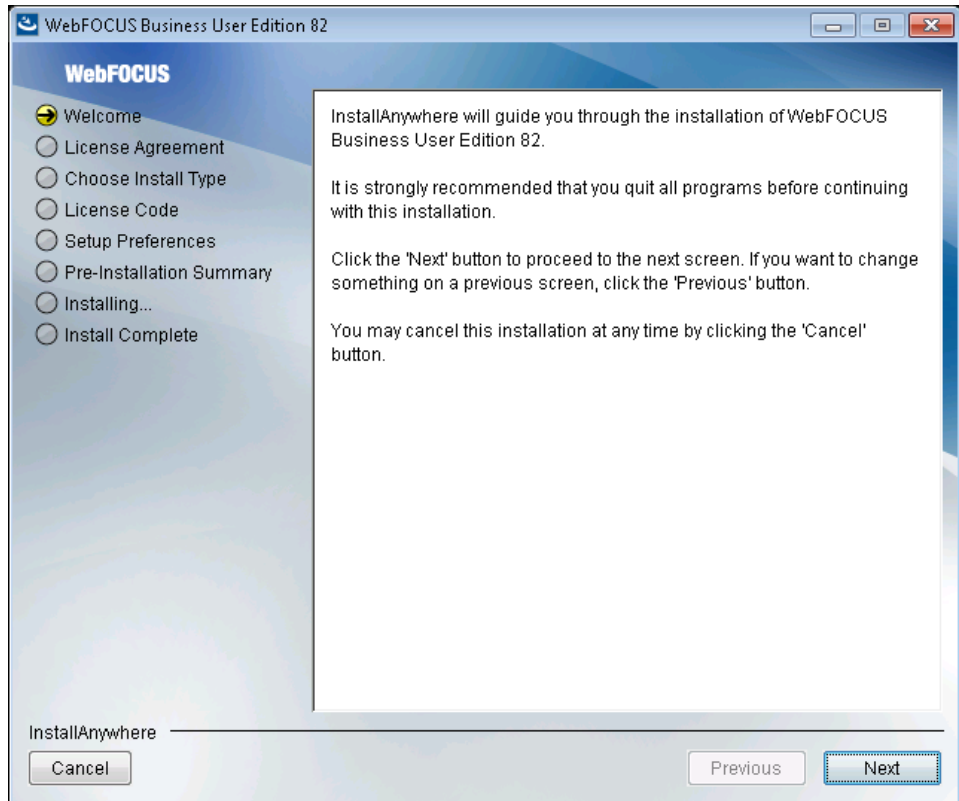
3. Choose the appropriate language from the drop-down list and click **OK**.

The list of languages depends on the regional language settings of the Windows Operating System where WebFOCUS BUE is being installed.

Note: The language selected will be the language used during the installation program. This language is also used to set the code page of the Reporting Server, which is responsible for data access.

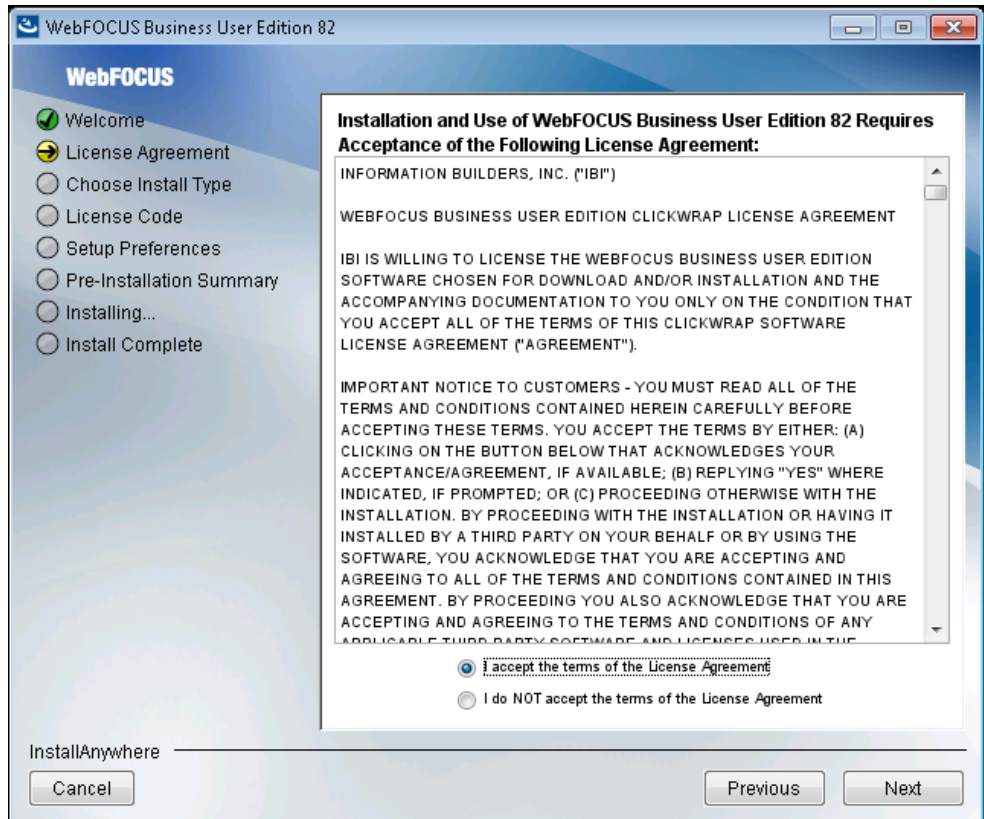
If your data sources are in a different language, you will have to change the code page of the Reporting Server.

The Welcome window opens.



4. Click *Next* to continue.

The License Agreement window opens.

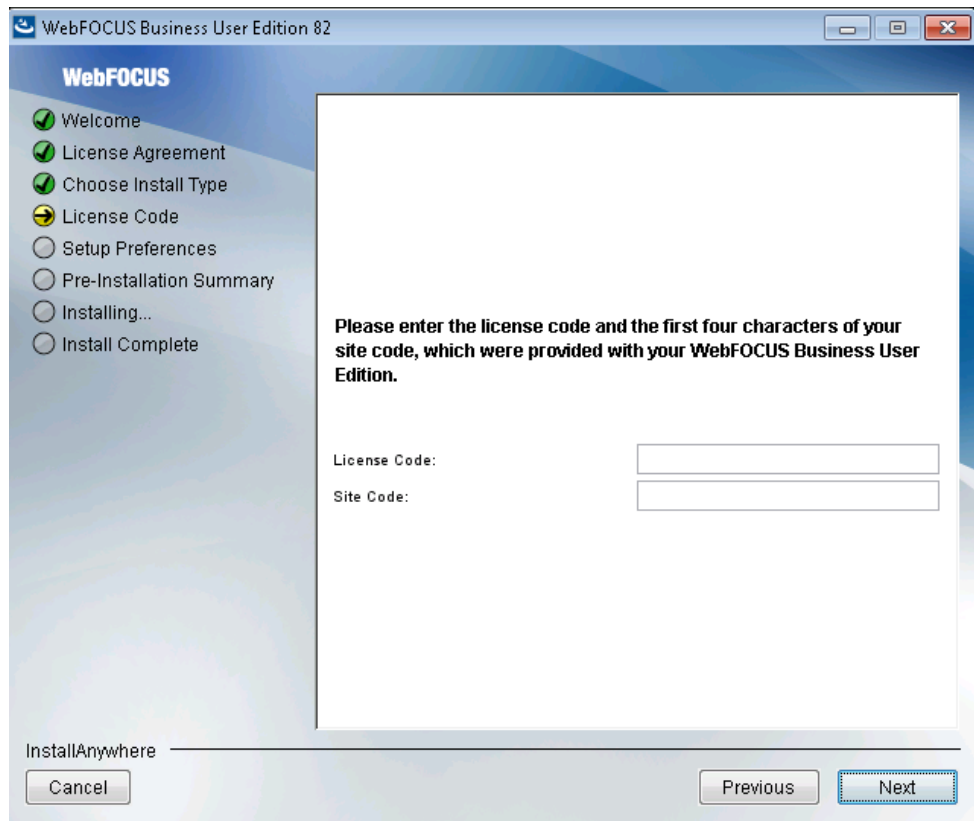


5. Click *I accept the terms of the License Agreement*, and then click *Next*.

Note:

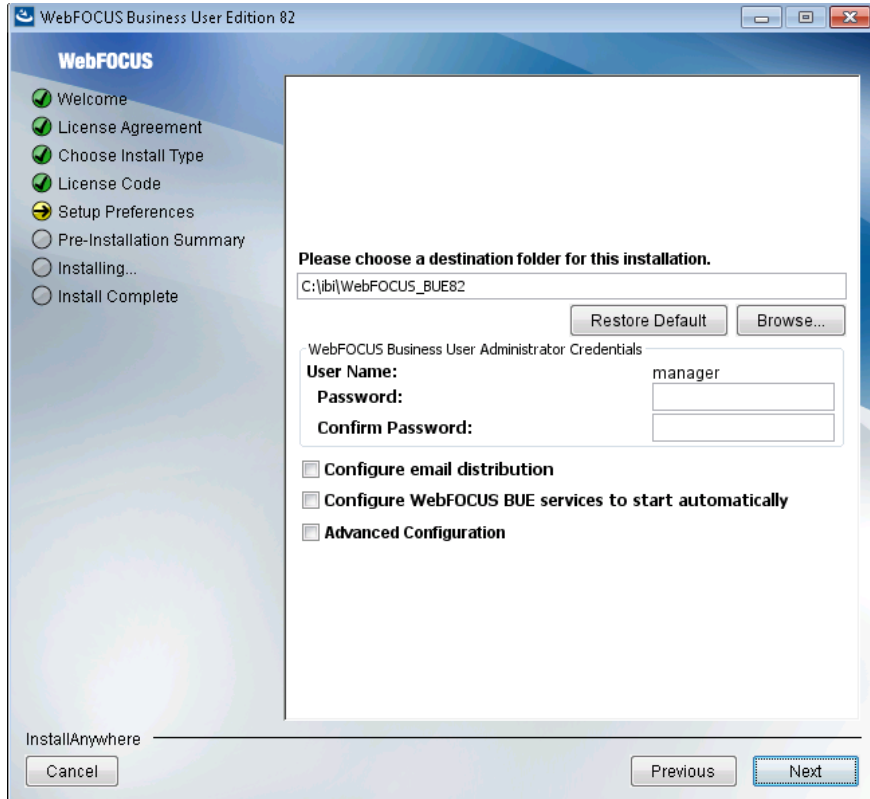
- ❑ Only one installation per platform is supported.
- ❑ If you have an earlier release of WebFOCUS BUE installed on your machine, you will be prompted to update your installation. To perform a full or new installation, you must quit the installation program, uninstall the existing version, and perform a new installation. When you uninstall WebFOCUS BUE, all components are removed from your directory structure, except the WebFOCUS BUE Reporting Server applications. For more information, see [Uninstalling WebFOCUS Business User Edition](#) on page 33.

The License Code window opens, as shown in the following image.



6. Type the WebFOCUS BUE license code and first four digits of the site code that were provided with your copy of WebFOCUS BUE, and then click *Next*.

The Setup Preferences window opens, as shown in the following image.



7. In the Setup Preferences window, complete the following steps:
 - a. The installation program preconfigures a user name, but requires that you type a password. You must complete the User Name credentials section to proceed with the installation. In the Password field, type a password, and then confirm your password.

Note: Passwords must be between 4 and 20 characters and cannot include double quotation marks (") or a percent sign (%). Leading blanks and trailing blanks will be removed.
 - b. Optionally, select the *Configure email distribution* check box to select the type of Mail Server to be used for report distribution. Valid options are:
 - ☐ *Google*. This requires a Google Account Name and Password.

Note: Google may block sign-in attempts when delivering reports through WebFOCUS BUE. To allow WebFOCUS BUE to connect to the Google email account that was configured for email distribution, sign in to the Google account, navigate to *My Account* settings, select *Sign-In & security*, and turn the Allow less secure apps option On.

For more information, see:

<https://www.google.com/settings/security/lesssecureapps>

☐ *Yahoo*. This requires a Yahoo Account Name and Password.

☐ *Other*. Select this for all other mail servers. This option requires you to type the host name or IP address for your SMTP mail server.

- c. Select the *Configure WebFOCUS BUE services to start automatically* check box to start the WebFOCUS BUE services. This sets the services Startup Type to Automatic. This will ensure that the services required to run the product are always running, and will not require that you start the services manually, even if you restart the machine.

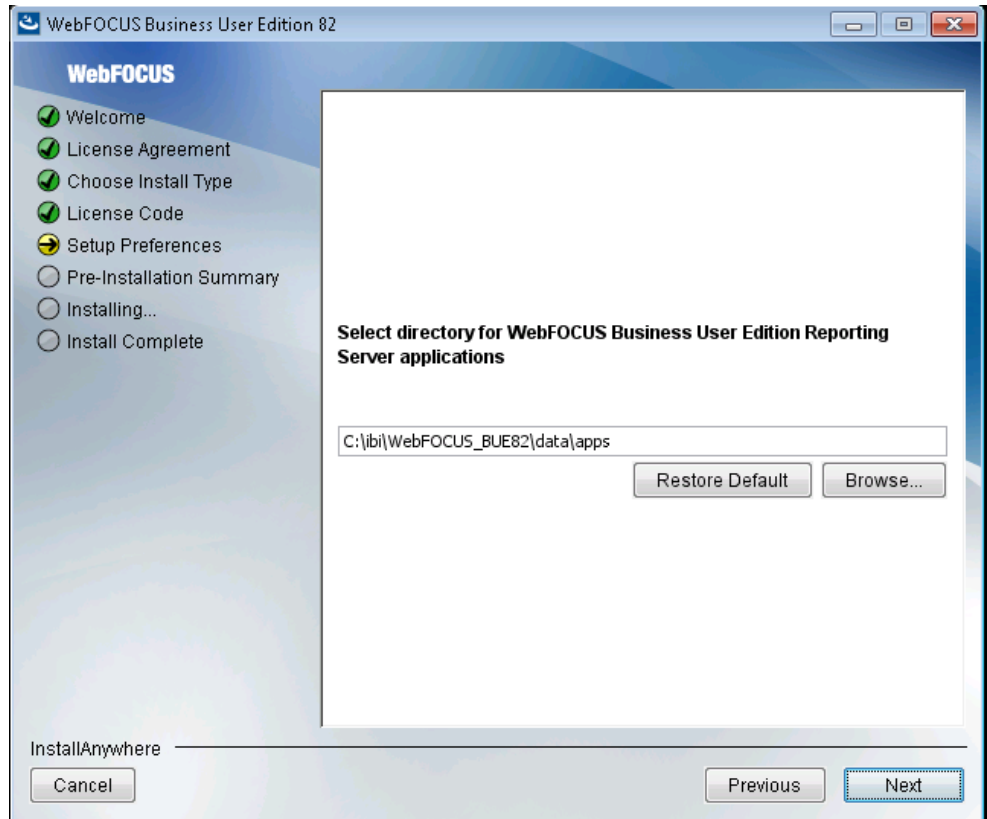
If you do not select this option, you can manually start and stop the WebFOCUS BUE services from the shortcut options found in the WebFOCUS Business User Edition 82 group, or through the Windows services program.

- d. Select the *Advanced Configuration* check box if you want to customize the location of your Reporting Server applications directory. The default location is `\WebFOCUS_BUE82\data\apps`.

8. Click *Next* to continue.

The Pre-Installation summary window opens.

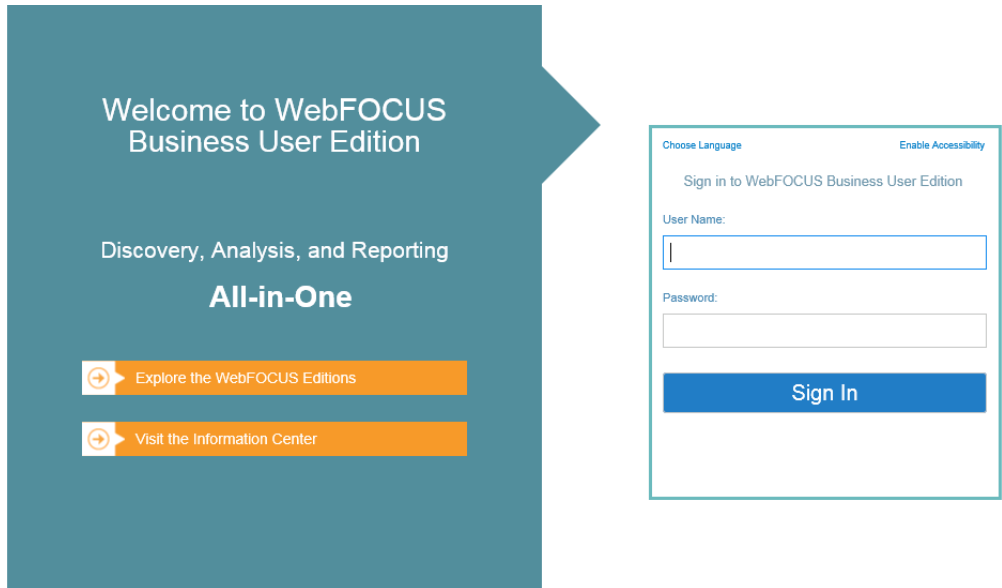
Note: If you selected the Advanced Configuration check box, the following window opens.



Server applications can be created in the default location or in a new location outside of the `drive:\ibi\WebFOCUS_BUE82\` folder. Select the location of your applications and then click *Next*.

9. Confirm that the summary information is correct, and then click *Install*.
10. When the installation completes, click *Done* to close the installation program.

The Sign In page opens, as shown in the following image.



11. Sign in using the manager user credentials that you configured during the installation program.

Running WebFOCUS Business User Edition Services Manually

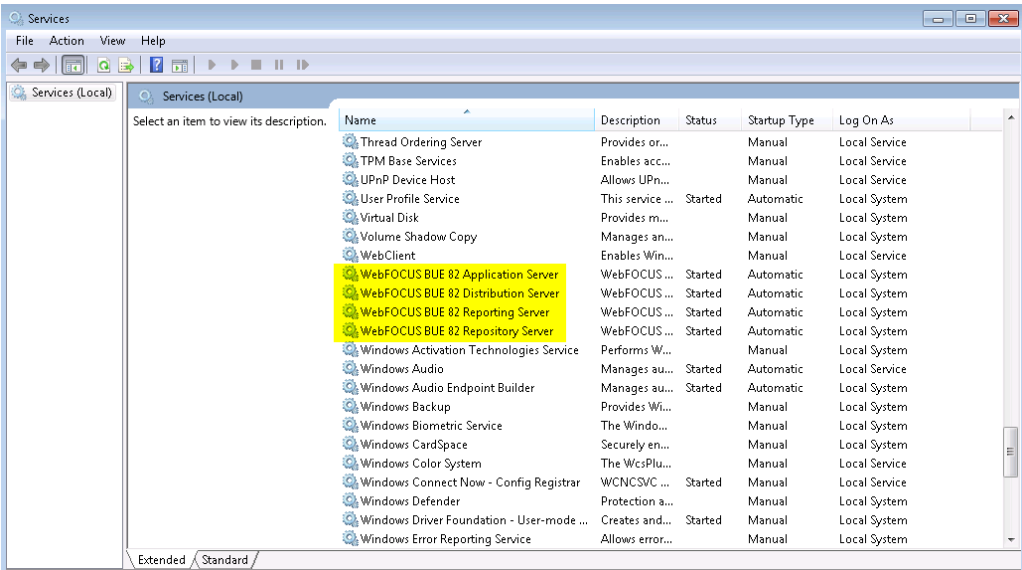
You should refer to this topic if you did not select the *Configure WebFOCUS BUE services to start automatically after a reboot* check box during the installation program.

The installation program installs the following services:

- ☐ An application server
- ☐ A Distribution Server
- ☐ A repository server
- ☐ A Reporting Server

Note: If you are updating an earlier release of WebFOCUS BUE, you must manually stop all WebFOCUS BUE services before beginning the installation program. For more information, see [Running WebFOCUS Business User Edition Services Manually](#) on page 31.

To view the status of these services, launch your Windows Services program from the Control Panel, and click the *Standard* tab. The following image highlights the WebFOCUS BUE services as they appear in the Windows Services program.



Display name	Service Name
WebFOCUS BUE 82 Application Server	WfBUE82AppSrv
WebFOCUS BUE 82 Distribution Server	WfBUE82DistSrv
WebFOCUS BUE 82 Reporting Server	WebFOCUS BUE 82 Reporting Server
WebFOCUS BUE 82 Repository Server	WfBUE82DbSrv

Procedure: How to Start or Stop WebFOCUS Business User Edition Services Manually

From the Start menu, navigate to the WebFOCUS Business User Edition shortcut, under the Information Builders program group.

- ❑ Click *Start WebFOCUS_BUE Services* to start the WebFOCUS BUE services.
- ❑ Click *Stop WebFOCUS_BUE Services* to stop the WEBFOCUS BUE services.

Uninstalling WebFOCUS Business User Edition

If you want to remove an existing WebFOCUS BUE installation and want to perform a full installation of a newer release, you can run the uninstall program.

Note: Before you uninstall WebFOCUS BUE, it is recommended that you save the cleanup utility to a different location on your machine in case the uninstall fails and you need to manually uninstall any components. For more information, see [Troubleshooting WebFOCUS Business User Edition](#) on page 35.

Procedure: How to Preserve Your Data

The following steps can be performed if you want to preserve your data and perform a new installation, if you need to move your data to another location, or if an upgrade installation fails and you need to perform a new full installation.

1. Back up the WebFOCUS BUE repository server and Reporting Server application folders to a different location on your machine.

By default, these are located in the following folders:

- ❑ `drive:\ibi\WebFOCUS_BUE82\derby\WFBUE82\`
- ❑ `drive:\ibi\WebFOCUS_BUE82\data\apps\`
- ❑ `drive:\ibi\WebFOCUS_BUE82\srw\wfs\hs\`

2. Uninstall WebFOCUS BUE. You can uninstall WebFOCUS BUE from the Start menu, or manually, as described in [How to Manually Uninstall WebFOCUS Business User Edition Components](#) on page 36.

Procedure: How to Restore and Move Backed Up Data

1. Install WebFOCUS BUE.
2. After the installation program is complete, do the following:
 - a. Manually stop the repository server service.
 - b. Replace the new repository server and Reporting Server application folders, using the backup directories that you made in step 1 of [How to Preserve Your Data](#) on page 33.
 - c. Restart the repository server service.

Procedure: How to Uninstall WebFOCUS Business User Edition

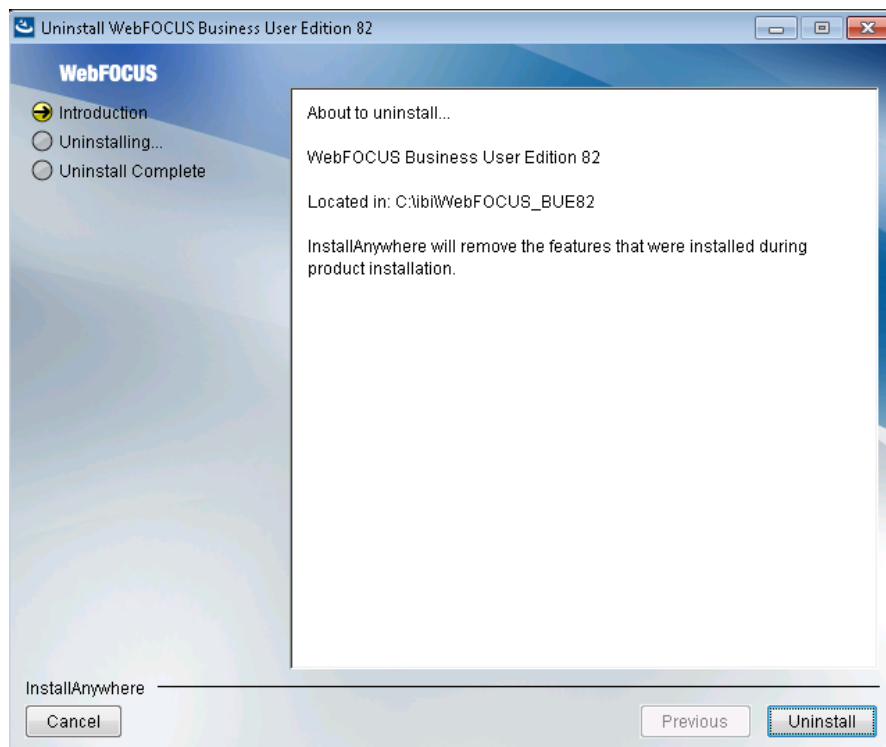
1. From the Start menu, navigate to the Uninstall shortcut, under the Information Builders Program group.

2. Click *Uninstall*.

The InstallAnywhere installation program opens.

Note: If the uninstall program is not visible from the Windows Start menu, uninstall the software using the Programs and Features option on the Control Panel. Alternatively, create a shortcut to C:\ProgramData\Microsoft\Windows\Start Menu\Programs. This will open Windows Explorer with a classic view of all installed programs. You can use this to navigate to WebFOCUS Business User Edition 82 in the Information Builders folder, and then run the uninstall program.

3. In the Uninstall WebFOCUS Business User Edition 82 window, click *Uninstall*, as shown in the following image.



The installation program begins and uninstalls the WebFOCUS BUE components.

Troubleshooting WebFOCUS Business User Edition

If you are unable to launch WebFOCUS BUE, ensure that all services are running. If they are not, do the following:

- ☐ Stop all services.
- ☐ Restart the services, and launch WebFOCUS BUE.

If this does not work, stop the services again. Before you restart the services, ensure that the Hyperstage processes, *ibengine.exe* and *postgres.exe*, are also stopped. Restart your machine if you are still unable to restart the services.

Port Assignment

By default, ports in the range of 26000 to 26040 are checked for availability. If the installation program detects that no ports in that range are available, it then increases the range by 10 and checks again for availability. For example, if ports 26000 to 26040 are unavailable, ports 26010 to 26050 are then checked.

Default Port Assignment.

- ☐ Ports 26000 to 26003 are used by the application server.
- ☐ Port 26010 is used by the repository server.
- ☐ Ports 26020 to 26023 and port 26040 are used by the Reporting Server .
- ☐ Port 26030 is used by the Distribution Server.

WebFOCUS Business User Edition Log Files

WebFOCUS BUE creates a log file in the following location for Windows 7:

drive:\Users\user_id\WebFOCUS_BUE82_Install_date_time.log

where:

user_id

Is your Windows user ID.

date_time

Is the date and time the log file was created. This log file provides information about the WebFOCUS BUE installation. If you contact Customer Support Services with an installation problem, have this file available.

Troubleshooting the Uninstall Process

If you choose to uninstall WebFOCUS BUE, and the uninstall process fails, you can follow one of the procedures in this section to clean up your machine before reinstalling the product.

Before performing one of the following procedures, ensure that you have uninstalled WebFOCUS BUE, as described in [Uninstalling WebFOCUS Business User Edition](#) on page 33.

Procedure: How to Manually Uninstall WebFOCUS Business User Edition Components Using the Cleanup Utility

If the WebFOCUS BUE uninstall process fails, you can run the wf_cleanup.bat file to clean a damaged installation on your machine.

1. Navigate to the following location on your machine:
`drive:\ibi\WebFOCUS_BUE82\WebFOCUS\utilities\install\`
2. Copy the `wf_cleanup.bat` file from the install directory to a different directory (for example, `drive:\ibi\`).
3. Right-click `wf_cleanup.bat`, and then click *Run as administrator*.
4. When prompted, type the location of the install directory to proceed.

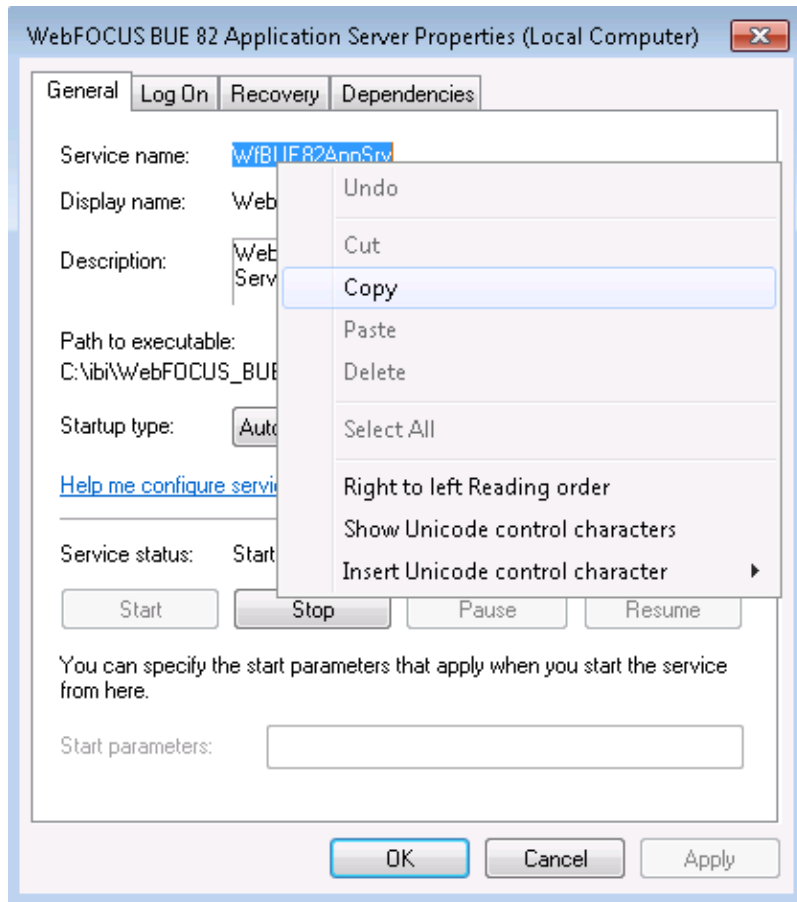
Note: By default, the utility will back up the Reporting Server apps, data, and the Derby database.

Procedure: How to Manually Uninstall WebFOCUS Business User Edition Components

If the WebFOCUS BUE uninstall process fails, you can do the following to manually cleanup the remaining WebFOCUS BUE components on your machine.

1. Stop any running WebFOCUS BUE services on the system, as detailed in [How to Start or Stop WebFOCUS Business User Edition Services Manually](#) on page 32.

2. Remove any remaining services from your machine.
 - a. Locate the existing WebFOCUS BUE services. To get the service name, right-click the service in the Windows Services program and click *Properties*. The Properties dialog box opens, where you can right-click the service name and click *Copy*, as shown in the following image.



- b. To remove a service, open the Command Prompt as an Administrator and issue the following command:

```
sc delete "service name"
```

where:

service name

Is the name of the service you are manually deleting. For example:

```
C:\sc delete "WfBUE82AppSrv"
```

Note: The service name must be enclosed in double quotation marks (").

Repeat this for the remaining services.

3. Open Task Manager and ensure that the process *ibengine.exe* is not running.
4. Remove files from disk, except for the Reporting Server application folders.
For example, if WebFOCUS BUE is installed on the C:\ drive, remove all folders under C:\ibi\WebFOCUS_BUE82\, except for the following folders:

☐ C:\ibi\WebFOCUS_BUE82\data\

☐ C:\ibi\WebFOCUS_BUE82\sr\wfs\hs\

☐ C:\ibi\WebFOCUS_BUE82\Derby\WFBUE82\

5. Click Start, point to *All Programs*, and expand the *Information Builders* folder.
6. Right-click the *WebFOCUS Business User Edition 82* folder and click *Delete*.
7. Launch the Windows Registry Editor, and remove the following registry keys.

☐ HKEY_LOCAL_MACHINE\SOFTWARE\WebFOCUS_BUE82

☐ HKEY_LOCAL_MACHINE\SOFTWARE\Information Builders\Derby\WfBUE82DbSrv

☐ HKEY_LOCAL_MACHINE\SOFTWARE\Information Builders\WebFOCUS BUE 82
Reporting Server

☐ HKEY_LOCAL_MACHINE\SOFTWARE\Wow6432Node\Information Builders
\ReportCaster\WfBUE82DistSrv

☐ HKEY_LOCAL_MACHINE\SOFTWARE\Wow6432Node\Apache Software Foundation
\Procrun 2.0\WfBUE82AppSrv

Installing the WebFOCUS Business User Edition on a Linux Environment

The following topic describes how to install WebFOCUS Business User Edition (BUE) on a Linux environment. WebFOCUS Business User Edition (BUE) is supported on:

- ☐ Intel Linux - Kernel-2.6.32-glibc-2.6 and higher x86_64 64-bit

Note: Java Version 8 Update 102 (8u102) and Tomcat 8.0.36 are included with the WebFOCUS BUE product installation.

- ☐ Intel Linux - Kernel-3.10.0-glibc-2.17 and higher x86_64 64-bit

Note: Java Version 8 Update 102 (8u102) and Tomcat 8.0.36 are included with the WebFOCUS BUE product installation.

- ❑ Linux on IBM Power Systems - Kernel-3.16.0-libc6-2.19 and higher ppc64le 64-bit (Little Endian)

Note: Java Version 8 Update 101 (8u101) and Tomcat 8.0.36 are included with the WebFOCUS BUE product installation.

WebFOCUS Business User Edition System Requirements on a Linux Environment

The following section describes the system requirements for installing WebFOCUS BUE on a Linux environment.

Note: Only one installation per platform is supported.

Installation Prerequisites

The following are installation prerequisites for installing WebFOCUS BUE on Linux:

- ❑ WebFOCUS BUE cannot be installed as user root, due to requirements of the PostgreSQL database used by Hyperstage. If user root is detected, the installation will exit and the following message displays:

```
The installation cannot proceed under the root user account and will
terminate. Restart the installation as a non-root user.
```

- ❑ When installing WebFOCUS BUE on Linux machines, if the unzip application is not found, the following message displays:

```
Invalid unzip command found.
```

This is generated by the installation program and does not prevent the installation from proceeding and completing successfully.

- ❑ WebFOCUS BUE requires a minimum of 5GB of available disk space to successfully install on the target system.
- ❑ An additional 8GB of required disk space is used during the installation of the product. This temporary use of disk space will be removed at the end of the installation.

If the temp directory on the host does not meet the minimum size requirements, the InstallAnywhere environment variable IATEMPDIR can be set to use an alternate directory.

You may specify an alternate /tmp location by setting the following environment variable:

```
IATEMPDIR=/large_tmp
export IATEMPDIR
```

System Resource Limits

The operating system provides ways of limiting the amount of resources that can be used. These limits can affect the installation process.

Use the `ulimit` command to set process memory-related resource limits for your session.

The WebFOCUS BUE installation program requires `ulimit` to be set to 8192.

Limits can be *hard* or *soft*. Hard limits are set by the root user. Only the root user can increase hard limits, although other users can decrease them. Soft limits can be set and changed by other users, but they cannot exceed the hard limits. To view the current limits, enter the following command:

```
ulimit -a
```

Use the `/etc/security/limits.conf` file to store `ulimit` settings. Changes to this file should be made by a system administrator.

Installing WebFOCUS Business User Edition Using Console Mode

The following section describes how to install WebFOCUS BUE using Console Mode.

Procedure: How to Installing WebFOCUS Business User Edition Using Console Mode

1. Download the self extracting installation program, `installWebFOCUSBUE8201.bin`.
2. Apply the permissions necessary for the owner of the file to execute the installation by using the following command:

```
chmod u+x installWebFOCUSBUE8201.bin
```

3. Run the installation program using the following command:

```
./installWebFOCUSBUE8201.bin -i console
```

The Welcome to WebFOCUS Business User Edition 82 prompt displays.

4. Press Enter to continue.

The License Agreement prompt displays.

5. Read the License Agreement. Press Enter after each screen until you receive the License Agreement Acceptance prompt.
6. Type Y to accept the terms of the agreement and press Enter to continue.

The Choose Install Type prompt displays.

7. Type 2 to perform a full installation of WebFOCUS BUE.
8. Press Enter.

The License Code prompt displays.

9. Type the license code (17 digits) for the WebFOCUS BUE Installation.

This code was sent to you as part of the email message that contained the download link for the software.

You are prompted to enter the site code.

10. Type the site code (4 digits) for the WebFOCUS BUE Installation.

This code was sent to you as part of the email message that contained the download link for the software.

11. Press Enter.

The Setup Preferences prompt displays.

12. Accept the default destination folder for the installation. If you want to install WebFOCUS BUE in a different folder, type the path to the location where you want to install the software.

Note: Spaces are not allowed in the destination path.

You are prompted to enter the WebFOCUS BUE Administration credentials.

13. Type a password to use with the Administration ID, *manager*.

Note:

☐ The *manager* Administration ID cannot be changed.

☐ Passwords must be between 4 and 20 characters and cannot include double quotation marks (") or a dollar sign (\$). Leading blanks and trailing blanks will be removed.

The Advanced Configuration prompt displays. Type *Y* and press Enter if you want to change the default directory location of the WebFOCUS BUE Reporting Server application. Type *N* and press Enter if you want to use the default directory location of the WebFOCUS BUE Reporting Server.

If you chose to use the default WebFOCUS BUE Reporting Server directory, the Pre-Installation Summary prompt displays.

14. Verify that the information in the Pre-Installation Summary window is correct, and press Enter to continue.

The installation begins.

During installation, the activity window indicates which installation program tasks are being performed.

When installation is complete, a summary is provided displaying the servers that were installed and the ports that have been assigned.

Configuring Java Memory Allocation for the Application Server

For UNIX installations, the default Tomcat Java memory configuration is set to use an initial Java heap size of 256MB and a maximum Java heap size of 512MB.

These settings are located in the `/ibi/WebFOCUS_BUE82/tomcat/bin/catalina.sh` file. For example:

```
JAVA_OPTS="-XX:MaxPermSize=256m -Xms256m -Xmx512m -  
Djava.awt.headless=true"
```

For better product performance, it is recommended to increase these values. The initial Java heap size should be at least 1024MB (1GB). The size is normally set in Megabytes. For example:

```
-Xms1024m  
-Xmx2048m
```

These values need to be set based on the physical memory of the machine.

It is recommended to set the minimum/initial memory value to one eighth (1/8) of the physical memory. The maximum value needs to be one quarter (1/4) of the physical memory.

Launching WebFOCUS Business User Edition

To launch WebFOCUS BUE, type the following:

```
http://host:port/ibi\_apps/
```

Note: Enter the application server port number when using this command to start WebFOCUS BUE. The application server port number is the lowest port number displayed in the summary at the end of the WebFOCUS BUE installation.

The default port for the WebFOCUS BUE application server is 26000. It is listed during the end of the installation and also available in the installation log file.

Installation log files are under the user home directory. An example of an installation log file is shown in the example below:

```
/\$HOME/WebFOCUS\_BUE82\_Install\_date\_time.log
```

where:

date_time

Is the date and time the log file was created. This log file provides information about the WebFOCUS BUE installation. If you contact Customer Support Services with an installation problem, have this file available.

Running WebFOCUS Business User Edition Servers Manually

The following commands are used to manually start and stop the required WebFOCUS BUE servers.

Note: The double periods (..) before each of the entries below indicates that the full path of the items below should be entered.

Repository Server

- ❑ To start the WebFOCUS BUE Repository Server, run:

```
nohup ../ibi/WebFOCUS_BUE82/derby/bin/start.sh &
```

- ❑ To stop the WebFOCUS BUE Repository Server, run:

```
../ibi/WebFOCUS_BUE82/derby/bin/stop.sh
```

Reporting Server

- ❑ To start the WebFOCUS BUE Reporting Server, run:

```
../ibi/WebFOCUS_BUE82/srv/wfs/bin/edastart -start
```

- ❑ To stop the WebFOCUS BUE Reporting Server, run:

```
../ibi/WebFOCUS_BUE82/srv/wfs/bin/edastart -stop
```

Application Server

- ❑ To start the WebFOCUS BUE Application Server, run:

```
../ibi/WebFOCUS_BUE82/tomcat/bin/startup.sh
```

- ❑ To stop the WebFOCUS BUE Application Server, run:

```
../ibi/WebFOCUS_BUE82/tomcat/bin/shutdown.sh
```

Distribution Server

- ❑ To start the WebFOCUS BUE Distribution Server, navigate to the ../ibi/WebFOCUS_BUE82/WebFOCUS/ReportCaster/bin directory and run:

```
nohup ./schbkr &
```

- ❑ To stop the WebFOCUS BUE Distribution Server, navigate to the `../ibi/WebFOCUS_BUE82/WebFOCUS/ReportCaster/bin` directory and run:

```
./stopit
```

Updating WebFOCUS Business User Edition

You can update an earlier release of WebFOCUS BUE using the following steps.

Procedure: How to Update WebFOCUS Business User Edition

1. Manually stop all WebFOCUS BUE servers, including the Application Server, Repository Server, Distribution Server, Reporting Server, and Hyperstage Server.
2. Ensure that all ports that are used by WebFOCUS BUE processes are stopped. You can run a command, such as `netstat`, to check the ports.

Note: If the ports are busy, run a command, such as `lsof`, to identify what processes are still running, and stop those processes. Before you proceed, you must rerun the `netstat` command.

3. Download the self extracting installation program, `installWebFOCUSBUE8201.bin`.
4. Apply the permissions necessary for the owner of the file to execute the installation by using the following command:

```
chmod u+x installWebFOCUSBUE8201.bin
```

5. Press Enter to continue.

The License Agreement prompt displays.

6. Run the installation program using the following command:

```
./installWebFOCUSBUE8201.bin -i console
```

The Welcome to WebFOCUS Business User Edition prompt displays.

7. Read the License Agreement. Press Enter after each screen until you receive the License Agreement Acceptance prompt.
8. Type `Y` to accept the terms of the agreement and press Enter to continue.
The Choose Install Type prompt displays.
9. Type `1` to update an existing WebFOCUS BUE installation to a new Service Pack level.
10. Press Enter.

11. Verify that the information in the Pre-Installation Summary window is correct, and press Enter to continue.

The installation begins.

During installation, the activity window indicates which installation program tasks are being performed.

12. After the install program ends, change the file permissions for the Hyperstage Server data folder by issuing the following command:

```
chmod -R 0700 ../ibi/WebFOCUS_BUE82/srv/wfs/hs/pg_data
```

13. Before launching WebFOCUS BUE, start the WebFOCUS BUE Servers as outlined in [Running WebFOCUS Business User Edition Servers Manually](#) on page 43.

Uninstalling WebFOCUS Business User Edition

To uninstall WebFOCUS BUE, run the uninstall program using the following command:

```
../ibi/WebFOCUS_BUE82/Uninstall/Uninstall
```

Configuring for Email Distribution

An email server is required for email distribution and notifications. The following procedures provide the steps for configuring the supported mail servers.

Procedure: How to Configure the Mail Server

1. From the BUE Portal Menu bar, click *Administration*, then click *Administration Console*. Click the *ReportCaster* tab.

or

From the BUE Portal Menu bar, click *Tools*, then click *ReportCaster Status*.

2. Click the *Configuration* option on the ribbon.
3. Select the *Email Distribution* folder.
4. Type the name of your mail server in the Mail Server text box.

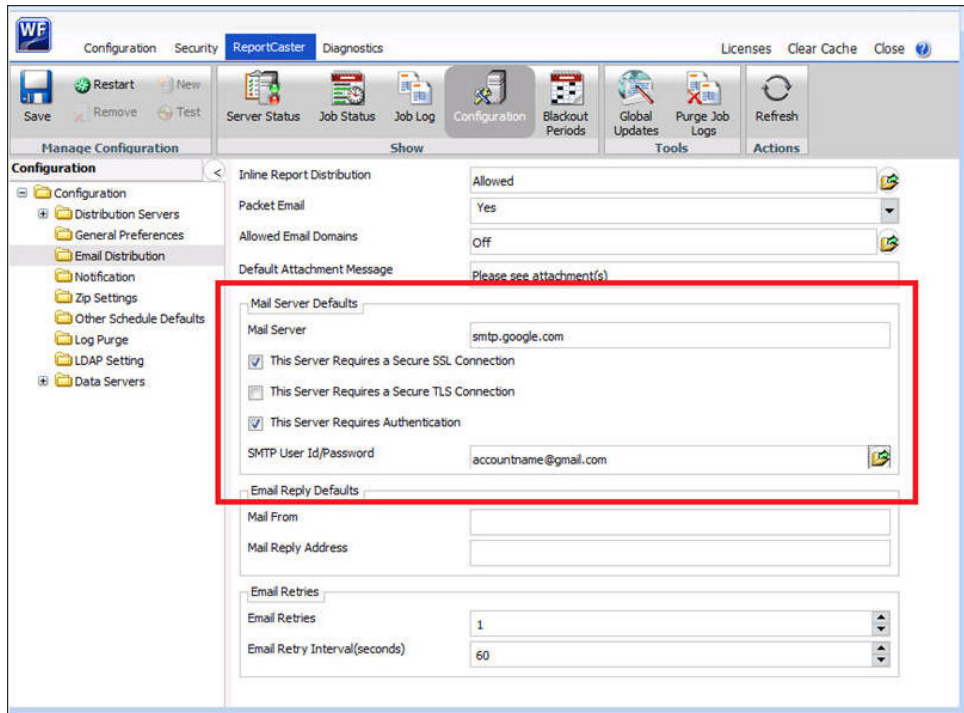
In addition, if your mail server uses a secure connection or requires authentication, select the appropriate options and type the required credentials. You need to save your configuration changes and restart ReportCaster to make the changes effective.

If you use Google as your email server, see [How to Configure Google as the Email Server](#) on page 46.

If you use Yahoo as your email server, see [How to Configure Yahoo as the Email Server](#) on page 47.

Procedure: How to Configure Google as the Email Server

1. Type the name of the Google SMTP Server in the Mail Server text box, as shown in the following image.

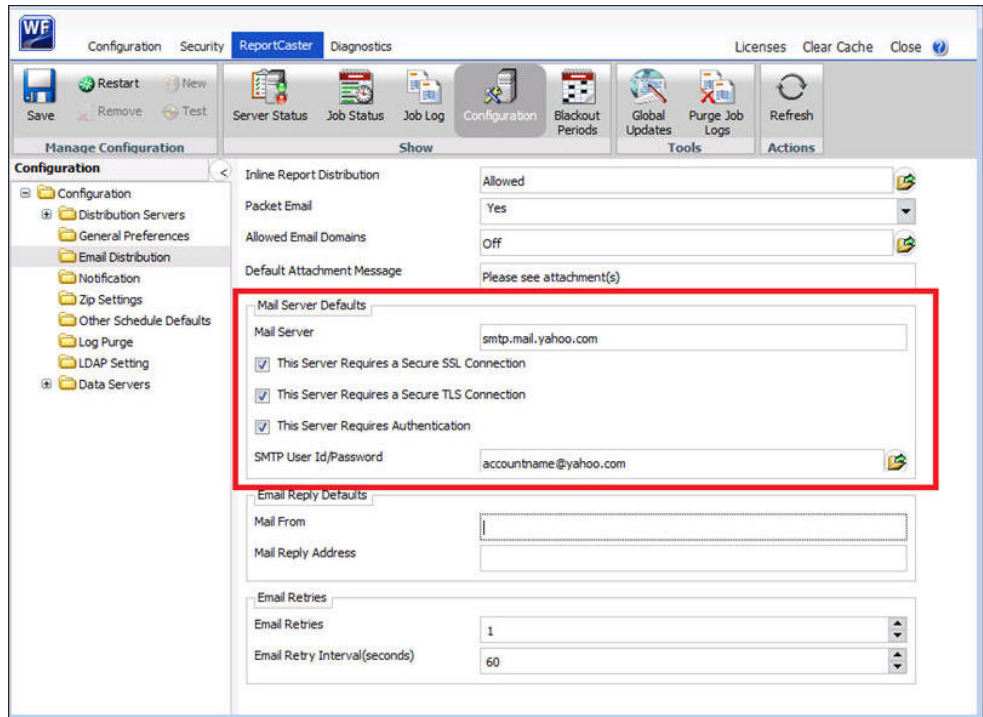


2. Select the *This Server Requires a Secure SSL Connection* check box.
3. Do not select the *This Server Requires a Secure TLS Connection* check box.
4. Select the *This Server Requires Authentication* check box.
5. Click the button to open the SMTP User ID/Password dialog box.
6. Type your Google account name.
7. Type the password for this account.
8. Confirm the password for this account.
9. Click OK to close the dialog box.
10. Click Save in the Manage Configuration group on the ribbon.

- Click *Restart* in the Manage Configuration group on the ribbon.

Procedure: How to Configure Yahoo as the Email Server

- Type the name of the Yahoo SMTP Server in the Mail Server text box, as shown in the following image.



- Select the *This Server Requires a Secure SSL Connection* check box.
- Select the *This Server Requires a Secure TLS Connection* check box.
- Select the *This Server Requires Authentication* check box.
- Click the button to open the SMTP User ID/Password dialog box.
- Type your Yahoo account name.
- Type the password for this account.
- Confirm the password for this account.
- Click OK to close the dialog box.
- Click *Save* in the Manage Configuration group on the ribbon.
- Click *Restart* in the Manage Configuration group on the ribbon.

Navigating the Environment

This topic introduces the WebFOCUS Business User Edition environment. It describes how to navigate the WebFOCUS Business User Edition (BUE) portal, the Administration Console, and InfoAssist+.

It is also important to understand that Manager users, Developers, Advanced users, and Basic users have different access to environment options. That access is determined by the group of users to which you are assigned.

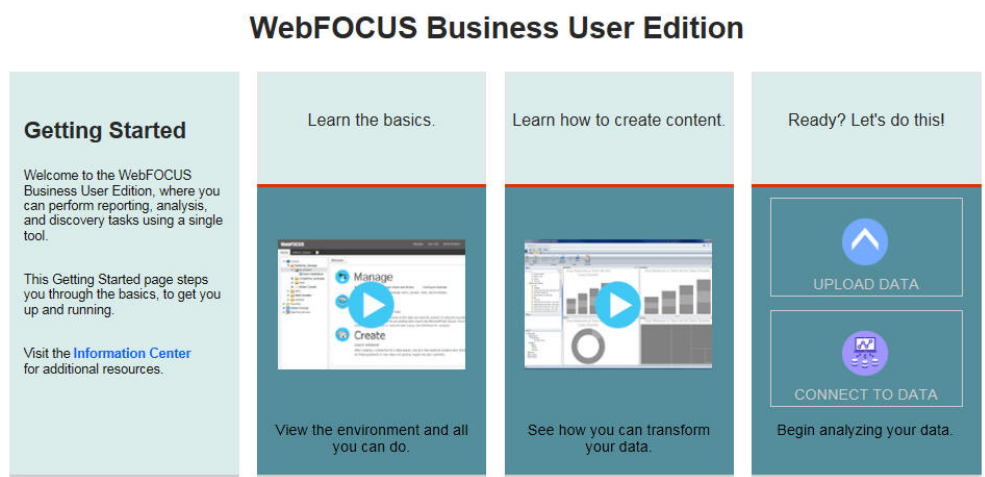
Navigating the WebFOCUS Business User Edition Portal

When you access the WebFOCUS Business User Edition (BUE), using the Sign-in page or browser URL, you are presented with the BUE Portal. From the portal, depending on your user role, you can create and edit content, view content that other users published or shared, create and edit personal pages, manage users, configure the system, upload files to the repository, connect to data and edit data, and access interactive help and instructional materials.

The main components of the BUE Portal include a Getting Started Page, Home Page, Resources tree, and Menu bar.

Getting Started Page

The Getting Started page, shown in the following image, is the first page that you see when you sign in to WebFOCUS BUE with the login and password that you created during the installation.

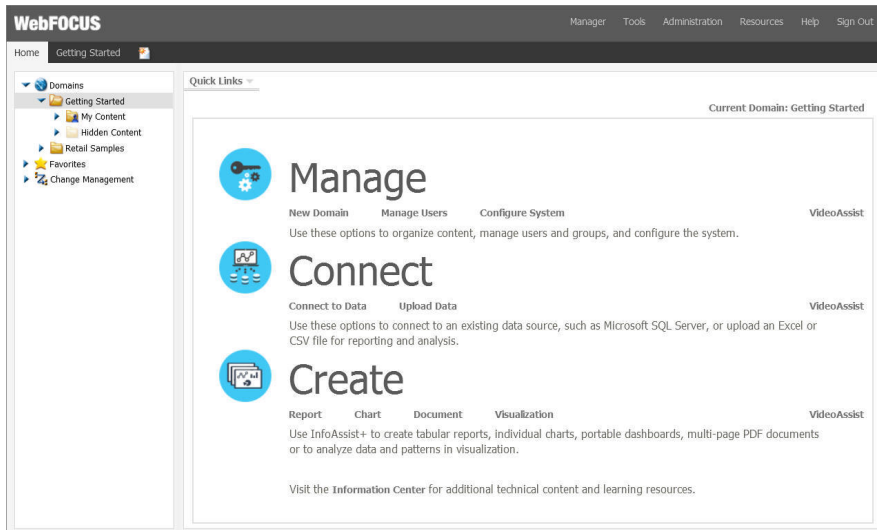


The Getting Started page provides access to resources that get you started with the software. You can access the Information Center, a series of videos, and the Upload wizard and Connect to Data wizard features.

Only Manager users can access this page.

Home Page

The Home page, which is shown in the following image, is the first page that you see when you sign in as a Basic User, Advanced User, Developer, or Group Administrator.



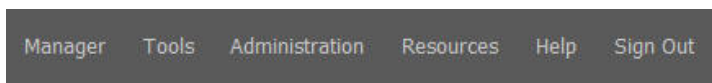
The Home page has three sections that you can interact with:

- ☐ Menu bar
- ☐ Resources tree
- ☐ Quick Links tab

The content in these sections varies, depending on your user type.

Menu Bar

The Menu bar is located in the upper-right section of the screen, inside the top banner. The following image shows the Menu bar that appears when you are signed in as a Manager.

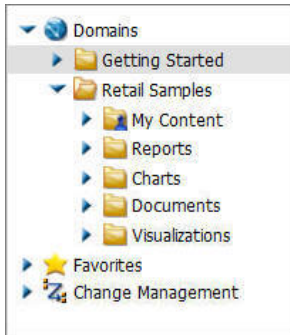


The Menu bar gives you access to the following options:

- ☐ Change your password (click your username).
- ☐ Tools:
 - ☐ View the status of deferred reports, and manage them.
 - ☐ Stop your outstanding interactive requests that are running.
 - ☐ Access Session Viewer, where you can visualize the exploratory analysis of the current web session.
 - ☐ Access the ReportCaster Status, where you view the status of your scheduled jobs and log reports.
- ☐ Administration:
 - ☐ Access the Security Center, where you define users and groups and grant access permissions.
 - ☐ Access the Administration Console, where you configure, administer, and monitor the components of the WebFOCUS application.
 - ☐ Access the Reporting Server Console, where you can view and manage the server environment.
 - ☐ View and administer the private resources for groups and users.
 - ☐ Switch between Normal mode (seeing your own content) and Manager mode (managing other user content).
- ☐ Open the Resources tree on the right side of the window, so you can manage content on Personal Pages.
- ☐ Access the online Help, Information Center, a library of Help topics, self-service videos, Community, Information Builders Home page, as well as information about the WebFOCUS Business User Edition application, version, and available licenses.
- ☐ Sign out

Resources Tree

The Resources tree, located on the left side of the Home page, contains your available resources, such as domains, folders, and content. It also contains functionality that allows you to save content as a favorite, or perform a Change Management import or export. The following image shows the Resources tree that appears when you are signed in to WebFOCUS Business User Edition.



The resources are secured using the security model, that is folder-based, similar to that of desktop operating systems. The resources differ for every user. For example, the Basic User sees only public content.

The Resources tree contains the following nodes:

- ☐ **Domains.** Lists the domains and folders that you have created. Yellow folders are published folders. Gray folders are unpublished.
- ☐ **Favorites.** Lists the items that you have designated as Favorites.
- ☐ **Change Management.** Lists the Import and Export facilities.
 - ☐ **Import.** Enables users to move a change package into an acceptance test.
 - ☐ **Export.** Enables a developer to move changes made to application files, to the user acceptance test environment.

Node Context Menu Options

Each node in the Resources tree has a shortcut menu. The available options include:

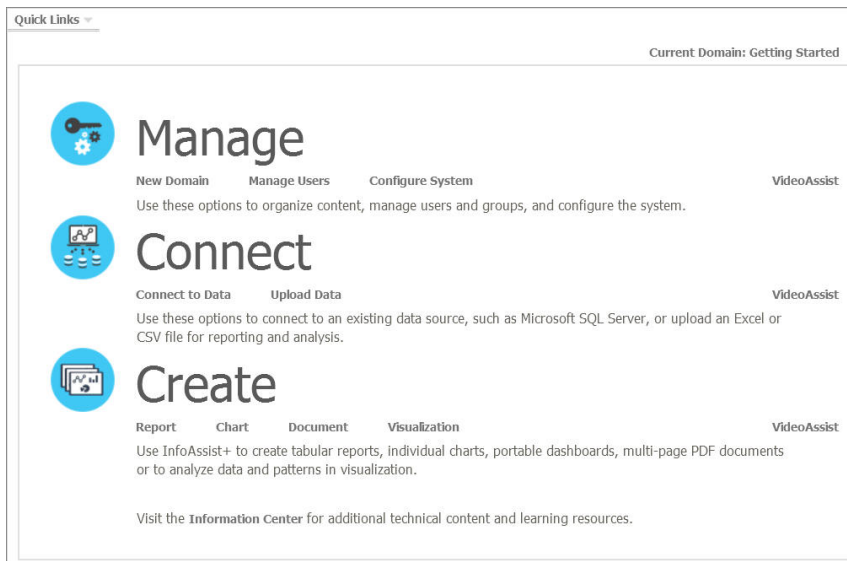
- ☐ **View.** This option is only available for the Domains node, and includes:
 - ☐ **Display By Title.** Displays folders by title. The default display.

- ☐ **Display By Name.** Displays folders by name.
- ☐ **New.** This option is only available to managers, and includes:
 - ☐ **Domain.** Creates a new domain.
- ☐ **Refresh.** Refreshes the Resources tree. The option is available for all nodes.
- ☐ **Paste.** Pastes the item under the applicable node.
- ☐ **Clear Favorites.** Clears the Favorites list. This option is only available for the Favorites node.

Quick Links

The Quick Links on the Home page, include links for uploading or connecting to data, creating content, managing domains and users, and configuring your system. It also shows the name of the domain that is currently highlighted in the Resources tree. Whenever you navigate to a different domain, the Quick Links refresh to reflect the new active directory.

The available Quick Links also change depending on the user. Each user only sees the options that they have privileges to access. The following image shows an example of the Quick Links that are available for a Manager.



Understanding Domain Content

A domain node is the highest level of folder organization in the Resources tree. All of the domains that a user is authorized to access are displayed in the tree. Domains contain all of the components and content that you can create.

Selecting a Domain

A list of the domain folders, populated from the WebFOCUS Repository, appear in the Resources tree. The items you can access, and the options available to you, are permitted by the Manager of your environment. When the contents of a folder change, such as when you create and save a new report or a report is published, the Resources tree automatically refreshes. You can also use the Refresh button in the toolbar to update the Domain contents.

Identifying Types of Content in the Resources Tree

A domain can contain a wide range of items, including reports, charts, visualizations, documents, dashboards, Internet hyperlinks, schedules, and distribution lists.











These items are located in folders. Both the items and the folder may be private, published, or shared. If you are permitted to create your own reports, you also have the My Content folder. My Content may contain the Custom Reports, Shared Reports, and folders for each My Report report you created.







- ☐ **Published content.** Domain content that is published is considered authoritative and has usually undergone quality assurance and testing before being published for the user community.
- ☐ **Private content.** This content can be in the following forms:
 - ☐ **My Content.** Personal content created by a user. This content remains private to a user unless the user chooses to share it with others or unless an administrative user has the ability to manage private content for that user.
 - ☐ **Other private content.** Domain content which is not published. It is accessible only to its owner, which may be a specific user or a group.

When you create a folder, it is private content. Private content appears as a gray folder in the Resources tree. If you share a folder, the folder includes a link icon and is available either to all users, specific groups, or users permitted to access the folder location. If you publish the folder, it appears as a yellow folder in the Resources tree and is available to all users.

Identifying Content in the Resources Tree

The icons located next to each item represent the item type. In the following table, the first column lists the icons, and the second column describes what they represent.

Icon	Identifies
	Folder. Allows users to organize their private or published application content.
	My Content folder. Provides the user with a location to create private reports, output, and schedules. This is created when the parent folder has the Auto create My Reports folder property selected and the user is permitted to create private content in a My Content folder. The private content within this folder cannot be published.
	Report procedure. Contains the core language commands to tell the Reporting Server how to create the report.
	Visualization. Represents your data graphically, and is comprised of one or more visuals. Visuals can be charts, maps, or grids.
	Chart procedure. Contains the core language commands to tell the Reporting Server how to create the chart.
	Document or dashboard. Customized document or dashboard that contains more than one item.
	Schedule. Specifies when to run a report procedure, how to distribute the report (Email, FTP, Managed Reporting, Printer), and the destination to which the report will be distributed. This is available when licensed for the feature that creates schedules.
	Distribution List. Used with a ReportCaster schedule (Email, FTP, or Printer) to specify the email addresses, directory locations, or printers to distribute.
	Microsoft® Office Excel Worksheet
	Adobe® Acrobat PDF file

Icon	Identifies
	GIF image file
	HTML file
	JPG image file
	PNG image file
	SVG image file
	Webpage or a URL to run a report

The following topics describe the shortcut menu options for each type of item.

Options for Folders and Items

When you right-click an item, the options displayed in the shortcut menu are determined by the selected item type, and the tools and features you are permitted to access.

Reference: Options for Folders

The following options are available when you right-click a folder:

New

Provides access to the following options:

Report

Opens InfoAssist+, where you can create a report.

Chart

Enables you to create a chart in InfoAssist+.

Document

Enables you to create a document in InfoAssist+.

Visualization

Opens InfoAssist+, where you can create visualizations.

Sample Content

Opens the Open dialog box where you can select a Master File for generating sample content.

Shortcut

Creates a shortcut to a Repository Resource or a Master File.

URL

Launches the Create URL dialog box, where you can create a link to a webpage.

Distribution List

Launches the Distribution List tool, where you can create distribution lists.

Folder

Creates a new folder.

Duplicate

Makes a copy of the item with an underscore and number appended to the title, for example, duplicating Sales_Report creates Sales_Report_1 in the same location. The number is incremented each time a duplicate is created.

Cut, Copy, and Paste

Cuts, copies, or pastes the folder and its contents.

Delete

Deletes the folder and its contents.

Change Title

Launches a dialog box, where you can change the title of a folder. The Title is the value that displays in the Resources tree and does not change the folder *Name* value.

Refresh

Refreshes the content of the folder.

Upload

Enables you to upload a data file, document, or image.

Metadata

Enables you to create or edit a synonym.

Publish/Unpublish

Allows or suppresses public access to the content.

Show/Hide

Shows or hides the item.

Security

Provides access to the following option:

Owner

Allows you to set the resource owner, either Published (no specific owner), or a specific group or user.

Properties

Shows the properties of the folder.

Note: The options available for a folder are also available for other types of items.

Reference: Options for Content

Like folders, content, such as report, chart, dashboard, and visualization, have options that include the Duplicate, Cut, Copy, Delete, Change Title, Publish/Unpublish, Show/Hide, Security, and Properties options.

Note: Documents and dashboards have the same options as report and chart procedures with the exception of Publish/Unpublish.

Report and chart procedures also include the following right-click options:

Run

Runs the report procedure or schedule.

Run Deferred

Submits a report procedure or a schedule for deferred execution.

Run With SQL Trace

Runs a report and includes an SQL trace of the report results. This option is not available for visualizations.

Schedule

Schedules a report procedure. Scheduling options include Email, Printer, and Repository. This option is not available for visualizations.

Edit

Edits an item with the tool that was used to create it.

Add to Favorites

Adds the item to the Favorites folder.

Note: The options available for content are also available for other item types.

***Reference:* Options for Schedules**

Like folders, Schedule options include the Duplicate, Cut, Copy, Delete, Change Title, Security, and Properties options.

Like content, Schedule options include the Edit and Run options.

Schedules also include the following right-click option:

View Log

Launches the Schedule Log Options dialog box from which you can select whether to view the most recent log report for the schedule or specify a time range for which to obtain log reports.

***Reference:* Options for Distribution Lists**

Like folders, Distribution List options include the Duplicate, Cut, Copy, Delete, Change Title, Publish/Unpublish, Show/Hide, Security, and Properties options.

Like content, Distribution List options include the Edit option. The Edit option invokes the Distribution List tools.

***Reference:* Options for URLs**

Like folders, URL options include the Duplicate, Cut, Copy, Delete, Change Title, Publish/Unpublish, Show/Hide, Show/Hide, Share/Unshare, Share with, Security, and Properties options.

Like content, URL options include the Edit, and Add to Favorites options.

URLs also include the following right-click option:

View

Enables you to launch the URL link.

***Reference:* Options for Excel Workbooks, PDF, and Image Files**

Like folders, these options include the Duplicate, Cut, Copy, Delete, Change Title, Publish/Unpublish, Show/Hide, Security, and Properties options.

These files also include the following right-click option:

View

Enables you to download a copy of the file to your browser, which will launch the File Download dialog box to allow you to open or save the file. If you choose open, the file will open in its corresponding application.

***Reference:* Options for Favorites**

These files include the following right-click option:

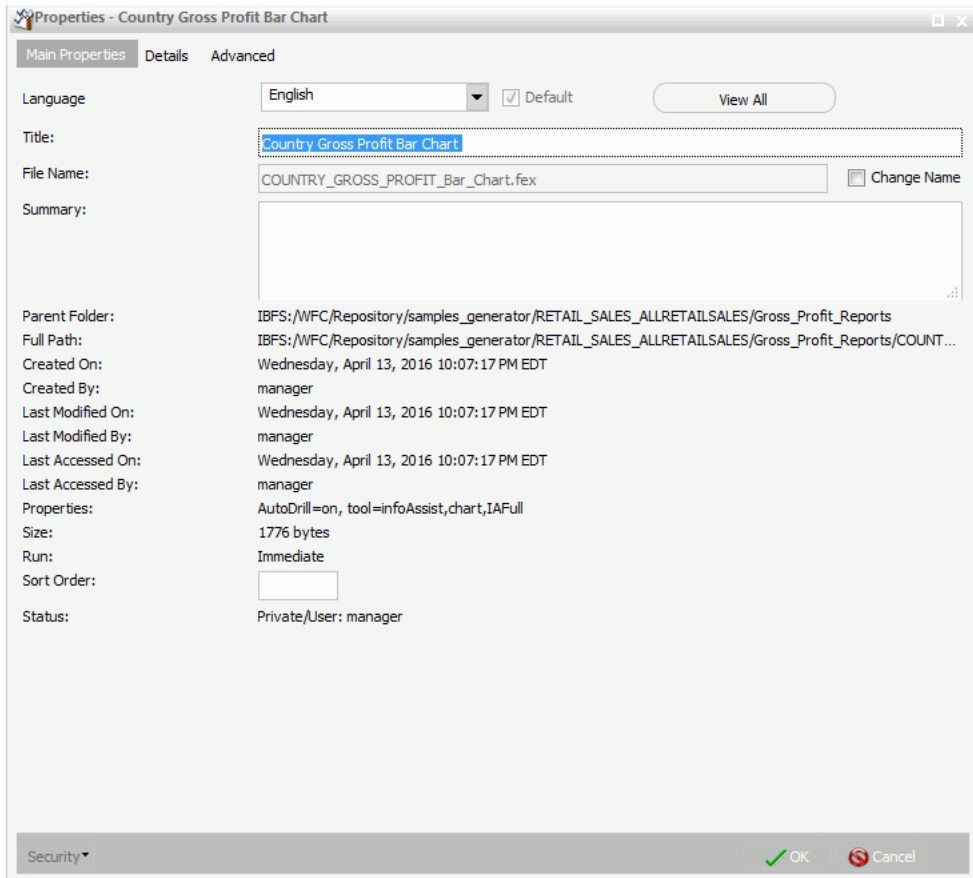
Remove Favorite

Removes the Favorite from the list.

Viewing Folder and Item Properties

Authorized users can select the Properties option to review information about a folder or item, including when it was created, last modified, title, name, and settings that control the Reporting Server and applications it will access. There are also settings that control the functionality available to users when accessing the folder or running or scheduling an item.

The following image displays the Properties dialog box for an item.



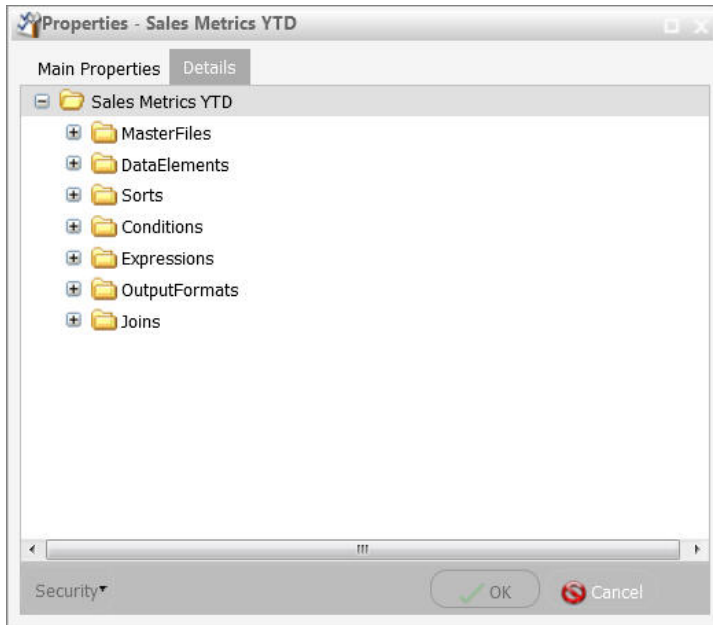
The Main Properties tab includes the following settings:

- ☐ **Language.** Provides a list of languages that you can select.
- ☐ **View All.** Opens the Language Properties dialog box, where you can select a language.
- ☐ **Title.** Displays a value in the Resources tree users can see which identifies the content within the folder.
- ☐ **Folder/File Name.** Displays a unique reference to the folder or item within the folder. Select the *Change Name* check box to change the name of the folder or item.
- ☐ **Summary.** Provides a detailed explanation with additional information about the folder or item.

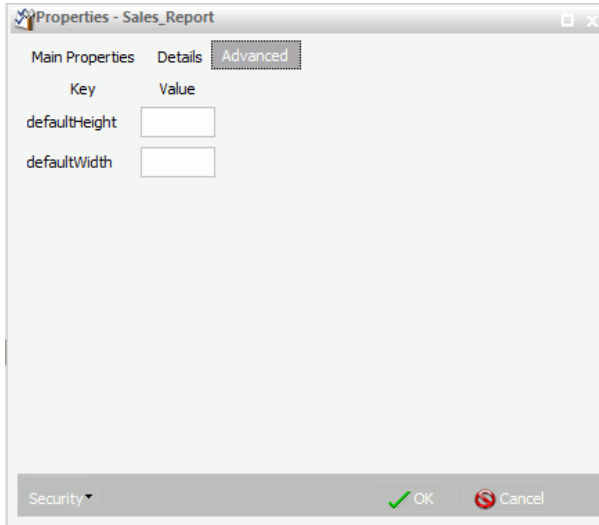
- ☐ **Parent Folder.** Shows the previous folder in the folder path of the WebFOCUS repository.
- ☐ **Full Path.** Displays full path of the folder or item in the WebFOCUS repository.
- ☐ **Created On.** Identifies the date the folder or item was created.
- ☐ **Created By.** Identifies the user who created the folder or item.
- ☐ **Last Modified On.** Shows the date the folder or item was last modified.
- ☐ **Last Accessed On.** Shows the date the folder or item was last opened.
- ☐ **Last Accessed By.** Identifies the user who last opened the folder or item.
- ☐ **Properties.** Shows a string of name value pairs that identify the properties of the folder or item.
- ☐ **Size.** Displays N/A bytes for folders. For items, displays the size in bytes.
- ☐ **Run.** Displays N/A, as this property is not applicable to folders. For items, displays Immediate.
- ☐ **Sort Order.** Specifies the order to list the folder in the Resources tree or list items within a folder.
- ☐ **Status.** Indicates whether the folder or item is Published or Private. Private will also display the owner information.

Authorized users can view or edit the access rules, effective policy, or sharing permissions by clicking *Security*.

The Details tab, shown in the following image, displays the components of the item in a tree structure to allow a user to review the request information prior to running or opening the item in a tool. The tree structure displays the request components, such as folders, Master Files, Data Elements, Sorts, Conditions, Expressions, Output Formats, and Joins, which can be opened to show the fields referenced.



The Advanced tab, shown in the following image, allows you to set default height and width values for the item, when it is displayed inside a portal at run time.



Navigating the Administration Console

The Administration Console contains four tabs and a menu bar that help you navigate to its settings and other features.

These tabs organize administration activities into the following categories:

- ☐ **Configuration.** Configures Reporting Server connections, Application Settings, Custom Settings, NLS Settings, the Dynamic Language Switch, Redirection Settings, and InfoAssist + Properties.
- ☐ **Security.** Configures general security settings for Internal and External Authentication.
- ☐ **ReportCaster.** Opens the ReportCaster Console where you can configure ReportCaster, restart the Distribution Server, configure environment parameters, and turn traces on and off. The Administration Console authenticates to ReportCaster with the value of the user ID, IBIMR_RC_SVCUSER. If this authentication fails, users are prompted for their credentials.
- ☐ **Diagnostics.** Displays component installation and configuration details, turns session logging on or off, and enables Managers to view or delete log files.

You can view traces with the Session Viewer, which is accessible from the Tools option on the Menu bar in the Portal.

Options on the menu bar connect you to basic tasks, such as reviewing WebFOCUS BUE and third party licensing information, clearing the cache, closing the Administration Console, or opening the online Help.

To update or review an Administration Console setting, click a tab, and then click the folder or page icon from the main menu on your selected tab. The main window refreshes and the individual settings assigned to your selected page become available.

Navigating the Configuration Tab

The Configuration tab contains settings and features that describe Reporting Server connections and other application settings, which are listed and described in the following table.

Folder [Page]	Available Functionality
Reporting Servers	<p>The Reporting Servers subfolders contain the tools that manage all connections from the client to remote servers. Using the following sub-folders, you can:</p> <ul style="list-style-type: none"> <input type="checkbox"/> Server Connections. Add and change Remote Services settings. In the Business User Edition, this feature is limited to the main reporting server, EDASERVE. <input type="checkbox"/> Alternate Server Mappings. Configure alternate mappings to individual Remote Servers. This feature is not available in the Business User Edition.
Configuration [Application Settings]	<p>Pages in the Application Settings folder of the Configuration tab manage settings for the WebFOCUS BUE application. Pages in this folder include settings for the following functional areas:</p> <ul style="list-style-type: none"> <input type="checkbox"/> Change Management <input type="checkbox"/> Deferred Reporting <input type="checkbox"/> ESRI <input type="checkbox"/> Parameter Prompting <input type="checkbox"/> Text Generation Server

Folder [Page]	Available Functionality
Configuration [Custom Settings]	<p>The Custom Settings page of the Configuration tab contains a text-based input field where you can define advanced customization settings for the Client. Using this page, you can:</p> <ul style="list-style-type: none"> <input type="checkbox"/> Customize Client settings. <input type="checkbox"/> Create a Client site profile. <input type="checkbox"/> Create a Client universal profile.
Configuration [NLS Settings]	The NLS Settings page of the Configuration tab contains National Language Support settings.
Configuration [Dynamic Language Switch]	<p>The Dynamic Language Switch page of the Configuration tab contains a list of languages that can be included in this feature.</p> <p>In the Business User Edition, Dynamic Language Switch settings are stored in the ibimultilanguage.js file located in the <i>drive:/ibi/WebFOCUS_BUE82/WebFOCUS/ibi_html/javaassist/intl</i> directory.</p>
Configuration [Redirection Settings]	The Redirection Settings page of the Configuration tab contains settings that manage the redirection of report output.
Configuration [InfoAssist+ Properties]	The InfoAssist+ Properties page of the Configuration tab contains properties that configure reporting options in the InfoAssist+ reporting tool.

Navigating the Security Tab

The Security tab contains internal and external security settings, which are listed and described in the following table.

Folder [Page]	Available Functionality
Security Configuration [Internal]	The Internal page of the Security tab contains login and password settings for authentication and authorization managed within WebFOCUS BUE.

Folder [Page]	Available Functionality
Security Configuration [External]	The External page of the Security tab contains settings that define the method and location of authentication and authorization activities managed by third-party applications outside of WebFOCUS BUE.

Navigating the ReportCaster Tab

When you click the ReportCaster tab, the ReportCaster Server Status page opens, by default. Features on that page identify the current status of ReportCaster Server operations. For more information, see [Using the ReportCaster Console](#) on page 137.

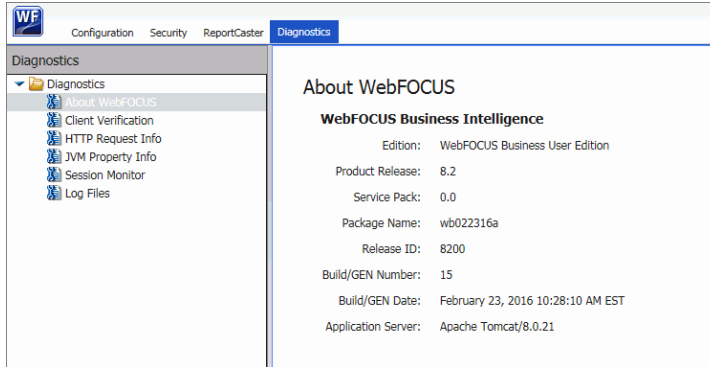
Navigating the Diagnostics Tab

The Diagnostics tab contains settings and features that describe system performance and activities, which are listed and described in the following table.

Folder [Page]	Available Functionality
Diagnostics	<p>The Diagnostics section of the Administration Console contains the following functional areas:</p> <ul style="list-style-type: none"> ❑ About WebFOCUS. Displays version and release information about your installation of WebFOCUS BUE. ❑ Client Verification. Displays your directory permissions and the status of your ability to perform common WebFOCUS and WebFOCUS Client operations. ❑ HTTP Request Info. Displays information about HTTP request headers. ❑ JVM Property Info. Displays information about your Java VM environment. ❑ Session Monitor. Displays session monitor events and links to detailed traces. ❑ Log Files. Displays links to WebFOCUS log files.

Reviewing Version Information

The About WebFOCUS page displays information about the release you are using and the optional components installed with it, as shown in the following image.



You can use the information on this page to identify your current product release and service pack when corresponding with the Information Builders Customer Support team.

When you click *About WebFOCUS*, the following information appears in the main window:

Edition. Product edition, for example, WebFOCUS Business User Edition.

Product Release. Release number, for example, 8.2.

Service Pack. Service pack number, for example, 0.0.

Package Name. Installation file package name, for example, wb022316a.

Release ID. Product release number, for example, 8200.

Build/GEN Number. Specific product build number, for example, 15.

Build/GEN Date. Date and time the build number was generated, for example, February 23, 2016 10:28:10 PM EST.

Application Server. Application Server, for example, Apache Tomcat/8.0.21.

This version of the About WebFOCUS page, that opens from the Diagnostics tab, is only available to those users whose privileges allow them to open and review the Administration Console.

However, the information on this page is also available to those users whose privileges allow them to view the About WebFOCUS Business Intelligence window. To open this other window, in the BUE Portal, on the Menu bar, click *Help*, and then click *About*.

Reviewing Client Verification

The Client Verification page displays the current status of client configuration and application settings. When you open this page, the client automatically conducts the test required to verify that configuration and application settings are open and available.

Settings marked pass are available for use. Settings marked fail are not available for use.

The automated verification process includes checking web server aliases and directory permissions for each type of client communication mode (CGI, WFServlet, or ISAPI).

The Client verification tools, by default, place the verification logs in the `drive:\ibi\WebFOCUS_BUE82\WebFOCUS\logs` directory. These tools test read, write, and remove permissions for the logs directory. They also test read and write permissions for the `drive:\ibi\WebFOCUS_BUE82\WebFOCUS\config` directory.

To test a Reporting Server connection and the current status of the Graph or Table functionality it can deliver, you must open the Configuration tab, and right-click the icon for the Reporting Server you wish to test. All three tests are available on the shortcut menu assigned to each server icon.

Procedure: How to Verify the WebFOCUS Client

From the Administration Console, click *Diagnostics*, and then click *Client Verification*.

The Client Verification page displays your directory permissions, such as creating and deleting applications, signing in as the administrator, reading and writing from the standard directories, creating and deleting domains, and creating and deleting reports.

Note: If you run this test immediately after performing the installation procedure, there might be a delay resulting from a slow or delayed first time initialization of the Tomcat Web Application that supports client operations.

HTTP Request Headers and JVM Property Information

The Diagnostics tab also contains information about HTTP request headers and your Java VM environment.

- ☐ Settings on the HTTP Request Info page display information about the HTTP or HTTPS headers returned to your browser. This is useful for troubleshooting and configuring HTTP headers, especially if you integrate web or application server security with WebFOCUS BUE, or if your web or application server uses virtual hosts (HTTP headers).

- ❑ Settings on the JVM Property Info page display information about your Java VM environment. This is useful for troubleshooting and configuring the Java environment for WebFOCUS BUE web applications and for resolving memory or resource issues.

Navigating the InfoAssist+ Interface

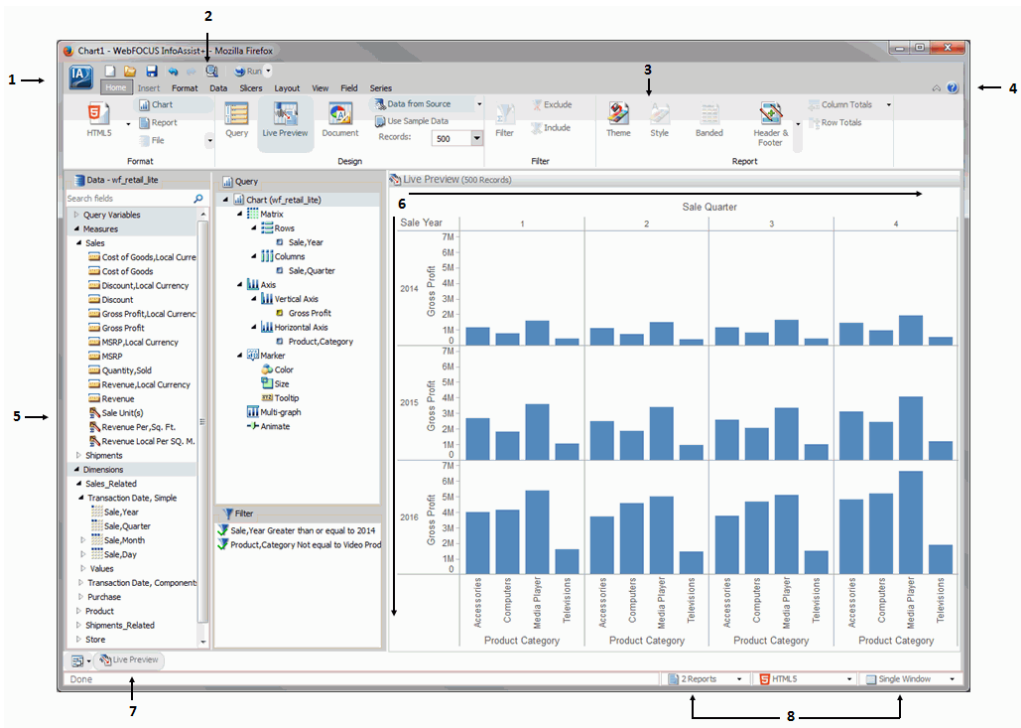
InfoAssist+ provides an intelligent, flexible layout for creating reports, charts, visualizations, and documents.

The application window provides intuitive menus and toolbars, a versatile ribbon that provides access to specialized groups for different functional areas of report design, a taskbar for selecting output, and a status bar for directing output.

The application window also provides a resources area for selecting and sorting data, and a multi-faceted results area that can display report design, a preview of report output, or actual report output.

InfoAssist+ Application Window

The components of the InfoAssist+ Application window are shown in the following image.




The main interface components are explained as follows:

1. **Application button.** Provides access to the Application menu of procedure-related commands.
2. **Quick Access Toolbar.** Displays frequently-used commands, such as New, Open, Save, Undo, Redo, View code, Run, and Preview, in a toolbar that remains visible. For more information, see [Quick Access Toolbar](#) on page 73.
3. **Ribbon.** Displays the commands you need to create reports, charts, visualizations, and documents. It also displays the Help menu. For details, see [Ribbon](#) on page 74.
4. **Help.** Provides access to the online technical content for InfoAssist+ in a new browser window.
5. **Resources panel.** Displays the Data pane, Query pane, and Filter pane.
6. **Canvas.** Displays the results of your report. In Live Preview mode, the canvas displays a preview of the file as it is developed.

Depending on the type of file that you are developing, the canvas may display differently. For example, when creating a document, the canvas includes rulers around the development area. This is to assist you in placing the components on to the document.

7. **Navigation taskbar.** Displays groups and icons that provide different views and quick access to all active reports and report output. For details, see [Using the Navigation Taskbar](#) on page 85.
8. **Status bar.** Provides an output format button that you click to see the selected format and an output target button that you click to view the selected option for displaying new output windows or tabs. For more information, see [Using the Status Bar](#) on page 86.

Application Main Menu

In the upper-left corner of the InfoAssist+ interface, click the IA button  to open the Application main menu.

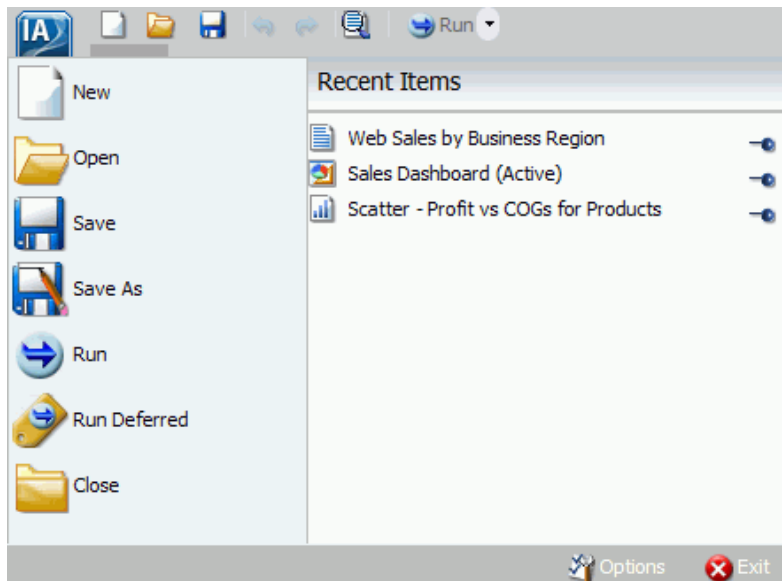
You can run the following commands from the Application main menu:


- ☐ **New.** Opens the InfoAssist+ splash screen, where you can create a new report, chart, visualization, or document. The exact functionality of the New command depends on your current InfoAssist+ session.


When you open a new session of InfoAssist+ and then click *New*, a new report is created. You are prompted to choose a data source.

- ☐ **Open.** Opens an existing report, chart, visualization, or document.

- ❑ **Save.** Saves a report, chart, visualization, or document.
- ❑ **Save As.** Saves a report, chart, visualization, or document with a new name.
- ❑ **Run.** Runs a report, chart, visualization, or document.
- ❑ **Run Deferred.** Submits a report, chart, or document for processing in the background while you continue to work on other tasks.
- ❑ **Close.** Closes the currently active report, chart, visualization, or document.
- ❑ **Recent Items.** Displays recent reports, charts, visualizations, and documents and those items that have been pinned to the menu. The Recent Items area of the InfoAssist+ Application main menu is shown in the following image.



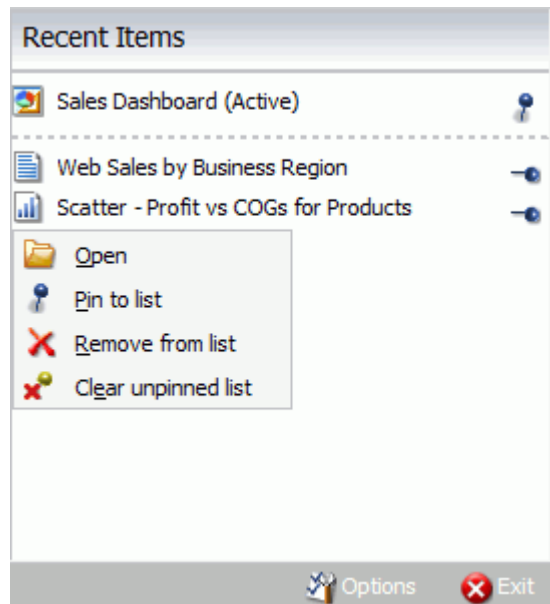
Pinned reports, charts, visualizations, and documents are represented by a blue pin  icon. Pinned items appear at the top of the list in alphabetical order.

Recent reports, charts, visualizations, and documents are represented by a sideways blue pin  icon. Recent items appear below the separator bar in the order in which they were created, with the most recently created item at the top.

You can pin important reports, charts, visualizations, and documents to the top of the Recent Reports section of the Application main menu for quick and easy access. To promote a recent report to pinned report, click the sideways blue pin icon. The icon turns upright and the report is moved to the pinned report area of the Recent Items window, where it stays until it is unpinned.

To demote a pinned report, click the blue pin icon. The icon turns sideways and the report is moved below the separator bar.

A shortcut menu is available when right-clicking a recent item, as shown in the following image.



The options in the menu are:

- ☐ **Open.** Opens the selected report, chart, visualization, or document.
- ☐ **Pin to list.** Pins or unpins a recent report, chart, visualization, or document to the pinned reports list above the separator bar.
- ☐ **Remove from list.** Unpins a pinned report, chart, visualization, or document from the pinned reports list.
- ☐ **Clear unpinned list.** Clears all unpinned reports, charts, visualization, or documents from the Recent Items list.

- ☐ **Options.** Opens the Options window to customize your user preferences.
- ☐ **Exit.** Exits the application.

Reference: Open Dialog Box

The Open dialog box displays when you launch InfoAssist+. It also displays when you join and blend data. You can use this dialog box to do the following:

- ☐ Select data sources for creating reports, charts, visualizations, and documents.
- ☐ Select data sources for joining and blending data.
- ☐ Configure data adapters for connecting to an existing database and creating synonyms.
- ☐ Upload user data.

The Open dialog box contains the following options:

Connect to Data

Opens the Connect to Data wizard, where you can configure a Data Adapter to connect to an existing database and create synonyms.

Upload Data

Opens the Upload wizard, where you can upload user data from a machine to which you have access. This data can be used with WebFOCUS reporting.

Open

Opens the selected Master File.

Quick Access Toolbar

The Quick Access Toolbar provides access to the most commonly used functions. It is located to the right of the Application main menu button and is always visible no matter which options are selected.

- ☐ **Undo and Redo.** The Undo icon undoes your last action. The Redo icon repeats your last action.

The Undo icon is enabled (blue) when there is an action to undo. Otherwise, the icon is unavailable (gray). The Redo icon is enabled (blue) when there is an action to redo. Otherwise, the icon is unavailable (gray).

For example, if you add a database field to a report, the Undo icon turns blue. You can now click the Undo icon to remove the database field from the report. The Undo icon turns gray, and the Redo icon turns blue. To restore the field to the report, click the blue Redo icon.

You can also press Ctrl+Z to undo an action, or Ctrl+Y to redo an action.

You can undo and redo up to 25 actions per session. InfoAssist+ maintains the undo and redo list of actions even when you switch between reports.

When a dialog box is open in the application window, you cannot use the Undo and Redo icons. However, when you click OK and close the dialog box, the icons become available for use. With a single click of Undo, you can undo all the actions that you performed in the dialog box, and you can reinstate them with a single click of Redo.

- ☐ **View code.** View the underlying code for the report.
- ☐ **Run.** Click the Run button to run a report, chart, visualization, or document immediately. Click the down arrow to open the Run menu, which contains the following additional options:
 - ☐ **Preview.** Runs the report, chart, or document in the selected format with the limited number of records as set in the Design group on the Home tab.
 - ☐ **SQL Trace.** Returns the SQL commands for the request.
 - ☐ **SQL Preview Trace.** Returns the SQL commands for the Live Preview run.

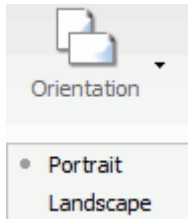
Ribbon

The ribbon is a rectangular area of distinct groups of buttons that spans the top of the InfoAssist+ application window.

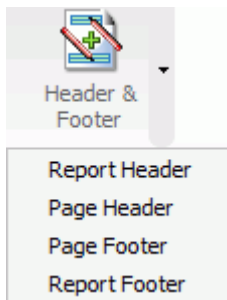
The ribbon is contextual and changes depending on the type of file that you are developing. For example, in Chart mode, the ribbon is made up of nine tabs, whereas when working in Visualization mode, the ribbon consists of five tabs. Each tab contains a subset of InfoAssist+ functionality organized in logically-related groups of controls and commands graphically represented by distinctive icons.

Working With the Ribbon

The ribbon contains two types of arrow-based buttons. The first type of button opens a menu when you click it. The Orientation button is an example of this type of button. Clicking the Orientation button opens a menu of options, as shown in the following image.



The second type of button is a split button. Clicking the left side of the split button performs a default action. Clicking the down arrow, on the right side of the button, opens a menu of options. The Header & Footer button is an example of a split button. It is shown in the following image.



Clicking the Header & Footer button on the left opens the Header & Footer dialog box. Clicking the down arrow on the right opens a menu of options.

Some options open dialog boxes of additional commands and option lists.

The ribbon displays all controls and commands using different sized icons and groups, depending on the size of the monitor and application window.

When you reduce the size of the application window, some groups are collapsed into single icons as determined by the amount of available space.

When a group is collapsed into a single icon, the individual icons are removed from view, but are still available. Clicking a collapsed group icon restores the group to its normal full size and displays all of the individual icons.

When expanding a collapsed group, the ribbon collapses a neighboring group to make enough room to expand the selected group.

When an icon, button, or option in a menu or dialog box appears dimmed, that functionality is not available for that report as it currently exists. Some functionality is available for reports only, charts only, visualizations only, or documents only.

In some situations, selecting one or more options makes other options incompatible with the previously selected ones. As a result, the incompatible options are dimmed and unavailable for selection. InfoAssist+ automatically makes incompatible options unavailable as you create and modify a report.

Note: If you do not have access to a particular option, contact your administrator.

Home Tab

The Home tab contains the most commonly used commands and options, which you can use when developing reports, charts, documents, or visualizations. The following list highlights the features on the Home tab for each mode.

- ❑ For reports, the Format, Design, Filter, and Report groups offer options for changing the output format of your report, as well as styling it.
- ❑ For charts, the Format Design, Filter, and Report groups enable you to change the theme or your chart or add a header and footer.
- ❑ For documents, options are enabled in the Format and Design groups only. Available options for documents include changing the data source for the document and changing the output format.
- ❑ For visualizations, the Clipboard, Data, Visual, and Storyboard groups enable you to perform some of the more basic functions, such as copy and paste. You also use the Home tab to change your visual type.

Related Information for Reports, Charts, and Documents:

- ❑ [Using Filters to Customize the Display of Data](#) on page 645

Related Information for Visualizations:

- ❑ [Building Visualizations](#) on page 381
- ❑ [Using Storyboards](#) on page 642
- ❑ [Joins](#) on page 326

Insert Tab

The Insert tab contains options to add reports, charts, existing reports, text, images, and active form controls (for active reports only) to a canvas in Document mode.

Note: The Insert tab is only available in Document mode.

Related Information for Documents:

- ❑ [Building a Document](#) on page 367

Format Tab

The Format tab provides options for formatting your report, chart, document, or visualization. Depending on the mode you are using, you can perform tasks such as selecting a chart type, enabling Auto Linking, or adding navigational aspects to a report.

The following list highlights the features on the Format tab for each mode.

- ❑ For reports, the Format tab provides access to the Destination, Navigation, Features, and Auto Linking groups. These options enable you to perform many functions related to reports, including Auto Drill or any other report navigation feature.
- ❑ For charts, the Format tab provides access to the Destination, Chart Types, Features, Labels, Interactive, and Navigation groups. These options enable you to perform a variety of charting tasks, including changing the chart type and adding interactive options.
- ❑ For documents, the Format tab enables you to change to Report or Chart mode. You can also access InfoMini.
- ❑ For visualizations, the Format tab provides access to the Report, Features, Label, and Interactive groups. These options enable you to format your visualization. Also included are grid commands, as well as various features and label commands.

Related Information for Reports, Charts, and Documents:

- ❑ [Customizing Reports](#) on page 553
- ❑ [Formatting a Series](#) on page 573
- ❑ [Formatting Data Labels](#) on page 587
- ❑ [Formatting a Legend](#) on page 594
- ❑ [How to Style and Customize a Report](#) on page 376
- ❑ [How to Style and Customize a Chart](#) on page 376

Related Information for Visualizations:

- ❑ [Building Visualizations](#) on page 381

- ❑ [Customizing Charts and Visualizations](#) on page 571

Data Tab

The Data tab contains data manipulation and data display options. For example, you can access Join functionality and work with Defines and Computes. The Data tab is available in the Report, Chart, and Document mode. It is not available in Visualization mode.

The following list highlights the features on the Data tab for each available mode.

- ❑ For reports, the Data tab provides access to the Calculation, Join, Filter, Display, and Data Source groups. When working with reports, you can join data sources or create Defines and Computes. You can also filter data.
- ❑ For charts, the Data tab provides access to the Calculation, Join, Filter, Display, and Data Source groups. When working with charts, these options enable you to join data sources, create a Define or Compute, and filter data.
- ❑ For documents, the Data tab provides access to the Calculation, Join, Filter, Display, and Data Source groups. When working with a document, you can create a define or join data sources. You can also add or switch data sources.

Related Information for Reports, Charts, and Documents:

- ❑ [Joins](#) on page 326
- ❑ [Using Filters to Customize the Display of Data](#) on page 645

Slicers Tab

The Slicers tab provides the ability to create and edit slicers. Slicers are dynamic filters that you can use with reports, charts, and documents.

Note: The Slicers tab is unavailable in Visualization mode.

The features on the Slicers tab are the same for reports, charts, and documents. You can perform the following functions with the slicer functionality:

- ❑ Add slicers to your report, chart, or document to create dynamic filters
- ❑ Set a record limit for the display of your data
- ❑ Create groups of slicers to filter your data
- ❑ Add fields to existing slicers

- ❑ Clear slicers to clear all selected values from existing slicers

Related Information for Reports, Charts, and Documents:

- ❑ [Working With Slicers](#) on page 670

Note: The slicer capabilities are only available in the full version of InfoAssist+.

Layout Tab

The Layout tab provides access to page display and layout options. These include page orientation and AutoFit, which is a feature that limits the width of columns in a report to be no wider than the largest value in each column.

The Layout tab is available in Report, Chart, and Document mode.

Note: The Layout tab is unavailable in Visualization mode.

The following list highlights the features on the Layout tab for each available mode.

- ❑ For reports, the Layout tab provides access to the Page Setup and Report groups. These options allow you to change the size or orientation of your report. You can also add page numbers.
- ❑ For charts, the Layout tab provides access to the Page Setup and Size & Arrange groups. These options allow you to change the orientation or size of your chart. You can also set a height and width for your chart, and enable the AutoFit functionality.
- ❑ For documents, Layout tab provides access to the Page Setup and Size & Arrange groups. You can change the size of your document and adjust its orientation. You can also lock the height and width aspect ratio of your document.

Related Information for Reports, Charts, and Documents:

- ❑ [Customizing Reports](#) on page 553

View Tab

The View tab provides access to reporting options, including design and display options. Some of the options on the View tab control how information is displayed, whether it be in structured view or list view. The View tab is available in all modes of InfoAssist+.

The following list highlights the features on the View tab for each mode.

- ❑ For reports, the View tab provides access to the Design, Show/Hide, Data Panel, Query panel, Output Window and Report groups. When creating your report, you can change the way your data displays or switch to another mode, such as document.

- ❑ For charts, the View tab provides access to the Design, Show/Hide, Data Panel, Query panel, Output Window and Report groups. When creating your chart, you can change the output location for the chart or switch to another report or chart.
- ❑ For documents, the View tab provides access to the Design, Show/Hide, Data Panel, Query panel, Output Window and Report groups. When working with your document, you may want to change the display of data in the Data pane or Query pane.
- ❑ For visualizations, the View tab provides access to the Show/Hide and Data Panel groups. When creating your visualization, you can use these options to show or hide the resources panel. You can also change the display of your data to either logical, list or structured.

Related Information for Reports, Charts, and Documents:

- ❑ [Using the Query Pane and Filter Pane on the Canvas](#) on page 82

Field Tab

The Field tab is enabled in the ribbon when you select a data source field. The options available in the Field tab are specific to the data type that is selected. The options available for numeric fields are different from the options available for non-numeric and date fields. The Field tab provides access to the Filter, Sort, Break, Style, Format, Display, and Links groups.

The following list highlights the features on the Field tab for each mode.

- ❑ For reports, the Field tab provides options for filtering and styling your report. It also provides you with access to the Traffic Lights and Within functionality, making your reports more robust and customized.
- ❑ For charts, the Field tab enables you to hide a particular field or group values in your chart. You can also use the filter options to limit the display of information in your chart.
- ❑ For documents, the Field tab provides options for filtering, as well as Traffic Light functionality. You can also hide fields and add aggregations.
- ❑ For visualizations, the View tab provides access to the Show/Hide and Data Panel groups. When creating your visualization, you can use these options to show or hide the resources panel. You can also change the display of your data to either logical, list or structured.

Related Information for Reports, Charts, and Documents:

- ❑ [How to Apply Traffic Light Conditional Styling to a Report \(By Constant\)](#) on page 562
- ❑ [How to Apply Traffic Light Conditional Styling to a Report \(By Field\)](#) on page 564

- ❑ [Changing a Field Format](#) on page 673

Series Tab

The Series tab provides access to chart options through the Select, Style, Properties, Line, and Pie groups. The Series tab is available in Document, Chart, and Visualization mode. It is not available in Report mode.

The following list highlights the features on the Series tab for each available mode.

- ❑ For charts, the Series tab enables you to add such features as a trendline or data labels to your chart. If multiple series are specified, you can select a series and specify options specific to that series.
- ❑ For documents, the Series tab displays once you have specified fields within your document or embedded chart. You can use the options on the Series tab to add markers, smoothlines, or trendlines to your data.
- ❑ For visualizations, the Series tab enables you to switch between series, style the selected series, or add any other property or line features, as required.

Related Information for Reports, Charts, and Documents:

- ❑ [Formatting a Series](#) on page 573
- ❑ [Formatting Data Labels](#) on page 587

Related Information for Visualizations:

- ❑ [Formatting Axis Labels](#) on page 608

Understanding the Canvas

The canvas displays a preview of the report being created or modified on the canvas when you are in the default Live Preview. To select Live Preview, go to *Home* tab, and in the *Design* group, click *Live Preview*, or on the *View* tab, in the *Design* group, click *Live Preview*. The canvas is always fully maximized and cannot be minimized, cascaded, or tiled. However, a blank canvas opens when there is no report.

The canvas displays either live data or sample data.

- ❑ To display live data, on the *Home* tab, in the *Design* group, click *Data from Source* (default).
When you select Data from Source, a live preview of the report being built is refreshed on the canvas as you add, remove, and style data source fields in the report.

- ❑ To display sample data, on the *Home* tab, in the *Design* group, click *Use Sample Data*.

When you select *Use Sample Data*, the canvas displays sample data based on the field type, with the same formatting and styling used to display live data.

When you execute a report, the canvas displays the Output window. If the Output is minimized, or if there is no report to preview, the canvas is blank. To change the canvas view, you can select *Query Design* view or *Live Preview* from the *Design* group of the *View* tab or *Home* tab.

When working with a report in *Live Preview*, you can rearrange the columns of the report by dragging them to the left or right, depending on your display preference. If the column is numeric, you are presented with the option to *Drop as Sum* or *Drop as Sort*.

Note: If you select *Drop as Sort*, the values are listed individually and they are sorted. If you select *Drop as Sum*, the values are summed for that particular row. The default is *Drop as Sum*.

When placing the column in a new location, a red indicator bar displays, indicating the location of placement for the moved column. This allows you to see where the column will be placed before you actually perform the operation.

Note: Column sort order can also be rearranged by dragging the fields in the *Query* field containers of the *Query* pane to reorder them.

If you create a filter, then the filter appears in the *Filter* pane.

Using the Query Pane and Filter Pane on the Canvas

If you click *Query* on the *Home* tab, in the *Design* group, the *Query* and *Filter* panes expand in to the canvas.

This provides a larger area for displaying the *Query* and *Filter* panes. This feature is useful when you are designing a report with multiple filters or numerous fields.

Depending on whether you are creating a report or a chart, the *Query* pane displays selected data source fields using different types of *Query* field containers.

Related Information:

- ❑ [Using Filters to Customize the Display of Data](#) on page 645
- ❑ [Using Field Containers](#) on page 91

Understanding Output Options

When you run a report, the output appears, either in a tab on the canvas, or in a new browser window. You can create and display output in several different ways, depending on the following options. You can select these options on the *View* tab, in the *Output Window* group:

- ☐ Cascade
- ☐ Tile Horizontally
- ☐ Tile Vertically
- ☐ Single Tab
- ☐ New Tab
- ☐ Single Window
- ☐ New Window
- ☐ Switch

Output window and tab options are also available in the status bar, and output window display options are also available in the Navigation taskbar.

Note: When you run a report, tab focus is not on the output window and pressing the Tab key does not move the selection. To move the Tab focus out of the output window, press F6.

Reference: Output Target Options

The following are output target options that you can select.

- ☐ **Single Tab.** When you click *Single Tab* and run a report, a new output window is created on the canvas, a report instance is created, and an output tab is placed on the Navigation taskbar. As you modify a report, the same output window is refreshed each time the report is run. This option, which is the default, is ideal when you are working with just one report.
- ☐ **New Tab.** When you click *New Tab*, each time you run a report, a new output window is created on the canvas. A new report instance is also created and preserved by the addition of a new output tab on the Navigation taskbar. Each output tab maintains the output of the report that generated the corresponding output window. Selecting an output tab on the Navigation taskbar loads the associated output instance into the output window.

- ❑ **Single Window.** When you click *Single Window* and run a report, a new browser window is opened and populated with the report output. As you modify the report, the same browser window is refreshed each time the report is run. If the browser window is closed and the report is run, a new browser window is opened again and is refreshed for each subsequent run. The output is not displayed on the canvas, and an output tab is not added to the Navigation taskbar.
- ❑ **New Window.** When you click *New Window*, each time you run a report, a new browser window is opened and populated with the report output. The output is not displayed on the canvas, and an output tab is not added to the Navigation Taskbar.

Reference: Output View Options

The following are output view options (Arrange) that you can select.

- ❑ **Cascade.** When you click *Cascade*, if multiple output windows exist, they are cascaded diagonally across the canvas. This option does not affect open browser windows when you select New Window or Single Window.
- ❑ **Tile Horizontally.** When you click *Tile Horizontally*, if multiple output windows exist, they are tiled horizontally, one above another, across the canvas. This option does not affect open browser windows when you select New Window or Single Window.
- ❑ **Tile Vertically.** When you click *Tile Vertically*, if multiple output windows exist, they are tiled vertically, side by side, across the canvas. This option does not affect open browser windows when you select New Window or Single Window.
- ❑ **Switch Output.** When you click *Switch Output*, a drop-down menu opens, where you can select any active report to view the output. The selected report is loaded into the output window or browser window, depending on the selected output window or tab option.

Reference: Output Format Options

The Output window can display report output in the following formats: HTML, HTML5, active report, and PDF.

Note: Excel and PowerPoint formats open in their native programs in a window external to InfoAssist+.

Using the Navigation Taskbar

The Navigation taskbar provides quick access to all active output windows and to the report design that generated the output. You can return to the last report that you edited in the Query pane by clicking *Display Design View* on the taskbar. The Navigation taskbar is always visible. It is located near the bottom of the application window, just above the status bar.

Each of the active output windows displays a tab on the Navigation taskbar. Selecting the tab displays that output window on the canvas. If you save a report with a unique name each time you modify it, when the report is executed and a new output window is generated, the unique name appears as a tab on the taskbar. If you continue to modify and execute a report without saving it with a unique name, a number in parentheses is appended to the original, saved report name to differentiate it among the multiple output windows.

The ability to select report output from the Navigation taskbar depends on the output window option selected from the status bar or from the Output Window group of the View tab. For more information, see [Using the Status Bar](#) on page 86, or [Understanding Output Options](#) on page 83.

Each report tab on the Navigation taskbar has a shortcut menu with the following options:

- ☐ Restore
- ☐ Minimize
- ☐ Maximize
- ☐ Close
- ☐ Auto Resize

Restore, Minimize, Maximize, and Close are standard options available in any browser window or software application. Minimize, Maximize (if the window is not maximized), or Restore (if the window is maximized), and Close can also be found in the top-right corner of the output window.

The Auto Resize option enables automatic resizing of an output window as needed when you add or remove fields.

The Window Options menu at the far left of the Navigation taskbar provides options for displaying all active output windows on the canvas. The display options that you can select are Cascade, Tile Horizontally, Tile Vertically, Restore All, Minimize All, Maximize All, or Close All. The name of each active output window appears at the bottom of the menu.

The Window Options menu options are directly linked to the options available in the Output Window group of the View tab.

Using the Status Bar

The status bar displays the status of the last selected action, a reports button that shows the number of open reports, an output format button that shows the selected format, as well as a list of the available formats, and an output target button that shows the selected option for displaying new output windows or tabs.

When you click the reports button, a menu opens, with options for selecting any of the open reports. Each report is listed by name and an icon which represents the report type (report, chart, visualization, document). The report you select becomes active.

When you click the output format button, a menu opens, with options for selecting a different output format. For a complete list of options, see [Understanding Output Options](#) on page 83.

When you click the output target button, a menu opens, with options for Single Tab (default), New Tab, Single Window, and New Window. For more information, see [Understanding Output Options](#) on page 83.

Understanding the Resources Panel

The Resources panel is comprised of the Data pane, the Filter pane, and the Query pane. It is located on the left side of the InfoAssist+ interface and is visible by default. You can customize how the Resources panel displays by using the options on the Home tab and View tab. For example, on the *View* tab, in the *Design* group, click *Query* to display only the Query and Filter panes.

The Data pane, which contains all the fields from the selected data sources, is always displayed.

You can manually adjust the size of the Resources panel. To do so, move the mouse pointer over the border. When the pointer changes to a two-way arrow, drag the border.

Reference: Field Image List

In the Resources panel, each field has an image associated with it. The following table displays each image and describes what it represents.

Note: This list of icons changes based on the type of database that is in use.

Reference: Filter Pane

The Filter pane displays all filters that have been created for the selected report.

For more information on filtering, see [Field Tab](#) on page 80 and [Data Tab](#) on page 78.

Note: The Filter pane displays all created filters, both active (included) and inactive (excluded).

Using the Data Pane

The Data pane contains the data fields that are used to construct reports, charts, or visualizations. The default structure of data in the Data pane displays your data by field type, for example, measure field or dimension field. You can explore this structure using the sample retail database that is included with InfoAssist+. The structure of this sample database, which is a cube database, includes measure groups, measure fields, dimension hierarchies, dimension fields, and attributes.

The fields in the database structure are displayed in Logical view, by default. You can use the commands in the View tab to change how your data fields are organized in the Data pane. These commands display the data fields in a Logical, List, or Structured view. All three views provide options for displaying each data source field as a Title, Description, Field, or Alias. The List view also includes options to show the Alias, Format, and Reference of each field.

Field list search functionality is also available in the Data pane, which you can use to search for specific fields within a tree or list. When conducting a search in the Tree view, the search functionality searches only the attribute that is displayed.

When searching in List view, all attributes are searched at once.

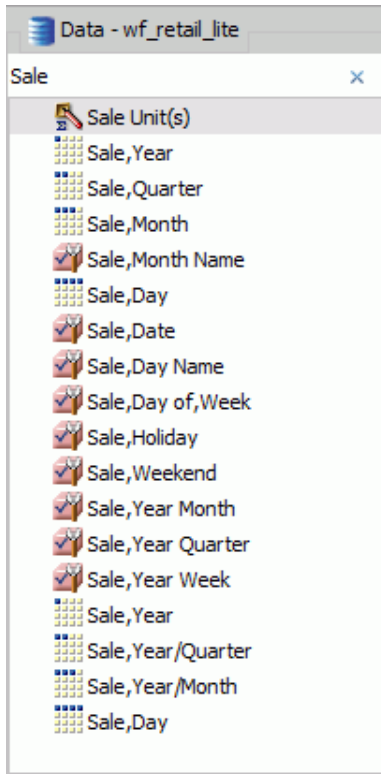
Note: There are additional options on which you can search, including Name, Title, Alias, Format, Segment, Filename, Description, and Reference.

As you enter search criteria, InfoAssist+ begins the search process. When you enter just a few letters, the list of records that is returned is typically long. As your search criteria gets more specific, the list of returned fields narrows.

If you are conducting a search that you wish to cancel, click the X icon in the field list search tool to abort the search.

Note: The X icon displays only when you have search criteria specified in the Search fields box.

The following is an example of the default Logical view, displaying the Title of each field.



Using the Data Pane to Add Fields to a Report

There are several ways that you can add data source fields to a report. You can drag, double-click, or right-click data source fields in the Data pane to add them to a Query field container or the Filter pane.

After you add data source fields to a Query field container, you can change the order of the fields by dragging and dropping one field above or below another field.

Drag. This method provides the most control. You can drag data source fields from the Data pane to the appropriate Query field container or Filter pane.

For a larger work area in which to drop data source fields in the appropriate Query field container, make sure that Query Design view is selected, and then, on the View tab, in the *Query Panel* group, click *Areas 2x2* or *Areas 1x4*.

You can select Query Design view on the *Home* tab, in the *Design* group, by clicking *Query* or on the *View* tab, in the *Design* group by clicking *Query*. That selection expands the Query and Filter panes.

Multi-Select. You can multi-select data source fields that you want to add to a report in Live Preview, Query Design view, and Document mode. To select multiple data source fields to add to a report, click the appropriate fields while holding the Ctrl key on the keyboard. You can drag fields onto the canvas, or add them to the Query field container.

Double-Click. To automatically add a field to the appropriate Query field container in the Query pane, you can double-click a data source field in the Data pane.

- ☐ When you double-click a measure field in the Data pane, it is automatically added to the Sum Query field container.
- ☐ When you double-click a dimension field in the Data pane, it is added to the By (Row Label) Query field container for a report, or to the X-axis Query field container for a chart.

You cannot automatically add a field to the Across (Column Label) Query field container for a report, or to the Legend (Series) and Multi-graph Query field containers for a chart.

Right-Click. You can right-click a field in the Data pane to add it to the Filter area or a Query field container in the Query pane. For reports, the available shortcut menu options are as follows:

- ☐ **Across.** For dimension fields.
- ☐ **Add to Query.** Opens a menu of available Query field containers, to which you can add your data.
- ☐ **Create Group.** Allows you to create a group of elements based on the field data type that you select.
- ☐ **Include as Category Axis.** For dimension fields.
- ☐ **Include as Coordinated.** Only available in Document mode.
- ☐ **Include as Legends Series.** For dimension fields.
- ☐ **Filter.** For all types of fields.
- ☐ **Slicers.** For all types of fields.
- ☐ **Sort.** Adds the field to the Sort field container. This is for all types of fields.
- ☐ **Sum.** For measure fields.

A measure is a numeric value, such as Gross Profit or Cost of Goods Sold, that you may want to aggregate. All numeric values that can be summed are measures. Numeric fields that cannot be summed, such as product number and miles per gallon, 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 can 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.

Using the Query Pane and Filter Pane

The Query and Filter panes appear below the Data pane, except when you select Query Design view, which expands the size of the Query and Filter panes. There are different Query field containers for reports and charts.

Note: If you have more than one item in either Sum, By, or Across Query field containers (for reports) or Measure (Sum) or X-axis Query field containers (for charts), you can drag them up or down in the Query pane to rearrange the order in which they display in your report or chart. When you drag fields to rearrange them, an indicator line displays, providing guidance as to where the field will be placed. The color of this line is determined by the theme. Once you have performed the rearrangement, the Live Preview refreshes based on the newly indicated order.

Reports. For all reports, the Query field containers in the Query pane include Sum, By, and Across.

- ☐ Use the Sum  field container to aggregate or display numeric measure fields. Its context menu provides options to Sum (default), Print, Count, or List the fields in the report.
- ☐ Use the By Query field container to vertically sort dimension fields to produce row labels in the report output. Dimension fields are normally non-numeric or date fields.
- ☐ Use the Across Query field container to horizontally sort dimension fields to produce column labels in the report output.

Using Field Containers

Field containers are used to hold fields that you select based on the function that you want it to perform. For example, in Chart and Visualization mode, you can add a field to the Color field container, which will color your data based on the selected field.

For reports, the available field containers are different than those presented for charts and visualizations. This section reviews all of the field containers, providing you with a reference point when working with field containers.

Field Containers for Reports

The following field containers display when creating reports.

Across

Enables you to display column headings across the top of the report for the measure or dimension that is placed in this field container.

By

Enables you to specify sort fields for your report.

Note: You can rearrange the display order of your By sorts by dragging them into the desired order in the Query pane.

Sum

Displays numeric totals for numeric (measure) fields that are added to this field container.

Field Containers for Charts and Visualizations

The following field containers display when creating charts and visualizations.

Animate

Enables you to animate time progression using a slider control. As you move the control along the slider bar, an animation effect results. The slider control has a Play button that allows you to play and pause the animation. When you click Play, the Pause option is activated, enabling you to pause the progression and analyze your data. Slider controls are limited to one sort field and should be time or sequence related, such as month or year. This field container displays for many different chart types.

By

Use this field container to differentiate data types in a field that contains multiple categorical values. For example, when creating a gauge chart, the field that you specify in this field container becomes a By field, by which all values are displayed as a separate chart.

Category

Enables you to display data categorically in a chart. This field container displays for Tag Cloud charts, which are available in the Other charts category.

Category Axis

Use this field container to specify a field that contains categorical data. These categories display on the categorical portion of a 3D bar chart. The other axis, Series Axis, is plotted against the categorical data.

Close

Enables you to specify a numeric field to indicate the closing value of a stock. Other required values include: Open, High, and Low. This field container displays for Stock charts, which are available in the Other charts category.

Color

Enables the application of different colors based on the underlying data set for the field in this field container. When you place a numeric field in the Color field container, the legend displays as a heat scale. When you place an alphanumeric or a date field (non-measure) in the Color field container, the legend displays as colored markers. To change how the legend displays, right-click the Color field container and click *Color BY*.

Columns

Enables you to specify a field to display column data in a matrix chart in a visual. The use of measure fields is supported. Column data is displayed at the top of the visual, along the X axis. This field container is available for charts and visualizations.

Detail

Use this field container to add detail to your visual by adding a data field to it. For example, if you add Sale,Quarter to the Detail field container in your Scatter plot, the points on the plot are quadrupled, one for each quarter. In addition, the field that you specify in the Detail field container also displays on the hover menu for each point in the plot.

Grouping

Enables you to specify a field by which to present your data in categories or groups. For example, this data field controls how a treemap is grouped.

High

Enables you to specify a numeric field to indicate the high value of a stock. Other required values include: Open, Low, and Close. This field container displays for Stock charts, which are available in the Other charts category.

Horizontal Axis

Displays the data for the selected along the X axis. This field container is available for charts and visualizations.

Note: You can specify up to three fields on the horizontal axis.

Layer

Enables you to specify a Geolocation field for use in a map.

Note: Geolocation fields are listed in the Data pane with a Geolocation icon .

Legend (Series)

Enables you to add a field to specify for the legend of the chart. This field container displays for a number of charts that are available in the Other chart category.

Low

Enables you to specify a numeric field to indicate the low value of a stock. Other required values include: Open, High, and Close. This field container displays for Stock charts, which are available in the Other charts category.

Lower Limit

Enables you to specify a numeric (measure) field by which to establish a lower limit. This field container displays for Vertical Box Plot charts, which are available in the Other chart category.

Lower quartile

Enables you to specify a numeric (measure) field by which to establish a lower quartile. This field container displays for Vertical Box Plot charts, which are available in the Other chart category.

Measure

Use this field container to specify a measure that will serve as the basis for a pie chart. The Measure metric is used with the Color field container for pie charts to create sections based on your field selections. It is also used with Gauge charts.

Measure (Sum)

Enables you to specify a numeric (measure field) by which to sum data. This field container displays for a number of charts that are available in the Other chart category.

Median

Enables you to specify a numeric (measure) field by which to establish an median. This field container displays for Vertical Box Plot charts, which are available in the Other chart category.

Multi-graph

Enables the creation of multiple graphs based on the field that you place in this field container. This field container displays for many different chart types, including Spectral Heatmaps, which are available from the Other dialog box.

Open

Enables you to specify a numeric field to indicate the opening value of a stock. Other required values include: High, Low, and Close. This field container displays for Stock charts, which are available in the Other charts category.

Rows

Enables you to specify a field to display row data in a matrix chart in a visual. The use of measure fields is supported. Row data is displayed on the left side of the visual, along the Y axis. This field container is available for charts and visualizations.

Series Axis

Use this field container to specify a series to plot against the data presented on the Categorical Axis. This field container displays for 3D bar charts.

Size

Controls the size of markers or other data points.

Size (Sum)

Enables you to specify a numeric (measure) field, which will control the size of the data presented in a chart. This field container displays for Tag Cloud charts, which are available in the Other charts category.

Slices

Enables you to specify a field whose data values will dictate the size of the slices in a chart. This field container is available for Funnel charts, which are available in the Other chart category.

Tooltip

When you place a field in this field container, InfoAssist+ enables you to view additional information in the tooltip for a chart or visual.

Upper limit

Enables you to specify a numeric (measure) field by which to establish an upper limit. This field container displays for Vertical Box Plot charts, which are available in the Other chart category.

Upper quartile

Enables you to specify a numeric (measure) field by which to establish an upper quartile. This field container displays for Vertical Box Plot charts, which are available in the Other chart category.

Vertical Axis

Displays the data for the selected field along the Y axis. This field container is available for charts and visualizations.

X Axis

Enables the specification of a field on the X Axis of certain chart types. For example, XY Polar. This field container displays for a number of charts that are available in the Other chart category.

Y1 Measure (Sum)

Enables you to specify a measure (numeric) field for the Y Axis in a Pareto and certain line charts, which can be accessed from the Other chart category.

Y2 Measure

Use this field container to indicate a measure to serve as the basis for second Y axis. This field is used in various line charts, which can be accessed from the Other chart category.

Using Shortcut Menu Options in the Query Pane

In the Query pane, you can right-click any field and select from a list of available options that are displayed in the menu that appears. The options that you can select vary, depending on the type of Query field container in which the field is located and the type of report that you are creating.

You can also rename the content type in the Query pane. You can right-click the component and click *Rename* to change the title.

This allows you to customize the title in the Query pane. In addition, when working in Excel format, the name that you provide is used as the worksheet name in the workbook that is generated at run-time.

Note: If you are working with a report or chart in HTML format, you can also access Sum, Print, List, and Count aggregations from the shortcut menu on the report or chart component.

Related Information:

- ❑ [Using Field Containers](#) on page 91
- ❑ [Right-Click Field Options in the Query and Filter Panes](#) on page 96

Reference: Right-Click Field Options in the Query and Filter Panes

The following table lists and describes all the right-click field options available in the Query and Filter panes for a selected field.

Option	Description
# of Columns	Opens the Number of Columns dialog box, where you can indicate the number of columns in which you wish to display multiple graphs.
Aggregation Functions	Provides a menu for selecting options to assign an aggregation type value to a selected numeric field in a report.
Break	Provides a menu of options to enable page breaks and subtotals, and resetting page numbers.
Create Group	<p>Allows you to create a group of elements based on the field data type that you select. Once you define a new group, a higher-level field is created that contains the selected elements.</p> <p>For more information, see How to Create a Dynamic Group on page 98.</p>
Change Title	<p>Opens the Edit Title dialog box, where you can change the title of the selected field by typing the new title in the Enter Title field.</p> <p>Note: When changing the title of a column heading, you can place a comma between words in the Enter Title field to create a multi-line header.</p>
Data Bars	Provides a menu for enabling the data bar representation functionality. Selecting <i>On</i> adds a data visualization column to the right of a selected numeric field. The column displays values in each row using horizontal bars that extend from left to right in varying lengths, depending on the corresponding data values.

Option	Description
Delete	Deletes the selected field. Note: In the Query and Filter panes, you can select and delete multiple data fields at one time. Using the CTRL key, select two or more data fields, right-mouse click, and then click <i>Delete</i> .
Drill Down	Opens the Drill Down dialog box, where you can create multiple drilldown links on a data field to external procedures or websites.
Edit Format	Opens the Field Format Options dialog box, where you can edit the field type and display options.
Filter Values	Opens the Filter dialog box for creating WHERE statements, which enable you to select only the data that you want and to exclude all unwanted data.
Missing	Allows you to show or hide fields with no value.
Sort	Provides access to the Sort, Rank, and Limit menus. The Sort menu allows you to sort your data either in ascending or descending order. The Limit menu allows you to specify the number of unique values displayed for a sort group that has been added. The Rank menu allows you to insert a rank column immediately to the left of a Sort By field if a Sort By field is selected and adds a rank column to the left of the Sort By field if a Measure field is selected. Ranking a Measure field results in two copies of the field, the original Measure field and the Sort By field that is created during ranking.
Sub Footer	Opens the Sub Header & Sub Footer dialog box where you can edit and style your footers.
Sub Header	Opens the Sub Header & Sub Footer dialog box where you can edit and style your headers.
Traffic Light Conditions	Opens the Traffic Light Condition dialog box, where you can add new conditional styling or modify existing conditional styling by applying traffic light (and other) colors to a selected field in the report output when the field meets specified criteria.

Option	Description
Visibility	Provides a menu for controlling the visibility of the selected field. Clicking <i>Hide</i> removes the selected field from the report output. Clicking <i>Show</i> (default) displays the selected field in the report output.

All of the shortcut menu options that are available in the Query and Filter panes are also available in the Field tab. For more information, see [Field Tab](#) on page 80.

Dynamic Grouping

Dynamic grouping allows you to create groups of elements based on the field data type that you select. For example, in the wf_retail_lite database, there are a number of brands of televisions. Using the dynamic grouping functionality, you can create groups based on the popularity of a particular brand. The first group might include top sellers such as LG and Sony. The second group might contain the remaining brands (Panasonic, GPX, Supersonic, Tivax, and Audiovox). This would allow you to group top sellers into one group, and the remaining brands into another group.

Note: The Create Group option is only available for dimension fields of non-numeric format or attribute.

You can also specify multiple, unique groupings in the same session. For example, you might want to group the data to indicate groups or products, or specific regions.

Note: If you want to exclude a specific data element from your analysis, you can use the filter functionality.

The grouping that you specify is applied and this new group then replaces the original field that you selected in the Query pane. The name that you specify when creating the group is reflected in the Query pane.

Note: You can edit the group once it has been created by right-clicking on the higher-level field and clicking *Edit Group*.

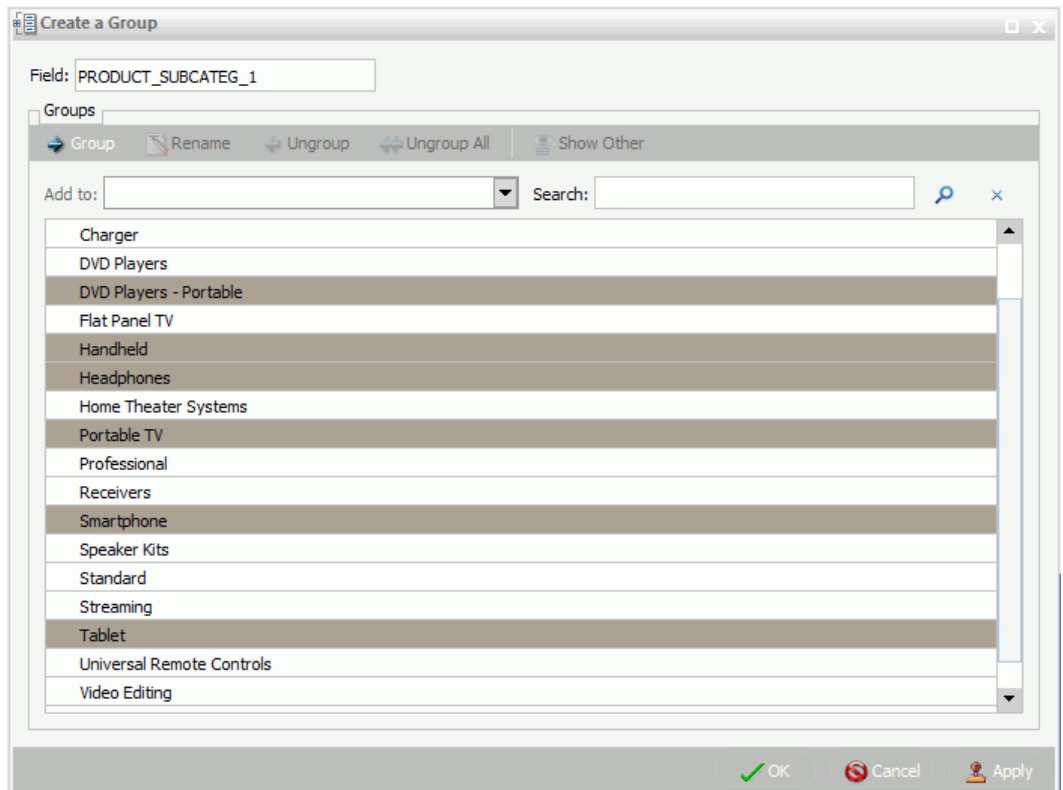
Procedure: How to Create a Dynamic Group

1. From the Data pane, add one or more data fields to your report or chart.
Note: You can use the dynamic grouping functionality on non-numeric dimension fields only.
2. In the Query pane, right-click the data field for which you want to apply dynamic grouping.
Note: The Create Group option is also available in the Data pane shortcut menu.

3. Click *Create Group*.

The Create a Group dialog box displays.

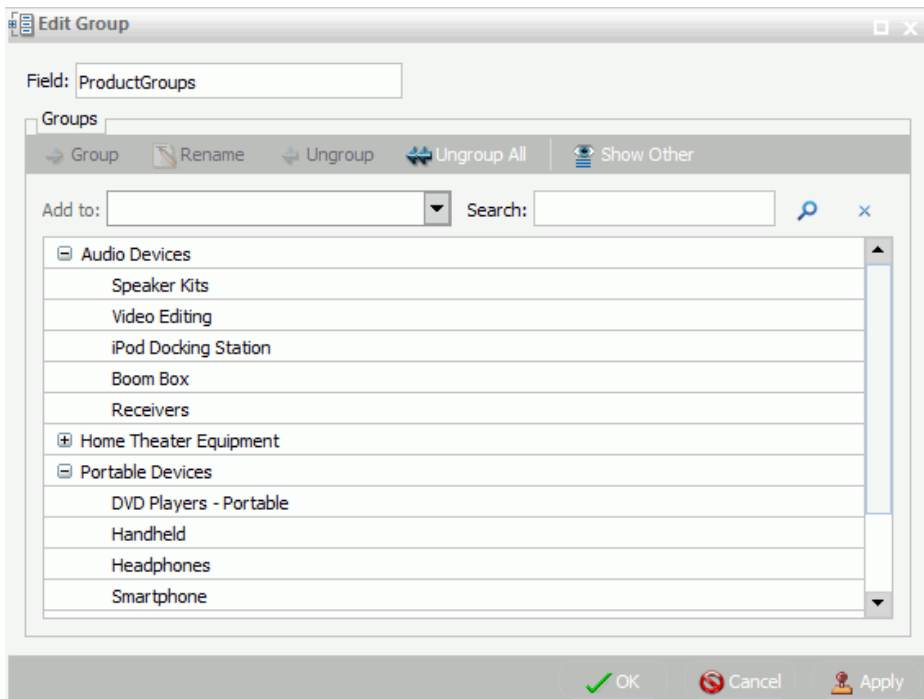
4. In the Field text box, optionally type a name for the new group.
5. Select the data values that you want to group. Use CTRL + click to select more than one value, as shown in the following image.



6. Click *Group*.

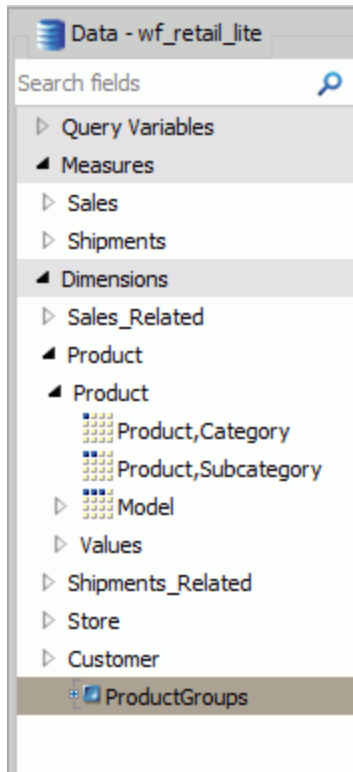
Note: To ungroup values, click a group and then click *Ungroup*.

7. Optionally, create additional groups, as shown in the following image.



8. Click OK.

Your grouped data will display in the Data pane when your report, chart, or visualization refreshes, as shown in the following image.



Setting Up Your Environment

Configure and customize your environment, create users and groups, and manage your resources using the available functionalities.

In this chapter:

- ☐ [Using the Administration Console](#)
 - ☐ [Changing Application Settings](#)
 - ☐ [Changing Client Settings](#)
 - ☐ [Using the ReportCaster Console](#)
 - ☐ [Working With Domains](#)
 - ☐ [Working With Folders](#)
 - ☐ [Managing Users](#)
 - ☐ [Managing Groups](#)
 - ☐ [Managing Private Resources](#)
 - ☐ [Authenticating Users to Your Active Directory](#)
 - ☐ [Configuring WebFOCUS BUE for SSL](#)
 - ☐ [Creating a Change Management Package](#)
 - ☐ [Changing InfoAssist+ User Preferences](#)
 - ☐ [Configuring Hyperstage](#)
 - ☐ [Distributing the Quick Data Add-In File](#)
-

Using the Administration Console

The Administration Console contains the settings that configure the WebFOCUS Client, customize internal or external authentication settings, connect you to ReportCaster, and support diagnostic research.

Opening the Administration Console

Because the Administration Console contains settings that can alter the operation of your entire WebFOCUS installation, it is available only to those users with the privileges to update or reconfigure system settings. In the Business User Edition, the Administration Console is available only to the Manager User.

Before you can open the Administration Console, you must sign in with a User ID that has the privilege to do so. Once you have signed in with the appropriate privileges, the Administration Console is visible and available for review and updates. You can then open the Administration Console from the Menu bar, or from a link on the Getting Started page.

You can also open the Administration Console directly from the address bar of your browser window by entering the URL of the Administration Console and then signing in with a User ID that has the appropriate privileges.

Procedure: How to Sign In to the Administration Console From the Start Menu

1. If WebFOCUS Business User Edition has been installed on a Windows machine, click the *Start* button, and then click the *Information Builders* folder.
2. Click the *WebFOCUS Business User Edition 82* folder, and then click *Run WebFOCUS Business User Edition*.
3. In the Sign In page, type the *ID* and *Password* of a User that has privileges to open the Administration Console, and then click *Sign In*.
4. To open the Administration Console from the Menu bar, click *Administration*, and then click *Administration Console*.
5. To open the Administration Console from the Quick Links Page, in the Manage section, click *Configure System*.

Procedure: How to Sign in to the Administration Console From a Browser Window

1. Go to the URL:

`http(s)://machine:port/context/admin`

where:

`machine`

Is the network id of your computer.

`port`

Is the number of the port that connects your computer to the server hosting WebFOCUS BUE.

`context`

Is the local address for WebFOCUS BUE. For example, *ibi_apps*.

2. On the Sign In page, type the *ID* and *Password* of a User that has privileges to open the Administration Console, and click *Sign In*.

The Administration Console opens automatically.

To display the Administration Console using a different language, you can use the Dynamic Language Switch on the Configuration menu.

Using the Administration Console Menu Bar

The Administration Console menu bar appears above the Administration Console tab display. Its commands and features are available to all of the Administration Console tabs.

Using the Licenses Menu

The Licenses Menu links you to information about your current product license, an audit of User and Group licenses and roles, and to information about licenses for all third-party software products included in the installation. Using Licenses menu commands you can:

- ☐ View the current license number, product edition, license key expiration date, and the number of licensed users. You can also add new license numbers.
- ☐ Access license information for all third-party software packaged with WebFOCUS BUE.

Reviewing Client License Information

The WebFOCUS Client command opens the License Information dialog box. This dialog box identifies the current license key and the individual product components made available by that key. You can also use it to replace the current license key with a new license key when your current license expires or changes.

The License Information dialog box provides the following information:

- ☐ **Product Edition.** The name of the current product edition.
- ☐ **License Key.** The license key currently in use.
- ☐ **License Key Expiration Date.** The date the license key will expire. By default, a warning message for the client license key expiration date begins to appear fourteen (14) days before the actual expiration date. This message displays the expiration date and the number of days remaining until that date. The License Expiration Warning message appears only to Administrators during sign on, and it is written to the event.log file located in the logs directory of the WebFOCUS Client installation.
- ☐ **User Licenses.** The total number of available user licenses and the number of licenses used for each user category. For example:
- ☐ **Product Components.** The product components your license entitles you to use. If the check box to the right of an entry is visible and selected, you are entitled to use that product component.
- ☐ **New License Key.** Opens the Update License dialog box, where you can add a new license key and site code.

Reference: Managing Client Licenses

Access to WebFOCUS features and the number of licensed users is based on your license key and site code.

When the number of users exceeds the number of licensed users, the User Licenses Used count displays, in red, a message that the user license count has been exceeded, which is written to the event.log trace file. Users that are authorized to access the Administration Console will receive a message upon signing in.

User licenses are maintained or enforced when the following product components are licensed:

- ☐ **Total Users.** The total number of named users in the WebFOCUS Repository.
- ☐ **Portal Users (PR).** The number of users with portal privileges.
- ☐ **InfoAssist+ Users.** The number of users with portal privileges and InfoAssist+ privileges.

Procedure: How to Configure License Codes

Access to the WebFOCUS features and the number of Managed Reporting users is based on your license key and site code. You can change these values from the License Management page.

1. In the Administration Console menu bar, click *Licenses*, and then click *WebFOCUS Client*.
The License Information window opens, displaying features available under the current license.
2. Click *New License Key*.
3. Type your new license key and site code.
4. Click *Validate*.

The License Management page displays the current license key, the new license key, and the features that the new license key provides.

5. Click *Save* to implement the new license.

You must reload your web application in order for your changes to take effect. In addition, users must sign out and sign back in to obtain access to any new features.

Reviewing User Audit Information

The User Audit command evaluates the repository license usage for Total Users, Portal Users, and InfoAssist+ Users. The command produces a License Analysis report that identifies the total number of licenses by license type, the number of licenses in use by license type, and analyzes license assignments by Group and by User.

You can run the User Audit utility (license_audit.bat) from your local WebFOCUS installation directory, which is available in the following location:

`drive:\ibi\WebFOCUS_BUE82\WebFOCUS\utilities\mr`

When you run this program, it generates the License Analysis report and transfers it to the auditUserCounts.htm file, in the same directory.

The License Analysis report contains the following information:

License Analysis	
License Key	Displays your current license key.
User License	Displays the user license types that are authorized under your current license key. This can include the following: <ul style="list-style-type: none"> <input type="checkbox"/> Total Named Users <input type="checkbox"/> Portal Users <input type="checkbox"/> InfoAssist+ Users
Code	Displays the code for each user license, such as TU for Total Users.
Maximum	Displays the maximum number of user licenses that are available with your license key.
In Use	Displays the number of user licenses that are currently in use.
Available	Displays the number of user licenses that are available for each license.
Analysis of Groups	

License Analysis	
Group Path	<p>Displays the Groups stored in the repository. The following groups are created by the WebFOCUS Repository Creation utility, by default:</p> <ul style="list-style-type: none"> <input type="checkbox"/> /EVERYONE <input type="checkbox"/> /Getting_Started <input type="checkbox"/> /Managers <input type="checkbox"/> /Retail Samples
License Type(s)	Displays the license types for each Group, such as TU.
Role	Displays the role of each Group, such as SystemFullControl.
On Resource	Displays the resource to which the Role is applied for the Group.
Former Type(s)	Displays the former types of licenses for each Group.
Groups Summary	<p>Displays counts for the following:</p> <ul style="list-style-type: none"> <input type="checkbox"/> Number of groups <input type="checkbox"/> Number of groups with license types <input type="checkbox"/> Number of groups with no license types <input type="checkbox"/> Number of groups with changed user types <input type="checkbox"/> Number of groups with cleared user types <input type="checkbox"/> Number of groups with unchanged types
Analysis of Users	
User Name	<p>Displays the users stored in the repository. The following users are created by the WebFOCUS Repository Creation utility, by default:</p> <ul style="list-style-type: none"> <input type="checkbox"/> manager
License Type(s)	Displays the license types assigned to each user.

License Analysis	
# Group w/Licenses	Displays the number of groups with licenses of which the user is a member.
Former Type(s)	Displays the license types that have been changed or cleared for each user.
User Summary	Displays counts for the following: <ul style="list-style-type: none"> <input type="checkbox"/> Number of users <input type="checkbox"/> Number of users with license types <input type="checkbox"/> Number of users with no license types <input type="checkbox"/> Number of users with changed user types <input type="checkbox"/> Number of users with cleared user types <input type="checkbox"/> Number of users with unchanged types

Procedure: How to Run the User Audit for the WebFOCUS Client from the Administration Console

From the Administration Console menu bar, click *Licenses*, and then click *User Audit*.

The License Analysis report opens in a separate browser window.

Reviewing Third Party Licenses

The Third Party command opens the 3rd Party Information page that displays the following information for each third-party software application used to support different features in WebFOCUS BUE:

- ☐ Description
- ☐ Version
- ☐ File(s)
- ☐ License
- ☐ Third-Party Links

The WebFOCUS BUE product uses various third-party software applications that support a variety of features.

Clearing the Cache

The Clear Cache command refreshes the state of the WebFOCUS application by applying saved changes that are not applied dynamically. Even though some changes are dynamic or only require the administrative user to clear the cache to take effect, others require an administrative user to recycle the web application.

Closing the Administration Console

The Close command closes the Administration Console. After the console closes, you remain signed in to your work session.

Opening Administration Console Help

When you click the Help icon, the online Help file opens to a topic that describes the tab, setting, or feature currently on display.

Configuring Security

In the Administration Console Security Tab, you can adapt the default security settings to your local environment.

Security can either be configured internally in the WebFOCUS BUE repository, or externally in a Microsoft Active Directory or an LDAP directory that is not part of WebFOCUS BUE. To accommodate this, you can use either the Internal page or the External page on the Security tab in the Administration Console.

Internal authentication and authorization is enabled by default.

Using Internal Security

Internal authentication and authorization is enabled by default. Optionally, you can use the settings in the Internal page to configure password policies.

Maximum Sign-In Attempts (IBI_Max_Bad_Attempts)

Specifies the number of unsuccessful sign-in attempts allowed before the account status is changed to locked. The default value is 0, which allows unlimited attempts.

Lockout Duration (Minutes) (IBI_Account_Lockout_Duration)

Specifies the number of minutes before the status of an account changes from locked to active. The default value is 0 (off).

Lockout Duration Reset (Minutes) (IBI_Account_Lockout_Duration_Reset)

Specifies the number of minutes that must elapse after the number of failed sign-in attempts specified by the Maximum Sign-in Attempts setting before the allowed sign-in attempt counter is reset to 0. The available range is from 1 to 99,999 minutes. The default value is 0 (off).

Enforce Password Complexity (IBI_Password_Complexity)

Specifies password requirements. The default value is False (check box clear).

If set to True (check box selected), passwords must:

- ☐ Not contain the user account name or parts of the full name of the user that exceed five consecutive characters.
- ☐ Be at least six characters long or at least the number of characters specified in Minimum Password Length, whichever is greater.
- ☐ Contain characters from three of the following four categories:
 - ☐ Uppercase English characters (A through Z).
 - ☐ Lowercase English characters (a through z).
 - ☐ Base 10 digits (0 through 9).
 - ☐ Non-alphabetic characters (for example, !, \$, #, %).

- ☐ Complexity requirements are enforced when passwords are changed or created.

Minimum Password Length (IBI_Password_Minimum_Length)

Defines the required minimum length of a password. The default value is 0 characters.

Password Reuse (IBI_Password_Reuse)

Specifies the number of recent passwords that cannot be reused. The default value is 0 changes. If Password Reuse is set to 25, for example, WebFOCUS BUE will track the 25 most recent password changes.

Days Until Password Expires (IBI_Password_Expire)

Specifies the number of days that a password will remain active. The default value is 0, which prevents passwords from expiring. Once the password has expired, the user must take the action specified by the IBI_Password_Expire_Action setting.

Days Until Password Expiration Warning (IBI_Password_Expire_Warning)

Specifies the number of days prior to expiration that a warning will be displayed to the user. The default value is 0, which provides no warning. The value should be less than or equal to the IBI_Password_Expire setting.

Password Expiration Result (IBI_Password_Expire_Action)

Specifies the action required when a password expires. You can choose one of the following options:

- ☐ *To force users with expired passwords to change their passwords before signing in. (MUSTCHANGE) This is the default value.*
- ☐ *Change the status of users with expired passwords to inactive. Such users cannot sign in until an administrator resets the password in the Security Center. (DISABLE-USER)*

Using External Security

Use the External page if you have elected to configure security in a Microsoft Active Directory (AD) or Lightweight Directory Access Protocol (LDAP) directory that is not part of WebFOCUS BUE.

Enable External Security

When you select this check box, internal security settings are overridden. WebFOCUS BUE directs all authentication activities and approvals to the external system you identify on this page.

External Security Type (IBI_Authentication_Type)

The drop-down list box for this field contains the following values:

- ☐ **Reporting Server.** Authenticates users against an AD or LDAP directory.
- ☐ **Legacy LDAP.** This value is not used with the Business User Edition.
- ☐ **Custom Java Plug-In.** This value is not used with the Business User Edition.

Understanding Custom Settings

The Custom Settings page allows you to customize WebFOCUS BUE by typing customized values for standard settings.

When you save updates to settings that you type into the Customized Setting text box, they are transferred to the site.wfs file, located at `drive:\ibi\WebFOCUS_BUE82\WebFOCUS\client\wfc\etc\`. When you use this page to assign new values to settings, they override the default values assigned to them. These overrides are carried over as you upgrade to new versions.

After you save a custom setting, the text continues to display on this page. You can use comments to identify specific updates and additional information about them.

Procedure: How to Configure Custom Settings

Only a manager can configure settings on the Custom Settings page.

1. In the BUE Portal, on the Menu bar, click *Administration*, and then click *Administration Console*.
2. On the Configuration tab, click *Custom Settings*.
3. Under the final comment statement at the top of the Custom Settings text box, or the most recent custom setting entry, type the variables, settings, commands, or comments that comprise the custom settings.

Use the format required by the application or operating system that will execute the command.

To help track changes to custom settings, use comments to identify and separate individual changes.

4. To store your custom settings in an encrypted format, select the *Encrypt* check box.

Note: Even when you select this check box, settings continue to appear in an unencrypted format in the Custom Settings text box.

5. When your configuration is complete, click *Save*.
6. When you receive a confirmation message, click *OK*.
7. When the Custom Setting page clears, click *Custom Settings* under the Application Settings folder to see your updated comments, settings, or commands in the Custom Settings text box.

Procedure: How to Configure Collation Sequence Settings

Only a manager can configure settings on the Custom Settings page.

1. In the BUE Portal, on the Menu bar, click *Administration*, and then click *Administration Console*.
2. On the Configuration tab, click *Custom Settings*.
3. Under the final comment statement at the top of the Custom Settings text box, type the comment line:

```
# Collation Sequence Settings
```

4. Under the comment line, type the command:

```
_site_profile=&_site_profile\nSET COLLATION={BINARY|SRV_CI|SRV_CS|  
CODEPAGE}
```

where:

BINARY

Bases the collation sequence on binary values.

SRV_CI

Bases the collation sequence on the LANGUAGE setting, and is case-insensitive.

SRV_CS

Bases the collation sequence on the LANGUAGE setting, and is case-sensitive.

CODEPAGE

Bases the collation sequence on the code page in effect, and is case-sensitive.

CODEPAGE is the default value. In most cases, CODEPAGE is the same as BINARY.

The only differences are for Danish, Finnish, German, Norwegian, and Swedish in an EBCDIC environment.

5. To store your custom settings in an encrypted format, select the *Encrypt* check box.
Note: Your settings will continue to appear in the Custom Settings text box in an unencrypted format.
6. When your configuration is complete, click *Save*.
7. When you receive a confirmation message, click *OK*.
8. When the Custom Setting page clears, click *Custom Settings* under the Application Settings folder to see your updated comments, settings, or commands in the Custom Settings text box.

Understanding NLS Settings

You can use the Administration Console to configure National Language Support and enable the Dynamic Language Switch.

Separate message files exist for every national language that WebFOCUS supports. If you want to customize the set of characters used in your report output, you must select the code page for every language you use.

These settings do not carry over during updates. You must repeat this customization step for each new release that you install.

Procedure: How to Configure National Language Support

1. In the Administration Console, on the Configuration tab, click *NLS Settings*.
2. On the NLS settings page, click the option for the operating system on which the WebFOCUS Client resides.

The list adjusts to display the code pages that are available to the selected operating system.

3. From the list, select a code page that configures the client for the correct display of report output in the browser.

Tip: The language selected for the Client usually corresponds to the language selected for the Server from the Reporting Server Console.

If the language chosen from the Reporting Server Console does not appear in the list, click *User Defined Code Page* and type the number of the user-defined code page.

Use this option, for example, when the server adds support for a new code page that is not yet reflected in the client software.

In the following sample configuration window, the administrator specified code page 437.

Select the operating system where the WebFOCUS Client resides

- ☒ User Defined Code Page
- ☐ Windows, UNIX and AS/400
- ☐ OS/390

Save

Cancel

Unicode (UTF-8) is available for the Windows, UNIX, or AS/400 operating systems.

4. Click Save to store your NLS settings.

The Administration Console will generate and update the client configuration file (nlscfg.err), found in *drive:\ibi\WebFOCUS_BUE82\WebFOCUS\client\home\etc*, with the CODE_PAGE setting. Note that if you click NLS Settings again, your new setting is highlighted as the active code page.

Reference: WebFOCUS BUE Client Code Page Settings

The following code page settings are available:

- ☐ * 137 - U.S. English/Western European
- ☐ 874 - Thai
- ☐ * 942 - Japanese
- ☐ * 946 - Simplified Chinese
- ☐ 949 - Korean
- ☐ 1250 - Eastern European
- ☐ 1251 - Russian
- ☐ * 1252 - Western European
- ☐ 1253 - Greek
- ☐ 1254 - Turkish

- ☐ 1255 - Hebrew
- ☐ 1256 - Arabic
- ☐ 1257 - Baltic
- ☐ * 10942 - Japanese EUC
- ☐ * 10948 - Traditional Chinese
- ☐ * 65001 - Unicode (UTF-8)

Note: Only those code page settings marked with an asterisk are fully supported in the current release.

Customizing the Dynamic Language Switch

You can customize the languages that are made available on the sign in pages by activating the Dynamic Language Switch.

Procedure: How to Customize the Dynamic Language Switch

1. On the Administration Console, on the Configuration tab, under the Application Settings folder, click *Dynamic Language Switch*.

The Dynamic Language Switch window appears with a list of the languages made available by the code page selected in the NLS Settings page. By default, the Enable Dynamic Language check box is unselected, and all of the language check boxes are deactivated.

The Dynamic Language Switch window also shows the Client Code Page setting specified in [How to Configure National Language Support](#) on page 115.

2. Select the *Enable Dynamic Language* check box to activate the check boxes for all of the available languages displayed in the panel, as shown in the following image:

Client Code Page: 65001 - Unicode (UTF-8)

☒ Enable Dynamic Language

<input checked="" type="checkbox"/>	Locale	Language Code
<input checked="" type="checkbox"/>	Chinese - Simplified GB	zh
<input checked="" type="checkbox"/>	Chinese - Traditional Big-5	tw
<input checked="" type="checkbox"/>	English	en
<input checked="" type="checkbox"/>	French - Canadian	fc
<input checked="" type="checkbox"/>	French - Standard	fr
<input checked="" type="checkbox"/>	German	de
<input checked="" type="checkbox"/>	Italian	it
<input checked="" type="checkbox"/>	Japanese	ja
<input checked="" type="checkbox"/>	Portuguese - Brazilian	br
<input checked="" type="checkbox"/>	Spanish	es

Selecting the Enable Dynamic Language check box and one or more languages activates the display of the Select Languages button on all of the sign in pages. It also activates the Language menu on the BUE Portal Menu bar.

3. Select the check box next to the Locale heading if you want all of the languages to appear in the Select Languages drop-down list on the sign in pages and in the Language menu.

OR

4. Clear the check box next to the Locale heading and select the individual check boxes next to the individual languages that you want to appear on the sign in pages and in the Language menu.
5. Click Save to save your changes.

Note: To remove individual languages from the Select Languages drop-down list on the sign in pages, clear the check boxes next to the languages you want to remove.

Understanding Redirection Settings

You can view or edit redirection settings for the WebFOCUS Client through the Redirection Settings area of the Administration Console. However, you should not alter Redirection Settings without consulting Customer Support Services.

Redirection allows users to save report output in a temporary directory when a request is executed. Then an HTTP call is made from the browser to retrieve the temporary stored output for display in the browser.

If redirection is turned off, the report output displays in the browser immediately after the request is executed.

To change redirection settings in the Administration Console, click the *Configuration* tab and then *Redirection Settings*. The Redirection Settings panel opens, as shown in the following image.

Redirection Settings

WebFOCUS Extension	Content Type	Format	Redirect	Server Extension	Save Report	Client Extension	IBFS Format
.acx	text/plain	ascii	no ▼	ACCESS	no ▼	.acx	ascii
.bmp	image/bmp	binary	no ▼	BMP	no ▼	.bmp	binary
.cfg	text/cfg	ascii	no ▼	N/A	no ▼	.cfg	ascii
.class	java/*	binary	no ▼	N/A	no ▼	.class	ascii
.css	text/css	binary	no ▼	CSS	no ▼	.css	ascii
.csv	application/csv	ascii	yes ▼	N/A	no ▼	.csv	ascii
.dif	application/x-dif	ascii	yes ▼	N/A	no ▼	.dif	ascii
.doc	application/msword	ascii	yes ▼	DOC	no ▼	.doc	ascii
.docx	application/vnd.openxml	binary	no ▼	DOCX	no ▼	.docx	binary
.e97	application/vnd.ms-excel	ascii	no ▼	N/A	no ▼	.e97	ascii
.err	text/plain	ascii	no ▼	ERRORS	no ▼	.err	ascii
.fex	text/plain	ascii	no ▼	FOCEXEC	no ▼	.fex	ascii
.foc	application/foc	binary	no ▼	FOCUS	no ▼	.foc	binary
.for	text/plain	ascii	no ▼	N/A	no ▼	.for	ascii
.ftm	application/x-ftm	ascii	no ▼	FOCTEMP	no ▼	.ftm	ascii
.gfa	application/gfa	binary	no ▼	N/A	no ▼	.gfa	binary
.gif	image/gif	binary	no ▼	GIF	no ▼	.gif	binary
.hex	text/plain	ascii	no ▼	N/A	no ▼	.hex	ascii

Procedure: How to Change Redirection Settings

It is recommended that you do not change these settings unless instructed to do so by Customer Support Services.

1. In the Administration Console on the Configuration tab, click *Redirection Settings*.
2. In the Redirect column, select yes on a row to redirect the output to a temporary directory for the specified extension.

3. In the Save Report column, select yes to prompt users in the browser to open or save report output. When Save Report is set to yes, the report output retains the Save As name, if specified in the request.

For example, specifying `ON TABLE PCHOLD AS MYREPORT FORMAT PDF` in a request and setting Save Report to yes for the .pdf extension enables a user to open or save the output as MYREPORT.pdf. The Save As name specified is returned to the browser in uppercase. If Save Report is set to yes and no Save As name is specified in the request, a random file name is generated.

Important: You must do the following to use the Save Report functionality for GRAPH requests (specified with a PNG, SVG, GIF, JPEG, or JPG format in the procedure):

- ☐ Set Save Report to yes for the .htm Extension.

Running a server-side GRAPH request creates an HTM file that contains a link to the actual graph output, which is stored as a temporary image file with a .jpeg, .jpg, .gif, .svg, or .png extension.

- ☐ When you execute a GRAPH request, if you select the Save option when prompted to open or save the output, the output is saved to an HTM file using only a reference to the graph image, which will eventually expire and be deleted from the server (according to the temporary file expiration settings in the Client Configuration).
- ☐ To preserve the output of the GRAPH request, open the saved HTM file, right-click the graph image, and select *Save Picture As* to save it to disk permanently. You can then substitute an absolute reference to the saved image file in the returned HTM output file.

4. If you want to encrypt the redirection settings, select the Encrypt check box at the bottom of the screen.
5. Click Save to save your changes in the Redirection Settings panel.

Changing Application Settings

The Application Settings enables Managers to update configuration settings that are required by various components of the WebFOCUS BUE web application. You can view or edit the application settings through the Administration Console.

Initially, WebFOCUS BUE reads the configuration file to locate parameters and their values. If a parameter does not exist in the file, the BUE will then obtain the parameter and its value from the web.xml file of the web application currently deployed by the BUE.

Reference: Understanding Change Management Settings

The Change Management Settings determine which files are exported during the change management process, the format of the exported file, and whether some legacy functionality is preserved.

File Types Included in Export Package (IBI_CM_EXPORT_WFRS_FILE_EXTENSIONS)

Determines, by file extension, which files from the WebFOCUS Reporting Server or the web will be included in exports created by the change management feature. The values included by default are acx, bmp, css, fex, gif, htm, html, ico, jpeg, jpg, mas, png, sty, and svg.

Retain Handles (IBI_CM_RETAIN_HANDLES)

This option is not relevant to the BUE.

Zip Change Management Package (IBI_CM_ZIP)

When selected (True), the export package is compressed and delivered in a zip file.

Name format of Zip export files (IBI_CM_ZIP_FILE_FORMAT)

Select an option from the drop-down menu to specify the name format of the zip file.

Verify Signature on Import (IBI_CM_ZIP_VERIFY_SIGNATURE)

When selected (True), the export package is flagged for signature verification, which ensures both that the code has not been tampered with and that the signature was produced with the expected key.

Reference: Understanding Deferred Reporting Settings

Deferred Reporting Settings determine how deferred reports are processed.

Prompt for Custom Deferred Report Description (IBI_DEFERRED_CUSTOM_DESCRIPTION)

When this check box is selected, the default setting, users are prompted to optionally customize the description for the deferred report, which defaults to the title of the report being run deferred. This occurs whenever the description defined in the parameter (IBIMR_defer_description) has not been submitted with the run deferred report request.

When this check box is clear, the title of the report being submitted to run deferred is assigned to the deferred report description automatically, and no prompt appears.

Display Deferred Request Submitted Notification (IBI_DEFERRED_NOTIFY_SUBMITTED)

When this check box is selected, the default setting, the Deferred Request Submitted window displays to confirm a successful deferred request. The user clicks OK to close the window.

When this check box is clear, the Deferred Request Submitted window does not display.

Display Deferred Ticket Delete Confirmation (IBI_DEFERRED_TICKET_DELETE_CONFIRM)

Activates an automated message that prompts the user to confirm the deletion of a deferred report. When this check box is selected, which is the default, a message prompts the user to confirm the deletion, so a deletion requires two clicks. When this check box is clear, the user is not prompted to confirm the deletion, so a deletion requires only one click. Making a large number of deletions is faster when suppressing the confirmation message.

***Reference:* Understanding ESRI Settings**

The ESRI Settings page defines the connection to the local application that supports ESRI-based maps.

[ESRI On Premise \(IBI_ESRI_ON_PREMISE\)](#)

This information is not yet available.

***Reference:* Understanding Parameter Prompting Settings**

The Parameter Prompting Settings determine parameter prompting behavior in the WebFOCUS BUE Client.

Managed Reporting (IBIMR_PROMPTING)

Enables or disables parameter prompting for all Managed Reporting requests. Possible values are:

- ☐ **Off.** Turns off parameter prompting at the site level.
- ☐ **Run with Default Values. (XMLRUN)** Prompts for amper variables created with the - DEFAULT command and any other amper variable that does not have a value. This is the default value.
- ☐ **Always Prompt. (XMLPROMPT)** Prompts for amper variables created with the - DEFAULT command when there is another amper variable that does not have a value assigned.

Managed Reporting when Prompt Parameters Property Unset (IBIMR_PROMPTINGUNSET)

Enables or disables parameter prompting for Managed Reporting procedures (FEXes) when IBIMR_prompting is set to XMLPROMPT or XMLRUN, and the Prompt for Parameters setting is unchecked in the FEX Properties dialog box. Possible values are:

- ☐ **OFF.** Turns off parameter prompting.

- ☐ **Run with Default Values. (XMLRUN)** Prompts for amper variables created with the - DEFAULT command and any other amper variable that does not have a value. This is the default value.
- ☐ **Always Prompt. (XMLPROMPT)** Prompts for amper variables created with the - DEFAULT command when there is another amper variable that does not have a value assigned.

Self Service (IBI_WFDESCRIBE_DEFAULT)

Enables or disables amper auto prompting for self-service reporting. Possible values are:

- ☐ **OFF.** Turns off auto prompting. This is the default value.
- ☐ **Run with Default Values. (XMLRUN)** Prompts for amper variables created with the - DEFAULT command and for any other amper variable that does not have a value.
- ☐ **Always Prompt. (XMLPROMPT)** Only prompts for amper variables created with the - DEFAULT command when there is another amper variable that does not have a value assigned and, therefore, will be prompted for.
- ☐ **Display XML (Debug with syntax error checking). (XML)** The XML document describing the amper variables is displayed in the browser. This setting is used internally, and is recommended for debugging and syntax error checking purposes only.
- ☐ **Display XML (Debug). (XMLCHECK)** The XML document describing the amper variables is displayed in the browser. This setting is used internally, and is recommended for debugging purposes only.

Note: Managed Reporting uses a separate variable setting, which is IBIMR_prompting.

Default Template (IBI_WF_DESCRIBE_HTML)

The HTML template that defines the auto prompt layout.

Null Behavior (IBIF_DESCRIBE_NULL)

Specifies the value (_FOC_NULL or FOC_NONE) that the client assigns (in a -SET command) to the amper variable when the dynamic multi-select list *No Selection* value is selected. The default value is _FOC_NULL.

Reference: Understanding Text Generation Server Settings

The Text Generation Server Settings define the connections to an independent server that provides narrative descriptions for chart headers, footers, and tool tips.

Text Generation Server URL (IBI_TEXT_GENERATION_SERVER_URL)

This information is not yet available.

Text Generation Server Key (IBI_TEXT_GENERATION_SERVER_KEY)

This information is not yet available.

Changing Client Settings

The client configuration settings are grouped into categories under the Configuration menu in the WebFOCUS console. The term *Initial Value* next to a setting means that the value shown initially is the installation default value and that it can be overridden by setting the variable explicitly in the URL request.

InfoAssist+ Properties

Settings in the InfoAssist+ Properties page of the Administration Console determine the display and use of the InfoAssist+ tool that opens when Advanced Users, Developers, or Managers create or update content.

To enable or disable reporting options for the InfoAssist+ tool, click *Utilities*, scroll down to the bottom of the Configuration tab menu, and then click *InfoAssist+ Properties*.

Reference: Understanding InfoAssist+ Home Tab Properties

The InfoAssist+ Home tab enables you to control the most commonly used properties and options from the Format, Design, Filter, and Report groups. These properties are:

Use Live Preview Mode

Determines whether InfoAssist+ opens in the Live Preview mode or the Query Design View by default. When Yes is selected, InfoAssist+ opens in the Live Preview mode as the default. When Yes is not selected, InfoAssist+ starts with the Query Design View. If *Allow User Override* is checked for this option, users can change the setting specified by the Manager.

Record Limit

Enables the Record Limit menu of the Home tab. If *Show* is not selected, the Record Limit menu is removed from the InfoAssist+ interface.

Themes

Provides InfoAssist+ users with various color-coded StyleSheet themes that can be used to style reports and charts. Users can select standard InfoAssist+ themes, or select customized cascading style sheet themes created by your organization.

Page Heading

Enables the Head/Foot menu of the Home tab. InfoAssist+ users can use the Head/Foot menu to add a heading or footing to each page of the report output.

Report Heading

Enables the Head/Foot menu of the Home tab. InfoAssist+ users can use the Head/Foot menu to add a heading or footing to the first page of the report output.

Reference: Understanding InfoAssist+ Format Tab Properties

For reports or charts, InfoAssist+ displays a list of output file format options, such as, HTML, PDF, or Excel, in the Format Group of the Home tab. Other options that make additional layouts and display features available when creating a report or chart appear on the Format tab itself. You can control the display of both types of options through the settings contained in this section. The settings that affect the Format tab display are InfoMini Run Immediate, Other Chart Types, Pages on Demand, Stack Measures, User Selection.

Note: Settings in this section do not affect the display of Format tab features for visualizations.

active report Format

Enables the use of the HTML active report format. An HTML active report is a self-contained report that is designed for offline analysis. It contains all of the data and JavaScript within the HTML output file and it includes analysis options, such as filtering, sorting, and charting.

Select the check box to ensure that this option appears in the drop-down lists for the Report Output Format, Chart Output Format, and Document Output Format properties under the Tool Options Dialog Defaults section.

Additional HTML Formats for Chart

Enables the use of the PNG, JPEG, GIF, and SVG output formats. The default value is PNG. PNG is not available as a format for chart output.

Additional PDF Formats for Chart

Enables the use of the PDF/SVG and PDF/GIF output formats. The default value is PDF/SVG.

Excel 2000 Format

Enables the use of the Excel 2000 spreadsheet output format. The Excel 2000 format supports most StyleSheet attributes, allowing for full report formatting. The computer on which the report displays must have Microsoft Excel 2000 installed.

When this check box is selected, this output format option is available to select in the Output Format drop-down menus in the Tool Options Dialog Defaults section.

This check box is selected, by default.

Excel 2000 Formula

Enables the use of the Excel 2000 formulas when the *Excel 2000 Format* option is selected.

This check box is selected, by default.

Excel 2007 Format

Enables the use of the Excel 2007 spreadsheet output format. The computer on which the report displays must have Microsoft Excel 2007 installed.

When this check box is selected, this output format option is available to select in the Output Format drop-down menus in the Tool Options Dialog Defaults section.

This check box is selected, by default.

Excel 2007 Formula

Enables the use of the Excel 2007 formulas when the *Excel 2007 Format* check box is selected.

This check box is selected, by default.

Excel Pivot

Enables the use of the Excel 2000 PivotTable output format. PivotTable is an Excel tool for analyzing complex data, much like OLAP.

This check box is clear, by default.

HTML Format

Enables the use of the HTML page report format.

Select the check box to ensure that this option appears in the drop-down lists for the Report Output Format, Chart Output Format, and Document Output Format properties under the Tool Options Dialog Defaults section.

InfoMini Run Immediate

If *Enable* is selected, reports run immediately when InfoMini first launches. This setting is enabled by default.

Other Chart Types

Allows the creation of more complex graph output types, such as Spectral Maps, Gauge Charts, and Pareto Charts.

Pages on Demand

Enables the display of report output one page at a time. InfoAssist+ users can use the navigation menu at the bottom of the output screen to view each page. This option is activated only when HTML or active report output format is selected.

PDF Format

Enables the use of the PDF report format.

Select the check box to ensure that this option appears in the drop-down lists for the Report Output Format, Chart Output Format, and Document Output Format properties under the Tool Options Dialog Defaults section.

PowerPoint 2000 Format

Enables the use of the PowerPoint® 2000 document output format. The computer on which the report appears must have Microsoft PowerPoint 2000 or higher installed.

Select the check box to ensure that this option appears in the drop-down lists for the Report Output Format, Chart Output Format, and Document Output Format properties under the Tool Options Dialog Defaults section.

PowerPoint 2007 Format

Enables the use of the PowerPoint® 2007 document output format. The computer on which the report appears must have Microsoft PowerPoint 2007 or higher installed.

Select the check box to ensure that this option appears in the drop-down lists for the Report Output Format, Chart Output Format, and Document Output Format properties under the Tool Options Dialog Defaults section.

Stack Measures

Displays all numeric measure field names in the first column of the report output, with the corresponding numeric data values displayed across time in a column for each selected time period. The Stack Measures feature is activated only when HTML, Excel, or PowerPoint output format is selected. If *Allow User Override* is checked for this option, users can change the setting specified by the Manager.

User Selection

Allows users to change the output type of their reports at run time.

Reference: Understanding InfoAssist+ View Tab Properties

Enables InfoAssist+ users to customize the view of different report components in the InfoAssist+ tool, such as the design mode, output location, and data view. You can configure the following properties in the InfoAssist+ View tab:

Display View Tab

Enables the View tab and all of its menu options. If this is not selected, the View tab is removed from the InfoAssist+ interface.

Data Panel

Allows the user to customize Data Panel settings. Values are *Logical* (default), *List*, and *Structured*.

Query Panel

Allows the user to customize the view of the query components, such as Filters, Column and Row labels, and Measures when building a report. Values are *Tree* (default), *Area 2x2* (2 columns by 2 rows), *Area 1x4* (1 column by 4 rows). If *Allow User Override* is checked for this option, users can change the setting specified by the Manager.

Reference: Understanding InfoAssist+ Tool Options Dialog Defaults Properties

Enables Managers to specify default tool settings. If *Allow User Override* is checked for an option, users can change the setting specified by the Manager. However, the Manager cannot specify a default value that has already been disabled in one of the other groups. For example, if you have disabled the active report format in the Format Tab section, you will see an error message if you attempt to set that format as a default Compose Output Format in the Dialog Defaults section.

Report Output Format

Sets the default format for reports. Valid values are *HTML*, *active report*, *PDF*, *EXL07*, *EXL2K*, *PowerPoint 2000*, *PowerPoint 2007*. To ensure that these options are available, a Manager must select the check box for each under the Format Tab section. The default value is *HTML*.

Chart Output Format

Sets the default format for charts. Valid values are *HTML*, *HTML5*, *active report*, *PDF*, *EXL2K*, *PowerPoint 2000*, and *PowerPoint 2007*. To ensure that these options are available, a Manager must select the check box for each under the Format Tab section. The default value is *HTML*.

Document Output Format

Sets the default format for documents that are generated in InfoAssist+. Valid values are *HTML*, *active report*, *PDF*, *EXL2K*, *PowerPoint 2000*, and *PowerPoint 2007*. To ensure that these options are available, a Manager must select the check box for each under the Format Tab section. The default value is *active report*.

Page Orientation

Sets the default page orientation for reports and charts. Valid values are *Portrait* and *Landscape*. The default value is *Portrait*.

Page Size

Sets the default page size for reports and charts. Valid values are *A3*, *A4*, *A5*, *Letter*, *Tabloid*, *Legal*, *PPT-SLIDE*, and *Large Size*. The default value is *Letter*.

Data Preview Method

Sets the default action for whether reports are previewed using sample data or actual data from the data source. Valid values are *Sample* and *Live*. The default value is *Live*.

Record Limit

Sets the default maximum number of rows retrieved from the data source when Interactive Design view is selected. This feature is useful in reducing response time if users are working with a large amount of data. It is applicable only when developing the report. The record limit setting will not affect the report output at run time. Valid values are 0 to 10,000 rows. The default value is 500 rows.

Output Target

Sets the default location for reports and charts. Valid values are *Single tab*, *New tab*, *Single window*, and *New window*. The default value is *Single tab*.

InfoAssist+/Portal StyleSheet

Sets the style sheet to be used for InfoAssist+ and the Portal. Click *Change Stylesheet* to open the Browse predefined template files window.

Visualization StyleSheet

Sets the style sheet to be used when creating visualizations. Click *Change Stylesheet* to open the Browse predefined template files window.

Encode HTML

Encodes script tags within data, so that the tags are replaced and not executable in a browser. The default value is *Yes*. This includes the `ON TABLE SET HTMLENCODE ON` command in the procedure.

Enable Pages On Demand

Allows InfoAssist+ users to view report output one page at a time. The user can use the navigation menu at the bottom of the output screen to view each page. This option is activated only when HTML or active report output format is selected.

Rows retrieved from cache

Establishes how many rows of cached data stored in a binary file are returned to the output window at one time. The default value is 100 rows.

***Reference:* Understanding InfoAssist+ File Options**

Determines which of the following file types can be selected by InfoAssist+ users when creating and saving HOLD files:

Binary

Stores report or chart data as binary numbers in numeric fields. Binary files use the extension (*.ftm).

FOCUS

Stores report or chart data as text in a segment structure that conforms to FOCUS database requirements. FOCUS files use the extension (*.foc).

Comma Delimited with Titles

Stores report or chart data as text in sequence by field. Alphanumeric fields are enclosed in quotation marks. Fields are separated by commas and are preceded by Field Names. Comma Delimited with Titles files use the extension (*.csv) (Comma Separated Values).

Plain Text

Stores report or chart data as text in sequence by field without delimiters or field names. Plain Text files use the extension (*.ftm).

Tab Delimited

Stores report or chart data as text in sequence by field. Fields are separated by tab characters. Tab Delimited files use the extension (*.tab).

Tab Delimited with Titles

Stores report or chart data as text in sequence by field. Fields are separated by tab characters, and are preceded with field names. Tab Delimited with Titles files use the extension (*.tab).

Database Table

Stores report or chart data as text in a field structure that conforms to a Structured Query Language (SQL) Database format. Database Table files use the extension (*.sql).

Database Table output is only available when working against an SQL database.

Hyperstage

Stores report or chart data as text in a field structure that conforms to the Hyperstage database table format. Hyperstage files use the extension (*.bht).

Hyperstage output is only available when the reporting server has a Hyperstage adapter configuration.

SQL script

Stores report or chart data as text in a sequential field structure that can be imported into a database table that conforms to the Structured Query Language (SQL) Database format. SQL Script files use the extension (*.sql).

SQL Script output is only available when working against an SQL database.

XML

Stores report or chart data as text in a field structure that conforms to the rules of the Extensible Markup Language. Fields are separated by tags that identify content. XML files use the extension (*.xml).

Understanding InfoAssist+ Auto Drill Properties

Settings in this section enable the use of drill-down navigation options, which are part of the Auto Drill functionality.

Single Click Navigate

Enables the use of single click navigation, which is an automatic drill down to the next level of a dimension within the body of a report or chart made in response to a single click on a top-level entry or feature.

By default, this check box is not selected, meaning that single click navigation is disabled, and top-level Auto Drill entries or features display the Drilldown menu in response to a single click. If this check box is selected, single click navigation is enabled, and instead of displaying the Drilldown menu, top-level Auto Drill entries or features automatically refresh the report or chart with results based on the next lower level of your selected dimension in response to a single click.

Breadcrumbs

Enables the display of a breadcrumb trail at the top of an Auto Drill report or chart.

By default, this check box is selected, and Auto Drill reports and charts display a breadcrumb trail. If this check box is cleared, Auto Drill reports and charts do not display a breadcrumb trail.

In an Auto Drill report or chart, a breadcrumb trail displays a series of links to previous versions that were generated as you drilled through each level of your selected dimension to reach the version currently on display.

Restore Original

Enables the display of the Restore Original option in the Drilldown menu.

By default, this check box is selected, and the Restore Original option appears in the Drilldown menu. If this check box is cleared, the Restore Original option does not appear in the Drilldown menu. In an Auto Drill report or chart, the Restore Original option returns you directly to the original version.

Drill Up

Enables the display of the Drill up option in the Drilldown menu.

By default, this check box is selected, and the Drill up option appears in the Drilldown menu. If this check box is cleared, the Drill up option does not appear in the Drilldown menu. In an Auto Drill report or chart, the selection of the Drill up option refreshes the display with results based on the next level above the current level of your selected dimension.

Drill Down

Enables the display of the Drill down option in the Drilldown menu.

By default, this check box is selected, and the Drill down option appears in the Drilldown menu. If this check box is cleared, the Drill down option does not appear in the Drilldown menu. In an Auto Drill report or chart, the selection of the Drill down option refreshes the display with results based on the next level below the current level of your selected dimension.

Note: In addition to disabling the Drill down option, clearing this setting also removes hyperlinks from top level report entries and the breadcrumb trail display from reports and charts. If the Single Click Navigate setting is also cleared, clearing the Drill Down setting effectively disables Auto Drill navigation tools in reports and charts that contain only the top level of a dimension value in their design. If the Single Click Navigate setting is selected, and the report or chart contains entries below the top level, clearing the Drill Down setting shifts the Single Click Navigation feature to those lower-level entries. However, because this setting also suppresses the display of the Drilldown menu, users will neither be able to restore the original version of the report or chart, nor will they be able to drill back up to a higher level.

Reference: Understanding InfoAssist+ Miscellaneous Options**Use two-part file name**

If selected, this option requires the use of two-part file names, which specify the path to the Master File location. If not selected, a one-part file name must be used instead. The default value is selected.

Expand Data Source Tree

Determines whether the initial view of the data source tree is expanded or collapsed. If selected, the tree is expanded. If not selected, the tree is collapsed. The default value is selected.

Join Tool

Displays the Join menu option on the InfoAssist+ Data tab. If not selected, the Join menu option is removed from the Data tab. The default value is selected.

Layout Tab

Enables the Layout tab in the InfoAssist+ control panel. If not selected, the Layout tab is removed from the InfoAssist+ control panel. The default value is selected.

Series Tab

Enables the Series tab in the InfoAssist+ control panel. The Series tab displays only when working with chart queries. It provides access to charting properties and options in the Properties, Line, and Pie menus. If not selected, the Series tab is removed from the InfoAssist+ control panel. The default value is selected.

Reporting Server Configuration Settings

Reporting Server configuration settings are available on the Configuration tab of the Administration Console. To view them you must expand the Reporting Server folder and the Server Connections folder, and click the Reporting Server icon.

Reference: Reporting Server Node Properties

The Reporting Server Node properties from the Basic pane are explained below.

Basic**Node Name**

The logical name of the node. The name cannot be the same as any other node name. It must begin with a letter and cannot be more than eight characters. The Client will use this name when it accesses this server.

Node Description

Optional. The description of the node that appears in the Configuration pane. If this is omitted, the node name will be used.

Host

The Host name or IP address of the Server.

TCP/IP Port

The Port number for the TCP listener. The default port is 8120.

HTTP(S) Port

The Port number for the HTTP listener. This is typically one port after the TCP/IP port.

The default HTTP port is 8121.

Security

The security options for the reporting server connection.

- ☐ **Prompt for Credentials.** This option is not relevant for the BUE.
- ☐ **HTTP Basic.** This option is not relevant to the BUE.
- ☐ **Kerberos.** This option is not relevant for the BUE.
- ☐ **SAP Ticket.** This option is not relevant to the BUE.
- ☐ **Service Account.** This option is not relevant to the BUE.
- ☐ **Trusted.** Allows you to connect to the Reporting Server with only a user ID. This option is useful when no password is available for the user. Controls should be placed on the Server to ensure that connections from unauthorized clients are rejected. For example, you can employ the Reporting Server RESTRICT_TO_IP setting or configure a network firewall so that only a particular client can connect to the Server.

Note: When configuring the Client to make trusted connections to the Reporting Server, you must also enable the Reporting Server to accept trusted connections.

Advanced

The Reporting Server Node properties from the Advanced pane are explained below.

Service Name

Description for the Reporting Server node. This description displays to end users.

Use HTTPS

Enables encrypted communication between the Client and the Reporting Server HTTP listener. The default value is off.

This option must be selected if the Reporting Server HTTP listener is configured to use SSL. If you are using a self-signed certificate to enable HTTPS communication with a Reporting Server, the certificate must be configured in the Java environment where the Client is installed. This enables HTTPS communication between the Reporting Server and the Administration Console.

Compression

Enables data compression. By default, data compression is disabled.

Encryption

Sets data encryption ability and the cryptography symmetric method used.

Select one of the following options from the drop-down list:

- ☐ **0.** Off. This is the default value.
- ☐ **AES.** Advanced Encryption Standard. The AES selections are in the format

CIPHER(x) (-MODE)

where:

CIPHER

Is AES128, AES192, AES256.

x

Is optional and defines an RSA key length of 1024 bits. When this is not specified, the RSA key is 512 bits.

CBC

Is optional and defines the use of Cipher Block Chaining (CBC) mode. When the mode is not specified, Electronic Code Book (ECB) is used.

For example, AES256x-CBC is the AES256 cipher with a 1024-bit RSA key in CBC mode. AES128 is the AES128 cipher with a 512-bit RSA key in ECB mode.

Connect Limit

Specifies the number of seconds that the Client will hold the pending connection. Other possible values are 0 (no wait) and -1 (infinite wait). The default value is -1.

Maximum Wait

Specifies the time, in seconds, that the Client will wait before timeout. You can optionally specify different return times for the first row and other rows. A single number indicates the return time is valid for any row. If two numbers are separated by a comma, the first number specifies the return time for the first row and the second number specifies the return time for the subsequent rows. The default value is -1, which indicates an infinite wait time.

Security Object

For any security option, a Manager can specify one or more HTTP header names and/or cookie names as follows:

- ☐ **Cookie.** Specify each HTTP cookie name separated by a comma (,). For example:

cookie_name1, cookie_name2

- ☐ **Header.** Specify each HTTP header name separated by a comma (,). For example:

header_name1, header_name2

Note:

- ☐ HTTP cookie and header names must not contain commas (,) or colons (:). These are reserved delimiters.
- ☐ REMOTE_USER is a special type of HTTP header variable whose contents will not be sent to the Reporting Server. Therefore, it is not a valid HTTP header value. Instead, specify the WF_REMOTE_USER variable.

Reference: Reporting Server Node Security Options

The security options from the Client Configuration pane are explained below.

Prompt for Credentials

WebFOCUS makes an explicit connection to the Reporting Server with the user ID and password specified in the Web Security tab. This is the default value.

HTTP Basic

WebFOCUS extracts the user ID and password from the authorization header. These credentials are then used to make an explicit connection to the Reporting Server. You should only select this option when your web tier is performing Basic Authentication.

Note: You can verify that the authorization header is available in by selecting *HTTP Request Info* in the *Diagnostics* tab.

Kerberos

WebFOCUS passes a Kerberos ticket for the user to the Reporting Server. This option enables an end-to-end single sign on solution from the desktop to WebFOCUS, from WebFOCUS to the Reporting Server, and from the Reporting Server to supported relational DBMS systems. To use Kerberos authentication, the Reporting Server must run in security OPSYS mode.

SAP Ticket

WebFOCUS passes the user MYSAPSSO cookie, which is created on SAP Enterprise Portal, to the Reporting Server. The Reporting Server then validates the cookie using the SAP security API. This option enables single sign on from WebFOCUS to a Reporting Server configured with the Data Adapter for SAP for environments using Open Portal Services in SAP Enterprise Portal.

Service Account

Allows you to specify a user ID and password to be used for all connections to the Reporting Server.

The service account credentials are encrypted and stored in the SECURITY keyword of the odin.cfg file. When defined, the service account overrides any other credentials that may be presented to WebFOCUS for this Reporting Server node, and all users connect to the Reporting Server using the same credentials. This approach does not make it possible to identify which user is running a given request on the Reporting Server in Managed Reporting deployments, and therefore is not recommended for them.

Trusted

Allows you to connect to the Reporting Server with only a user ID. This option is useful when no password is available for the user. Controls should be placed on the Server to ensure that connections from unauthorized clients are rejected. For example, you can employ the Reporting Server RESTRICT_TO_IP setting or configure a network firewall so that only a particular client can connect to the Server.

Note: When configuring the WebFOCUS Client to make trusted connections to the Reporting Server, you must also enable the Reporting Server to accept trusted connections.

Using the ReportCaster Console

The ReportCaster Console is the interface that provides access to the tools that administer the Distribution Server and manage schedule job logs, blackout dates, and execution IDs.

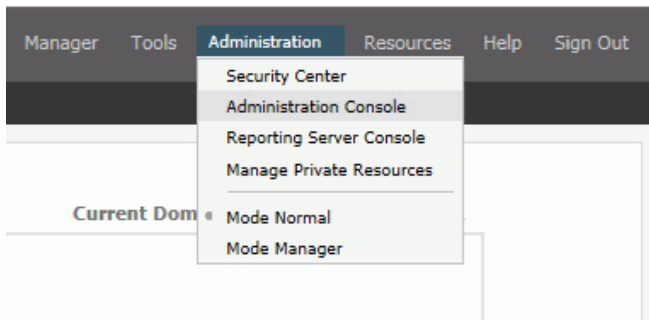
Note: Throughout this section, the ReportCaster Console will be referred to as the Console.

Using the ReportCaster Console

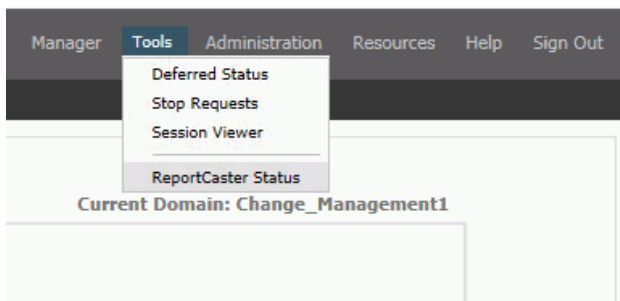
The following tools are accessible from the Console.

- ☐ Server Status
- ☐ Job Status
- ☐ Job Logs
- ☐ Configuration
- ☐ Blackout Periods
- ☐ Global Updates
- ☐ Purge Logs

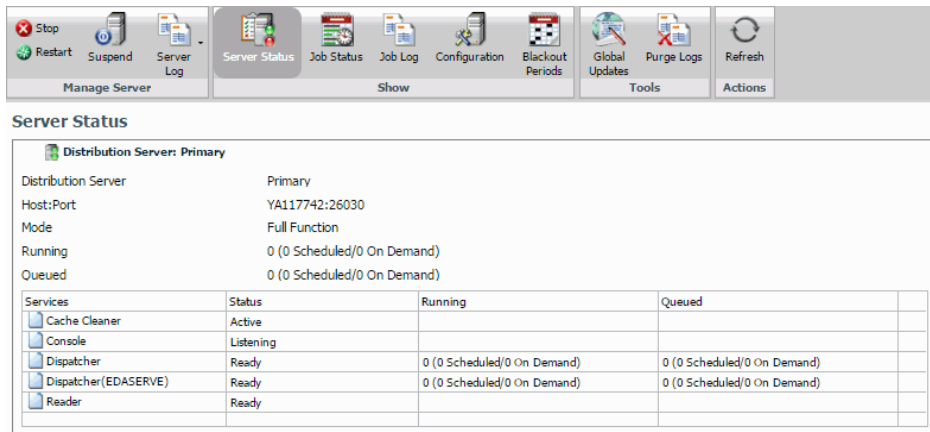
In WebFOCUS Business User Edition configurations licensed for ReportCaster, authorized users can access the ReportCaster Console through the Administration Console, from the Administration option on the menu bar, as shown in the following image.



Depending upon your privileges, these options are also available from the ReportCaster Status option on the Tools menu, as shown in the following image.

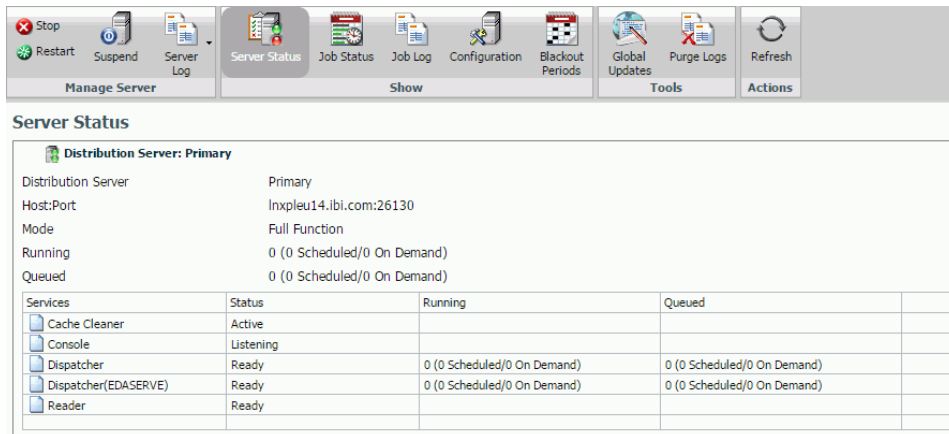


The Console intuitive ribbon displays in a new browser window, as shown in the following image. The ribbon displays a tab for each tool the user is authorized to access.



Server Status

The Server Status tool allows Managers to select a Distribution Server to restart, suspend, or stop. The tool also enables Managers to view traces or refresh data, as shown in the following image.



Viewing the Distribution Server Status

The Server Status tool, accessed by selecting the Server Status tab in the Console, enables you to view the status of the Distribution Server. The Server Status tool also provides details about the Distribution Server, such as the host name and port number, the status, and the number of jobs that are running and in the queue. The Distribution Server information includes:

- ❑ **Distribution Server.** The name used to identify the server in the Console. Primary is the name given to the server listed in the ReportCaster Configuration tool setting (*Primary Distribution Server*).

Note: When the Distribution Server attempts to make an SMTP connection to an Email server, the connection will timeout after five minutes.

- ❑ **Host and Port.** The host name and port number where the Distribution Server is installed.

- ❑ **Mode.** The state and function of the Distribution Server. Options include:

- ❑ **Full Function.** Indicates that the Distribution Server is up and functioning. When you configure ReportCaster with a secondary Distribution Server, this indicates that the server is acting as the primary Distribution Server.

- ❑ **Down.** Indicates that the Distribution Server is stopped.

- ❑ **Running.** The number of scheduled and on demand jobs that are currently running.

- ❑ **Queued.** The number of scheduled and on demand jobs that are in the Distribution Server queue.

- ❑ **Services.** The services currently running on the Distribution Server. Options include:

- ❑ **Cache Cleaner.** The Distribution Server uses this service to refresh the IBFS Cache. The frequency at which the cache is refreshed is controlled by the setting `IBI_Repository_Sync_Interval` in the Administration Console.

- ❑ **Console.** The Distribution Server uses this service to listen for communication from the ReportCaster application or API.

- ❑ **Dispatcher.** The Distribution Server uses this service to execute scheduled jobs.

Note: Depending on your server configuration, you may have one or more additional Dispatchers listed in the Server Status Dialog box.

- ❑ **Reader.** The Distribution Server uses this service to poll the Repository.

- ☐ **Status.** The status of each service currently running on the Distribution Server. Options include:
 - ☐ **Active.** The service is active.
 - ☐ **Ready.** The service is available.
 - ☐ **Standing By.** The service is standing by.
 - ☐ **Suspend.** The service is suspended.
 - ☐ **Listening.** The Console service is actively listening.
 - ☐ **Polling.** The Reader service is active.
 - ☐ **Monitoring.** The Repository Monitor is active.
 - ☐ **Waiting.** Displayed when a job in the running queue is waiting for a connection to the reporting server. Occurs when a multi-task schedule is started because a connection is available to the reporting server in the first task but then waits because a connection is not available to the reporting server in the second task.

From the Server Status interface, you can perform the following tasks:

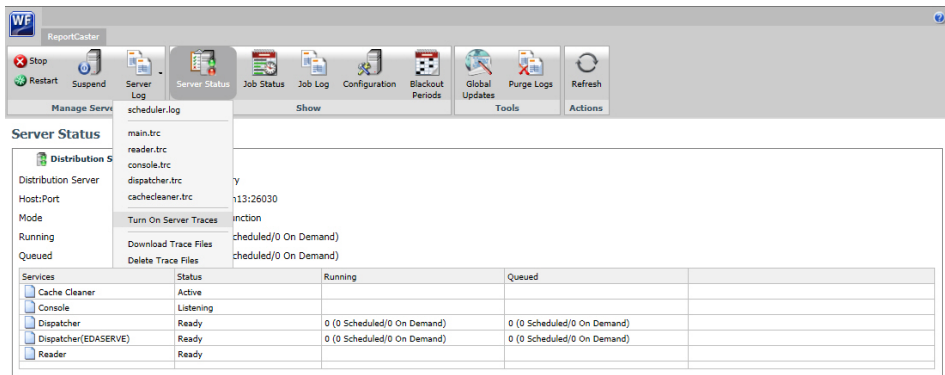
- ☐ **Refresh.** Retrieves the most current information and refreshes the Distribution Server status with this information.
- ☐ **Restart.** Recycles the Distribution Server and the Application Server.
- ☐ **Suspend.** This option is always available. Suspends the Distribution Server services, but the server remains running. When you suspend a server, the Suspend button label will change to Resume.
- ☐ **Stop.** Brings the Distribution Server completely down.

Note: When you stop the Distribution Server using this option, you must restart it from the machine where it resides. You cannot restart the Distribution Server remotely.
- ☐ **View Trace.** Allows you to view trace information for the scheduler.log, main.trc, reader.trc, console.trc, and dispatcher.trc files. You can also turn the Distribution Server traces on or off. For more information, see [How to Turn Distribution Server Traces On or Off](#) on page 142.

Note: Distribution Server traces are tracked separately from Job traces using this functionality. You do not need to turn on job traces to see distribution server traces.
- ☐ **Help.** Opens the Console Server Status online documentation.

Procedure: How to Turn Distribution Server Traces On or Off

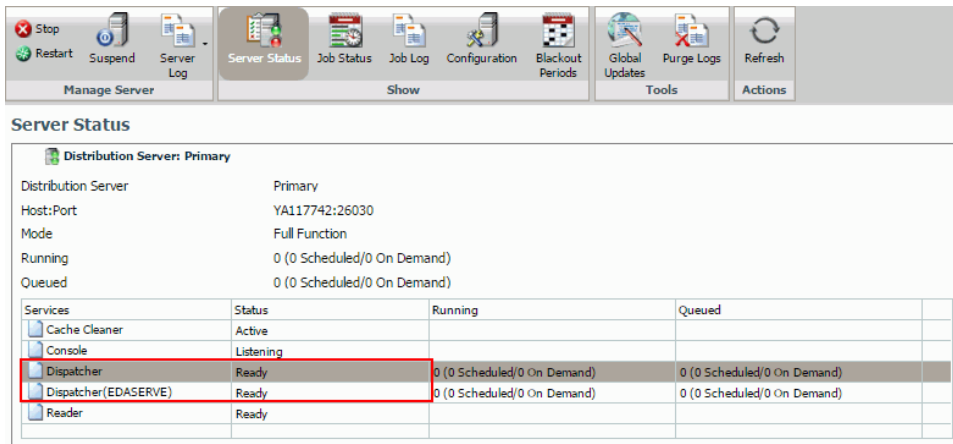
- 1. From the ReportCaster Console, select *Server Status*.
- 2. From the Distribution Server list, select a server.
- 3. On the toolbar, click the down arrow on the Server Log button.
- 4. Click *Turn On Server Traces* to turn the Distribution Server traces on, as shown in the following image.



To disable the Distribution Server traces, follow steps 1 and 2, and then hover over Turn On Server Traces and click *Turn Off Server Traces*.

Separate Job Queues for Each Data Server

The Distribution Server has separate job queues for each Reporting Server, as well as an additional queue for tasks that do not require a Reporting Server, as shown in the following image. Therefore, there will always be at least one job thread available for each Reporting Server, in addition to at least one thread for non-server based jobs. No one Reporting Server can have all of the available job threads consumed by jobs associated with that server.



Viewing, Stopping, Suspending, and Restarting the Distribution Server

The Server Status interface enables you to stop and restart the Distribution Server, as follows:

- ❑ To stop the Distribution Server, click the *Stop* button. A window opens, asking you to confirm that you want to stop the server. Click *Yes*.

Note: When you stop the Distribution Server using this option, the Server Status *Restart* option cannot be used to restart the Distribution Server. To start the Distribution Server, log on to the machine on which the Distribution Server is installed.

- ❑ To suspend the Distribution Server, click the *Suspend* button. A window opens, asking you to confirm that you want to suspend the server. Click *Yes*.
- ❑ To restart the Distribution Server, click the *Restart* button. A window opens, asking you to confirm that you want to restart the server. Click *Yes*.

When you restart the server, the Distribution Server and the ReportCaster Application are restarted.

- ❑ To obtain the most current information about the Distribution Server, click *Refresh*.

Procedure: How to Restart a Server

1. Select a server that is running.
2. From the toolbar, click *Restart* to restart the server.

A window opens, asking you to confirm that you want to restart the server.

3. Click Yes.

Procedure: How to Suspend and Resume a Server

1. Select the server in Full Function mode.
2. On the toolbar, click *Suspend*.

A window opens, asking you to confirm that you want to suspend the server.

3. Click Yes.

The reader service is suspended and the toolbar button changes to *Resume*.

To resume polling the repository for schedules, click *Resume*. Click Yes to confirm.

Procedure: How to Stop a Server

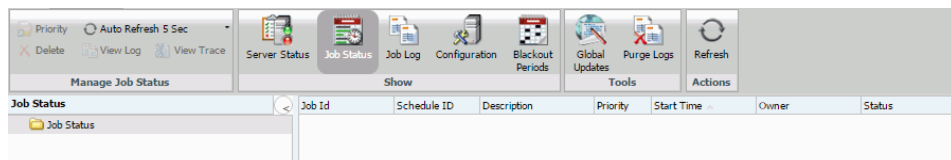
1. Select a running server and click *Stop*.

A window opens, asking you to confirm that you want to stop the server.

2. Click Yes.

Job Status

Another resource for tracking schedules is the schedule job status. The schedule status provides a list of scheduled jobs that are in the Distribution Server queue. Status information includes the Schedule ID, the time it started running, and the status of the job.



The schedule information includes:

- ☐ **Job Id.** The ID assigned to the job.
- ☐ **Schedule ID.** A unique ReportCaster generated key assigned to the job when it was scheduled.

- ☐ **Description.** The description provided when the schedule was created.
- ☐ **Priority.** The priority level of the schedule. 1 is the highest and 4 is the lowest priority.
- ☐ **Start Time.** The time that the schedule run began.
- ☐ **Owner.** The name of the user who owns the schedule.
- ☐ **Status.** The status of the scheduled job. It contains one of the following values:
 - ☐ **Running.** The scheduled job is currently running.
 - ☐ **Queue.** The scheduled job is waiting for a thread to become available to run the request.
 - ☐ **Server Name.** The Reporting Server to which the job has been submitted.

Job Log

The Job Log tab enables you to view the logs of executed jobs belonging to you or to users whose job logs you are authorized to see. You can view log and trace information, open job logs, delete job logs, refresh job logs, or access related help for job logs on the Job Log tab. You can also view log and trace information for the daily and on-demand Log Purge jobs. The Log Purge log can be accessed from the system folder. The logs for other jobs can be accessed under the folder of the user to whom the job belongs.

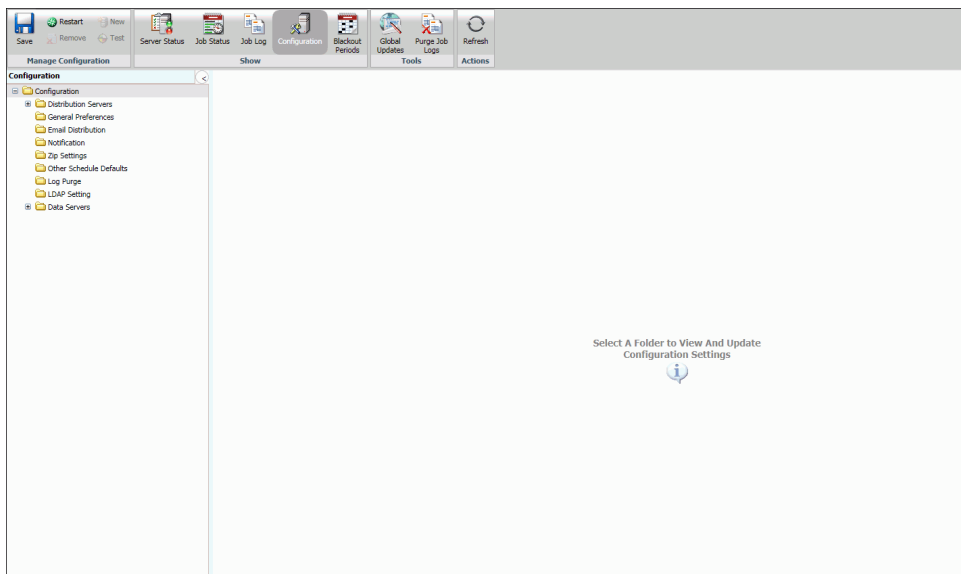
Note: The Job Log tab supports the functionality of a multiple selection, whereby you can open or delete multiple files simultaneously.

The job logs information includes:

- ☐ **Job Id.** The ID assigned to the job.
- ☐ **Start Time.** The time that the schedule was run.
- ☐ **Job Duration(seconds).** The amount of time needed to complete a job.
- ☐ **Job Status.** The status of the job when it completed processing.
 - ☐ **Success.** No errors occurred during the scheduled job processing.
 - ☐ **Error.** One or more errors occurred during the scheduled job processing. No report was generated or distributed.
 - ☐ **Warning.** One or more warnings occurred during the scheduled job processing. A report was generated and distributed.

ReportCaster Configuration

The Configuration tool enables Managers to view and manage the configuration of the Distribution Server, Servlet (deployed in WebFOCUS Business User Edition web application) interfaces and tools. Options that Managers can modify include Distribution Servers, Repository Settings, General Preferences, Email Distribution, Notification, and other options, as shown in the following image.




Configuration Icons

In the Configuration tab on the ReportCaster Console ribbon, a series of icons enable you to perform the following tasks.

Note: When you select *Configuration* in the ReportCaster Console ribbon, the Manage Configuration group displays on the left-hand side. Clicking *Server Status*, *Job Status*, *Job Log*, or *Blackout Periods* will change the name and functionality of this group, depending where you are in the interface. These options all appear in the Show group of the ribbon.

Manage Configuration group

-  **Save.** Saves any changes made to the configuration. You will receive a message asking you to confirm the save. You must save the changes to the configuration settings as part of the process to implement the change. For details, see [Changing Configuration Settings](#) on page 148

- ☐ **New.** Creates a new Reporting Server. This option is only active when working in the Data Servers folder.
 - ☐ **Remove.** Deletes a Reporting Server from the ReportCaster configuration. You will receive a message asking you to confirm the deletion. This option is only active when working in the Data Servers folder.
 - ☐ **Test.** Tests the connection to the selected server or repository. This option is only active when working in the Data Servers, Repository Settings, and LDAP Settings folders. You may be prompted for a user ID and password to connect to a specified server. You will receive a message describing whether the test succeeded or failed.
 - ☐ **Restart.** Restarts the Distribution Server and ReportCaster to implement changes to the server configuration. You can also select *Restart* from the Action menu. For more information, see [Changing Configuration Settings](#) on page 148.
- ☐ **Tools group**
- ☐ **Global Updates.** Authorized users can make global updates in the Global Updates interface for the Mail Server, Printer, Email Address, Email from, and Data Server values stored in schedules and distribution lists.
 - ☐ **Purge Job Logs.** Provides on-demand capability for purging logs. You can also specify a number of days (past) for which to purge logs. For example, if you want to purge logs for the past month, you would use the default number of days, which is 30. You can also specify an option for traces: Default Traces, No Traces, or Trace On. For more information, see [How to Purge Logs on Demand](#) on page 210.
- ☐ **Actions group**
- ☐ **Refresh.** Refreshes the settings to reflect the last saved configuration settings.
 - ☐ **Help.** Opens the online Help file. This option is only available when accessing the configuration tool through the ReportCaster web application. It is not available when accessing the tool from the Windows Programs menu or by running editit.bat or the UNIX editit script file from the /utilities directory within the ReportCaster Distribution Server installation.

Configuration Tab Folders

The Configuration tab provides access to configuration settings through the following folders:

- ☐ **Distribution Servers.** Defines and configures the Distribution Server.

- ❑ **General Preferences.** Specifies which distribution formats and methods will be available to a user creating a schedule. In this folder, you can also specify if ReportCaster scheduling options will include the ability to distribute a PDF file directly to a printer. For details on these settings, see [General Preferences](#) on page 152.
- ❑ **Email Distribution.** Configures email settings, such as the default email host, number of attempts to distribute to an email host, and security information. For details on these settings, see [Email Distribution](#) on page 158.
- ❑ **Notification.** Configures notification settings, such as the notification email host and default notification type. For details on these settings, see [Notification](#) on page 163.
- ❑ **Zip Settings.** Provides settings to configure how Zip files will be created and named. For details on these settings, see [Zip Settings](#) on page 166.
- ❑ **Other Schedule Defaults.** Defines the default end date and time of a ReportCaster schedule. For details on these settings, see [Other Schedule Defaults](#) on page 168.
- ❑ **Log Purge.** Allows you to specify the time and period that log files will be automatically purged. For details on these settings, see [Log Purge](#) on page 169.
- ❑ **LDAP Setting.** Allows you to configure ReportCaster self-service users to be authenticated and authorized against an LDAP data source rather than the WebFOCUS Business User Edition Repository. You can also configure ReportCaster to retrieve email address information from an LDAP data source. For details on these settings, see [LDAP Setting](#) on page 170.
- ❑ **Data Servers.** Allows you to configure multiple Reporting Servers, including cluster servers. For details on these settings, see [Data Servers Settings](#) on page 173.

Changing Configuration Settings

To implement changes to the ReportCaster server configuration settings, you must save the changes, and then restart the Distribution Server and the ReportCaster web application. Use the following steps to save any configuration changes.

1. After you make changes to any of the configuration settings within a Configuration folder, click the Save icon in the Manage Configuration group toolbar.

A window opens, asking you to confirm the save.


2. Click *OK*.

A window opens, explaining that you must restart the Distribution Server and the ReportCaster Web Application in order for the changes that you saved to take effect.

3. Click *OK*.
4. Click the *Restart* icon in the Manage Configuration toolbar.
A window opens, asking you to confirm that you want to stop all running jobs and restart.
5. Click *Yes*.
6. Notify all users that are currently logged that they must restart their sessions to obtain the new configuration information. This is required because user sessions have information cached pertaining to the ReportCaster configuration. The user interfaces must be restarted to obtain any updated configuration information.

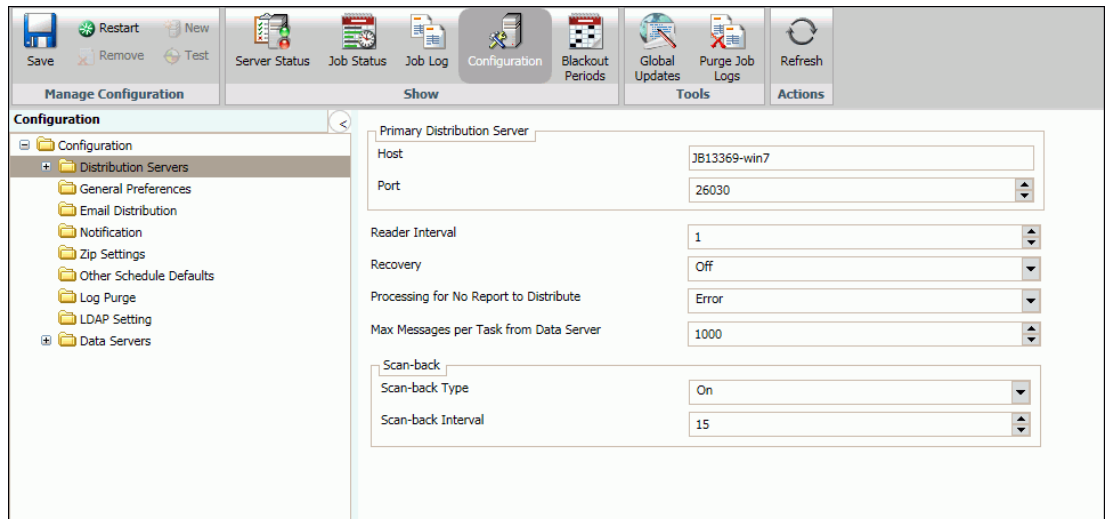
Changing Default User IDs

Several of the Configuration tab folders provide the option to change the default user ID

and/or password. To change a default user ID settings, click the  icon that appears to the right of the setting field. This opens the User dialog box, where you can type a new user ID and password. When your changes are complete, click *OK* to exit the User dialog box. Remember to save your changes and restart the Distribution Server to implement any changes.

Distribution Server Settings

The Distribution Servers folder, shown in the following image, contains the settings that define and configure the Distribution Server. The Server Status option, located on the ribbon, provides the ability to restart the Distribution Server, suspend distribution, stop distribution, switch servers, and refresh data. For more information, see [Server Status](#) on page 139.



The Distribution Servers folder contains the following configuration settings.

Setting	Optional or Required/ Default Value	Descriptions and Possible Values
Primary Distribution Server section		
Host	Required.	Host name of the primary Distribution Server.
Port	Required.	Port number of the primary Distribution Server.
Reader Interval	Required. Default value is 1 minute.	<p>Polling interval (in minutes) for the ReportCaster Distribution Server to check for scheduled jobs.</p> <p>An acceptable value is any positive integer from 1 to 999999. Negative numbers and zero are not allowed.</p>
Recovery	Default value is OFF.	<p>On. During startup, the ReportCaster Distribution Server recovers scheduled jobs that were processed but not completed.</p> <p>Off (Default). During startup, the ReportCaster Distribution Server does not recover any scheduled jobs.</p>

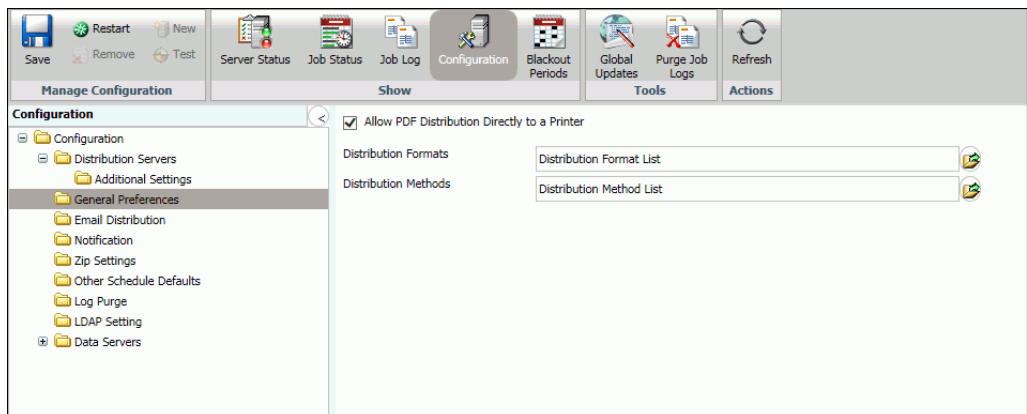
Setting	Optional or Required/ Default Value	Descriptions and Possible Values
Processing for No Report to Distribute	Required. Default value is Error.	<p>Specifies whether the <i>No Report to Distribute</i> message from the WebFOCUS Reporting Server is categorized as an error or a warning. This is a global setting, relevant to all schedules. Possible values are:</p> <p><input type="checkbox"/> Error. The <i>No Report to Distribute</i> message is categorized as an error and the message is written to the ReportCaster log report in red. When the schedule has the Notification option set to Error, the Notification is sent.</p> <p><input type="checkbox"/> Warning. The <i>No Report to Distribute</i> message is categorized as a warning and the message is written to the log report (in orange) as an informational message. When the schedule has the Notification option set to Warning, no error notification is sent.</p> <p>These settings also apply to burst reports when there is a <i>No Report to Distribute</i> message for an individual burst value.</p>
Max Messages per Task from Data Server	Required. Default value is 1000	Controls the number of messages for each task from the Data Server written to the ReportCaster log file.

Setting	Optional or Required/ Default Value	Descriptions and Possible Values
Scan-back section		

Setting	Optional or Required/ Default Value	Descriptions and Possible Values
Scan-back Type	Required. Default value is On.	Possible values are: <input type="checkbox"/> On <input type="checkbox"/> Off <input type="checkbox"/> Reset Next Run Time
Scan-back Interval	Default value is 15, 24 hour periods.	If the Distribution Server is unavailable for a period of time, this is the number of 24 hour periods (beginning with the Distribution Server restart time) that you want the Distribution Server to scan for jobs not yet run and run them.

General Preferences

The General Preferences folder in the Configuration tab, as shown in the following image, contains settings that determine which distribution formats and methods will be available to a user.



The General Preferences folder contains the following configuration settings.

Setting	Optional or Required/Default Value	Descriptions and Possible Values
Allow PDF Distribution Directly to a Printer	Required. By default, this option is selected.	When selected, PDF is a selectable format for the Printer distribution method. This enables a PDF file to be distributed directly to a printer. The printer must have the appropriate driver to print PDF files.
Distribution Formats	Optional. All formats are enabled, by default.	Specifies what report and graph formats are available to users.
Distribution Methods	Optional. All distribution methods are enabled, by default.	Specifies what distribution methods are available to users.

Specifying Schedule Format Settings

The Distribution Formats setting in the General Preferences folder enables users to specify what report and graph formats are available to users and groups.

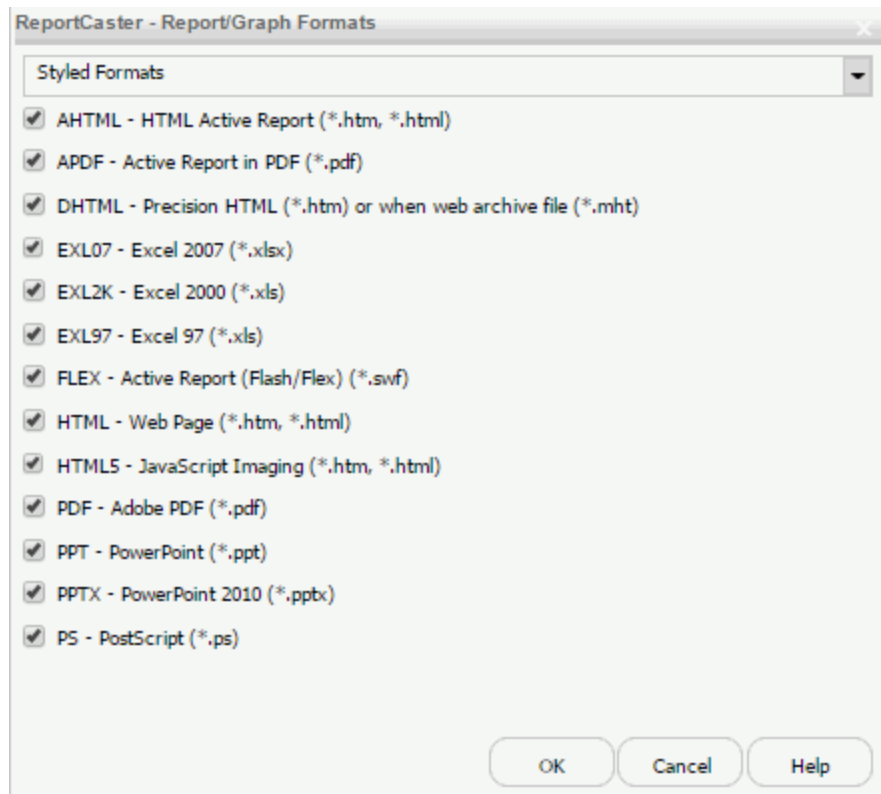
This setting only applies to WebFOCUS (Repository) procedures. By default, all report and graph formats are enabled (selected). A minimum of one report or graph format must be selected.

Note: If a Format is cleared after a schedule using that Format has been created, the schedule will fail when it is run. A message in the log will indicate how to fix the problem.

Procedure: How to Specify Schedule Format Settings

1. In the General Preferences folder, click the icon to the right of the *Distribution Formats* field.

The ReportCaster - Report/Graph Formats dialog box opens, as shown in the following image.

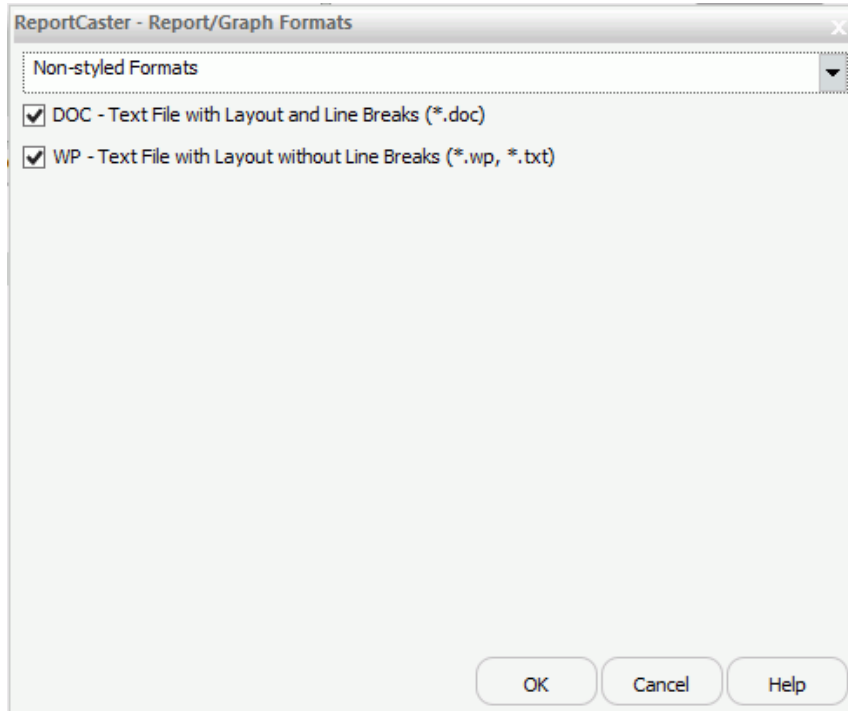


2. Using the Styled Formats drop-down list, you can select *Specialized Formats*, *Non-styled Formats*, or *Graph Images*. Optionally, you can keep the default, *Styled Formats*.

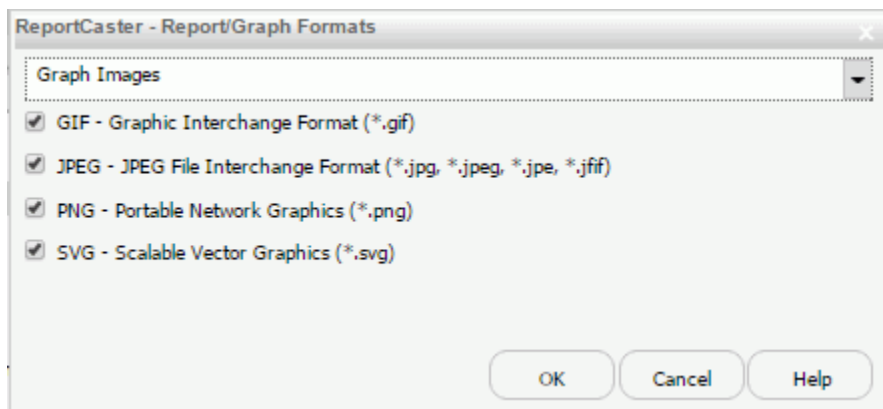
By default, *Styled Formats* are displayed with each format type selected.

3. To enable or disable styled formats, select or clear the check box for one of the listed formats.

4. To enable or disable Non-styled Formats, which do not support styling using WebFOCUS StyleSheet commands, select *Non-styled Formats* from the drop-down list. By default, the Non-styled Formats are displayed with each format type enabled, as shown in the following image.



5. To enable or disable graph images, which are created by a WebFOCUS BUE graph request, select *Graph Images* from the drop-down list. By default, the graph image formats are displayed with each format type enabled, as shown in the following image.



6. When your selections are complete in the Report/Graph Formats dialog box, click *OK*.
The changes are saved and the Report/Graph Formats dialog box closes.
7. To implement the configuration changes, restart the Distribution Server and WebFOCUS BUE web application.

Specifying Schedule Distribution Method Settings

The Distribution Methods setting, in the General Preferences folder, enables authorized users to specify the distribution methods that are available to users and groups. If a method is cleared after a schedule using that method has been created, and *Only run schedules for selected Distribution Method* is selected, the schedule processing will not run the scheduled tasks. A message in the log will indicate that the distribution method is not configured for use. In addition, a schedule that has multiple distributions will not run if one of the distribution methods is cleared in the Distribution Methods drop-down menu, in the General Preferences folder, of the Configuration tab.

By default, all distribution methods are enabled (selected).

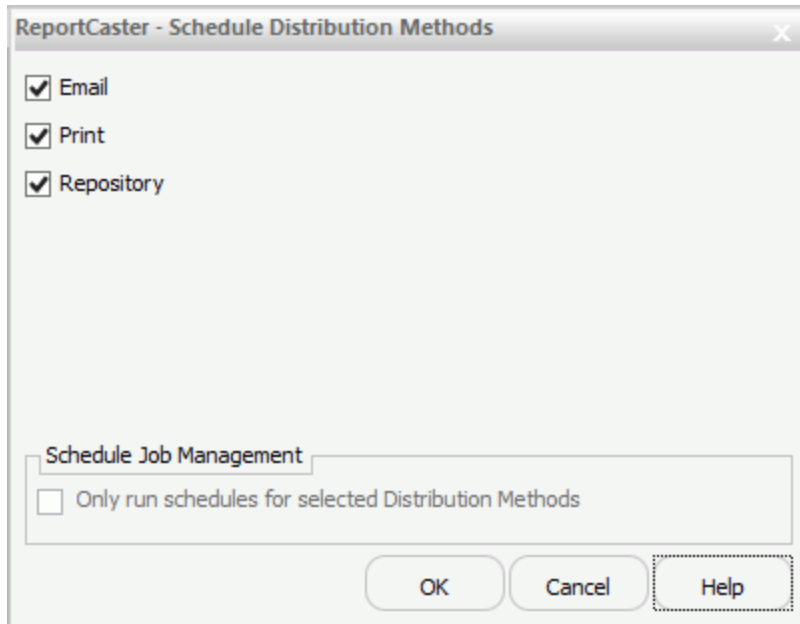
Note: A minimum of one distribution method must be selected.

The Repository option only appears if these products are enabled. Repository is an optional product component that is installed with the WebFOCUS BUE Client.

Procedure: How to Specify Schedule Distribution Method Settings

1. In the General Preferences folder, click the icon to the right of the *Distribution Methods* field.

The ReportCaster - Schedule Distribution Methods dialog box opens, as shown in the following image.



2. To enable or disable specific distribution methods, select or clear the check box for one of the listed options.

Note: At least one schedule distribution method must be selected.

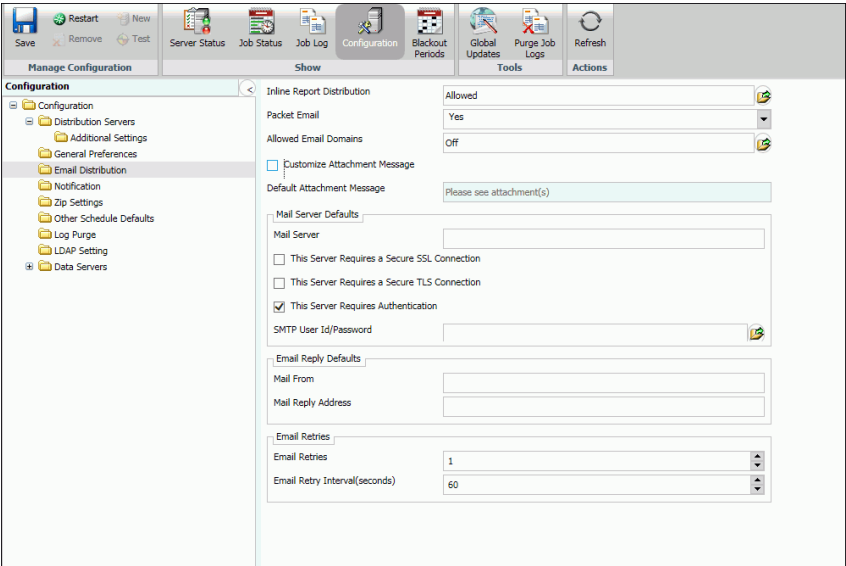
3. Optionally, select the *Only run schedules for selected Distribution Methods* check box to limit the schedules running to those that have at least one of the Distribution Methods selected.
4. When your selections are complete in the Schedule Distribution Methods dialog box, click *OK*.

The changes are saved and the Schedule Distribution Methods dialog box closes.

5. To implement the configuration changes, restart the Distribution Server and WebFOCUS BUE web application.

Email Distribution

The Email Distribution folder in the Configuration tab contains default email settings, email retry options, and email security. The Email Distribution window is shown in the following image.



The Email Distribution and Notification folder contains the following configuration settings.

Setting	Optional or Required/Default Value	Descriptions and Possible Values
Inline Report Distribution	Required. The default value is Allowed.	Specifies whether the Schedule tool will enable the email distribution option to send a report in the body of the email (inline). Note: If this option is cleared after a schedule using this option has been created, the schedule will fail when it is run. A message in the log will indicate how to fix the problem.

Setting	Optional or Required/Default Value	Descriptions and Possible Values
Packet Email	Required. The default value is Yes.	<p>Controls how task output and burst content are distributed through email.</p> <p>Valid values are:</p> <ul style="list-style-type: none"> <input type="checkbox"/> No. Each burst value or task output is distributed in a separate email. <input type="checkbox"/> Yes. People receiving burst values or output from tasks will receive a single email with multiple attachments. <input type="checkbox"/> Burst. Each burst value in a distribution list will generate a separate email for a given email address. There may be one or more attachments in the email, depending on the number of tasks in the schedule.
Allowed Email Domains	The default value is Off.	<p>Contains the email domains that are authorized in a distribution.</p> <p>Note: Files containing email addresses intended for distribution are also verified for valid domains at schedule execution time. If it contains a restricted address, then no delivery is made to that address and an error message is written to the log file.</p>
Customize Attachment Message	Optional	Provides the ability to specify a custom message.
Default Attachment Message	Required.	Defines the default message used in your Email Distribution. The message that you define here displays in the Basic Scheduling Tool.

Setting	Optional or Required/Default Value	Descriptions and Possible Values
Mail Server Defaults section		
Mail Server	Required.	Name of the default mail server used to distribute an email schedule. You can also specify a port for Mail Host using <i>hostname:port</i> . If you do not specify a port or the port you specify is not present, the default port is used.
This Server Requires a Secure SSL Connection	Optional.	Select this check box if the specified mail server uses SSL.
This Server Requires a Secure TLS Connection	Optional.	Select this check box if the specified mail server uses TLS.
This Server Requires Authentication	Optional.	Select this check box if the specified mail server requires authentication with a user ID and password.
SMTP User Id/ Password	Required if the mailhost is using SMTP Authorization. No default value.	User ID and password used to connect to the mailhost.
Email Reply Defaults section		
Mail From	Optional.	Default value for the email From field. This can be any value.
Mail Reply Address	Optional.	Default email reply address when creating an email schedule.
Email Retries section		

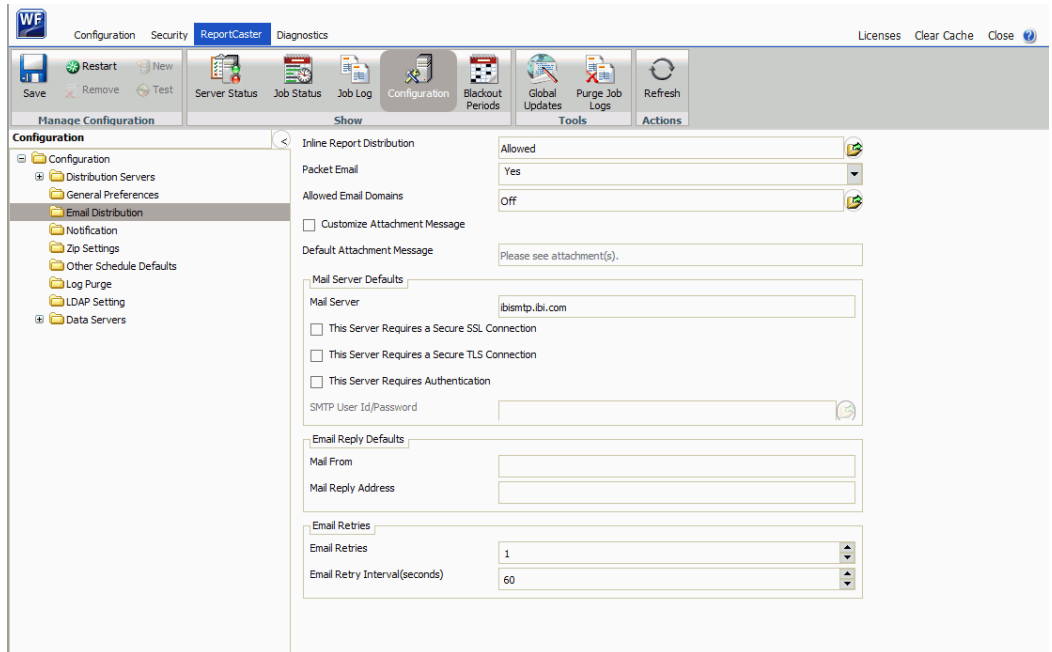
Setting	Optional or Required/Default Value	Descriptions and Possible Values
Email Retries	The default value is 1.	<p>Number of times the Distribution server will try to connect to the email server to deliver report output.</p> <p>If the Distribution server is unable to connect to the email server on the first try, it will attempt to connect again after the specified Email Retry Interval has passed. A message for each attempt is written to the log file.</p> <p>Valid values are 0 through 9.</p>
Email Retry Interval(seconds)	The default value is 60 seconds.	<p>Amount of time that the Distribution server will wait between retries.</p> <p>Valid values are 1 through 999.</p>

Procedure: How to Configure Email Distribution

1. From the Tools menu, click *ReportCaster Status*.

2. Click the *Configuration* button, as shown in the following image.

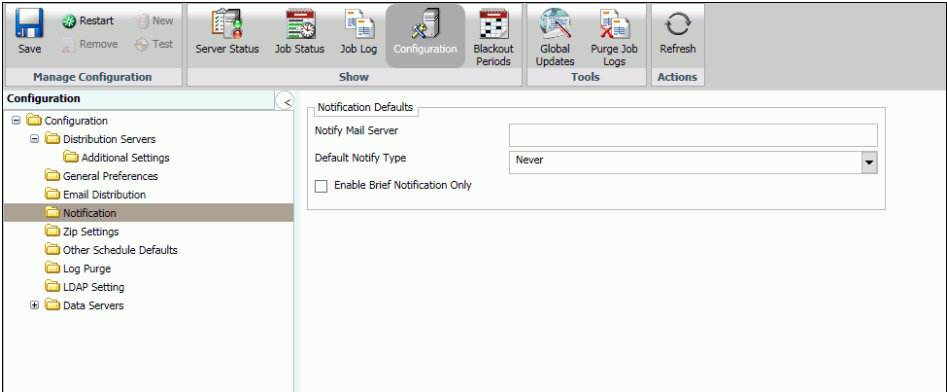
Note: Authorized users can also access the ReportCaster Configuration tool from the Administration Console.



3. In the left pane, select the *Email Distribution* folder.
4. Populate the email distribution fields. You can:
 - ☐ Make changes to the current email settings. For more information, refer to the preceding table.
 - ☐ Supply any default values that you would like to be available at schedule creation time.
5. If your mail server(s) require authentication or SSL or TLS, check the appropriate boxes and enter the required credentials. For more information, see the *Installation* topic in the Information Center.
6. Click **Save**.

Notification

The Notification folder in the Configuration tab contains default notification settings. The Notification window is shown in the following image.



The Notification folder contains the following configuration settings.

Setting	Optional or Required/Default Value	Descriptions and Possible Values
Notification Defaults section		

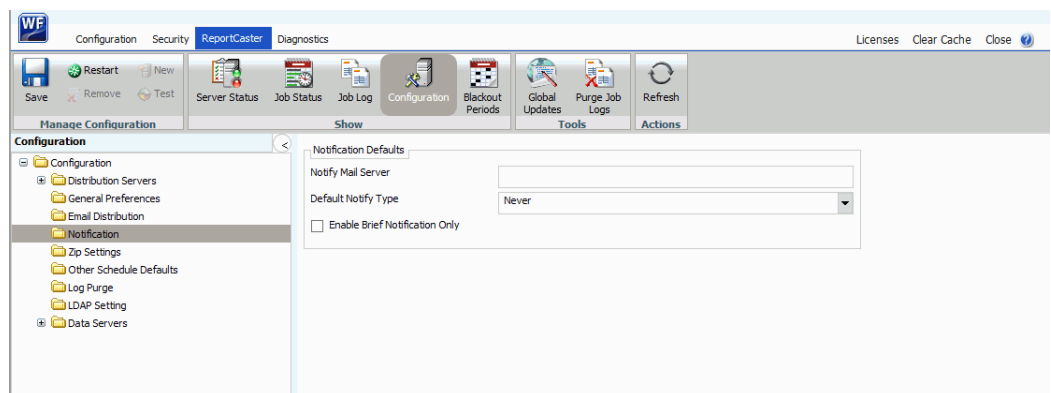
Setting	Optional or Required/Default Value	Descriptions and Possible Values
Notify Mail Server	Optional.	<p>Name of the mail server that distributes the notification email. If blank, ReportCaster uses the Mailhost setting as the notification mail server.</p> <p>Tip: Information Builders recommends using different mail servers for notification and email distribution. This way, if there is a problem with your Mailhost, notification will still be sent. Having separate mail servers ensures that you will be informed when the default mail server falters.</p> <p>You can also specify a port for the Notify Mailhost using <i>hostname:port</i>. If you do not specify a port or the port you specify is not present, the default port is used.</p>
Default Notify Type	The default value is Never.	<p>Specifies whether to send notification of the schedule status to a specified email address. Possible values are:</p> <ul style="list-style-type: none"> <input type="checkbox"/> Never. This is the default value. ReportCaster does not send notification of the schedule status under any circumstance. <input type="checkbox"/> On Error. The specified users are notified when errors are encountered while running the schedule. Information Builders recommends using the On Error notification option. <input type="checkbox"/> Always. The specified user is always notified when the schedule runs.

Setting	Optional or Required/Default Value	Descriptions and Possible Values
Enable Brief Notification Only	Optional.	<p>Controls whether Full Notification is available as a scheduling option. When this check box is selected, you can only select the Brief Notification option when you schedule a report. Full Notification is not available.</p> <p>Note: Messages are displayed in the log when Full Notification is disabled. If a schedule is created before Full Notification is disabled, when the schedule executes, a Brief Notification will be sent, and a warning will be displayed in the schedule log.</p>

Procedure: How to Configure Notification

1. From the Tools menu, click *ReportCaster Status*.
2. Click the *Configuration* button, as shown in the following image.

Note: Authorized users can also access the ReportCaster Configuration tool from the Administration Console.

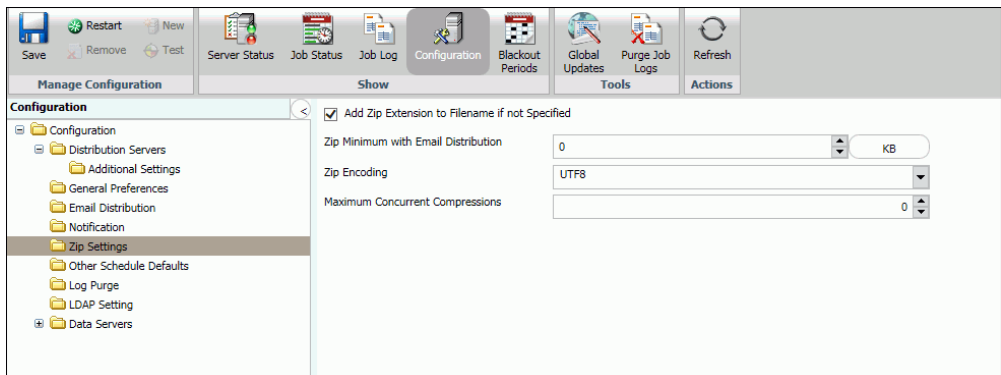


3. In the left pane, select the *Notification* folder.

4. Populate the notification fields. You can:
 - ☐ Make changes to the current notification settings. For more information, refer to the preceding table.
 - ☐ Supply any default values that you would like to be available at schedule creation time.
5. Click Save.

Zip Settings

The Zip Settings folder in the Configuration tab, as shown in the following image, contains settings for adding an extension to a distributed Zip file, the Zip encryption to use for distribution.



The Zip Settings folder contains the following configuration settings.

Setting	Optional or Required/Default Value	Descriptions and Possible Values
Add Zip Extension to Filename if Not Specified	Required. By default, this option is selected.	Controls whether or not the .zip file extension will be automatically appended to the Zip file name entered by the user in a schedule distributed by email or FTP. Select this option if you want to automatically append .zip to the entered file name. Do not select this option if you want to use the file name as entered by the user and not have .zip automatically appended to the file name.
Zip Minimum with Email Distribution	Required. The default size is in KB and set to 0.	Select either MB or KB and customize the size of your file using the up and down arrows.
Zip Encoding	Optional.	Specifies an encoding other than the default encoding of the ReportCaster Distribution Server platform. The encoding specified must match the encoding used by WinZip or any other Zip utility installed on the Distribution Server.
Maximum Concurrent Compressions	Optional The default value is zero (0).	The total number of compression operations that the Distribution Server will perform simultaneously. If many running jobs include a compression operation (for example, zipping the output before sending), the compression operations could consume all of the available resources on the Distribution Server. You can lower this number to prevent this from occurring.

Procedure: How to Configure Zip Settings

1. From the ReportCaster Tools menu, click *ReportCaster Status*.
2. Click the *Configuration* button.

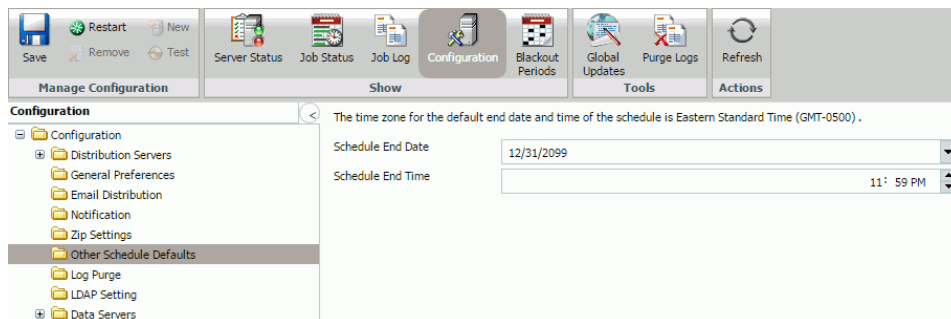
Note: Authorized users can also access the ReportCaster Configuration tool from the Administration Console.

3. In the left pane, select the *Zip Settings* folder.
4. Populate the Zip Settings fields using the information provided in the preceding tables.
5. If you want to produce zipped output that is encrypted and password protected, you can use the Zip Encryption Password Plug-in. To use your own program to supply the password, select *Custom* from the drop-down list and enter the name of the program in the Zip Encryption Password Plug-in Name field. To use the default, select *Default*.
6. Click *Save*.

Other Schedule Defaults

The Other Schedule Defaults folder in the Configuration tab, as shown in the following image, contains settings for Schedule End Date and Schedule End Time.

Note: Depending on your time zone, the default Schedule End Date may be set to Jan 1, 2100.

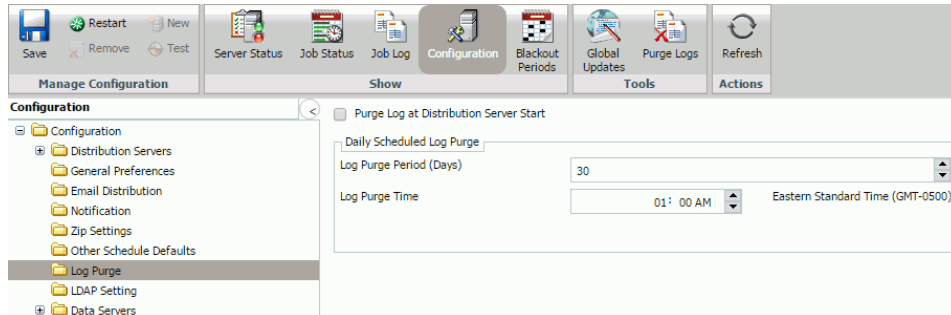


The following table lists and describes the configuration settings available in the Other Schedule Defaults folder.

Setting	Optional or Required/Default Value	Descriptions and Possible Values
Schedule End Date	Required.	Clicking the drop-down menu displays a calendar where you can select the schedule end date.
Schedule End Time	Required.	Manually, you can enter an end time for the schedule. Alternatively, use the arrows to assign a schedule end time.

Log Purge

The Log Purge folder in the Configuration tab, as shown in the following image, contains settings for purging log files, log purge periods, and log purge times.



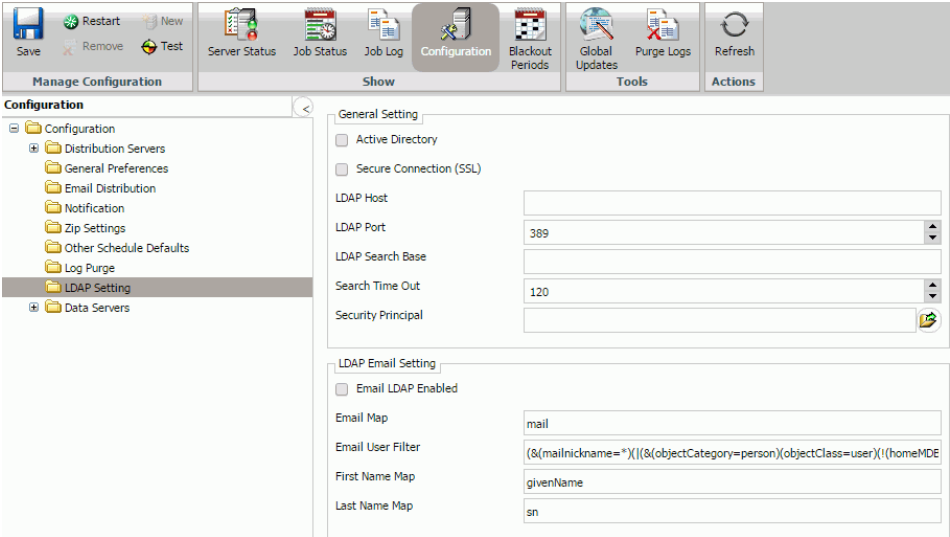
The Log Purge folder contains the following configuration settings.

Setting	Optional or Required/Default Value	Descriptions and Possible Values
Purge Log at Distribution Server Start	Optional. By default, the check box is unchecked.	When selected, log reports are automatically purged each time the Distribution Server starts. This is in addition to the scheduled log purging that is set using the Log Purge Period and Log Purge Time options.
Daily Scheduled Log Purge section		
Log Purge Period (Days)	Optional.	Automatically purges individual log reports when they are older than a set number of days.
Log Purge Time	Optional.	Time at which log purging occurs.

LDAP Setting

ReportCaster can be configured to retrieve email address information from an LDAP data source. The leading mail server software vendors use LDAP for storage of email information. This includes Active Directory on Windows platforms.

The LDAP Setting folder in the Configuration tab, as shown in the following image, contains settings to define LDAP server connection and security, as well as LDAP email settings.



The LDAP Settings folder contains the following configuration settings.

Setting	Optional or Required/Default Value	Descriptions and Possible Values
General Setting section		
Active Directory	By default, this option is not selected.	Select if you are using the Active Directory technology for LDAP. Do not select if you are using any other technology for LDAP.
Secure Connection (SSL)	By default, this option is not selected.	Indicates whether or not SSL (Secure Sockets Layer) is used to communicate with the LDAP server.
LDAP Host	There is no default value.	Host name or IP address of your LDAP directory server.

Setting	Optional or Required/Default Value	Descriptions and Possible Values
LDAP Port	Default value is 389.	Port on which the directory server listens.
LDAP Search Base	There is no default value.	Filter for LDAP searches. Only the subtree below the search base is available for LDAP queries. For example, dc=ibi, dc=com.
Search Time Out	Default value is 120.	Time, in seconds, that ReportCaster can search an LDAP data source before timing out.
Security Principal	No default value.	<p>Service account of the user performing authentication. The user must have sufficient access rights to locate user entries in the directory.</p> <p>Select the icon to the right of this option to open the User dialog box, where you can type a user name and password.</p>
LDAP Email Setting section		
Email LDAP Enabled	By default, this option is not selected.	Indicates whether ReportCaster is enabled to retrieve email address information from an LDAP data source.
Email Map	Default value is mail.	Attribute type for email entries.

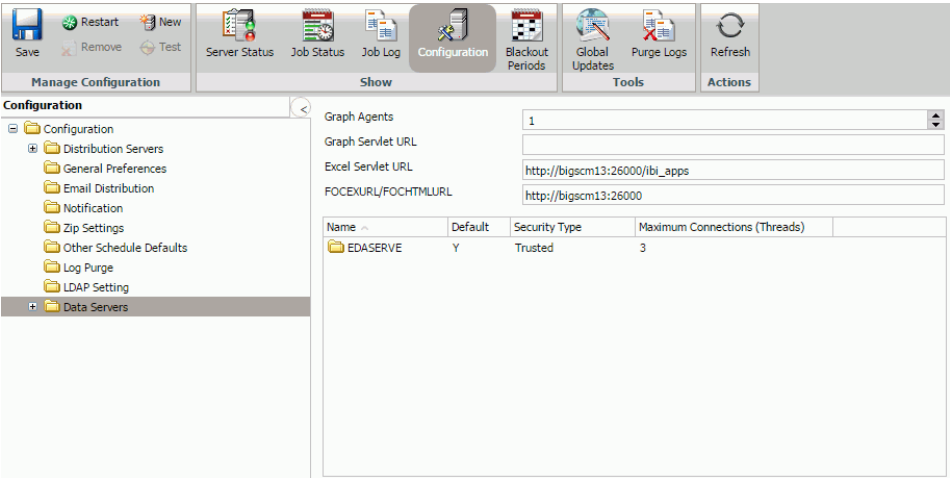
Setting	Optional or Required/Default Value	Descriptions and Possible Values
Email User Filter	Default value provided is dependent on the LDAP Type selected.	Filter for email entries to be retrieved. For example: <pre>(& (mailnickname=*) ((&(objectCategory=person) (objectClass=user) (!(homeMDB=*)) (!(msExchHomeServerName=*))) (&(objectCategory=person) (objectClass=user) ((homeMDB=*) (msExchHomeServerName=*))) (&(objectCategory=person) (objectClass=contact)) (objectCategory=group) (objectCategory=publicFolder)))</pre>
First Name Map	Default value is givenName.	Attribute for first name entries.
Last Name Map	Default value is sn.	Attribute for last name entries.

Data Servers Settings

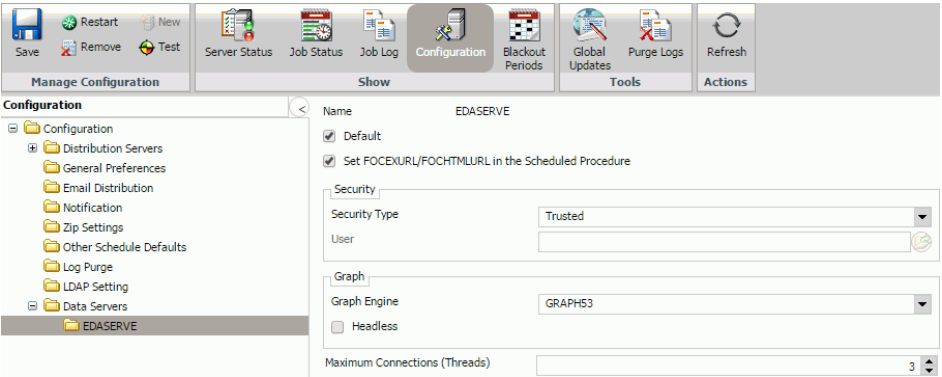
The Data Servers folder in the Configuration tab contains settings to configure the Reporting Servers associated with ReportCaster. Using the configuration settings in this folder, you can also configure multiple Reporting Servers with ReportCaster.

Note: Data Server connection information is stored in the WebFOCUS BUE Client, and not in ReportCaster. ReportCaster runs scheduled procedures through the WebFOCUS BUE Client which is installed with the Distribution Server. When a ReportCaster job is executed by the WebFOCUS BUE Client, the alternate deferred server is used if an alternate deferred server is defined.

The following image shows the display in the right panel when the Data Servers folder is selected. Initially, the default data server appears in this panel with information, such as the data server name, URL (used to connect to the Reporting Server), the type of server, whether or not it is the default server, and what security type it is using. When you add a data server that ReportCaster can access, it will appear in this list.



The following image shows the configuration settings when you select a specific data server under the Data Servers folder (in this example, EDASERVE).



The Data Servers folder contains the following configuration settings.

Setting	Optional or Required/ Default Value	Descriptions and Possible Values
Settings for the Data Servers folder:		
Graph Agents	The default value is 1.	Optimizes the processing of graphs. Due to performance considerations, Information Builders generally recommends configuring this setting to 1 Graph Agent for each concurrent graph report. However, your own internal testing should determine the Graph Agent value that best suits the business needs of your organization.
Graph Servlet URL	Optional. There is no default.	<p>Overrides the default graph server setting and configures graph image files to be created on the Application Server.</p> <p>Type the following value</p> <p><code>http://hostname/context_root/IBIGraphServlet</code></p> <p>where:</p> <p><code>hostname</code></p> <p>Is the host name of the Application Server where the WebFOCUS BUE Client is installed.</p> <p><code>context_root</code></p> <p>Is the site-customized context root for the WebFOCUS BUE web application deployed on your Application Server. <code>ibi_apps</code> is the default value.</p> <p>This setting is available for WebFOCUS BUE Server and WebFOCUS BUE procedures.</p> <p>This setting should not be used when web server security is enabled. This includes Basic authentication, IWA, SSL, and third-party security products (such as SiteMinder). In these cases, the web server security settings can prevent WebFOCUS BUE from creating the graph.</p>

Setting	Optional or Required/ Default Value	Descriptions and Possible Values
Excel Servlet URL	Default value is <code>http://localhost:8080/ibi_apps</code>	<p>Specifies the application server to be used to zip the file components that comprise an EXCEL[®] 2007 file (.xlsx) as follows:</p> <p><code>&URL_PROTOCOL://servername/alias/IBIEXCELSERVURL</code></p> <p>where:</p> <p><code>URL_Protocol</code> Is HTTP.</p> <p><code>servername</code> Is the name of the application server where the WebFOCUS BUE Client is installed.</p> <p><code>alias</code> Is the context root of the WebFOCUS BUE application. The default is <code>ibi_apps</code>.</p> <p>This setting is available for WebFOCUS BUE Server and WebFOCUS BUE procedures. This setting should not be used when web server security is enabled. This includes Basic authentication, IWA, SSL, and third-party security products (such as SiteMinder). In these cases, the web server security settings can prevent WebFOCUS BUE from creating the Excel 2007/2010 file.</p>
FOCEXURL/ FOCHTMLURL	Default value is <code>http://localhost:8080</code>	Specifies the host name and port of the FOCEXURL/ FOCHTMLURL.
Settings for an individual Data Server:		
Name	Required.	<p>Name of the selected Data Server.</p> <p>Note: Data Server names are case-sensitive. Data Servers are defined in the WebFOCUS BUE Client as uppercase so you should also define Data Servers as uppercase in ReportCaster.</p>

Setting	Optional or Required/ Default Value	Descriptions and Possible Values
Default	Required.	This option is selected by default.
Set FOCEXURL/ FOCHTMLURL in the scheduled procedure	Required.	This setting is selected, by default. If this setting is not selected, the Distribution Server will not set the value of FOCEXURL or FOCHTMLURL for a scheduled procedure. Therefore, when cleared, if FOCEXURL or FOCHTMLURL is already set in the edasprof.prf file, this setting remains in effect, unless it is overridden in the scheduled procedure.
Security section		
Security Type	By default, this option is set to User.	<p>Static. A valid Execution Id and password is supplied in the User setting. When creating a schedule, you cannot specify an Execution Id and password.</p> <p>User. A valid Execution Id and password must be specified when creating a schedule.</p> <p>Shared. When creating a schedule, the user ID and password is internally assigned as the Execution Id and password.</p> <p>Note: Since the actual password is not stored in the WebFOCUS BUE repository, the Shared configuration can only be used when a password is not required to connect to the Reporting Server.</p> <p>Trusted. The Execution Id is the schedule owner and no password is sent to the WebFOCUS BUE Reporting Server when schedules run.</p> <p>Note: The Reporting Server must be configured to accept a Trusted connection when the Trusted option is selected.</p>
Graph section		
Graph Engine	Required. The value is GRAPH53.	Controls which graph engine to use for server-side graphics. By default, this specifies the GRAPH53 setting.

Setting	Optional or Required/ Default Value	Descriptions and Possible Values
Headless	By default, this option is not selected.	Determines whether a graphics card exists on the WebFOCUS BUE Reporting Server. When not selected (the default), a graphics card exists on the server. When selected, no graphics card exists on the server.
Maximum Connection/ Threads	Required. The default connection is 3.	Maximum number of connections available to the Reporting Server. You can specify a maximum of 20 connections. This setting works in conjunction with the optional Weight setting, enabling you to prioritize the alternate servers in a cluster queue.

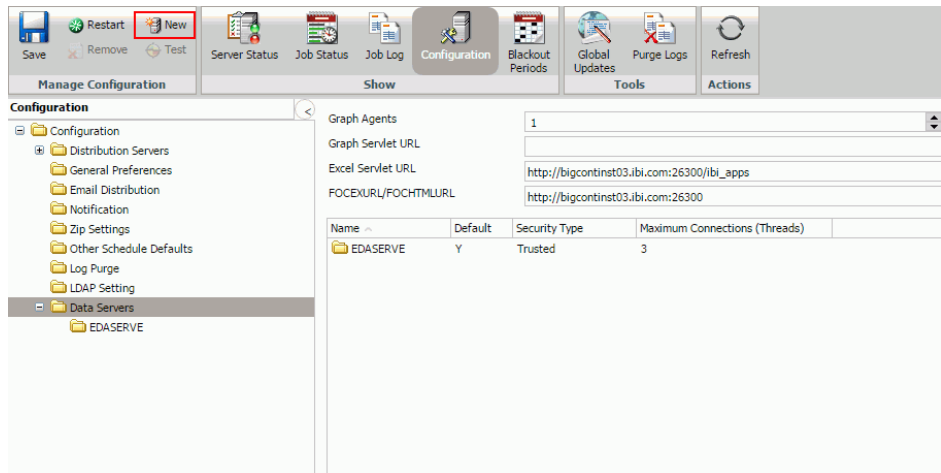
Configuring Reporting Servers With ReportCaster

You can configure multiple Reporting Servers with ReportCaster through the Data Server folder in the ReportCaster Configuration tab. Although the installation program automatically populates values for the default EDASERVE server, all additional Reporting Servers must be added to the configuration manually.

Note: If the default Reporting Server is changed on the Client, then the change will not be effective until the period specified by the IBFS Cache Cleaner service or until the Distribution Server is restarted. If the Distribution Server is installed on another machine with a separate copy of the Client, then a change to the default Reporting Server will not be effective until the same change is made on the Client installed on the same machine as the Distribution Server.

Procedure: How to Add a Reporting Server

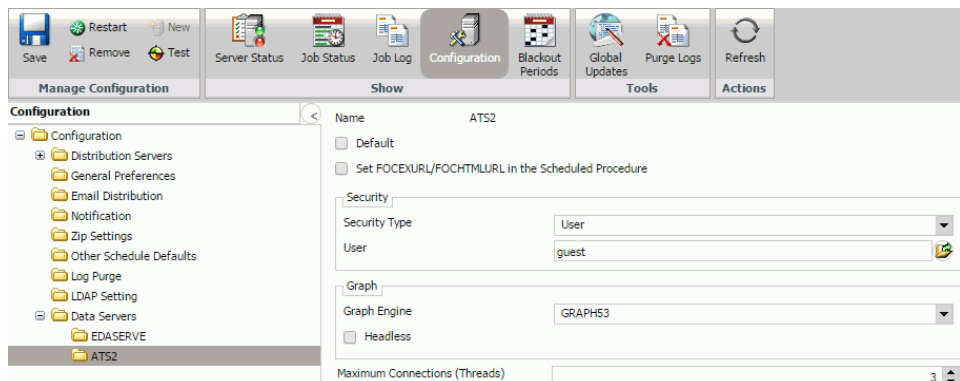
1. In the left panel of the Configuration tab, select the *Data Servers* folder, then select *New* in the toolbar, as shown in the following image.



Note: The default Data Server names are SERVER, SERVER 2, SERVER 3, and so on.

2. From the Available Data Servers dialog box, select one or more Data Servers, and click OK. You can optionally select the Select All check box.
3. Double-click the new server node in either the Data Servers folder in the left panel or the server list in the right panel.

The right panel displays the default properties for the selected server, as shown in the following image.



4. Provide values for the appropriate settings.

In the Name field, verify the name of the server you want to add to the ReportCaster configuration is the same as the NODE setting for that server specified in the WebFOCUS BUE Client Server Connections configuration. You can review the WebFOCUS BUE Client Server Connections by accessing the Administration Console and selecting Reporting Servers within the Configuration tab.

Optionally, provide values for the Security Type and Maximum Connections settings or leave the default values in place.

You can also optionally set the Data Server being added to be the default Data Server when new schedules are created, specify not to have the FOCXURL and FOCHTMLURL settings automatically added when scheduled jobs run, and specify that the Data Server does not have a graphics card, is Headless.

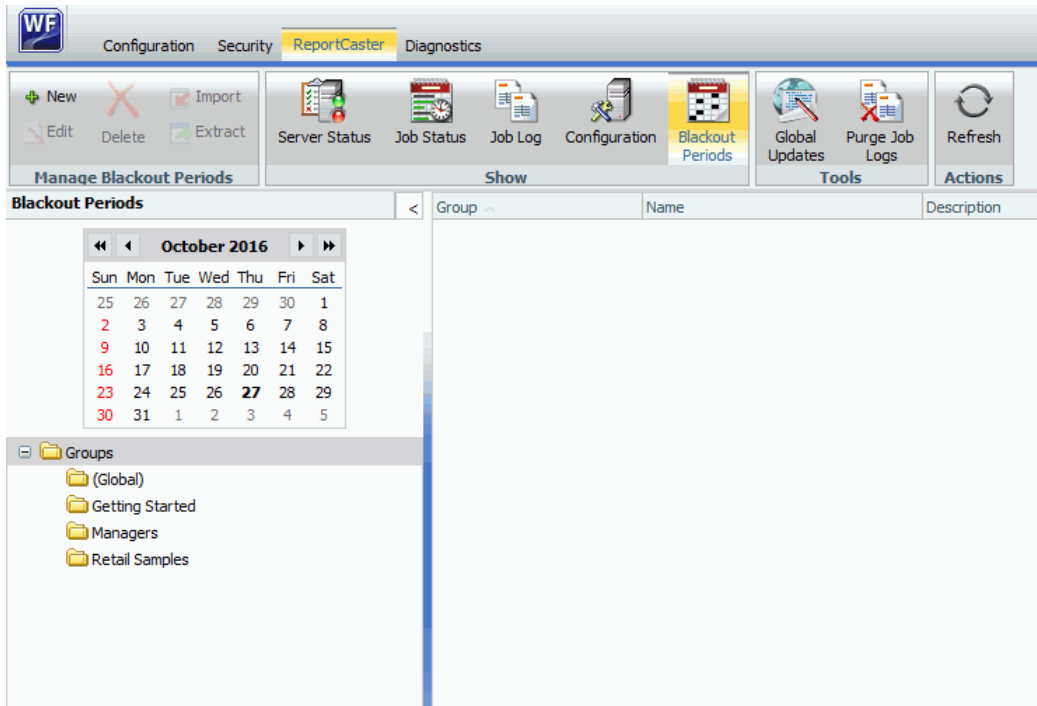
5. To implement the changes in ReportCaster, in the Configuration tab toolbar, select Save, then *Restart*.

Caution: Information Builders recommends that you avoid changing the name of a Reporting Server, since all existing jobs on that server will no longer run.

Using Blackout Periods

Blackout Periods are those dates and times on which schedules will not run and cannot be set to run. A user authorized to access the Blackout Periods tool can view, define, update, import, export, and delete blackout periods.

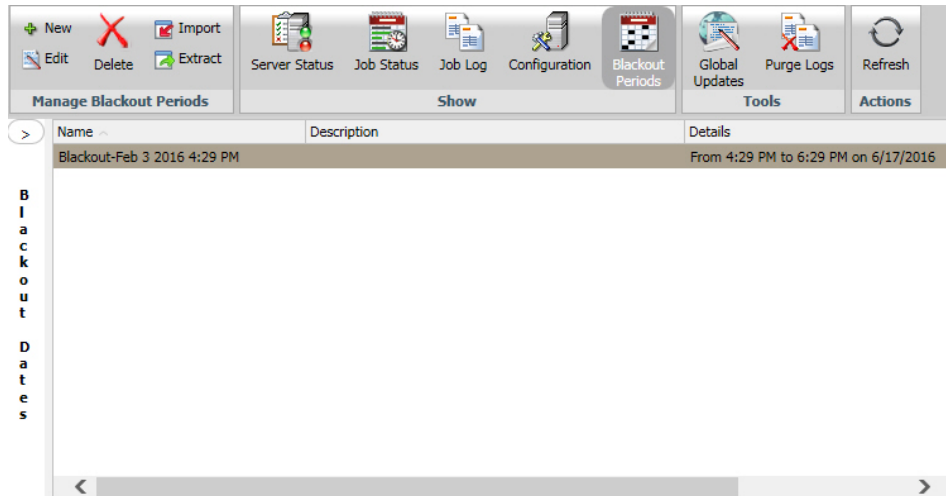
To view schedule blackout periods, select *Blackout Periods* from the Show group in the Console. The Blackout Periods interface, which is shown in the following image, provides a calendar in the left panel and the right panel lists the blackout dates you are authorized to manage.



Users authorized to manage blackout periods can add new blackout periods, delete blackout periods, replace the description of existing blackout periods, and extract existing blackout period information to a file for future use. In the ReportCaster Blackout Periods interface, these tasks can be completed using the following features:

- ☐ The Blackout Date and Time dialog box, accessed from the New and Edit buttons in the Manage Blackout Periods group.
- ☐ The Import Dates dialog box, accessed from the Manage Blackout Periods group.
- ☐ The Extract Blackout Dates dialog box, accessed from the Manage Blackout Periods group.

You can change the month or year using the arrows at the top of the calendar. Dates only appear as available or unavailable. You can show or hide the left panel by clicking the arrow in the top-right corner of the left panel. The following image shows the display with the left panel hidden.



Reference: Blackout Period Configurations

Every type of Blackout Period profile shares the same basic settings: Group Assignment, Name, Description, Details, and Blackout Time. Within that basic configuration, there are four types of Blackout Periods that accommodate differing scheduling requirements. These include:

- ☐ **Weekly Blackout Periods.** Recur on a specified day or days of the week.
- ☐ **Monthly Blackout Periods.** Recur on a specified monthly date or dates.
- ☐ **Single Day Blackout Periods.** Occur only once on a specified date.
- ☐ **Every Day Blackout Periods.** Recur at a specified time every day.

This variety of profiles enables you to build blackout periods into your reporting schedule that accommodate those regularly recurring and special one-day events that would cause you to suspend reporting.

Reference: Basic Blackout Period Settings

The Blackout Date and Time dialog box contains settings that define the timing and frequency of a scheduled Blackout Period. There are four variations of this dialog box to accommodate the Weekly, Monthly, Single Day, and Every Day frequency options.

The following table describes settings in the Global Blackout Date and Times profile.

Setting	Optional or Required/Default Value	Descriptions and Possible Values
Group	Displays the name (Global).	Displays the name (Global).
Name	Optional. By default displays the name: Blackout-[Current Date] [Current Time] For example: Blackout-Jun 26 2015 10:27 AM	A descriptive name for the Blackout Date and Time profile. You can modify the default name by clicking within the Name field.
Description	Optional.	A detailed description of the Blackout Date and Time profile.
Details Label	Assigned to the profile after creation.	A summary of the detailed times and frequency of the Blackout Period. This is a narrative description of the selections you make in the Weeks, Days, Months, and Blackout Time fields. ReportCaster creates this description automatically when you save the profile, and updates it when you save changes to it. You cannot create, edit, or delete it directly.

Setting	Optional or Required/Default Value	Descriptions and Possible Values
Blackout Time check box	By default this check box is cleared.	<p>Selected. The Blackout Period covers the hours specified in the Start (Time) and End (Time) fields.</p> <p>Not Selected. The Blackout Period covers the entire day.</p> <p>Note: If you change a profile from Single Date to any other frequency setting, you must clear this check box if the blackout period is to cover the entire day.</p>
Start (Time)	Optional. By default, displays the current hour and minute.	<p>The Hour and Minute in which a Blackout Period is to begin.</p> <p>This value is relevant only if the Blackout Time check box is selected.</p>
End (Time)	Optional. By default, displays the time two hours <i>after</i> the current hour and minute.	<p>The Hour and Minute in which a Blackout Period is to end.</p> <p>This value is relevant only if the Blackout Time check box is selected.</p>
Frequency	Required. By default, the Weekly option is selected.	<p>Weekly. Blackout periods that recur on a specified day of the week.</p> <p>Monthly. Blackout periods that recur on a specified monthly date.</p> <p>Single Day. Blackout periods that occur once on a specified date.</p> <p>Every Day. Blackout periods that recur at a specified time every day.</p>

Procedure: How to Configure a Blackout Period

To configure a blackout period:

1. From the Manage Blackout Periods group on the ribbon, click **New**.

The Blackout Date and Time dialog box opens, as shown in the following image.

Untitled - Blackout Date and Time

Group: (Global)

Name: Blackout-Apr 14 2016 7:02 PM

Description:

Details:

☒ Blackout Time (Uncheck this box to black out the entire day)

The time zone for the Start Time and End Time is Eastern Standard Time (GMT-0400) .

Start: 7:02 PM

End: 9:02 PM

☒ Weekly On: ☐ Select All ☐ Select All

☐ Monthly ☐ First ☐ Sunday

☐ Single Day ☐ Second ☐ Monday

☐ Every Day ☐ Third ☐ Tuesday

☐ Fourth ☐ Wednesday

☐ Last ☐ Thursday

☐ Friday

☐ Saturday

Months: ☐ Select All

☐ January ☐ February ☐ March ☐ April

☐ May ☐ June ☐ July ☐ August

☐ September ☐ October ☐ November ☐ December

OK Cancel

2. Accept the default Name assigned to the Blackout Period profile or enter a new value for Name.
3. Type a description for the Blackout Period profile in the Description field.
4. If this blackout period must cover the entire day, leave the Blackout Time check box cleared and skip to step 7.
5. If this blackout period must be limited to a range of hours within a day, select the Blackout Time check box and configure the start and end time for the Blackout Period.

For more information, see [How to Configure a Start Time and End Time for a Blackout Period](#) on page 194.

6. Click the appropriate frequency option:
 - ☐ **Weekly.** Establish the frequency for the blackout period. For configuration information, see [Configuring Weekly Blackout Periods](#) on page 186.
 - ☐ **Monthly.** Establish the frequency for the blackout period. For configuration information, see [Configuring Monthly Blackout Periods](#) on page 189.
 - ☐ **Single Day.** Select the date for the blackout period. For configuration information, see [Configuring Single Day Blackout Periods](#) on page 192.
 - ☐ **Every Day.** Establish the hours for the daily blackout period. For configuration information, see [Configuring Every Day Blackout Periods](#) on page 193.
7. Review your configuration.
 - ☐ If your configuration of dates and hours is unacceptable, the OK button will not respond when you attempt to click OK. Adjust your configuration and refresh the profile by clearing and reselecting the recurring check boxes.
 - ☐ If your configuration is acceptable, the OK button will be available, and the Details label will include a description summarizing your selections.
8. When your configuration is complete, click **OK**.

An entry for the Blackout Profile appears in the right pane of the Blackout Dates window and the Blackout Date Calendar in the left pane highlights the new blackout dates.

Configuring Weekly Blackout Periods

The Weekly Blackout Period configuration bases its recurrence on a specified day of the week. It suits events that recur as part of a weekly schedule, regardless of the date on which that day falls.

Even though the name implies that this Blackout Period occurs only once a week, Blackout Periods using the weekly configuration can occur more or less frequently.

The tools in this configuration enable you to specify:

- ☐ The day, or days, of the week on which the blackout period will occur.
- ☐ The week, or weeks, of the month in which the blackout period will occur.
- ☐ The month, or months, of the year in which the blackout period will occur.

Instead of requiring you to enforce blackout periods on the same day each and every week, this flexible configuration enables you to schedule Blackout Periods that occur more than once a week. It also enables you to configure Blackout Periods that skip one or more weeks in a month or skip one or more months in a year.

Weekly Blackout Period Settings

If you select the Weekly option, the Blackout Date and Time dialog box displays a unique set of options that enables you to schedule Blackout Periods for specified days of the week.

Setting	Optional or Required/ Default Value	Descriptions and Possible Values
On Week of the Month Check Boxes (Left Column)	Required (At least one selection from this column.)	<p>The week of the month check boxes list ordinal weeks of the month. When you select one, you define the week of the month in which this blackout period is to occur. That is, the first week of the month, the second week, and so on.</p> <p>You can select one or more individual weeks.</p> <p>The Select All check box automatically selects every week of the month, establishing a blackout period that takes place every week in a month.</p>

Setting	Optional or Required/ Default Value	Descriptions and Possible Values
On Days of the Week check boxes (Right Column)	Required (At least one selection from this column).	<p>The day of the week check boxes list the days of the week. When you select one, you define the day of the week in which this blackout period will occur. That is, on Monday, Tuesday, and so on.</p> <p>You can select one or more individual days.</p> <p>The Select All check box automatically selects every day of the week, establishing a blackout period that takes place every day of your selected weeks in the month.</p>
Months	Required (At least one selection from this group).	<p>The Month check boxes list the months of the year. When you select one, you define the month of the year in which this weekly black out period will occur. That is, in January, February, and so on.</p> <p>You can select one or more individual months.</p> <p>The Select All check box automatically selects every month of the year, establishing a blackout period that takes place during every month on your selected week and day.</p>

***Procedure:* How to Configure Weekly Blackout Period Settings**

To configure weekly blackout period settings:

1. From the Blackout Date and Time dialog box, click *Weekly*.

The dialog box displays check boxes that support the Weekly frequency schedule, as shown in the following image.

The dialog box shows the following options:

- ☒ Weekly
- ☐ Monthly
- ☐ Single Day
- ☐ Every Day

On:

- ☐ Select All
- ☐ First
- ☐ Second
- ☐ Third
- ☐ Fourth
- ☐ Last
- ☐ Sunday
- ☐ Monday
- ☐ Tuesday
- ☐ Wednesday
- ☐ Thursday
- ☐ Friday
- ☐ Saturday

Months:

- ☐ Select All
- ☐ January
- ☐ February
- ☐ March
- ☐ April
- ☐ May
- ☐ June
- ☐ July
- ☐ August
- ☐ September
- ☐ October
- ☐ November
- ☐ December

Note: In order to enable the OK button and save the profile, you must select at least one week, day, and month check box.

2. Select the check boxes for the week or weeks of the month in which the Blackout Period is to occur, or click *Select All* to select every week automatically.
3. Select the check boxes for the day or days of the week on which the Blackout Period is to occur, or click *Select All* to select every day of the week automatically.
4. Select the check boxes for the month or months in which the Blackout Period is to occur, or click *Select All* to select every month automatically.

Configuring Monthly Blackout Periods

The Monthly Blackout Period configuration bases its recurrence on a specified monthly date. It suits events that recur on the same date, regardless of the day of the week on which that date falls.

Even though the name implies that this Blackout Period occurs only once a month, Blackout Periods using this configuration can occur more or less frequently. They can occupy the entire day, or they can be limited to a range of hours within a single day.

The tools in this configuration enable you to specify:

- ☐ The day, or days, of the month on which the blackout period will occur.
- ☐ The month, or months, of the year in which the blackout period will occur.

Instead of requiring you to enforce blackout periods on the same day of the month, each and every month, this flexible configuration enables you to schedule Blackout Periods that occur more than once a month. It also enables you to configure Blackout Periods that skip one or more months.

Monthly Blackout Period Settings

If you select the *Monthly* option, the Blackout Date and Time dialog box displays a unique set of options that enables you to schedule Blackout Periods for specified dates of the month.

Setting	Optional or Required/Default Value	Descriptions and Possible Values
Days	Required.	<p>The Days check boxes list the dates within a month. When you select one, you define the day of the month on which this blackout period will occur. That is, on the first day of the month, the second, the third, and so on.</p> <p>You can select one or more individual dates.</p> <p>The Select All option automatically selects every day of the month, establishing a blackout period that takes place every day during your selected months.</p>
Months	Required.	<p>The Months check boxes list the months of the year. When you select one, you define the month in which this black out period will occur. That is, in January, February, and so on.</p> <p>You can select one or more individual months.</p> <p>The Select All option automatically selects every month of the year, establishing a blackout period that takes place during every month on your selected day or days.</p>

Procedure: How to Configure Monthly Blackout Period Settings

To configure monthly blackout period settings:

1. From the Blackout Date and Time dialog box, click *Monthly*.

The Blackout Date and Time dialog box displays the check boxes that support the Monthly frequency, as shown in the following image.

The screenshot shows a dialog box with the following sections:

- Frequency Selection:** Four radio buttons are listed: *Weekly*, *Monthly* (selected), *Single Day*, and *Every Day*.
- Days:** A grid of checkboxes for each day of the month (1-31) and a *Last* option. The *Select All* checkbox is also present.
- Months:** A grid of checkboxes for each month (January through December) and a *Select All* checkbox.

Note: At a minimum, you must select a check box for one day and a check box for one month. Until you do, the OK button will be unavailable, and you will be unable to save the profile.

2. Select the check boxes for the day or days of the month on which the Blackout Period is to occur, or click *Select All* to select every date in the month automatically.
3. Select the check boxes for the month or months in which the Blackout Period is to occur, or click *Select All* to select every month automatically.

Configuring Single Day Blackout Periods

Single Day Blackout Periods occur once, on a single, specified date. They can occupy the entire day, or they can be limited to a range of hours within a single day. The settings for Single Day Blackout Periods are shown in the following image.

Untitled - Blackout Date and Time

Group:

(Global)

Name:

Blackout-Feb 3 2016 2:41 PM

Description:

Details:

From 2:41 PM to 4:41 PM on 2/3/2016

☒ Blackout Time (Uncheck this box to black out the entire day)

The time zone for the Start Time and End Time is Eastern Standard Time (GMT-0500) .

Start:

2: 41 PM

End:

4: 41 PM

☐ Weekly

☐ Monthly

☒ Single Day

☐ Every Day

February 2016

Sun	Mon	Tue	Wed	Thu	Fri	Sat
31	1	2	3	4	5	6
7	8	9	10	11	12	13
14	15	16	17	18	19	20
21	22	23	24	25	26	27
28	29	1	2	3	4	5
6	7	8	9	10	11	12

OK

Cancel

Single Day Blackout Period Settings

If you select the *Single Day* option, the Blackout Date and Time dialog box displays a copy of the Blackout Dates calendar, which enables you to select a single date for a scheduled Blackout Period.

Procedure: How to Configure Single Day Blackout Period Settings

1. From the Blackout Date and Time dialog box, click *Single Day*.
- The dialog box appears, displaying the calendar automatically set to the current date.

2. Click the single arrows to change the month and click the double arrows to change the year.
3. When you have found the month and year for the scheduled single day blackout period, click on the date on which you want the Blackout Period to occur.

The calendar highlights the new date.

Configuring Every Day Blackout Periods

Every Day Blackout Periods recur every day. To prevent Every Day Blackout Periods from blocking the release of any reports, you must limit them to a range of hours within the day. The settings for Every Day Blackout Periods are shown in the following image.

Untitled - Blackout Date and Time

Group: (Global)

Name: Blackout-Feb 3 2016 2:41 PM

Description:

Details: From 2:41 PM to 4:41 PM

☒ Blackout Time (Uncheck this box to black out the entire day)

The time zone for the Start Time and End Time is Eastern Standard Time (GMT-0500) .

Start: 2: 41 PM

End: 4: 41 PM

☐ Weekly
☐ Monthly
☐ Single Day
☒ Every Day

Select the start time and end time to create a blackout time for every day of the year

OK Cancel

Every Day Blackout Period Settings

If you select the *Every Day* option, the Blackout Date and Time dialog box limits its display to the Beginning and End time for the Blackout Period. No other options are relevant.

Procedure: How to Configure Every Day Blackout Period Settings

1. From the Blackout Date and Time dialog box, click *Every Day*.

The Blackout Time check box is selected automatically, and the dialog box displays a note reminding you to select the start and end time.

The Start time is automatically set to the current time, and the End time is automatically set to a value two hours later.
2. To change the Start or End Time values, see [How to Configure a Start Time and End Time for a Blackout Period](#) on page 194.

Procedure: How to Configure a Start Time and End Time for a Blackout Period

From the Blackout Date and Time dialog box:

1. To change the hour of the start time, click in the *Hour* section of the Start field.
 - a. Click the up or down arrow to move the value ahead one hour or back one hour, respectively.
 - b. You can also type the hour in this section, but be careful to stay within the twelve hour range. If you enter a value outside of this range, the dialog box will automatically recalculate your value in terms of a twelve hour clock. For example, 44 hours would become 8. $(44 - (12 \times 3)) = (44 - 36) = 8$
 - c. Ensure that the start time is earlier than the end time.
2. To change the minutes of the start time, click in the *Minutes* section of the Start field.
 - a. Click the up or down arrow to move the value ahead one minute or back one minute, respectively.
 - b. You can also type the minutes in this section, but be careful to stay within the sixty minute range. If you enter a value outside of this range, the dialog box will automatically recalculate your value in terms of the number of minutes within an hour. For example, 88 minutes would become 28. $(88 - 60) = 28$.
 - c. Ensure that the start time is earlier than the end time.
3. To change the start time from AM to PM, click in the *AM/PM* section of the Start field.
 - a. Click the up arrow to move from AM to PM.
 - b. Click the down arrow to move from PM to AM.

- c. You can also type AM or PM directly into this section.
- 4. To change the hour of the end time, click in the *Hour* section of the End field.
 - a. Click the up or down arrow to move the value ahead one hour or back one hour, respectively.
 - b. You can also type the hour in this section, but be careful to stay within the twelve hour range. If you enter a value outside of this range, the dialog box will automatically recalculate your value in terms of a twelve hour clock. For example, 44 hours would become 8. $(44 - (12 \times 3)) = (44 - 36) = 8$.
 - c. Ensure that the end time is later than the start time.
- 5. To change the minute of the end time, click in the *Minutes* section of the End field.
 - a. Click the up or down arrow to move the value ahead one minute or back one minute, respectively.
 - b. You can also type the minute in this section, but be careful to stay within the sixty minute range. If you enter a value outside of this range, the dialog box will automatically recalculate your value in terms of the number of minutes within an hour. For example, 88 minutes would become 28. $(88 - 60 = 28)$.
 - c. Ensure that the end time is later than the start time.
- 6. To change the end time from AM to PM, click in the *AM/PM* section of the End field.
 - a. Click the up arrow to move from AM to PM.
 - b. Click the down arrow to move from PM to AM.
 - c. You can also type AM or PM directly into this section.

Important: You must use the right or left arrow keys to move between the Hour, Minutes, and AM/PM sections of the Start and End fields.

If you try to use the Tab key to move between sections, you will move down to the next option instead of across to the next section of the Start or End field.

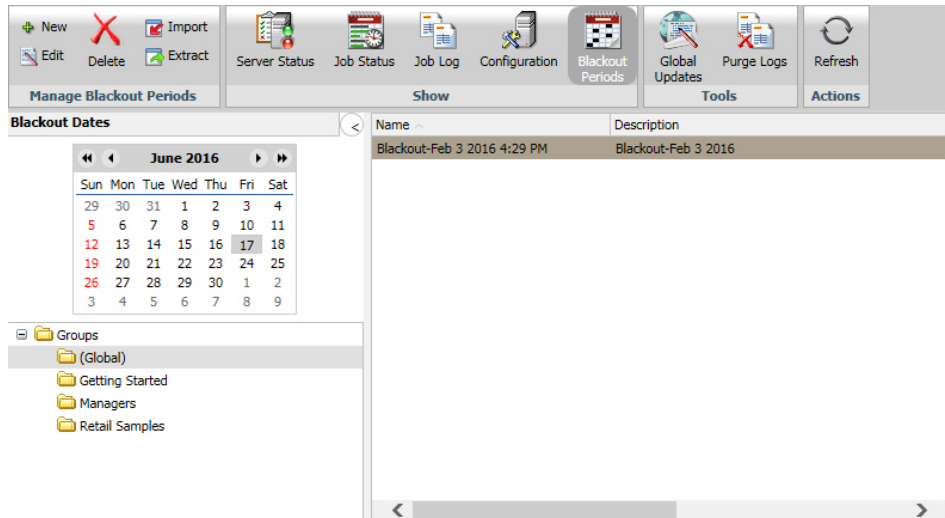
For example, if you use the Tab key to move from the Hours section of the Start field, you will move down to the Hours section of the End field, not over to the Minutes section of the Start field. To move right, you must use the right direction key instead.

Similarly, the Shift+Tab key combination will move you to the previous option, not back to a previous section within the Start or End field. To move left, you must use the left direction key instead.

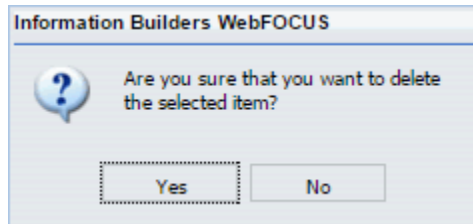
Procedure: How to Delete a Blackout Period Profile

1. In the left pane of the Blackout Dates dialog box, click the folder for the Group to which the Blackout Period profile you want to delete is assigned.

2. In the right pane, click the entry for Blackout Period profile you want to delete, as shown in the following image.



3. On the ribbon, in the Manage Blackout Periods group, click *Delete*.
4. When you receive a confirmation message, click Yes, as shown in the following image.



Your newly deleted entry disappears from the right pane and the dates assigned to that entry are no longer highlighted on the Blackout Dates calendar in the left pane.

Importing Blackout Periods

You can use the Blackout Period Import operation to automate Blackout Period management tasks. Using a properly formatted import file with this tool, you can:

- ☐ Create new Blackout Period profiles.
- ☐ Remove Blackout Period profiles.
- ☐ Enrich Blackout Period information.

Note: All imported files must contain UTF-8 (National Language Support) character encoding to upload correctly.

The source file for the import contains the following information about a Blackout Period Profile:

- ☐ **Date.** The date on which the Blackout Period profile becomes effective. This date can be omitted for Every Day Blackout Period profiles.
- ☐ **Start Hour.** The time (HH:MM:SS) at which the Blackout Period becomes effective. This value is optional for all but Every Day Blackout Period profiles.
- ☐ **End Hour.** The time (HH:MM:SS) at which the Blackout Period ceases to be effective. This value is optional for all but Every Day Blackout Period profiles.
- ☐ **Description.** A detailed description of the Blackout Period profile.
- ☐ **Name.** The unique name assigned to a Blackout Period profile.

Note: Additional Codes precede entries for Weekly or Monthly Blackout Period profiles. For more information, see [Weekly Blackout Period Import File Entry Layout](#) on page 198 and [Monthly Blackout Period Import File Entry Layout](#) on page 199.

The task you select when you run an import operation determines the way in which ReportCaster uses the information in the source file for the Import.

If you select:

- ☐ *Add*, the import creates new Blackout Period profiles from the records in the import source file.
- ☐ *Remove*, the import compares date and additional information in import file records to that of existing Blackout Period profiles and removes those profiles that match the file records.
- ☐ *Replace*, the import compares date information in import file records to that of existing Blackout Period profiles and adds Name and Description information to them.
- ☐ *Use the actions specified in the file*, the import runs the add, remove, and replace operations as required by commands included in the Import file along with entries for the Blackout Period profiles affected by them.

The automation of these tasks frees you from manually creating, updating, or deleting Blackout Period profiles when operational changes, such as monthly, quarterly, or annual operation schedule updates, require a sweeping review and revision of your reporting schedule.

For example, at the start of the year, you have a file containing the dates and names of all scheduled holidays that must be incorporated into your reporting schedule. The Blackout Period Import enables you to import that file and establish the holiday blackout schedule for an entire year in a single operation. Without the import you would have to create a Single Date Blackout Date and Time profile for each holiday in the coming year.

Viewing Blackout Period Import File Format

Even though entries in the Import File format for Blackout Period records use the same basic structure, each profile type contains minor variations that identify the type of Blackout Period profile ReportCaster must create from the entry details.

Single Date Blackout Period Import File Entry Layout

You can import Single Date Blackout Date and Time profiles from a flat file that lists the information for each profile on a separate line. Each line must follow the following layout:

```
[Date]/[Start Hour]| [End Hour] [Description]| [Name]
```

For example:

```
20150629/08:59:00|10:59:00 Report Blackout for June 29th Only|Special  
One Day Blackout - June 29th
```

- ☐ Dates use the format YYYYMMDD
- ☐ Hours use the format HH:MM:SS
- ☐ You can omit the hours to schedule an all day blackout.

The Weekly, Monthly, and Every Day Blackout Date and Time profiles use a variation of this format.

Weekly Blackout Period Import File Entry Layout

You can import Weekly Blackout Date and Time profiles from a flat file that lists the information for each profile on a separate line. Each line must follow the following layout:

```
[Blackout Pattern] [Description]| [Name]
```

For example:

```
[111111111111/10000:0000010]/17:00:00|23:59:00 Report Blackout First  
Friday of Every Month 5:00 PM to 11:59 PM|First Friday Afternoons
```

This information identifies the entry as a Weekly Blackout Period, and includes its Description and Name. The Blackout Pattern indicates the selected Month(s) and Selected Day(s) of the Week. A '1' means that a month or day is selected. A '0' means that a month or day is not selected.

Monthly Blackout Period Import File Entry Layout

You can import Monthly Blackout Date and Time profiles from a flat file that lists the information for each profile on a separate line. Each line must follow the following layout:

```
[Blackout Pattern]/[Start Hour]|[End Hour] [Description]|[Name]
```

For example:

```
[111111111111/10000:0000010]/17:00:00|23:59:00 Report Blackout First  
Friday of Every Month 5:00 PM to 11:59 PM|First Friday Afternoons
```

This information identifies the entry as a Monthly Blackout Period, and includes its Description and Name. The Blackout Pattern indicates the selected Month(s) and Selected Day(s) of the Month. A '1' means that a month or day is selected. A '0' means that a month or day is not selected.

Every Day Blackout Period Import File Entry Layout

You can import Every Day Blackout Date and Time profiles from a flat file that lists the information for each profile on a separate line. Each line must follow the following layout:

```
/[Start Hour]|[End Hour] [Description]|[Name]
```

For example:

```
/21:04:00|23:04:00 Every Day Blackout between 3:00 PM and Midnight|  
Daily Afternoon Blackout
```

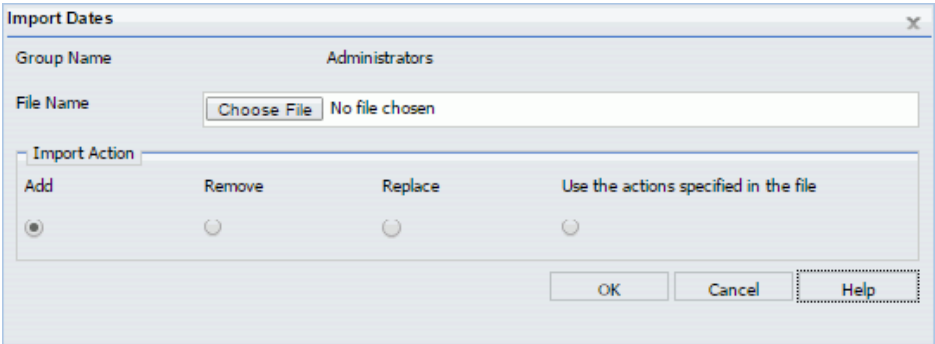
This information identifies the entry as an Every Day Blackout Period and includes its Description and Name.

Procedure: How to Add Blackout Period Profiles Using an Import File

To add a Blackout Period profile through the import is to create a new Blackout Period profile from information in the import file. Your Import file must therefore contain records for all Blackout Period profiles you want to add to ReportCaster through this import operation.

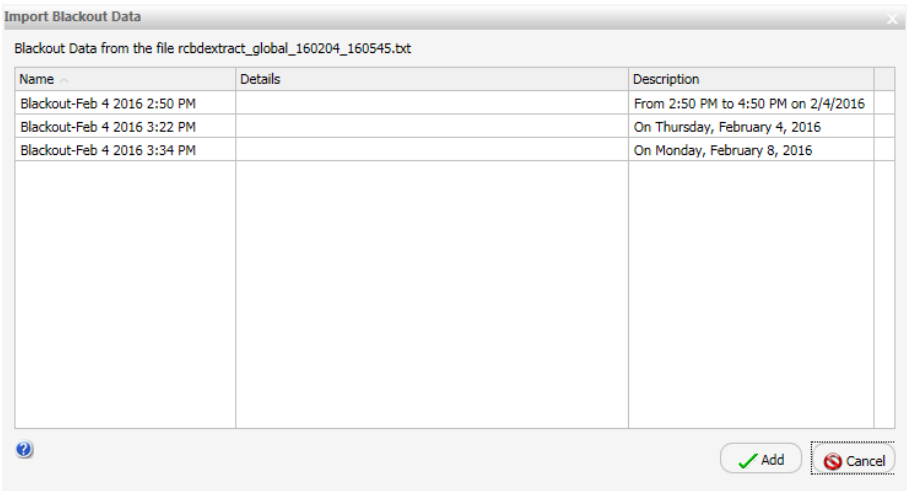
1. In the Blackout Dates pane, click the folder of the group for which you are importing Blackout Period profiles.

- 2. On the ribbon, in the Manage Blackout Periods group, click *Import*.



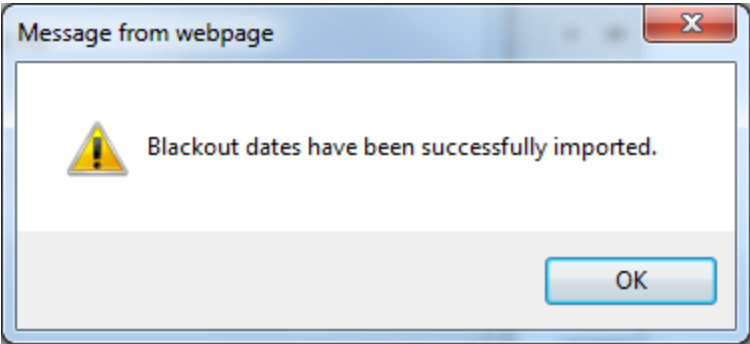
- 3. In the File Name field, type the full path to the file, or click *Browse* and navigate to the file you want to import.
- 4. Click *Add*, and click *OK*.

The Import Blackout Data dialog box opens, listing details of the new Blackout Profiles based on each entry in the Import file as shown in the following image.



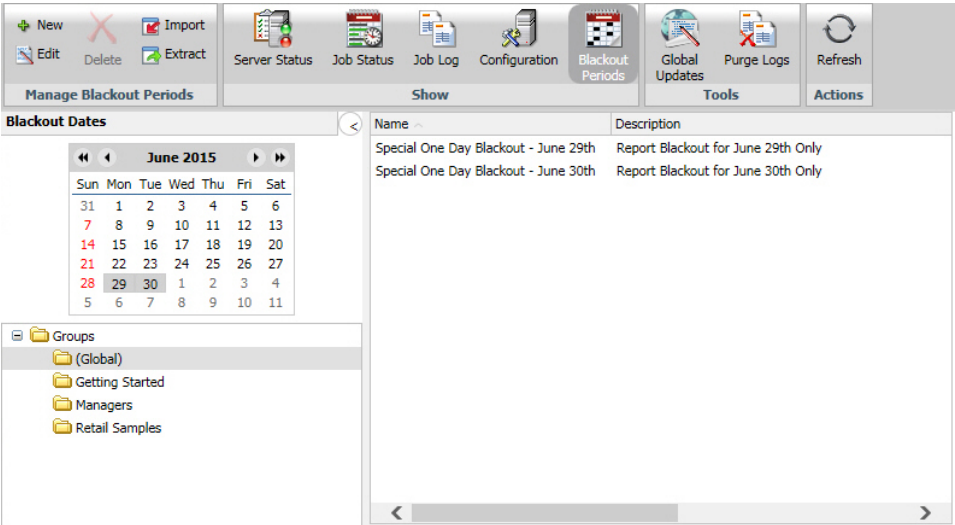
- 5. Click *Add* to add the new Blackout Period profiles listed in the Import Blackout Data dialog box to the Blackout Dates Calendar and Group entries.

A message displays, informing you that the blackout dates have been successfully imported, as shown in the following image.



- 6. Click OK.

The Blackout Dates calendar highlights the newly imported Blackout Period Dates in the left pane of the Blackout Dates window and entries for the new Blackout Period profiles appear in the right pane, as shown in the following image.



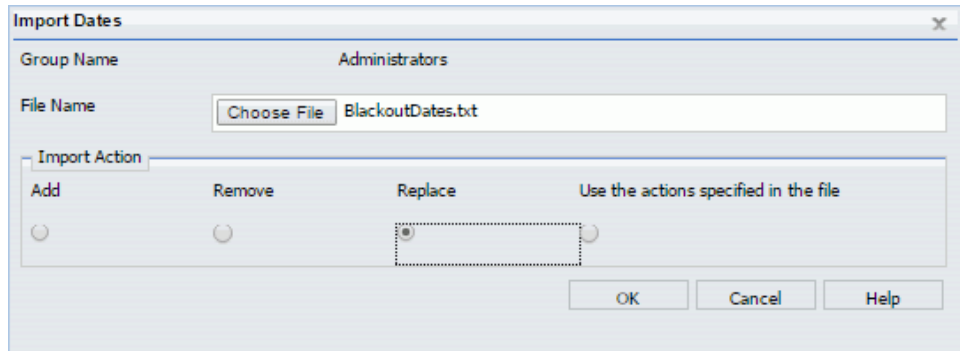
Procedure: How to Replace Blackout Periods Using an Import File

To replace a Blackout Period profile through the import is to add updated information to its Name and Description fields. To ensure that the import-based update affects the proper profiles, your Import file must contain records whose dates match those of the existing Blackout Period profiles that must be enriched by this Import operation.

Note: You *must* use this procedure to update or enrich information assigned to a Blackout Period profile created from an Import.

1. In the right pane of the Blackout Dates dialog box, select the group for which you are importing Blackout Period profiles.
2. On the ribbon, in the Manage Blackout Periods group, click *Import*.

The Import Dates dialog box opens, as shown in the following image.

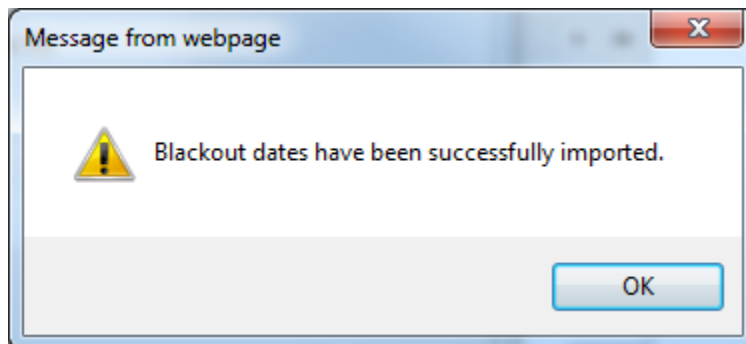


3. In the File Name field, type the full path to the file, or click *Browse* and navigate to the file you want to import.
4. Click *Replace*, and then click *OK*.

The Import Blackout Data dialog box opens, listing the details from each entry in the Import file that will be added to their corresponding Blackout Period profiles.

5. Click *Replace* to add new the information listed in the Import Blackout Data dialog box to the designated Blackout Period profiles.

A message displays, informing you that the blackout dates have been successfully imported, as shown in the following image.



6. Click *OK*.

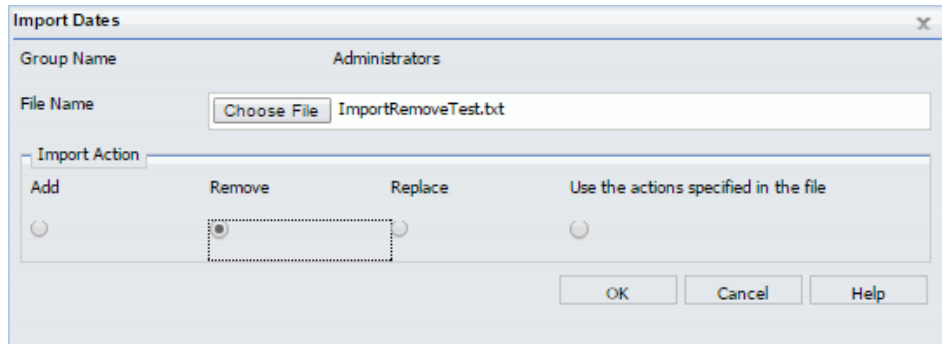
Enriched entries for the updated Blackout Period profiles appear in the right pane.

Procedure: How to Remove Blackout Periods Using an Import File

Removing a Blackout Period profile through the import deletes it. To ensure that the import-based deletion affects the proper profiles, your Import file must contain records whose dates match those of the existing Blackout Period profiles that are to be deleted by this Import operation.

1. In the Blackout Dates tab pane, select the group containing blackout period profiles that must be removed.
2. On the ribbon, in the Manage Blackout Periods group, click *Import*.

The Import Dates dialog box opens, as shown in the following image



3. In the File Name field, type the full path to the file, or click *Browse* and navigate to the file you wish to import.
4. Click *Remove*, and then click *OK*.

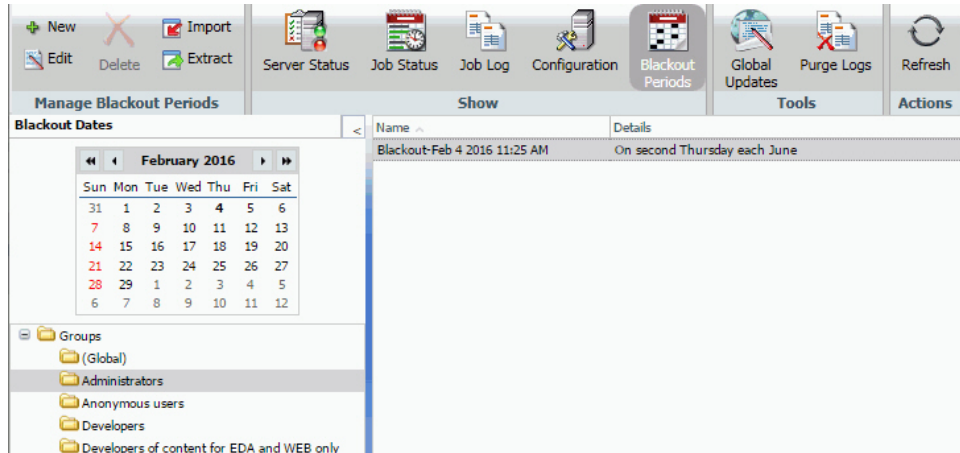
The Import Blackout Data dialog box opens, listing details from each entry in the Import file that successfully matched an existing Blackout Period profile.

5. Click *Remove* to delete those Blackout Period profiles listed in the Import Blackout Data dialog box.

A window opens, informing you that the blackout dates have been successfully imported.

6. Click *OK*.

Blackout Period profiles removed by this operation are no longer highlighted in the left pane of the Blackout Dates window or as entries for the new Blackout Period profiles in the right pane, as shown in the following image.



Procedure: How to Manage Multiple Blackout Period Updates Using an Import File

The *Use the actions specified in the file* option allows the import process to add, remove, and replace multiple Blackout Period profiles in a single operation. Your Import file must therefore contain the all necessary commands followed by entries representing Blackout Period profiles that must be added, removed, or enriched by the import operation.

1. In the right pane of the Blackout Dates dialog box, select the group for which you are managing Blackout Period profiles.
2. On the ribbon, in the Manage Blackout Periods group, click *Import*.

The Import Dates dialog box opens.

3. In the File Name field, type the full path to the file, or click *Browse* and navigate to the file you want to import.
4. Click *Use the actions specified in the file*, and then click *OK*.

The Import Blackout Data dialog box opens, listing details from each entry in the Import file and a note about its proposed update.

5. Click *OK*.
6. When you receive the *Blackout Dates have been successfully imported* message, click *OK*.

The Blackout Dates calendar highlights the dates of the newly imported Blackout Periods in the left pane of the Blackout Dates window. Entries for the new and enriched Blackout Period profiles appear in the right pane. Calendar highlights and entries for profiles removed by the Import no longer appear.

Note: You will be unable to use the Edit command to update Blackout Period profiles created from a direct file import. You will be required to update these profiles using the Replace option in the Import operation.

Extracting Blackout Period Profiles

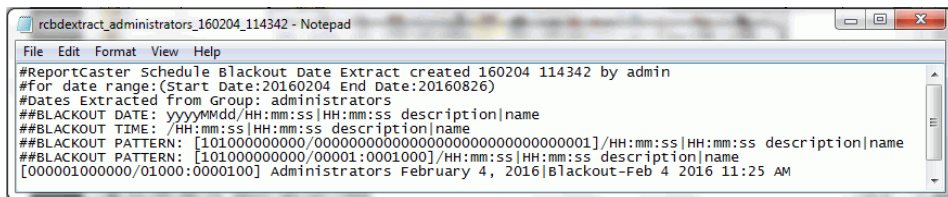
You can extract existing blackout period profile information from the ReportCaster Development Interface to a .txt file.

A direct file extract minimizes the time it would take to transfer information about multiple Blackout Periods to a text file that can serve as a backup of your Blackout Calendar configuration or as the source of an import to an external system for reporting or auditing purposes.

For example, if your group is required to maintain a backup copy of blackout dates for the coming six months, the extract operation enables you to transfer the Description, Date, and Name of all scheduled blackout periods during that time. You can recreate basic information about these profiles from the backup file in an emergency, shortening the time required to reconstruct your reporting schedule.

Extracting File Formats

The Import and Extract file operations use the same layout and format conventions. The Extract file contains additional comments that document the range of dates selected for the extract and templates for the format of the entries it contains, as shown in the following image.



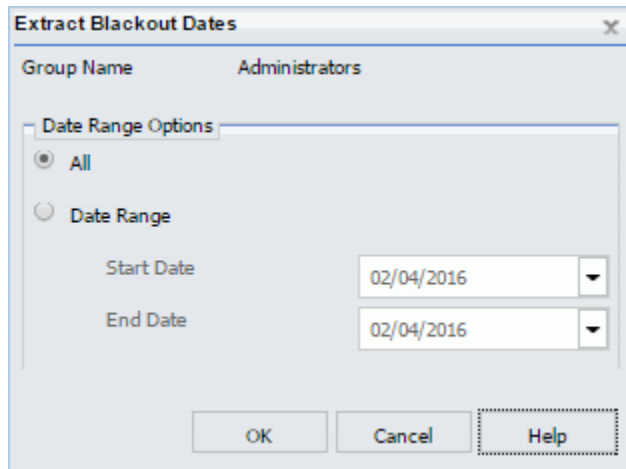
For more information about each entry type layout, see [Viewing Blackout Period Import File Format](#) on page 198.

Procedure: How to Extract Schedule Blackout Period Profiles to a File

To extract schedule blackout period profiles:

1. In the Blackout Dates pane, select the group from which you are extracting blackout period profiles.
2. In the Manage Blackout Periods Toolbar group, click *Extract*.

The Extract Blackout Dates dialog box opens. The Group Name from which you are extracting the blackout dates appears at the top of the dialog box, as shown in the following image.

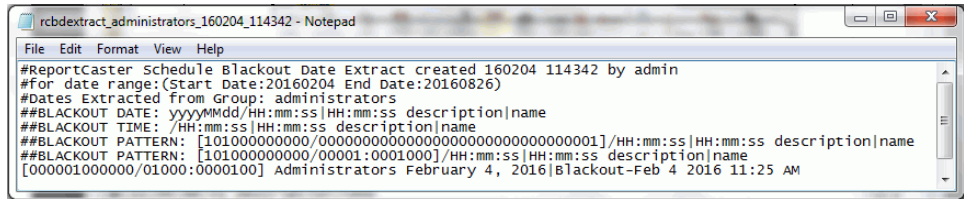


3. Select one of the following from the Date Range Options:
 - ☐ *All* to extract all currently saved blackout period profiles.
 - ☐ *Date Range* to specify the range of dates containing scheduled blackout period profiles that you want to extract.

If you select *Date Range*, type or select the Start Date and End Date. To select a date, click the down arrow next to the field. A calendar opens, from which you can choose a date.

4. Click *OK*.
5. To open the file, click the *Open* button when your browser presents it.

A window opens, displaying the contents of the extracted content profiles, as shown in the following image.



You can save and close the file using the commands in the File menu.

6. To accept an automatic download of the extract file, click the Save button (users of Google Chrome need only close the page displaying the new file name).

You can later retrieve the file from the Downloads folder of your computer, rename it if necessary, and save a copy of it in another folder for archival or other purposes.

7. To rename and save the file in a different location, click the *Open* or *Open With* button, and select the Save As command from the program that opens the file.
8. From the Save As window, navigate to the folder in which you want to save the file, rename the file if necessary, and click Save.

You can close any additional web pages that may remain open after you save the file.

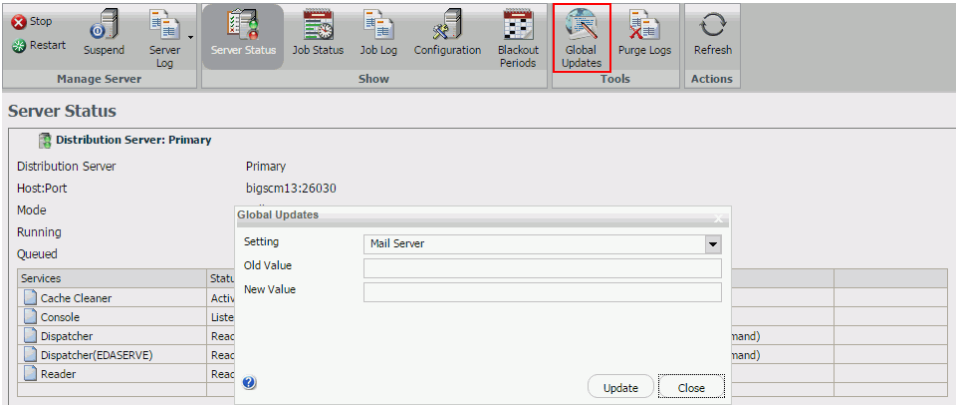
Note: The default extract file name is `rcbdextract_GroupName_YYMMDD_HHMMSS.txt`, where *GroupName* is the name of the group from which the blackout dates are extracted, YYMMDD and HHMMSS are the date (year, month, day) and time (hour, minute, second) that the file was created.

Global Updates

Authorized users can make global updates in the Global Updates interface for the Mail Server, Printer, Email Address, Email from, and Data Server values stored in schedules and distribution lists.

Procedure: How to Make a Global Update for a Mail Server

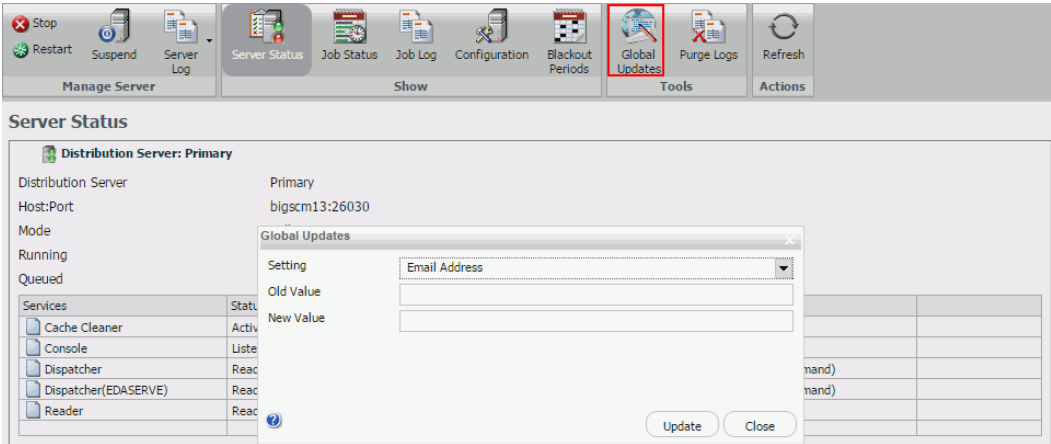
- 1. In the Global Updates interface, click the *Setting* drop-down list and select *Mail Server* (the default), as shown in the following image.



- 2. Type the existing Mail Server in the *Old Value* box.
- 3. Type the new Mail Server in the *New Value* box.
- 4. Click *Update* to update the new Mail Server value in schedules.

Procedure: How to Make a Global Update for an Email Address

- 1. In the Global Update interface, click the *Setting* drop-down list and select *Email Address*, as shown in the following image.

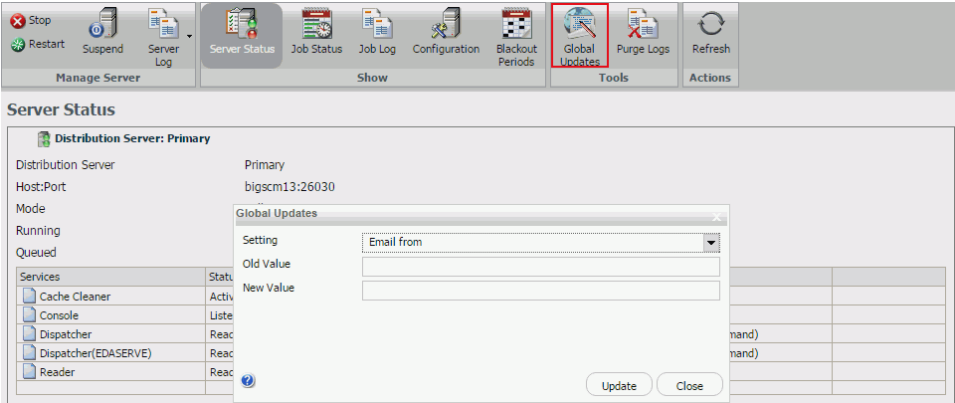


- 2. Type the existing Email address in the *Old Value* box.
- 3. Type the new Email address in the *New Value* box.

- Click *Update* to update the new Email address value in schedules and distribution lists.

Procedure: How to Make a Global Update for an Email From

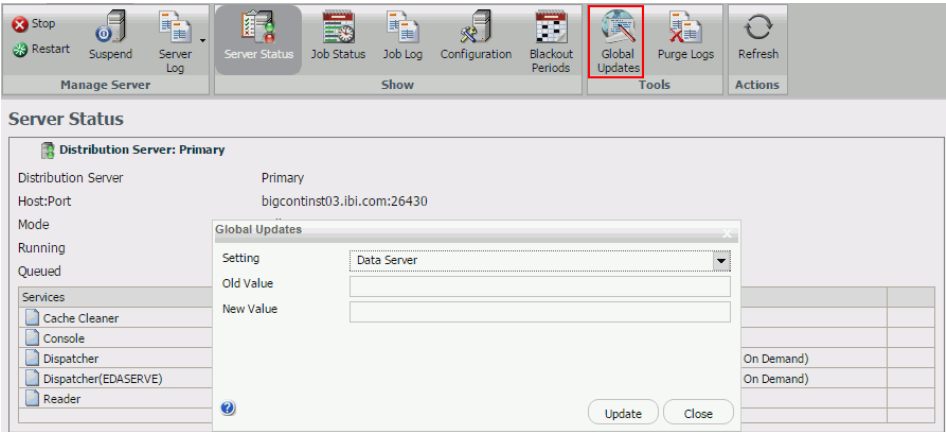
- In the Global Update interface, click the *Setting* drop-down list and select *Email from*, as shown in the following image.



- Type the existing Email from in the *Old Value* box.
- Type the new Email from in the *New Value* box.
- Click *Update* to update the new Email from value in schedules.

Procedure: How to Make a Global Update for a Data Server

- In the Global Update interface, click the *Setting* drop-down list and select *Data Server*, as shown in the following image.



2. Type the existing Data Server in the *Old Value* box.
3. Type the new Data Server in the *New Value* box.
4. Click *Update* to update the new Data Server value in schedules.

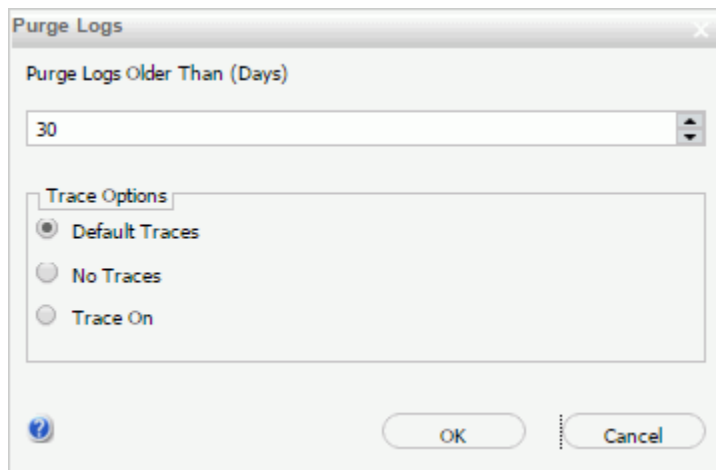
Purge Logs

The Purge Logs functionality provides on-demand options for purging logs, including the ability to specify a number of days (past) for which to purge logs. For example, if you want to purge logs for the past month, you would use the default number of days, which is 30. You can also specify an option for traces: Default Traces, No Traces, or Trace On.

Procedure: How to Purge Logs on Demand

1. From the Tools group in the ReportCaster Console, click *Purge Logs*.

The Purge Logs dialog box displays, as shown in the following image.



2. In the *Purge Logs older than (days)* field, accept the default number of days or specify another value. The default number of days is 30.
3. Under Trace Options, you can accept the default of *Default Traces*, or select *No Traces* or *Trace On*.
4. Click *OK*.

Working With Domains

The BUE uses Domains, and the groups of users they define, to support workgroups. Domains enable users to maintain private content, to share that content if their user role permits, and to access governed content published by others. Domains are available to multiple users, who, based on their role, have access to their own view of the Domain. This built-in workgroup security makes your data analysis and reporting activities easier to configure and manage.

In the Resources tree, Domains appear as root-level folders under the Domains node.

Domains comprise a collection of groups, an Application folder on the Reporting Server, and a set of rules that makes them all work together in a single workgroup. Domains partition content and metadata, and enable Managers to organize and store your content in the BUE portal. They are the place where Developers and Advanced Users create and manage content.

Procedure: How to Create a Domain

Only a Manager can create a Domain.

1. In the Resources tree, right-click *Domains*, point to *New*, and then click *Domain*.

The New Domain dialog box opens.

2. Type a title and a name.

Domain titles are shown to BUE users in the Resources tree. Managers can localize the title for different languages by going into the Property dialog after the Domain is created.

The Domain name is an internal identifier used by BUE, and is not shown to users.

3. Click *OK* to create the Domain.

A confirmation dialog appears. Click *OK*.

A folder that contains the title of the new Domain appears below the Domains node in the Resources tree.

Note: Within the new Domain, the BUE creates a Hidden Content folder as well as a My Content folder for each user. The Hidden Content folder can be used to store things that are related to the Domain but which you do not want users to see. For example, schedules that distribute output back to the Domain and image files or style sheets that are used by charts. The My Content folder is used by each user to store their personal content. Some users can share personal content with Domain users. Domain Developers and Managers can create folders and content within the domain and publish it so Domain users can access and interact with it.

Managing Domain Users

When a Manager creates a new Domain, the WebFOCUS Business User Edition automatically creates a new group for the Domain itself, and creates subgroups for each of the four user types within it. A Domain accommodates multiple users, maintaining independent views and levels of availability for each one. By assigning users to different groups within a Domain, a Manager or Group Administrator can set security options for each user at the Domain level.

To review these groups, open the Security Center. From there, a Manager can create new users for these Domain groups or add existing users to them. For more information about Groups, see [Managing Groups](#) on page 222.

Granting Access to a Domain

When a Manager first creates a Domain, it is available only to the Manager that created it.

To update this setting, the Manager must right-click a Domain, point to *General Access*, and then select one of the following settings.

- ☐ **Domain Groups.** Protects the Domain by limiting its availability to those users who are assigned to it and work within it. This is the default setting for all new Domains.
- ☐ **All Users.** Gives everyone Basic User access to the Domain. Using this setting, the Manager that created the Domain can make content and resources available to all users in a single action instead of individually assigning each user to the Domain.

The four Domain groups represent the four user types: Basic User, Advanced User, Developer, and Group Administrator. When the Manager or a Group Administrator assigns users to one of these groups, those users automatically obtain the privileges assigned to the user role represented by that group when working with content in that Domain.

Managing Domain Private Resources

The Manage Private Resources feature allows Managers and Group Administrators to identify and manage private resources owned by users or groups. In the BUE, Managers can manage private resources by user, group, or Domain.

Deleting Domains

A domain can only be deleted by a manager who has rights to delete all of the resources within it.

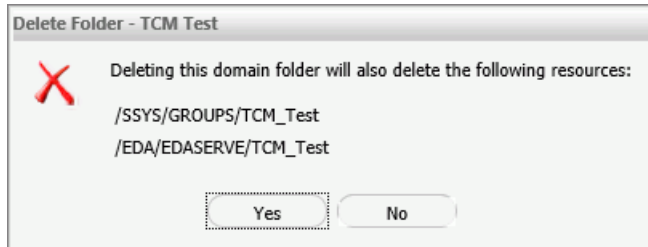
The process that deletes a domain first removes all users from groups in the domain, and then cascade deletes the groups and rules that were created with it.

Note: The cascade delete process only removes users from groups within a domain. It does not delete the users themselves.

Procedure: How to Delete a Domain

1. In the Resources tree, right-click an item, and then click *Delete*.

You receive a message advising you that this process will delete all resources created by this domain folder, as shown in the following Image.



2. Click Yes to delete the domain and associated resources.
3. If you receive a message warning that the folder contains private content, click *OK* to delete the domain and its private content.

If you need to protect the private content, click *Cancel* to end the process without deleting the domain.

When the process is complete, evidence of the domain and its resources disappear from the Resources tree and the Security Center.

Managing Domain Users After Deleting a Domain

Even though deleting a Domain automatically deletes the groups associated with it, this action does not delete the users assigned to those groups. Users from the deleted groups remain in the Users pane in the Security Center, but, unless they are assigned to another group, they are limited to view only privileges. To completely remove users from deleted groups, a Manager must delete them from the Security Center.

Working With Folders

Folders contain all repository content. Whenever a user creates a folder, it will always be created as a private folder. If the creator is permitted the necessary privileges, folders and their contents can be shared with other users or published for general use.

Folders have both titles and file names. The title is what is typically displayed to users. The file name is used as an internal reference by WebFOCUS BUE to provide an unambiguous context. Titles can be duplicated within a container, but file names cannot.

The folder path may have up to 1,040 characters for the path information (not including the object name), and up to 64 characters for the object name. For example, a folder may be named: /WFC/Repository/AmericaBank/Finance. In this example, /WFC/Repository/AmericaBank/ is 28 characters and Finance is 7 characters.

Procedure: How to View Folder Properties

Right-click the folder or item and click *Properties*. The Properties dialog box opens, displaying the information in the following table.

Item	Description
Language	Provides a list of languages that you can click.
View All	Opens the Language Properties dialog box, where you can click a language.
Title	Displays the value in the Resources tree, so users can identify the content within the folder.
Folder Name	Unique reference to the folder. click the <i>Change Name</i> check box to change the name of the folder.
Summary	Provides a detailed explanation with additional information about the folder.
Parent Folder	Shows the previous folder in the folder path of the WebFOCUS Repository.
Full Path	Displays full repository path of the folder in the WebFOCUS Repository.
Created On	Identifies the date and time the folder was created.
Created By	Identifies the user who created the folder.
Last Modified On	Shows the date the folder was last modified.
Last Modified By	Displays the user ID that last modified the folder.
Last Accessed On	Shows the date the folder was last opened.
Last Accessed By	Identifies the user who last opened the folder.

Item	Description
Properties	Shows a string of name value pairs that identify the properties of the folder.
Size	Displays N/A bytes for folders.
Run	Displays N/A, as this property is not applicable to folders.
Sort Order	Specifies the order to list the folder in the Resources tree or list items within a folder.
Status	Indicates whether the folder is Published or Private. Private will also display the owner information.

In addition, the following options are available to control the usage of content within the folder:

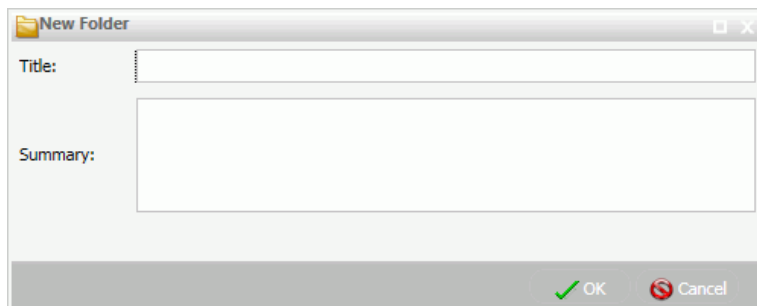
Authorized users can view or edit the access rules, effective policy, or sharing permissions by click *Security*.

The Server Properties tab shows the Assigned Server and available Application Paths. When a Reporting Server is not specified, the report request is submitted to the default Reporting Server specified in the Client configuration. When an Application Path is not specified, the Application Path defined during the processing of the Client configuration and the Reporting Server configuration is used.

Procedure: How to Create a Folder

1. In the Resources tree, right-click a domain or folder and click *New Folder*.

The New Folder dialog box opens, as shown in the following image.



2. Type the title and summary for the new folder.

Note: The unique folder name is derived from the title. The following characters are not permitted to be used within a folder name:

(Blank Space) & * () | : ; " , ? /

If you use restricted characters in the Title when creating a new folder or item, it is automatically removed from the Folder Name or File Name.

3. Click *OK*.

Procedure: How to Publish a Folder

In the Resources tree, right-click a folder, and click *Publish*.

Procedure: How to Duplicate a Folder

In the Resources tree, right-click a folder, and click *Duplicate*.

A duplicate folder appears. The name and title of the duplicate folder are the same as those of the original, appended with an underscore and an integer that increases each time the folder is duplicated.

Procedure: How to Cut or Copy and Paste a Folder

1. In the Resources tree, right-click a folder, and click *Cut* if you want to move the folder, or *Copy* if you want to leave the original folder in place.
2. Right-click the desired location and click *Paste*.

Folder names must be unique within a specific location. If the folder is pasted in a different parent folder, its name remains the same, as long as no folder of that name already exists in the new parent. If a folder of that name already exists, or if the pasted folder has the same containing folder as the original, the name of the copy is the same as the original, appended with an underscore and an integer that increases each time the folder is pasted. The folder titles update in the same way as the folder names.

Procedure: How to Change a Folder Title

1. In the Resources tree, right-click a folder, and click *Change Title*.

A cursor appears in the folder title field in the Resources tree.

2. Type the new name and press the Enter key.

The folder name is updated.

Procedure: How to Delete a Folder

1. In the Resources tree, right-click a folder, and click *Delete*.
A confirmation dialog box opens.
2. Click Yes to proceed with the deletion.

Managing Users

The Users pane of the Security Center lists all users. The Search field in this tab allows you to search the name and description fields for users. Simple wildcard searches are supported. The full range of features that manage users is available only to Managers. Group Administrators can see the Security Center, but can only assign existing users to groups. Basic Users, Advanced Users, and Developers cannot see the Security Center, nor can they create, update, or delete users or assign users to groups.

The toolbar above the User pane allows Managers to perform the following actions:

- ☐ Create, edit, or delete users.
- ☐ Import users.
- ☐ View when users last signed in.

Understanding Users

Users are those individuals who have access to the WebFOCUS Business User Edition (BUE). Managers and Group Administrators can assign users with similar responsibilities to one of the user type groups that the BUE automatically creates within a domain. This assignment allows users to take advantage of those features and content that supports their daily activities, but prevents them from using features or content that are beyond the range of their responsibilities and authority. The four user types are:

- ☐ **Basic Users.** Can view reports and content in the domains accessible to them. They can save deferred reports to their My Content folders, or copy parameters from a previously created report. They cannot share, publish, copy, or paste any folder or content.
- ☐ **Advanced Users.** Can do everything that Basic Users can do, and can also create original reports, charts and other content for their My Content folders. They can share folders and the content they contain with everyone or with selected users or groups.

- ❑ **Developers.** Can do everything that Advanced Users can do, and can view and publish content in their Hidden Folder. They can also copy and paste folders and content from their domain to another domain, but they must be sure that the domain they target for this operation maintains the same metadata as that used to create the content they are copying.
- ❑ **Group Administrators.** Can assign users to groups. They can also switch to Manager Mode and manage private resources.

Each user in the repository is defined by a unique name and may also be assigned a description, an email address, and a password. The user must be placed in a group at account creation and assigned a status. By default, the user is placed in the EVERYONE group, which is the group of all users in the system, and assigned the Active status.

Any of these characteristics, except the unique user name, may be edited later by an administrator.

Understanding User Name Requirements

Because user names are defined within the repository, they need to conform to the format rules and character limitations it imposes. If your installation supports external authentication, such as that provided by Microsoft Active Directory, user names also exist in an external repository, and must conform to the format rules defined in it.


The set of characters you can use to create user names is defined by the current character encoding setting established in the application server and the Client Code pages assigned to your NLS setting. For example, if the application server is configured to support UTF-8 encoding, and the NLS Setting is also configured to support the US Unicode (UTF) code page, you can use characters in the double-byte character set (DBCS) to create user names.

Given these considerations, when creating user names, take the following rules into account:

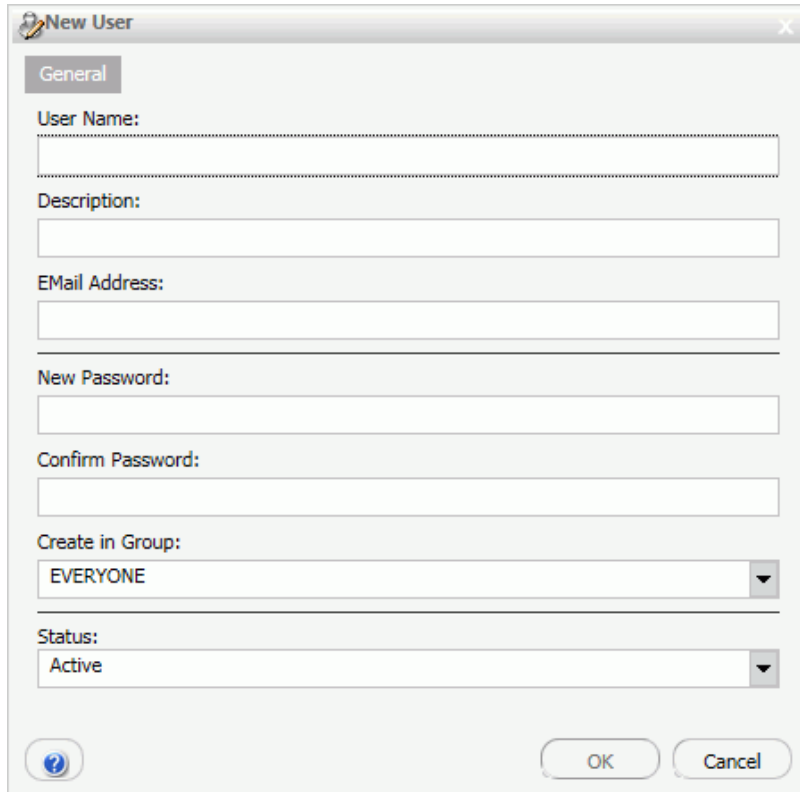
- ❑ User names may contain alphanumeric characters, spaces, and underscores.
- ❑ Depending upon the Client Code Page assigned to your NLS setting, user names can also include single-byte or double-byte NLS characters.
- ❑ The following characters are not supported in user names: “|:/*,?”
- ❑ It is recommended that you limit user names to 64 characters. Longer user names may cause problems during migration.
- ❑ If you support external authentication, avoid including characters in user names that your external authentication repository does not support. For more information about which characters to avoid, contact Customer Support Services.

Procedure: How to Create a User

Only a Manager can create a user.

1. In the BUE Portal, on the Menu bar, click *Administration*, and then click *Security Center*.
2. On the Users & Groups tab, click the *New User* button .

The New User dialog box opens, as shown in the following image.



The image shows a 'New User' dialog box with the following fields and options:

- General** tab selected.
- User Name:** Text input field.
- Description:** Text input field.
- E-Mail Address:** Text input field.
- New Password:** Text input field.
- Confirm Password:** Text input field.
- Create in Group:** Dropdown menu with 'EVERYONE' selected.
- Status:** Dropdown menu with 'Active' selected.
- Buttons at the bottom: Help (question mark icon), OK, and Cancel.

3. Type the user name, description, email address, password, and password confirmation, and if desired, select a group and a status for the user, and then click *OK*.

If you do not enter a description, the description defaults to the name. If you do not select a group and status for the user, the user will be created in the *EVERYONE* group and assigned the *Active* status by default.


Procedure: How to Import Users

A Manager can import a .csv file containing multiple users. This can be quicker and more efficient than individually creating multiple users with the New User dialog box.

The import file must contain the following columns: user name, password, description, email address, user status, and groups. A sample .csv import file, `getting_started_sample_users.csv`, is provided in the BUE file installation directory for your use as a template.


- ☐ Values are separated by commas.
- ☐ To use a comma in a description, enclose the description within double quotes.
- ☐ Multiple groups are separated by semicolons.

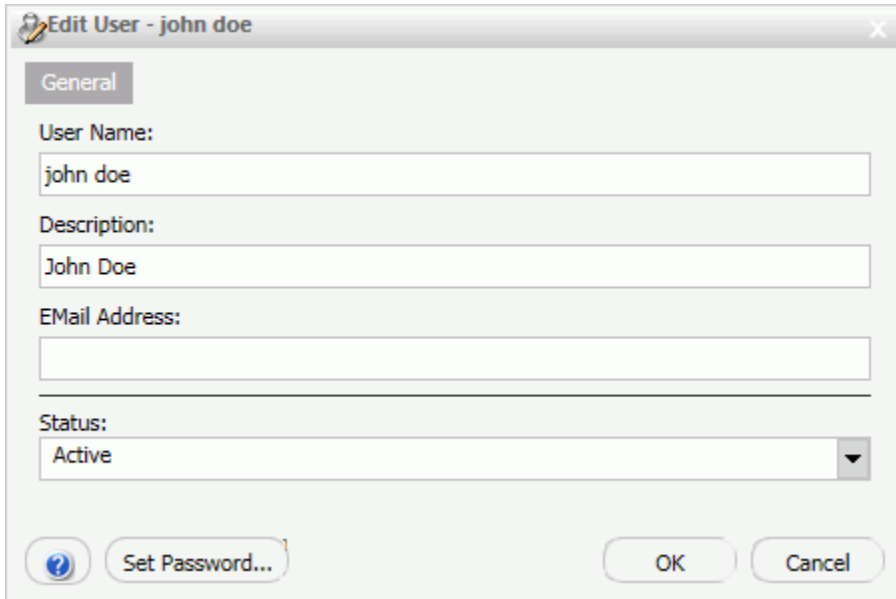
Note: Do not include a header in the user .csv file.

1. In the BUE Portal, on the Menu bar, click *Administration*, and then click *Security Center*.
2. On the Users & Groups tab, click the *Import Users* button .
The Import Users dialog box appears.
3. Browse to find the desired .csv file and click *Import*.
The new users are created within the specified groups.

Procedure: How to Edit User Details

Only a Manager can edit user details.

1. In the BUE Portal, on the Menu bar, click *Administration*, and then click *Security Center*.
2. On the Users & Groups tab, double-click a user, or right-click the user and select *Edit*, or click the user and then click the *Edit User* button . The Edit User dialog box opens, as shown in the following image.




3. If desired, type new information in the User Name, Description, or EMail Address field.
4. To change the status of a user, select *Active*, *Inactive*, or *Must Change Password* from the Status drop-down list.

Note: If you select *Must Change Password*, users will be prompted to change their password when they attempt to sign in.

Procedure: How to Delete a User

Only a Manager can delete a user. The action of deleting a user also deletes that user's private content. Be sure to publish or share any private content assigned to that user if it supports ongoing activities.

1. In the BUE Portal, on the Menu bar, click *Administration*, and then click *Security Center*.

2. On the Users & Groups tab, right-click a user and select *Delete*, or select the user and click the *Delete User* button .

A confirmation dialog box opens. Click Yes to delete the user.

Managing Groups

In the BUE, only Managers and Group Administrators can open the Security Center and review groups. The Groups field in the Users & Groups tab of the Security Center lists all the groups in the repository in alphabetical order. Subgroups appear indented below their parent groups. The Users in Group field lists the members of the selected group. If no group is selected, the field is blank. The Search field in this tab allows Managers and Group Administrators to search the name and description fields for groups. Simple wildcard searches are supported. A toolbar allows Managers and Group Administrators to perform the following actions:

- ☐ View the groups to which a user belongs.
- ☐ Add users to groups or remove them from groups.
- ☐ View the members of a group.

Understanding Groups

Groups are formed of users or subgroups that require similar capabilities or access to the same resources. All users are members of the EVERYONE group, which is the set of all named users in the system.

A group is a collection of similar users. Typically, users are permitted actions based on rules that apply to groups, although rules may also apply to individual user roles. Users may belong to more than one group.

Creating Groups From Domains

The BUE automatically creates groups whenever a Manager creates a new Domain and deletes them whenever a Manager deletes a Domain.

This feature simplifies group management by limiting it to the assignment of users to previously created groups. Managers and Group Administrators are not obligated to create or delete groups independently.

Each Domain creates one master group that uses its name. This group does not contain any users.

The Domain also creates four groups within that master group that correspond to the four basic user types within the BUE.

Members of the Basic User group can view content within their Domain and can save copies of reports they run deferred to their My Content folder for their own use.

Members of the Advanced User group have all the privileges of basic users, and they can create, share, and distribute their own content.

Members of the Developers group have all the privileges of basic and advanced users, and they can upload and connect to data, edit metadata, and create and organize Domain content. They can also manage the content other users see.

Group Administrators determine the role each user can have within a Domain by adding users to or removing users from groups and can change the General Access setting assigned to the Domain.

Users can be members of more than one of these groups and can have different privileges in different Domains. This capability is especially relevant to Group Administrators. In almost all cases, this group assignment, and the privileges it grants, is combined with membership in another group.

Procedure: **How to Add a User to a Group**

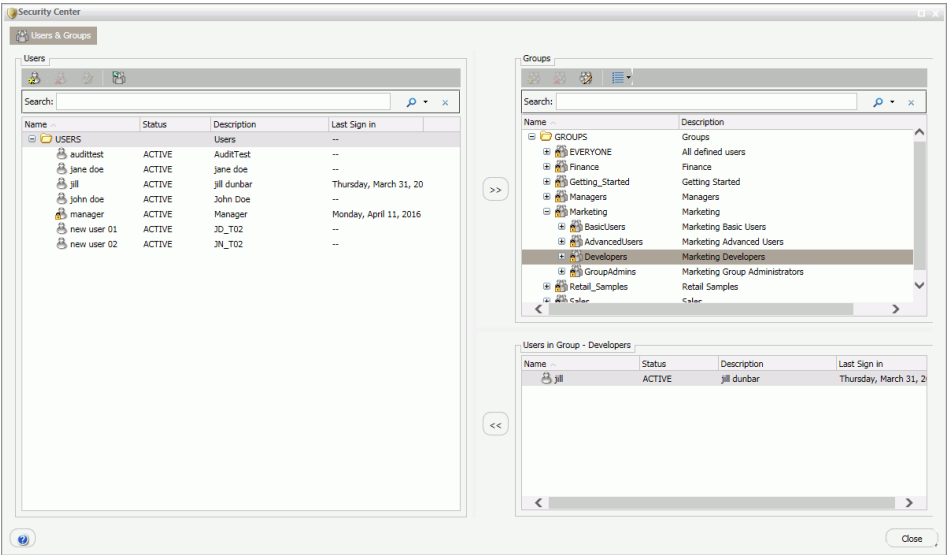
Only Managers and Group Administrators can add users to groups.

1. In the BUE Portal, on the Menu bar, click *Administration*, and then click *Security Center*.
2. On the Users & Groups tab, under Groups, select a group.

Note: When you select a group, the members of the group appear in the Users in Group field.

3. Drag a user from the Users field into the Users in Group field, or select the user and click the *Add selected users to group* button .

The user you added now appears in the Users in Group field, as shown in the following image.




- 4. Click OK.

Procedure: How to Remove a User From a Group

Only Managers and Group Administrators can remove users from groups.

- 1. In the BUE Portal, on the Menu bar, click *Administration*, and then click *Security Center*.
- 2. On the Users & Groups tab, under Groups, select a group.

- 3. Select a user and click the *Remove selected users from group* button , or drag the user into the Users field.

You can also remove a user from a group by right-clicking on the user and selecting *Groups*, then *Remove from*, then the group from which the user should be removed.

Managing Private Resources

Sometimes it is necessary for a manager to view or modify the private resources owned by another user. For example, when employees leave the company and their status is set to inactive, their private resources may need to be deleted or transferred to another user. It can also be useful for managers to have access to the private resources of the groups they manage in order to share resources or troubleshoot procedures. BUE Managers and Domain Group Administrators have access to this feature. Managers can access the private resources of all users. Group Administrators can access the private resources of the users in their Domain groups.

You can perform most actions on non-output resources owned by other users, such as FOCEXECs, and schedules. For output resources, such as PDFs or Libraries, your abilities are limited to deleting the resources or changing their titles.

You can view and manage private resources by domain or by user or group.

Authenticating Users to Your Active Directory

You can configure WebFOCUS BUE to authenticate users against your corporate Active Directory. The WebFOCUS BUE Client passes User sign in credentials to the WebFOCUS BUE Reporting Server, which in turn, validates them within an external source. WebFOCUS BUE can authenticate users against external Active Directory or LDAP directories. Users are externally authenticated whenever they access WebFOCUS BUE and when they access the Reporting Server Console.

The benefits of authenticating users to the Active Directory include:

- ☐ **Improved usability.** Users only need to remember a single user ID and password.
- ☐ **Reduced administration.** WebFOCUS BUE synchronizes user information with Active Directory. BUE Managers no longer need to maintain separate BUE passwords.
- ☐ **Improved maintenance.** Each time a user signs in, their user description and email is updated using the information found in the Active Directory, simplifying WebFOCUS BUE administration.

Configuring Active Directory/LDAP Authentication

To convert to external Active Directory or LDAP authentication, you must override the default setting of internal authentication in both the BUE Client and the Reporting Server, and establish a connection between the Reporting Server and LDAP provider that will support authentication activities.

Here is an overview of the configuration steps:

1. Create a new WebFOCUS BUE Manager account whose name matches an account in Active Directory.

Since the default Manager account manager generally does not exist in the external source, it cannot be authenticated once external authentication has been successfully configured.

The new Manager account that you create will exist in both WebFOCUS and in Active Directory so that you can use it for access to the BUE once you have restarted WebFOCUS in its new authentication configuration.

2. Configure the BUE LDAP provider to authenticate users to Active Directory.
3. Configure BUE to use the LDAP provider and restart the BUE services.

In the steps which follow, you will be required to provide credentials for two service accounts. The first is a BUE Reporting Server account, PTH\srvadmin, that is used by WebFOCUS BUE to delegate authentication to the Reporting Server. The password for this account is pre-configured during BUE installation to be the same as the password you supplied for the BUE Manager account.

The second is an Active Directory account of your choice that is used by the Reporting Server to authenticate users and retrieve their full description and email information, which in turn is passed back to the BUE to update the user account. This service account simply needs read access to Active Directory. Generally, any Active Directory account can be used for this purpose but you must make sure its password is set to never expire.

Procedure: How to Create the Externally Authenticated Manager Account

1. Sign in to WebFOCUS BUE as a Manager.
2. In the BUE Portal, on the Menu bar, click *Administration*, and then click *Security Center*.
3. In the Security Center, under Users, click *New User*.
4. Type the Active Directory ID of the person who will be the new Manager for the BUE after Active Directory authentication is established, in the User Name field.

You do not need to enter a description or email address because this information will be automatically updated during sign in based on information retrieved by the BUE from Active Directory.

5. Click *Managers* in the Create in Groups list.
6. Click *OK*.

An icon for the new user appears under Users and under Users in Group, when you click the Managers group.

Procedure: How to Establish LDAP as the Primary Security Provider on the Reporting Server

1. Sign in to WebFOCUS BUE as a Manager.
2. In the BUE Portal, on the Menu bar, click *Administration*, and then click *Reporting Server Console*.
3. In the Reporting Server Console, click the *Access Control* tab.
The Navigation pane displays an expandable LDAP folder.
4. Right-click the *LDAP* folder, and then click *New*.
5. In the LDAP Security Provider Configuration page, accept the default name, LDAP01, or type a new descriptive name for the LDAP security provider in the LDAP_PROVIDER field.
6. In the Connection Section, type the host name of your Active Directory server in the ldap_host field.

In some cases, you can also enter the domain name of your organization, for example: ibi.com.

7. Change the value in the LDAP port field only if your installation uses a different port number.

Most installations use the default port number, 389.

8. Click *Explicit* in the security list.

The section expands and displays the fields, ldap_principal, and ldap_credentials.

9. Type the name of a Service Account that has read access to the Active Directory, in the ldap_principal field.

It is important that this account has a non-expiring password to avoid disruption to the BUE.

10. Type the password of the Service Account in the ldap_credentials field.
11. Click *Next*.

If you receive a message that the Discover LDAP server attributes failed, click *OK*, and then review and update the settings you entered up to this point.

If all settings are correct, the page refreshes and displays additional headings. Fields in the User Search section contain values populated directly from the Reporting Server.

12. Click the *Trusted Connection* section heading.
13. In the Trusted Connection section, click *y* in the trust_ext list.
14. Click *Test User Authentication*.
15. Type the Active Directory User ID and Password of the person that you previously identified as the new BUE Manager, and then click *Continue*.

If you receive a message that the connection or password failed, review and update your settings if necessary, and try again.

If the password succeeded, continue with the next step.

16. Click *Save*.

17. In the *Activate Providers* page, in the LDAP entry that is identified by LDAP01, or by the descriptive name you typed in the LDAP_PROVIDER field, click *Primary* in the Status list.

The Status of the LDAP entry changes to *Primary*, and the Status of the PTH<internal> Security provider entry changes to *Secondary* automatically.

18. Click *Save Provider's Status*.

The screen refreshes and displays the *Change Effective Security Provider* page.

19. Click *Apply and Restart Server*.

When the confirmation dialog box opens, click *OK*.

The Reporting Server Console refreshes and displays the *Applications* tab.

20. Click the *Access Control* tab.

21. Review the new LDAP Security Provider and its *Primary* status.

22. Close the Reporting Server Console.

Procedure: How to Enable External Security in the WebFOCUS Client

1. Sign in to WebFOCUS BUE as a Manager.

2. In the BUE Portal, on the Menu bar, click *Administration*, and then click *Administration Console*.

3. In the Administration Console, click the *Security* tab.

4. Under the Security Configuration folder, click *External*.

5. On the External page, select the *Enable External Security* check box.

The External page displays the settings currently assigned to the Reporting Server.

6. Type *PTH\svradmin* in the *Server Administrator ID* field.

This is a Reporting Server administrator account that was installed automatically during the BUE installation.

7. Type the password for this account in the *Password* field.

The password was assigned during BUE installation, and is initially set to the same value that you entered for the manager account during installation.

Note: The placement of this ID and its associated Password in the Server Administrator ID field enables the Client to present them to the Reporting Server when sending User authentication requests.

8. Click *Connect* to verify the credentials you provided.
9. Leave User Authorization set to Internal and ignore the Account Creation on Sign In settings. The BUE does not support changes to these options.
10. Select the *Synchronize User Information with Authentication Provider* check box.
11. Click *Save*.

When the confirmation dialog box opens, click *OK*.

12. In the Administration Console menu, click *Close*.
13. Sign out of WebFOCUS BUE.
14. Stop and restart the Web Application to make these changes take effect. To do so:

If this installation of the BUE is based on the Windows operating system, stop and restart the WebFOCUS BUE 82 Application Server service in the Services Window.

If this installation of the BUE is based on the Linux operating system, navigate to `drive/ibi/WebFOCUS_BUE82/tomcat/bin` and run the `shutdown.sh` and `startup.sh` utilities.

15. When the Web Application restarts, sign in again using the Active Directory User ID and Password of the new BUE Manager that you identified at the beginning of the configuration.

The user description on the Menu Bar in the BUE Portal, and the Email Address of this account now reflect the values retrieved by the BUE from the Active Directory.

Creating BUE User Accounts When Configured for Active Directory Authentication

Now that you have configured BUE to authenticate users to Active Directory, you can create BUE accounts and assign them to the appropriate groups. This can be done in two ways:

Security Center. To use the Security Center to create and assign accounts to groups, create accounts the normal way including assigning them to the desired groups. However, since you are configured for Active Directory authentication you do not need to assign passwords for these users and you do not need to populate the Description and Email fields for them. As you have seen, this information will be automatically retrieved from Active Directory as each user signs in.

Import Users. To use the Import Users feature, simply define a CSV file containing one row for each user account. You can use the following file located in your BUE installation directory `getting_started_sample_users.csv` as a template. You can leave the password, user description and email values blank but you need to preserve the same number commas in the file to properly delimit all the required fields. You can adjust the group membership data in the CSV for each user account to suit your requirements or you can leave it blank and assign users in Security Center. The file should contain only data rows with the required number of commas on each row and contain no blank lines. Here is an example:

```
user1, , , ,ACTIVE,
```

```
user2, , , ,ACTIVE,Getting_Started/Developers;Retail_Samples/AdvancedUsers
```

Configuring WebFOCUS BUE for SSL

The Hypertext Transfer Protocol over Secure Socket Layer (https) establishes an encrypted Secure Socket Layer connection, and should be used to secure communications between WebFOCUS BUE and browsers assigned to end users. There are many configuration options that enable the use of this protocol, one of which is the Apache Tomcat configuration, as described in this section. WebFOCUS BUE uses this configuration by default.

To activate Secure Socket Layer-based communications, create a self-signed certificate for Java. You can optionally submit it to a Certificate Authority to establish it as a trusted certificate. The `keytool` utility that creates the certificate also modifies the connection type from open to SSL. Therefore, you must comment out the default Connector Protocol setting in the Tomcat `server.xml` file, and ensure that a setting for the new SSL Connector Protocol appears there instead.

Procedure: How to Create a Self-Signed Certificate

To create a Self-Signed Certificate with Java:

1. Open the command prompt window and redirect the command prompt to the `drive:\ibi\WebFOCUS_BUE82\jre\bin` directory.
2. Type the `keytool` command and values as shown in the following example.

```
keytool -genkeypair -alias mykey -ext  
san=dns:dnsName1,dns:dnsName2... -keyalg RSA -validity 720 -keystore /  
path_to_keystore/keystore -keysize 2048 -storepass MyPassword
```

where:

dnsName

Is the name, or alias, of the entity (the subject) that will present this certificate for authentication. You can include multiple names to ensure that all versions of the subject names are recognized. For multiple alternative names use the syntax, *dns:first_dnsName,dns:second_dnsName and so on*.

For example, *dns:bue,dns:bue.ibi.com*.

MyPassword

Is the password for this keystore. You can accept MyPassword, the default value, or you can replace it with a unique password by typing it in this field.

/path_to_keystore/keystore

Is the location information that specifies where the key file will be placed. This value is optional. If you do not specify a location for the key file, the Keytool utility places it in the default location.

Note: The name mykey is important if you need to issue a -certreq (certificate request) for a certificate signed by a Certificate Authority.

3. Press *Enter*.

The command prompt displays the first in a series of questions.

4. Respond to each question as follows:

- ☐ "What is your first and last name?" Type the first and last name of the certificate holder.
- ☐ "What is the name of your organizational unit?" Type the name of the organizational unit of the certificate holder.
- ☐ "What is the name of your organization?" Type the name of the organization of the certificate holder.
- ☐ "What is the name of your City or Locality?" Type the name of the city or locality of the certificate holder.
- ☐ "What is the name of your State or Province?" Type the two-letter abbreviation for the state in which the certificate holder is located.
- ☐ "What is the two-letter country code for this unit?" Type the two-letter abbreviation for the country in which the certificate holder is located.

5. When the command prompt displays the question, "Is CN=__, OU=__, O=__, L=__, ST=__, C=__ correct?", review the values and type *y* if they are correct.

If they are not correct, Type *n* and retype the keytool command from step 2.

If they are correct, the new Self-Signed Certificate is ready for use.

Establishing the Self-Signed Certificate as a Trusted Certificate

Until you identify the new self-signed certificate to the browser as a Trusted Certificate, the browser will display errors when you use it. During the initial testing period, you can add the new self-signed certificate directly to the Trusted Certificate Authority of those browsers included in the test. However, to fully establish the new certificate as a trusted certificate, you typically request certification for it from a Certificate Authority using the following request:

```
keytool -certreq -alias mykey -storepass MyPassword -file ./mykey.csr -  
keystore /path_to_keystore/keystore
```

where:

MyPassword

Is the password for this keystore. You can accept *MyPassword*, the default value, or you can replace it with a unique password by typing it in this field.

/path_to_keystore/keystore

Is the location information that specifies where the key file will be placed. This value is optional. If you do not specify a location for the key file, the Keytool utility places it in the default location.

You can then send the certificate request file (*mykey.csr*) to a Certificate Authority to sign, and when the authority returns the signed certificate, import it into the keystore.

Importing the Trusted Certificate into the Keystore

To import your certificate from the CA, type the following command:

```
keytool -import -alias mykey -file ./mykey.crt -keystore /  
path_to_keystore/keystore
```

where:

/path_to_keystore/keystore

Is the location information that specifies where the key file will be placed. This value is optional. If you do not specify a location for the key file, the Keytool utility places it in the default location.

If your CA is an internal CA, then type the following command to import the certificate from your Certificate Authority.

```
keytool -import -alias CA -trustcacerts -file ./ca.crt -keystore /  
path_to_keystore/keystore
```

where:

/path_to_keystore/keystore

Is the location information that specifies where the key file will be placed. This value is optional. If you do not specify a location for the key file, the Keytool utility places it in the default location.

Updating the Connector Protocols in the Tomcat Server.xml File

The Tomcat server.xml file is located in the following directory:

drive:\ibi\WebFOCUS_BUE82\tomcat\conf

The keytool utility disables the http connection assigned to port 26000. Therefore you must comment out the Connector tag in the server.xml file that defines this http-based connection by typing an exclamation point (!) after the open tag symbol (<).

```
<Connector connectionTimeout="20000" maxPostSize="-1" port="26000"
protocol="HTTP/1.1" redirectPort="26001" useBodyEncodingForURI="true"/>
```

The keytool utility also establishes an SSL connector on port 443. This connection replaces the old http based connection. Therefore, if it does not appear in the file, you must type this updated version of the connector tag, with its attributes and values, as shown in the following example:

```
<Connector protocol="org.apache.coyote.http11.Http11Protocol"
port="443" SSLEnabled="true" keystoreFile="C:/users/path_to_keystore/
keystore" keystorePass="MyPassword" maxThreads="150" scheme="https"
secure="true" clientAuth="false" sslProtocol="TLS"
ciphers="TLS_RSA_WITH_AES_128_CBC_SHA,TLS_RSA_WITH_AES_256_CBC_SHA"/>
```

where:

/path_to_keystore/keystore

Is the location information that specifies where the key file will be placed. This value is optional. If you do not specify a location for the key file, the Keytool utility places it in the default location.

MyPassword

Is the password for this keystore. You can accept MyPassword, the default value, or you can replace it with a unique password by typing it in this field.

Creating a Change Management Package

Many organizations do not grant developers write access to the user acceptance test and production environments. Access to these environments is strictly controlled and granted only to administrators, production control personnel, or automated change management processes.

Only developers know which changes are ready to be moved into the test environment. The Change Management Export facility presents developers with a graphical view of the resources they manage and allows them to build a change management package. This package is then loaded into another environment by production control personnel or automated processes.

***Procedure:* How to Create a Change Management Extract Package**

Only members of the BUE Managers group have access to the Change Management feature, which is available on the Resources tree as a node that contains two subfolders: Import and Export.

The steps required to create a Change Management Package are:

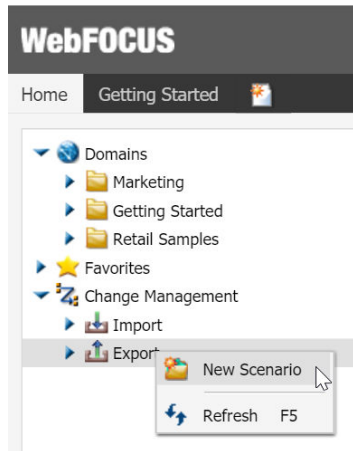
1. **Create a Scenario.** Utilizing the Change Management Export facility, an authorized user will create a scenario by selecting the resources to be exported. A scenario is a description of all the resources that will be exported into a Change Management Export Package.
2. **Export a Scenario.** After a scenario is created, a user can export this scenario into a Change Management Export Package. This Change Management Export Package is placed in the *drive:\ibi\WebFOCUS_BUE82\WebFOCUS\cm\export* directory in two formats: a zip file and a folder, which contains the expanded contents of the zip file.

The zip file or the exported folder is then copied to the target environment and placed in the *drive:\ibi\WebFOCUS_BUE82\WebFOCUS\cm\import* directory. For convenience the CM zip file can be downloaded from the BUE using a web browser and similarly uploaded to the target BUE where its content can then be imported and accessed.

***Procedure:* How to Access the Change Management Export Facility to Create a Scenario**

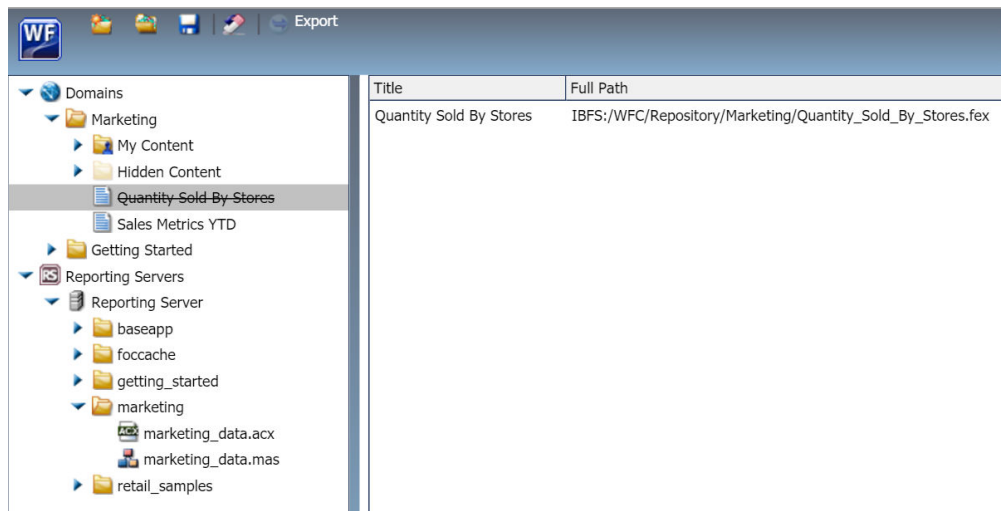
1. In the Resources tree, expand the *Change Management* node.

2. Right-click *Export* and click *New Scenario*, as shown in the following image.



3. You will then be prompted to enter the scenario name.

The Scenario Creation dialog box appears, as shown in the following image.



Selecting Resources

- ☐ You can select resources by dragging content from the Resources tree to the right pane. You can also right-click the content you want to move and select *Select With Subtree* or *Select Folder Only*.
- ☐ *Select With Subtree* selects that folder and all subfolders.

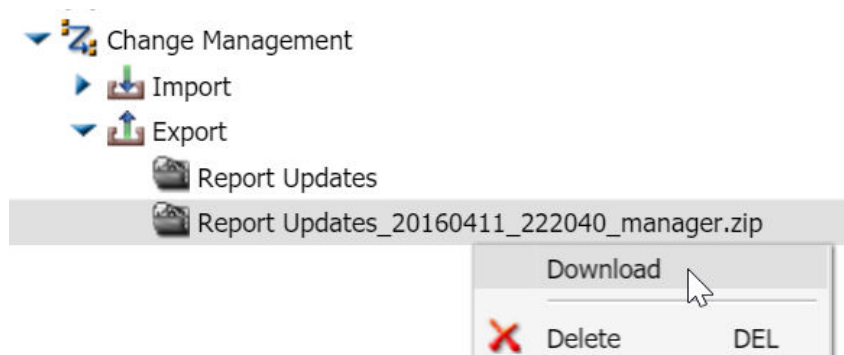
- ☐ *Select Folder Only* selects the specific folder, with no content. Typically, this is done to move rules on the folder.
- ☐ If a private resource is selected, the With Private Content check box is automatically selected and cannot be cleared.
- ☐ If a published folder is selected, you can include private content within that folder by selecting the With Private Content check box for that resource. This will export ALL of the private content in that folder and its subfolders, including user My Content folders, even if you do not have the privileges necessary to view that private content.
- ☐ Select resources only from a version that is equal to or greater than the currently established Change Management minimum version, typically 8.2.00.

Note:

- ☐ If private content is selected, it will only be imported if the owner of that private content already exists in the target environment.
- ☐ If a subfolder is selected, its parent folder must exist in the target system.

Now that resources are selected, click *Save* to save the scenario.

4. Click *Export* to create a Change Package based on the saved scenario file. You can do this from within the interface or by right-clicking the scenario file item in the Resources tree.
5. From the Resources tree, refresh the Export node under Change Management. If you do not see the zip file, refresh the Export node again. You can now download the zip file, so that it can be moved to the target system, as shown in the following image.



Procedure: How to Import a Change Management Package

Only members of the BUE Managers group can import Change Packages.

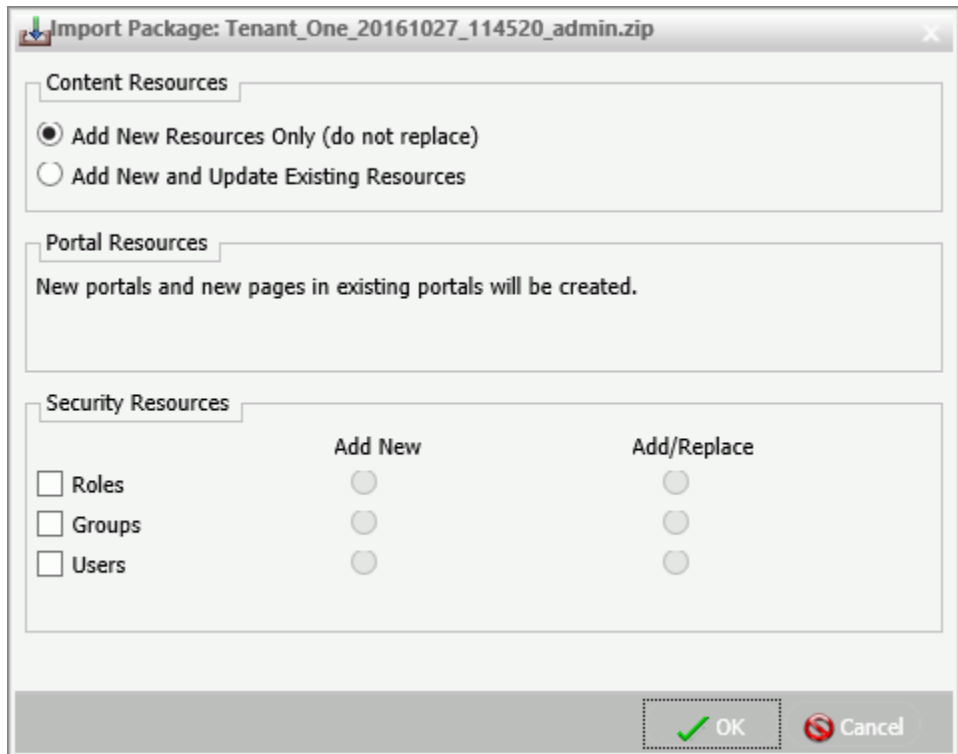
This procedure assumes that a change management zip package has been previously created on another system, the Manager is signed in to the target BUE environment, and the CM zip file is available to the Manager.

1. In the Resources tree, expand the *Change Management* node, then expand the *Import* node.
2. Right-click the package that you just uploaded and click *Import*.

If you receive a message stating that the resources in the Change Management Package are from a version that is below the minimum level, click *OK* and cancel the import.

You can recreate the Change Management Package using resources from a version that is greater than or equal to the version identified in the message.

The Import Package dialog box appears, as shown in the following image.



3. Select one of the following options:

Add New Resource Only (do not replace). This option will only add new resources to the target environment. For newly created items, the Created On and Last Modified On fields will be updated with the time at which they were imported. To view the Created On and Last Modified On fields, right-click an item and click *Properties*.

If an item already exists in the target environment but is also part of the change management export package, the target resource will be left alone and the Last Modified On field will not be updated.

Add New and Update Existing Resources. This option will add new resources to the target environment if they do not exist and update existing resources if they already exist. For newly created items, the Created On and Last Modified On fields will be updated with the time at which they were imported. For updated items, the Created On value for the target will be retained, but the Last Modified On field will be updated with the time at which it was imported.

The other options on this panel are not used in BUE and can be ignored.

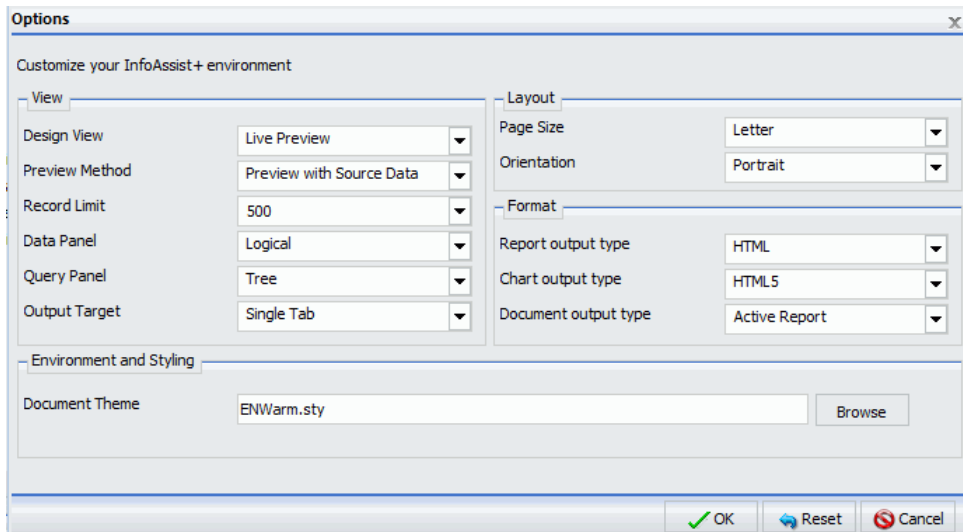
4. Click *OK*.

Changing InfoAssist+ User Preferences

You can change the default user preferences to customize the way that InfoAssist+ behaves when you create reports and generate output. The application theme, which is inherited from the BUE portal, customizes the InfoAssist+ interface, including all menus and dialog boxes.

You can style your reports by selecting a document theme independent from the interface. On the InfoAssist+ application main menu, click *Options*.

The Options window, as shown in the following image, opens to provide you with a user-friendly interface for customizing the InfoAssist+ application.



Note: If you make changes to the default selections in the Options dialog box, changes will take effect the next time InfoAssist+ launches.

If any of the options are unavailable, contact your administrator for assistance.

View

The View area provides settings for establishing the design view in which you will work, the type of data you will use when you preview your output, the limit you need to set on your record input, how your data and query panels will look, and the output target that you will use.

- ☐ **Design View.** Values are Live Preview and Query. Select *Live Preview* to activate the Preview Method drop-down menu. The default value is Live Preview.
- ☐ **Preview Method.** Values are Preview with Source Data and Preview with Sample Data. This menu becomes active when you select Live Preview from the Design View drop-down menu. The default value is Preview with Source Data.
- ☐ **Record Limit.** Values are All records, 1, 10, 50, 500, or you can type a numeric value directly in the menu. The default value is 500.
- ☐ **Data Panel.** Values are Logical, List, and Structured. The default value is Logical.
- ☐ **Query Panel.** Values are 2x2 (2 columns by 2 rows), 1x4 (1 column by 4 rows), and Tree. The default value is Tree.

- ☐ **Output Target.** Values are Single Tab, New Tab, Single Window, and New Window. The default value is Single Tab.

Layout

The Layout area provides settings for printing reports and charts.

- ☐ **Page Size.** Values are A4, A3, A5, Letter, Tabloid, and Legal. The default value is Letter.
- ☐ **Orientation.** Values are Portrait and Landscape. The default value is Portrait.

Format

The Format area provides settings for the output types for reports, charts, and documents.

- ☐ **Report output type.** Values are HTML, PDF, PowerPoint (pptx), Excel (xlsx), Excel (xlsx Formula), Excel, Excel (Formula), and active report. The default value is HTML.
- ☐ **Chart output type.** Values are HTML, HTML5, PDF, PowerPoint (pptx), Excel (xlsx), Excel, and active report. The default value is HTML5.
- ☐ **Document output type.** Values are HTML, PDF, PowerPoint (pptx), Excel (xlsx), Excel (xlsx Formula), Excel, Excel (Formula), and active report. The default value is active report.

Environment and Styling

The Environment and Styling area provides settings for styling reports and charts through the specification of a Document Theme.

You can use the default style sheet or select from a list of available themes. Click the *Browse* button to open the Templates - Browse predefined template files dialog box, in which you can search for an existing WebFOCUS StyleSheet. You can also click *Use Default Stylesheet* to revert back to the default style sheet.

Changing Global Preferences

You can change global preferences for InfoAssist+ through the Administration Console found on the BUE Portal. To do so, in the Administration Console, on the Configuration tab, click *InfoAssist+ Properties*.

Configuring Hyperstage

Hyperstage is a column-oriented, high performance analytic engine designed for analytic applications and data marts that need fast query response across large data volumes. Hyperstage was designed specifically for large volume data analytics applications with up to 50 Terabytes of data.

Hyperstage Overview

Hyperstage uses a unique and patent-pending approach to compressing, storing, and processing data that allows it to be installed and run on commodity hardware with little or no DBA intervention. Hyperstage requires little tuning to support ad hoc or complex business analytic queries.

Hyperstage is a database engine utilizing the PostgreSQL database environment. As such, Hyperstage is fully compatible with all PostgreSQL-compliant Business Intelligence tools and utilizes the PostgreSQL administrative interface to reduce the learning curve for system administrators.

Hyperstage provides a versatile, highly-compressed database system optimized for analytic-type queries. The ratio of possible compression and the speed of data import and retrieval are optimized at the expense of some transactional features of the engine performance, like the frequent data updating.

Hyperstage executes complex or ad hoc queries across vast amounts of data with a low cost of ownership.

Hyperstage and PostgreSQL

Hyperstage combines the Hyperstage storage engine with PostgreSQL server implementation. Hyperstage consists of several layers. The upper layers are provided by the PostgreSQL server implementation, and the lower layers are provided by Hyperstage.

Hyperstage includes both its own optimizer and executor along with the storage engine. The PostgreSQL query engine can be used with Hyperstage. However, since the PostgreSQL storage engine interface is row oriented, it cannot take full advantage of the column orientation or the Knowledge Grid and hence query execution through this path is reduced. Queries will be directed to the Hyperstage optimizer whenever possible.

Hyperstage ships with the full PostgreSQL binaries required. PostgreSQL is used to store catalog information (as with other storage engines). You can use the PostgreSQL instance for other purposes, but joining PostgreSQL and Hyperstage tables may result in reduced performance as the PostgreSQL query engine will be used.

PostgreSQL provides:

- ❑ Mature connectors, tools and resources.
- ❑ Interconnectivity and certification with BI tools.
- ❑ Management services and utilities.

Hyperstage provides:

- ❑ Load function that compresses data.
- ❑ Column-oriented storage engine.
- ❑ Knowledge Grid metadata layer that contains information about the compressed data.
- ❑ Optimizer/executor that uses the Knowledge Grid.

Configuring the Hyperstage Database

The following section describes the configuration steps for Hyperstage.

Configuring Hyperstage

The Hyperstage configuration file is called `infobright.cnf` and is located in the `ib_data` subdirectory within the Hyperstage Data installation directory (for example, `C:\ibi\WebFOCUS_BUE82\srv\wfs\hs\ib_data`). The configuration file is a text file containing the Hyperstage configuration parameters.

Each parameter is shown on a separate line.

If a parameter is not present in the configuration file or if the configuration file does not exist, the default values are used. Blank lines and comments (lines starting with `#`) are ignored.

Be sure to customize the following parameters to optimize performance. These tuning parameters are case sensitive and must be typed as shown in the following table.

Note: The values are commented out (preceded by `#`) in the `infobright.cnf` file, which causes them to default to the application minimum allowed values of 600 and 320 for `ServerMainHeapSize` and `LoaderMainHeapSize`, respectively.

Hyperstage Configuration Parameters

Parameter Name	Description
LicenseFile	Specifies the path or name of the newly required License file.
LogLevel	Controls how much information is written to logs. This is similar to the obsolete ControlMessages parameter.
LogRotateSize	Specifies how large the log file can be before it is rotated and archived.
LogRotateFiles	Specifies how many log archive files are kept.
KNFolder	Specifies the folder where Knowledge Grid is stored.
CacheFolder	Specifies the folder where temporary objects are stored.
ServerMainHeapSize	Specifies the size (in MB) of the main memory heap.
ThrottleLimit	Controls how many SELECT queries can run concurrently.

Configuration Tips and Examples

Important: You must properly configure your memory settings to ensure optimal performance. The following table shows sample, recommended memory configurations for different systems.

System Memory	Server Main Heap Size	Loader Main Heap Size
64GB	48000	800
48GB	32000	800
32GB	24000	800
16GB	10000	800
8GB	4000	800

System Memory	Server Main Heap Size	Loader Main Heap Size
4GB	1300	400
2GB	600	320

In most cases, the loader does not benefit from larger memory settings. However, increasing the LoaderMainHeapSize can help when:

- ❑ A table to be loaded has very long text values.
- or
- ❑ The table has many columns (for example, 1000 columns).

You can use more memory at import if you are planning to execute several concurrent load tasks to different data tables. However, disk access may become a bottleneck.

ServerMainHeapSize should be as large as possible, but safely smaller than the amount of physical memory on the machine. If performance decreases because of memory swapping by the operating system, try to set lower heap sizes. We also recommend decreasing the heap size if many users are running queries in parallel.

Note: Hyperstage may use additional memory for heavy loads or queries. Also, other applications on your server will use memory for their processes. It is important that the total of ServerMainHeapSize is less than the total available physical memory. If the system needs to swap memory, performance will be severely impacted.

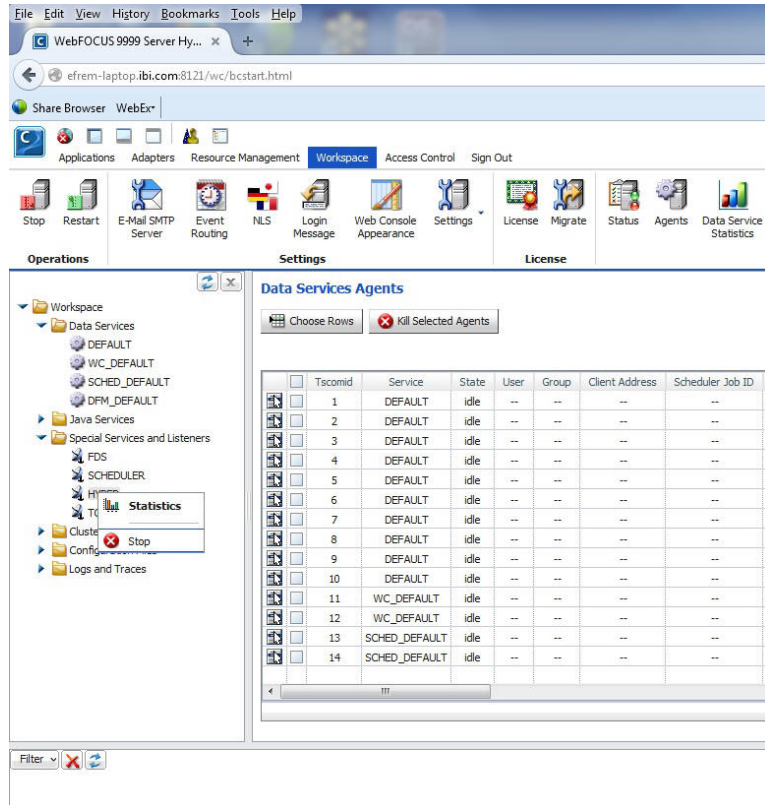
Using the Hyperstage Database Beyond WebFOCUS

The following section describes how to work with the Hyperstage server.

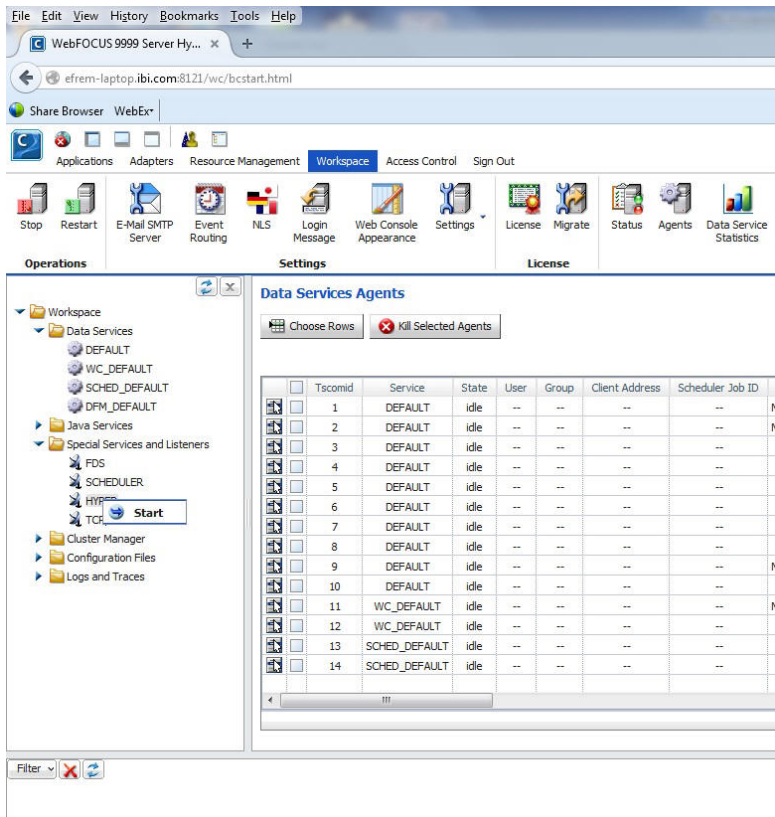
Starting and Stopping the Hyperstage Server

The Hyperstage Server starts and stops automatically when starting and stopping the Business User Edition Reporting Server. The Manager user ID is required to start and stop any component of the Business User Edition.

To manually stop the Hyperstage Server, from the Workspace/Select Special Services and Listeners section of the Reporting Server Web Console, right-click the *HYPER* service and select *Stop*, as shown in the following image.



To manually start the Hyperstage server, from the Workspace/Select Special Services and Listeners section of the Reporting Server Web Console, right-click the *HYPER* service and select *Start*, as shown in the following image.



Quick Copy For Hyperstage Using Extended Bulk Load Utility

Note: Hyperstage only supports Quick Copy, and Custom Copy as ETL tools.

The Quick Copy tool allows for the copying of all data from a Source table into Hyperstage. The *Bulk Load* option should be selected in order for data to be loaded quickly. If the *Bulk Load* option is cleared, the data will take much longer to load.

The Custom Copy tool allows for the copying of selected columns, presorting data within selected columns, and filtering of columns from a Source table into Hyperstage.

To access the Quick Copy tool, right-click the name of the synonym corresponding to the table or data you wish to copy into Hyperstage, and select *Quick Copy*.

The following configuration setting options are available:

Load Option

New. Recreate the target table before loading the data.

Append Existing Data. Data is loaded to an existing table.

Target Adapter

The list of adapters currently configured on the Reporting Server.

Target Connection

The Hyperstage connection used for the load operation.

Target Synonym Application

The target application on the Reporting Server where the target synonym will be stored.

Target Synonym

The name of the target synonym defining the Target Table Name.

Target Table Name

The name of the Hyperstage table where the data will be loaded.

Bulk Load

When selected, data will be loaded using the Hyperstage Bulk Load functionality. Bulk Load is the recommended approach for loading data into Hyperstage.

When cleared, data will be loaded using Insert/Update. Insert/Update is not recommended and will perform extremely slow.

Managing Hyperstage Tables

The following section describes how to work with the Hyperstage tables and lists the data types supported.




About the Hyperstage Database Files

Hyperstage tables are located in the `ib_data` subdirectory in your Hyperstage installation directory. Within the `ib_data` subdirectory, Hyperstage databases are stored in separate subdirectories.

Important: Do not manually copy a data table from one database to another by copying the database files. Internal table numbering errors and Knowledge Grid inconsistencies may occur. To copy a table, backup the entire database directory (see [Hyperstage Backup and Recovery](#) on page 252).

The following path and image shows the content of the ib_data directory, containing the Hyperstage databases webfocus and utf8test, as well as the BH_RSI_Repository directory, which holds the Knowledge Notes:

C:\ibi\WebFOCUS_BUE82\srv\wfs\hs\ib_data

Name	Date modified	Type
 BH_RSI_Repository	9/2/2016 1:14 PM	File folder
 cache	9/2/2016 12:00 PM	File folder
 webfocus	9/2/2016 1:10 PM	File folder

About Supported Data Types

The following data types are supported in Hyperstage. Note that numeric data types ranges are 1 less than the PostgreSQL minimums and maximums.

Numeric Types		
BOOLEAN	Values are either 0 or 1.	
SMALLINT	-32767	32767
INT (INTEGER)	-2147483647	2147483647
BIGINT	-9223372036854775807	-9223372036854775807
REAL	-3.402823466E+38	3.402823466E+38
DOUBLE PRECISION	-1.7976931348623157E+308	1.7976931348623157E+308

Numeric Types

Numeric(M, D) where: $0 < M \leq 18$ and $0 \leq D \leq M$	$-(1E+M - 1) / (1E+D)$	$(1E+M - 1) / (1E+D)$
--	------------------------	-----------------------

Date and Time Types

DATE	100-01-01	9999-12-31	YYYY-mm-dd
Time (without timezone)	00:00:00	24:00:00	HH:MM:SS
TIMESTAMP (without timezone)	100-01-01 00:00:00	9999-12-31 23:59:59	YYYY-mm-dd HH:MM:SS
TIME0053TAM P (with timezone)	1970-01-01 00:00:00 in UTC	2038-01-01 00:59:59 in UTC	
Interval	-178000000 years	178000000 years	

String Type

BYTEA (binary string)	$0 < N \leq 65536$
CHAR(N)	Fixed-length. Maximum length depends on character set (encoding). $0 < N * B \leq 65536$ where B is the maximum number of bytes for a single character.

String Type	
VARCHAR(N)	Maximum length depends on character set (encoding). $0 < N * B \leq 65536$, where B is the maximum number of bytes for a single character. For example, for UTF-8 it is 4 bytes, so the maximum number of characters that can be stored in a (VAR)CHAR column is $65536 / 4 = 16384$

Creating and Dropping Tables

Use the standard PostgreSQL commands to create and drop tables in Hyperstage, the same as you would with a PostgreSQL table. For detailed syntax information, see the *PostgreSQL 9.2 Documentation*.

Important: Do not manually copy a data table from one database to another by copying the database files. Internal table numbering errors and Knowledge Grid inconsistencies may occur. To copy a table from one database to another, back up the entire database directory (see [Hyperstage Backup and Recovery](#) on page 252). You can rename the entire database by renaming the folder. However, you should not copy a database folder from one active instance to another, or within the same active instance.

To create a table, enter the following command:

```
psql> create table <table_name> (<column(s)>) with (ENGINE=INFOBRIGHT);
```

Note:

- ❑ 'with (ENGINE=INFOBRIGHT)' syntax is necessary when creating tables manually, to specify that the table will be stored as part of the Hyperstage-specific Infobright engine. Without this syntax, the table will be created and stored as a regular PostgreSQL table.
- ❑ When creating a table, as a matter of practice, you should always use the ENGINE= option to ensure that the correct database engine is used. Hyperstage is shipped with DEFAULT ENGINE=INFOBRIGHT, but this can be changed. The name of the engine can be specified explicitly at the end of the create table statement.

To drop a table, enter the following command:

```
psql> drop table table_name;
```

For information on supported and unsupported options when creating columns, see *About Column Options*.

Character Set Support

The following section describes the character sets supported by Hyperstage.

Supported Character Sets

Hyperstage storage supports all ANSI and UTF-8 character sets. This means that Hyperstage can store and retrieve data encoded in 8-bit and multi-byte character sets.

Important: Queries that evaluate against UTF-8 character data columns will execute with less performance than an equivalent query against ASCII character data, due to ASCII support of Character Maps in the Knowledge Grid. UTF-8 specific Knowledge Grid extensions will be available in an upcoming release.

Collations and Comparisons

Hyperstage supports all custom UTF-8 collations supported by PostgreSQL:

<ul style="list-style-type: none">❑ utf8_bin❑ utf8_czech_ci❑ utf8_danish_ci❑ utf8_esperanto_ci❑ utf8_estonian_ci❑ utf8_general_ci (default)❑ utf8_hungarian_ci❑ utf8_icelandic_ci❑ utf8_latvian_ci❑ utf8_lithuanian_ci❑ utf8_persian_ci	<ul style="list-style-type: none">❑ utf8_polish_ci❑ utf8_roman_ci❑ utf8_romanian_ci❑ utf8_slovak_ci❑ utf8_slovenian_ci❑ utf8_spanish2_ci❑ utf8_spanish_ci❑ utf8_swedish_ci❑ utf8_turkish_ci❑ utf8_unicode_ci*
---	--

*utf8_unicode_ci properly handles both French and German collation, so specific collation types for these languages are not necessary.

For more information, see the *PostgreSQL 9.2 Documentation*.

The SQL standard does not define a default collation. Therefore, many DBMS engines have different default collations and produce different results. As a result, there are several differences between Hyperstage and other DBMS engines.

- ❑ For Hyperstage, character data types are case-sensitive. For example, the condition 'toronto'='Toronto' is not true in Hyperstage. Similarly, the condition, LIKE 'Abc%' is not true for 'abcde'.
- ❑ The Hyperstage sorting order is A...Z a...z (for example 'Zeta' < 'alfa'), which is the same sorting order as used by Oracle. The Hyperstage sorting order is different than the default PostgreSQL sorting order, which mixes lowercase and uppercase. The SQL Server order, which is aAbB...zZ; and the DB2 order, which is AaBb...Zz.
- ❑ The Hyperstage sorting order affects ORDER BY results, GROUP BY results (which is the order of groups and their definitions (for example, 'aaa' and 'AAA' define different groups) and DISTINCT results. WHERE conditions may also be affected if you are expecting a different sorting order than the one used by Hyperstage.
- ❑ To simulate Hyperstage collation in the PostgreSQL engine, set latin1_bin collation while creating a table (for more information, see the *PostgreSQL 9.2 Documentation*). Enter the following command:

```
psql> create table ... collate ascii_bin;
```

Padding

Hyperstage treats padding differently than other DBMS engines. Hyperstage assumes literal comparisons of text fields, including all whitespace characters. Therefore, a string containing two spaces is different than a string containing one space or an empty (0 length) string, which is also different than the NULL value.

The Hyperstage padding definition is compatible with the SQL standard. However, most DBMS systems have defined less restricted, customizable rules regarding text comparison. For example, 'abc ' = 'abc' may be true in some databases, but is not true in Hyperstage.

Note: In CHAR columns, trailing spaces are trimmed on LOAD, INSERT, and UPDATE, whereas in VARCHAR columns values are loaded with all spaces.

Hyperstage Backup and Recovery

The following section provides instructions on how to backup and restore the Hyperstage databases.

Backup Procedure

Use the following procedures to back up Hyperstage.

- ☐ To back up the Hyperstage databases, copy the `ib_data` and `pg_data` directories.
- ☐ You can take advantage of incremental backups, since only some of the database files are updated when new data is imported. Be sure to do a full backup occasionally.

Important: Some files in the KNFolder are updated when queries (using JOIN) are run, so be sure to back up the KNFolder on a regular basis.

Restore Procedure

To restore the Hyperstage databases from a backup copy, do the following:

1. Replace the `ib_data` and `pg_data` directories with the backup copy.
2. Replace the KNFolder with the backup copy (if the KNFolder is not inside the data directory).

Important: Do not manually modify database files or move them from one database to another. This may lead to data corruption and unpredictable results.

Functions and Operators

The following section lists the functions and operators supported by Hyperstage.

Hyperstage Optimizer Supported Functions and Operators

Comparison Functions and Operators

COALESCE	YES
----------	-----

Control Flow Functions

CASE	YES
COALESCE	TBD
NULLIF	YES

String Functions

BIT_LENGTH	YES
CONCAT	YES
LEFT	YES
LENGTH	YES
LOCATE	YES
LOWER	YES
LPAD	YES
LTRIM	YES
OCTET_LENGTH	YES
POSITION	YES
RIGHT	YES
RPAD	YES
RTRIM	YES
SUBSTR	YES
TRIM	YES
TRUNC	TBD
UPPER	YES

Numeric Functions

Modulo (%)	YES
ABS	YES

ACOS	YES
ASIN	YES
ATAN2, ATAN	YES
ATAN	YES
CEIL	YES
COS	YES
COT	YES
DEGREES	YES
EXP	YES
FLOOR	YES
LN	YES
LOG	YES
MOD	YES
PI	YES
POWER	YES
RADIANS	YES
RANDOM	TBD
SIGN	YES
SIN	YES
SQRT	YES
TAN	YES

Date and Time Functions

CURRENT_DATE	YES
CURRENT_TIME	YES
DATE	YES
DAY	YES
DAYOFYEAR	YES
FROM_UNIXTIME	YES
HOUR	YES
MINUTE	YES
MONTH	YES
NOW	YES
QUARTER	YES
SECOND	YES
TIME	YES
YEAR	No

Text Search and Other Functions

CAST	YES
MD5	TBD

Group By Aggregate Functions

AVG	YES
-----	-----

BIT_OR	No
BIT_AND	No
COUNT(DISTINCT)	TBD
COUNT	YES
MIN	YES
MAX	YES
STD, STDDEV	YES
STDDEV_POP	YES
STDDEV_SAMP	YES
SUM	YES
VAR_POP	YES
VAR_SAMP	YES
VARIANCE	YES

Hyperstage Data Tools

The following section describes the data tools used by Hyperstage.

Hyperstage Consistency Manager

Hyperstage provides a tool to validate Hyperstage-specific metadata structures. The Hyperstage Consistency Manager is an external stand-alone application that can be run against a Hyperstage instance to verify and repair most Hyperstage data structures, including the Knowledge Grid and Data Packs.

If you are seeing unexpected behavior with Hyperstage, such as server crashes, it can help to run the Hyperstage Consistency Manager for information for support and to perform repairs.

Note: Currently, the Hyperstage database must be offline in order to run the Hyperstage Consistency Manager.

Hyperstage Consistency Manager Tests

The Hyperstage Consistency Manager runs tests, as described in the following table.

Test	Description
Delete mask consistency check	Checks that the delete mask headers contain the proper sum for the delete mask body. If any inconsistency is found between the header and body, the Hyperstage Consistency Manager returns the list of blocks of delete mask where inconsistencies were found.
Number of objects in columns equality	Compares the stored number of objects in each column file related to the table. If any inconsistency is found in the number of objects, the Hyperstage Consistency Manager returns the first two columns with different object numbers.
Comparison of maximal value in DIMENSION dictionary versus DPN	Executes only for DIMENSION columns. Compares the maximal key value stored in the DIMENSION column dictionary and in DPNs. If the values differ, the Hyperstage Consistency Manager writes them to the log.
Comparison of number of objects in first-column DPN versus delete mask	Compares the metadata stored in the headers of the delete mask and DPN file related to the number of objects. If any inconsistencies are found, the Hyperstage Consistency Manager returns both numbers. The Hyperstage Consistency Manager compares only the first column because there is an independent test comparing this value between columns. If the test does not find the proper delete mask file or the proper DPN file, the Hyperstage Consistency Manager reports corruption.
Knowledge Grid consistency for column	Checks if the histograms report the proper value of the fixed parameter. A basic test of the Knowledge Node, ensuring the file has a proper format and the type of Knowledge Node corresponds to the column.
Knowledge Grid format for column	Each Knowledge Node is stored in a separate file. This test validates that the header data of each file is in the proper format.

Test	Description
Test for overlapping Data Packs in data files	Checks if there are Data Packs in files that overlap each other. If this situation occurs, the Hyperstage Consistency Manager returns a list of pairs of Data Packs numbers that are overlapping.
Tests of table metadata consistency	Verifies if the table metadata is valid. Includes verification of files used to store items, such as table name, number of columns and their names, types, and constraints like NOT NULL. These are the files created on CREATE TABLE and modified only on ALTER TABLE.
Test of DPNs for non-binary collation	Verifies Data Packs specifically for non-binary collation types (for example: Latin1_swedish.ci). If errors exist, they can be repaired using the Hyperstage Consistency Manager - -repair option.

Syntax: How to Run the Hyperstage Consistency Manager

To view the run options, run Hyperstage Consistency Manager with the - -help flag:

```
lcm-pure --help
```

To run Hyperstage Consistency Manager, use the following command:

```
lcm-pure --datadir=/data_directory_path [parameters]
```

For example:

```
c:\ibi\srv77\home06Hyperstage\hs\bin>lcm-pure.exe
--datadir=C:\HyperstagePG\ib_data --log-file=C:\temp\lcm-pure.log
```

Note: Hyperstage Consistency Manager should be run by the 'postgres' user. It should not be executed by 'root' or any rebuilt knowledge nodes will be owned by root (and cannot be edited), which will result in issues when loading any subsequent data into the 'corrected' tables.

The following table describes the Hyperstage Consistency Manager parameters.

Parameter	Description
- -help	Displays help message and exit.
-V [- -version]	Displays version information and exit.

Parameter	Description
- -basedir	Absolute path to Hyperstage installation directory.
arg	
- -datadir arg	Absolute path to directory. Mandatory.
- -database arg	Name of database chosen for data integrity testing. Optional. If specified, no other databases will be tested.
- -table arg	Name of table chosen for data integrity testing. Optional. If specified, no other tables will be tested.
- -log-file arg	Prints output to log file. Optional. If not specified, the logs will be printed to the console.
-F [- -full-check]	Runs full set of tests (may be time-intensive). Running Hyperstage Consistency Manager without the full-check option will result in a quicker test. However, the "Knowledge Grid consistency for column" test will not be run.
- -repair	Repairs found problems.
- -rebuild-kns	Rebuilds the Knowledge Grid. For more information, see About Rebuilding or Repairing Knowledge Nodes on page 261.
- -stop-on-error	Stops tests on first error and report.
- -cleanup	In case of an error in the Hyperstage Consistency Manager repair procedure, this option enables Hyperstage Consistency Manager to manually revert the datadir to its previous state. Running Hyperstage Consistency Manager with the - -cleanup option removes the old DPN files (containing incorrect DPNs) from the datadir and also makes the changes performed by Hyperstage Consistency Manager impossible to undo. If the - -cleanup option is not used, the old DPN files remain in the datadir.

About Rebuilding or Repairing Knowledge Nodes

Executing a rebuild of the Knowledge Nodes (using the `-rebuild-kns` option) will run the following tests:

- ☐ Test of table metadata consistency
- ☐ Test of Knowledge Grid format for column
- ☐ Test of Knowledge Grid consistency for column

The `-rebuild-kns` option will fix any issues found for the first two tests ("Test of table metadata consistency" and "Test of Knowledge Grid format for column").

You can also use the `-repair` option along with the `-full-check` option to achieve the same results as `-rebuild-kns`. Using either of these methods will rebuild any Knowledge Nodes that have been deleted.

About Cleanup Procedures

The Hyperstage Consistency Manager creates backup files when repairing problems related to "Test of DPNs for non-binary collation" (backup files are not created for any other tests). These backup files can be used to revert back to the original data if the Hyperstage Consistency Manager encounters an error during the repair procedure. To revert to the original data, copy or rename the `TAXXXXXDPN.icm_bck` files to the `TAXXXXXDPN.ctb` files (found in the `ib_data` directory).

Distributing the Quick Data Add-In File

WebFOCUS Quick Data add-in allows you to bring a large amount of enterprise information to the familiar Excel environment, and interact with this information without learning any additional software. You can install the WebFOCUS Quick Data Add-in on your desktop, and you can create and edit queries by accessing predefined data sources. You can save a query in an Excel document, and you can refresh it at any time.

Because all WebFOCUS connection and report information can be saved in an Excel workbook, users with the proper security and access rights can share spreadsheets throughout an organization. As a result, you spend less time recreating reports and more time analyzing information for effective decision-making.

Note: Quick Data is a WebFOCUS Business User Edition option, which requires a separate license and installation. For more information about licensing Quick Data, contact your Information Builders representative.

To distribute and enable the Quick Data Add-in in your environment, follow the procedures below.

Procedure: How to Distribute the Quick Data Add-in File

1. Copy the wfquickdata.xla add-in file and the configuration file, wfquickdata.cfg, located in one of the following directories:

❑ For Windows, *drive:\ibi\WebFOCUS_BUE82\WebFOCUS\utilities\quickdata*

❑ For Unix, */ibi/WebFOCUS_BUE82/utilities/quickdata*

2. Add the files to the following directory on the machines that use Quick Data.

C:\Users\userid\AppData\Roaming\Microsoft\AddIns

where:

userid

Is the name of the user logged on to the PC.

After the WebFOCUS Quick Data Add-in file is in the proper directory, you must open Excel and select the WebFOCUS Quick Data option in the Add-Ins dialog box, as described in the following procedure.

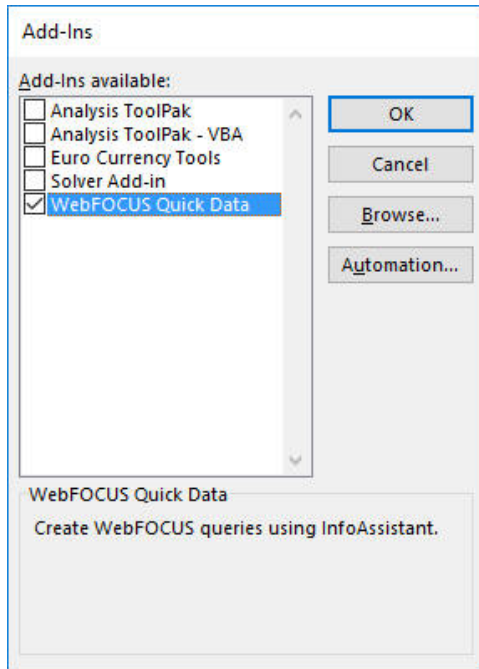
Note: If you already have Quick Data installed in your environment, be sure to backup your existing wfquickdata.cfg file to preserve all your customizations. You can then copy your edits from the existing wfquickdata.cfg file to the new file.

Procedure: How to Enable the Quick Data Add-In in Excel 2016

Use the following steps to enable the Quick Data Add-in in Excel 2016:

1. Launch Microsoft Excel 2016.
2. Click the File tab, click *Options*, and then click the Add-Ins category.
3. From the Manage dialog box, select *Excel Add-ins*, and then click *Go*.

The Add-Ins dialog box opens, as shown in the following image.



Note: If WebFOCUS Quick Data is not listed in the Add-Ins dialog box, make sure that the add-in is installed and placed in the correct directory.

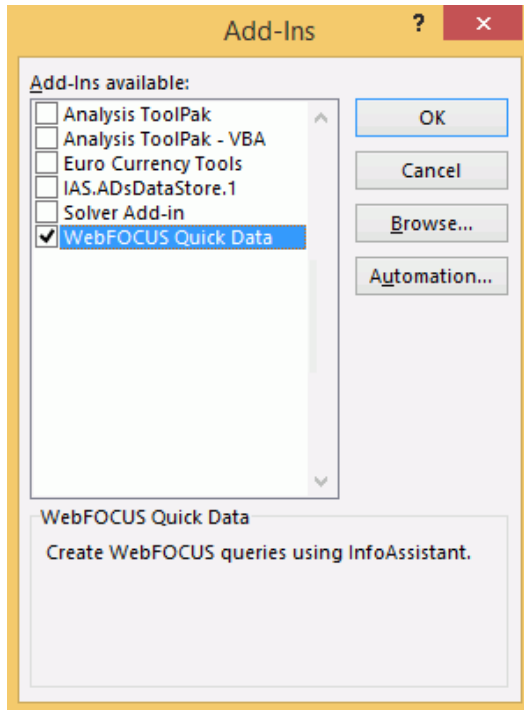
4. Select the WebFOCUS Quick Data check box and click *OK*.

Procedure: How to Enable the Quick Data Add-In in Excel 2013

Use the following steps to enable the Quick Data Add-In in Excel 2013:

1. Launch Microsoft Excel 2013.
2. Click the File tab, click *Options*, and then click the Add-Ins category.
3. In the Manage dialog box, select *Excel Add-ins*, and then click *Go*.

The Add-Ins dialog box opens, as shown in the following image.



Note: If WebFOCUS Quick Data is not listed in the Add-Ins dialog box, make sure that the add-in is installed and placed in the correct directory.

If the Quick Data Add-in was installed in a different directory, use the Browse button in the Add-Ins dialog box to locate it.

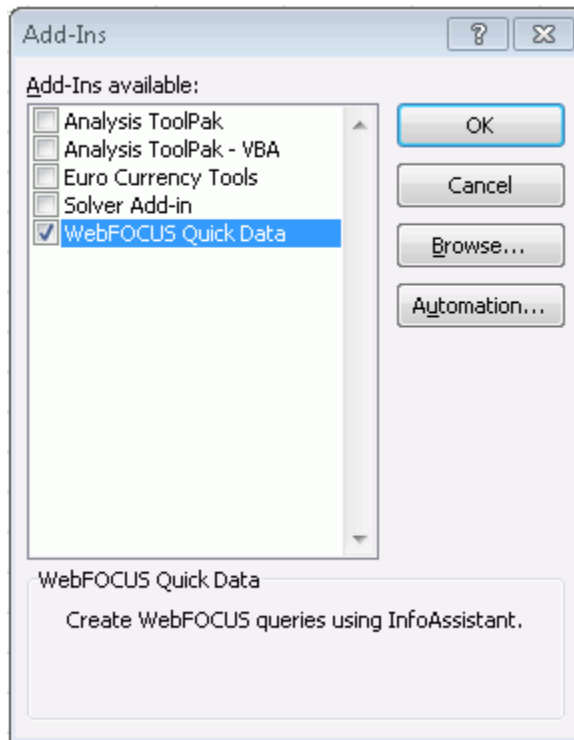
4. Select the WebFOCUS Quick Data check box and click *OK*.

Procedure: How to Enable the Quick Data Add-In in Excel 2010

Use the following steps to enable the Quick Data Add-In in Excel 2010:

1. Launch Microsoft Excel 2010.
2. Click the File tab, click *Options*, and then click the Add-Ins category.
3. In the Manage dialog box, click *Excel Add-ins*, and then click *Go*.

The Add-Ins dialog box opens, as shown in the following image.




Note: If WebFOCUS Quick Data is not listed in the Add-Ins dialog box, make sure that the add-in is installed and placed in the correct directory.

If the Quick Data Add-in was installed in a different directory, use the Browse button in the Add-Ins dialog box to locate it.

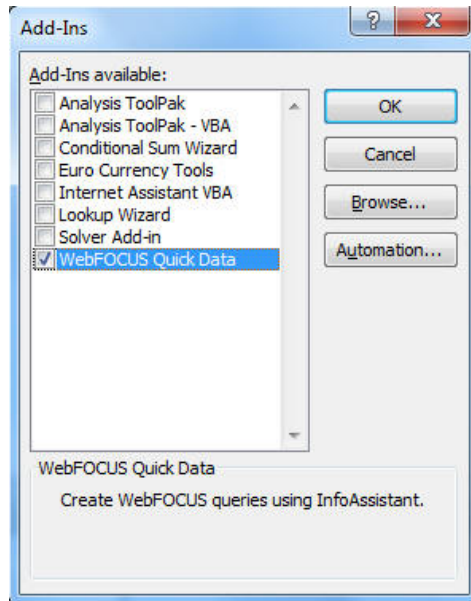
4. Select the WebFOCUS Quick Data check box and click *OK*.

Procedure: How to Enable the Quick Data Add-In in Excel 2007

Use the following steps to enable the Quick Data Add-In in Excel 2007:

1. Launch Microsoft Excel 2007.
2. Click the Microsoft Office Button  in the top-left corner.
3. Click *Excel Options* and then click *Add-Ins* in the Excel Options dialog box.
4. From the Manage drop-down list, select *Add-ins*, and click *Go*.

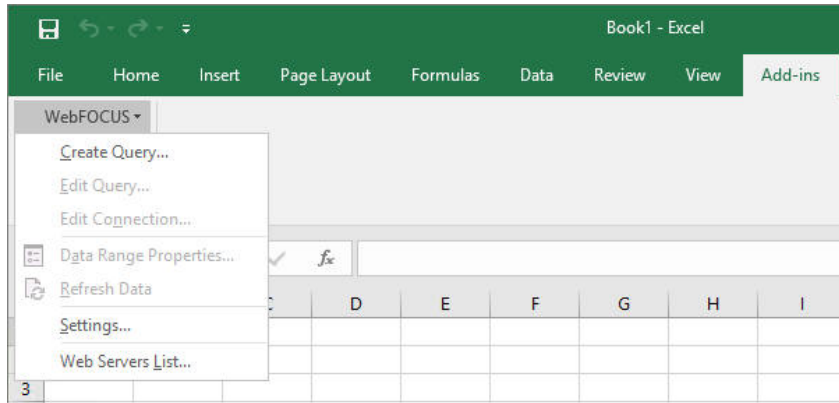
The Add-Ins dialog box opens, with WebFOCUS Quick Data listed as an add-in option, as shown in the following image.



Note:

- ☐ If WebFOCUS Quick Data is not listed in the Add-Ins dialog box, make sure that the add-in is installed and placed in the correct directory.
 - ☐ If the Quick Data Add-in was installed in a different directory, use the Browse button in the Add-Ins dialog box to locate it.
5. Select *WebFOCUS Quick Data*, and click *OK*.

A WebFOCUS menu opens in the Add-Ins tab. The following image shows an example of the WebFOCUS menu in Excel 2016.



WebFOCUS Quick Data Options

Once you have distributed the Quick Data Add-in, you can access the WebFOCUS Quick Data Options, which are described in this topic. Except for Settings and Web Servers List, these options are also available from Excel right-click context menus.

- ☐ **Create Query.** Available for new queries only, this option opens the Web Server Connection dialog box so that you can connect to WebFOCUS BUE. It continues by opening the Data Source Selection dialog box so that you can select a Master File. It then opens InfoAssist+, where you can create a query.
- ☐ **Edit Query.** Available for existing queries only, this option opens InfoAssist+, where you can edit a query.

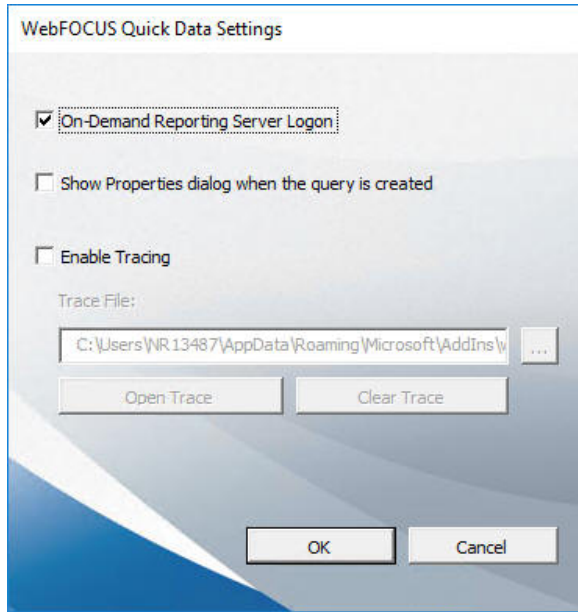
Edit Query is not enabled for password-protected cells.

- ☐ **Edit Connection.** Available for existing queries only, this option opens the Web Server Connection dialog box, where you can edit the connection settings including the Web Server URL, the HTML Alias, and the Client Path. The ability to edit connection information saves time when you are reusing reports and helps facilitate the sharing of workbooks across an organization.
- ☐ **Data Range Properties.** Available for existing queries only, this option opens the External Data Range Properties dialog box, where you can set Excel query properties.

- ☐ **Refresh Data.** Available for existing queries only, this option refreshes the data in the existing report query.

Refresh is not enabled for password-protected cells.

- ❑ **Settings.** This option opens the WebFOCUS Quick Data Settings dialog box, as shown in the following image.



The WebFOCUS Quick Data Settings dialog box provides the following settings:

- ❑ **On-Demand Reporting Server Logon.** This setting determines if the user will be prompted to log on to the Reporting Server the first time that a connection to the server is made during an Excel session (check this setting), or each time that a request is made to the Reporting Server during an Excel session (do not check this setting).
- ❑ **Show Properties dialog when the query is created.** When this setting is selected, a dialog box with options on how to insert data into Excel opens each time that a new query is executed.
- ❑ **Enable Tracing.** This option allows you to capture WebFOCUS Quick Data information in a trace file to troubleshoot communication problems and issues that may occur when you attempt to create and run report requests.

The captured information includes tasks performed by the tool when it attempts to connect to the web server and Reporting Server, when requests are made for data, and when data is retrieved. The default name of the trace file is wfquickdata.txt. It is created in the same directory as the WebFOCUS Quick Data Add-in file, for example:

`C:\Users\user_id\AppData\Roaming\Microsoft\Addins\`

Traces are captured for the duration of a single active Excel session. Tracing is automatically turned off when you close an Excel session. The trace file content is cumulative. Added to the file is trace information from each session in which tracing is enabled.

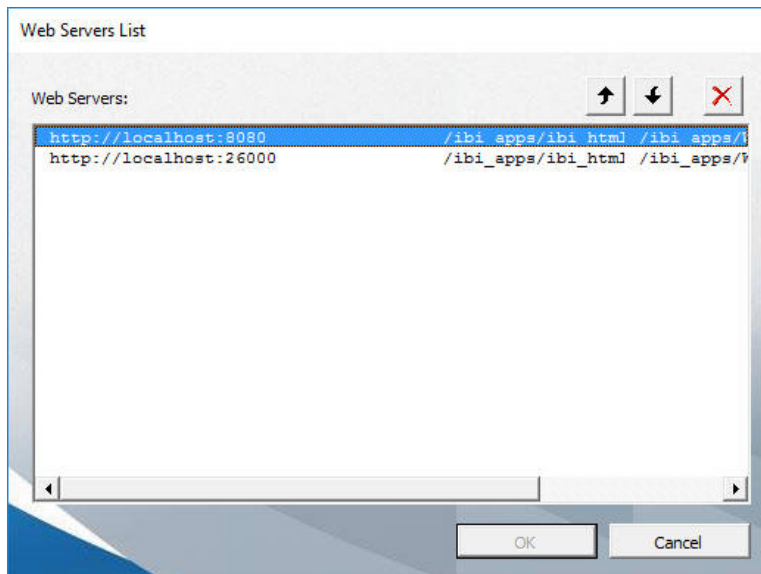
When you select Enable Tracing, the Trace File field is automatically populated with the full path to the trace file. The path includes the trace file name. You can change the location and name of the trace file by either typing the changes in this field, or by clicking the ellipsis and browsing to a new trace file location.

To view the current trace file, click *Open Trace*.

To delete the contents of the current trace file, click *Clear Trace*.

You can forward your trace file to Information Builders Customer Support Services (CSS) for analysis by the technical support team.

- ☐ **Web Servers List.** This option opens a dialog box that displays a list of the configured web servers, as shown in the following image. You can move the servers up or down in the list to change the order of appearance, and delete servers from the list.



Configuring a Default WebFOCUS BUE Environment

A configuration file is provided with the Quick Data add-in for the Administrator to use as a template when designing a default WebFOCUS BUE environment. The configuration file defines items such as the WebFOCUS BUE web server port number, alias, and client path.

Providing users with a default WebFOCUS BUE environment allows them to bypass the additional step of manually defining web server connection parameters.

The configuration file is named wfquickdata.cfg. It is originally located in the following directory: ..\ibi\WebFOCUS_BUE82\WebFOCUS\utilities\quickdata. After you have distributed the Quick Data Add-in in your environment, the configuration file is placed in the new directory, such as: C:\Users\userid\AppData\Roaming\Microsoft\AddIns.

The configuration file can contain multiple WebFOCUS configurations. Keep in mind that if the configuration file contains more than one WebFOCUS configuration, the last one appearing in the file is the configuration that is used when Quick Data is opened.

The configuration file provided as a template with the Quick Data add-in contains examples of configurations and instructions to help you create your own configuration. The following example displays a typical configuration:

```
SERVER_START
    PROTOCOL="http"
    HOST="localhost"
    PORT="26000"
    HTML_ALIAS="/ibi_apps/ibi_html"
    CLIENT_PATH="/ibi_apps/WFServlet.ibfs"
SERVER_END
```

Note:

- ❑ The use of double quotation marks around a parameter value, as shown in the example, is optional.
- ❑ Begin a comment line in the file with a number sign (#).

Use the following guidelines to create the configuration file:

- ❑ The configuration file must have the same name as the Quick Data Add-in file, and must have the extension .cfg, such as wfquickdata.cfg
- ❑ The configuration file must reside on the machine running the Quick Data Add-in, in the same directory as the .xla file.
- ❑ Each WebFOCUS configuration must begin with the delimiter, SERVER_START, and end with the delimiter, SERVER_END.
- ❑ Each configuration must contain the following parameters in order to connect to WebFOCUS BUE:
 - ❑ **PROTOCOL.** The protocol used in the environment running WebFOCUS BUE. If WebFOCUS BUE is running in a Secure Sockets Layer (SSL) environment, you must specify https as the protocol value. The default value is http.

- ❑ **HOST.** The name of the server on which the WebFOCUS BUE web application is installed.
- ❑ **PORT.** The port number of the application server on which WebFOCUS BUE is installed.
- ❑ **HTML_ALIAS.** The alias of the web server or application server on which the WebFOCUS BUE static pages are located. The default value is `/ibi_apps/ibi_html`.

Note: The leading slash is required.

- ❑ **CLIENT_PATH.** The path to the WebFOCUS BUE Servlet, as defined in the WebFOCUS BUE web application file, `web.xml`. The default value is

`/ibi_apps/WFServlet.ibfs`

where:

`/ibi_apps`

Is the default context root of the WebFOCUS BUE web application. The leading slash is required. You can configure this value.

`WFServlet.ibfs`

Is the name of the WebFOCUS BUE Servlet.



Chapter 3

Working With Data

Upload your own data sources and connect, join, and blend data, or learn about the sample data sources that are packaged with the WebFOCUS BUE software.

In this chapter:

- ☐ [Working With Data Overview](#)
 - ☐ [Preparing Data for Upload](#)
 - ☐ [Uploading and Appending Spreadsheets](#)
 - ☐ [Uploading Images and Documents](#)
 - ☐ [Connecting to Data and Editing Data](#)
 - ☐ [Using Data Preparation Options](#)
 - ☐ [Joining and Blending Data](#)
 - ☐ [Creating Virtual Fields](#)
-

Working With Data Overview

There are two ways that you can connect to data in WebFOCUS BUE. You can upload a Microsoft Excel spreadsheet or CSV file using the Upload wizard, or you can connect to an existing table in a data source of your choice, using the Connect to Data wizard. Both processes begin with identifying and preparing the data that you want to use. Once you have prepared your data, you can upload it or connect to it. After your file or table selection is complete, the wizard shows you the default breakdown of your data into measures, dimensions, and hierarchies.

At similar points during the upload and connect processes, you can access options to transform your data beyond the default settings. This includes grouping data into hierarchies, joining multiple tables into a cluster to create more fields and expand the scope of a synonym, editing geo roles and geo encoding to prepare the synonym for use in mapping and location analysis, data profiling and statistical analysis, changing columns or groups of columns into rows, and creating new or editing existing measures, dimensions, hierarchies, and attributes.

Once a synonym is complete, you can upload it to a target adapter environment or append it to an existing synonym. You can also save it as a template to allow repeated transformations if the same file is uploaded again in the future.

Understanding Your Data Structures

When you upload or connect to data in WebFOCUS BUE, you create a synonym that can be used to build analytical content. Synonyms define unique names (or aliases) for each object that is accessible from the server. Synonyms are useful because they hide the underlying data source location and identity from client applications. They also provide support for extended metadata features of the server, such as virtual fields and additional security mechanisms. Depending on the structure of the synonym that you are creating, the data inside a synonym is typically broken down into measures, dimensions, hierarchies, and attributes.

A measure is a numeric value, such as gross profit or cost of goods sold, that you may want to aggregate. All numeric values that can be summed are measures. Numeric fields that cannot be summed, such as product number and miles per gallon, 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 can 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 continent, country, state, and city dimensions. You can also define dimension fields that are not part of a dimension hierarchy.

You can assign dimension attributes to any dimension field, whether or not it is in a hierarchy. When applied to a field, attributes provide supplementary information that can be used for analysis or display. For example, in a geography hierarchy, which includes country, state, and city dimensions, population information can be assigned as an attribute of the city dimension. It is up to you to determine whether a measure as an attribute is useful. It depends on the design of the data source.

Preparing Data for Upload

Uploading data to WebFOCUS BUE can be made easier if you familiarize yourself with the data file first and ensure that it is properly formatted for upload, so geographic data, dimensional hierarchies, and other important aspects of your data are recognized. This is important so that the synonym created for your uploaded data provides the basis for quality analytical content.

The following topics detail tips and techniques to prepare your data for a smooth uploading process.

Naming Conventions and Excel Sheet Names

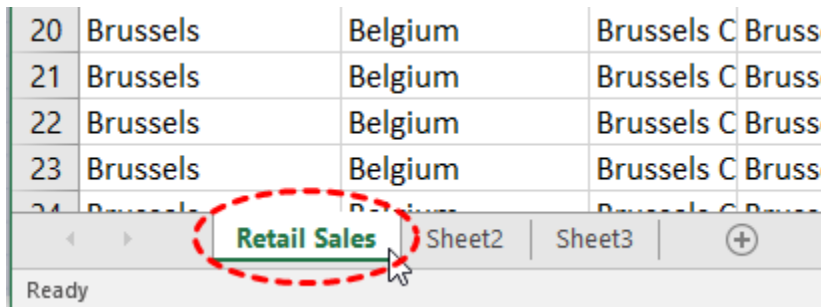
For Excel spreadsheets, the name of the file is not important but the name of the worksheet that contains your data is used to generate the synonym name. For the best results, follow the guidelines below:

- ☐ Ensure that the worksheet name is meaningful. For example, Store Sales is better than Sheet1.
- ☐ Remove special characters from the worksheet name. Spaces will be converted to underscores but all non-alphanumeric characters should be removed from the name, such as:

/^~!@#\$\$%^()-+={}|,;

- ☐ Remove or replace NLS characters with standard alphanumeric characters.

The image below shows a worksheet with a meaningful name, Retail Sales.



While you have an opportunity to edit the worksheet name within the Upload wizard, doing it in Excel may be preferable. Note that your spreadsheet data and column titles may contain National Language Support data and special characters.

For CSV files, there is no worksheet name, so the CSV filename is used to generate the synonym name. For this reason, all of the limitations identified for Excel worksheet names apply to the CSV filename, so be sure to check and adjust the filename prior to upload.

Removing Introductory Information

Sometimes an Excel spreadsheet contains formatted headings in the first few rows. This information cannot be imported into WebFOCUS BUE and should be removed. Delete the introductory rows and save the file before uploading. Alternatively, you can define a data range within your worksheet and leave the introductory information in place. The following image shows an example spreadsheet with a heading and subheadings highlighted.

	A	B	C	D
1	U.S. Energy Information Administration			
2	<i>May 2014 Monthly Energy Review</i>			
3				
4	Release Date: May 28, 2014			
5	Next Update: June 25, 2014			
6				
7	Annual Total	Primary Energy Consumed by the Residential Sector	Total Energy Consumed by the Residential Sector	Primary Energy Consumed by the Commercial Sector
8		(Trillion Btu)	(Trillion Btu)	(Trillion Btu)
9	1949	4460.434	5599.25	2668.869
10	1950	4829.337	5988.553	2834.094
11	1951	5104.476	6380.193	2737.684
12	1952	5158.193	6560.243	2672.911

Placing Column Titles in the First Row

For your data to be useful in WebFOCUS BUE, your data columns must be identified and properly described in the synonym that is generated during the upload process. You can make this easier by ensuring that the first few rows of your Excel spreadsheet contain column titles that are meaningful to you and to other users who will be using it, as an example shows in the following image.

	A	B	C	D	E	F	G	H
1	Store Name	Country	State	City	Sale Date	Revenue	Cost of Goods	Gross Profit
2	Des Moines	United States	Iowa	Des Moines	9/5/2013	3487.45	2288	1199.45
3	Dayton	United States	Ohio	Dayton	9/1/2013	13371.56	9459	3912.56
4	South Salt Lake	United States	Utah	South Salt Lake	9/2/2013	5642.79	4049	1593.79
5	Oslo	Norway	Oslo	Oslo	10/2/2013	13094.04	9206	3888.04
6	Des Moines	United States	Iowa	Des Moines	10/5/2013	4821.17	2962	1859.17
7	Dayton	United States	Ohio	Dayton	10/2/2013	13094.04	9206	3888.04
8	South Salt Lake	United States	Utah	South Salt Lake	10/5/2013	5564.52	3853	1711.52
9	Oslo	Norway	Oslo	Oslo	11/13/2013	18957.63	13293	5664.63
10	Des Moines	United States	Iowa	Des Moines	11/6/2013	6092.9	4513	1579.9

If your spreadsheet has more than one row of column titles, WebFOCUS BUE can merge the information when creating the synonym. You will be given an option to specify how many first rows of the Excel file contain title information in the Upload wizard.

	A	B	C	D
1	Annual Total	Primary Energy Consumed	Total Energy Consumed	Energy Consumption Balancing Item
2		(Trillion Btu)	(Trillion Btu)	(Trillion Btu)
3	1949	4460.434	5599.25	-0.165
4	1950	4829.337	5988.553	0.021
5	1951	5104.476	6380.193	0.188
6	1952	5158.193	6560.243	0.082
7	1953	5052.515	6558.748	-0.039
8	1954	5262.307	6846.058	0.091

Removing Aggregated Information

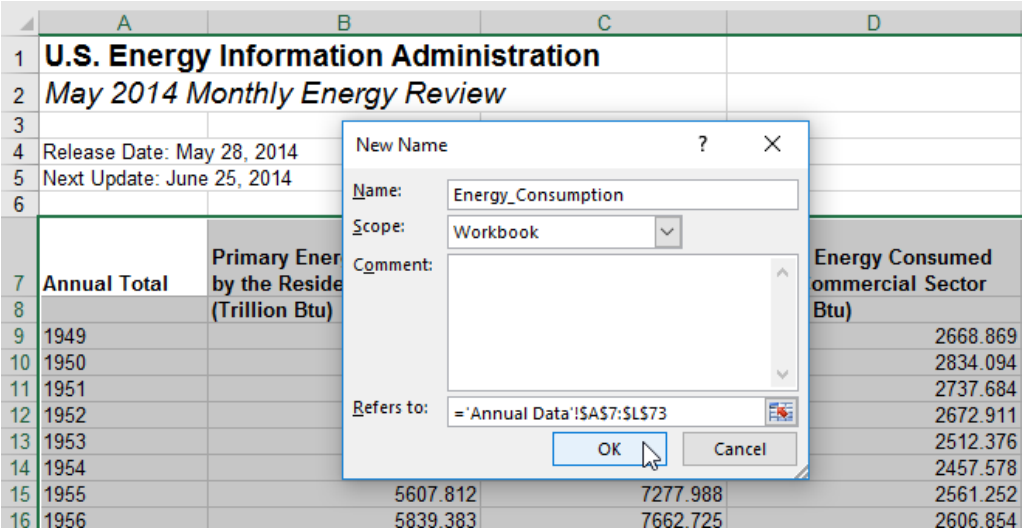
Excel spreadsheets may contain subtotals, grand totals, and other non-data row information. Data aggregation is performed by WebFOCUS BUE, so you should remove these kinds of rows from your spreadsheet and save the file before uploading it.

Using Excel Named Ranges

Data ranges defined within your Excel worksheet can be helpful for the following reasons:

- ☐ Your worksheet may have introductory information, such as formatted headings or non-data information, in the first few columns.
- ☐ You may not want to import all of the data columns found on your worksheet.

You can define a data range in your worksheet to remove the data that you want WebFOCUS BUE to process during upload, and leave your spreadsheet in its original format.



Preparing Hierarchical Data Columns

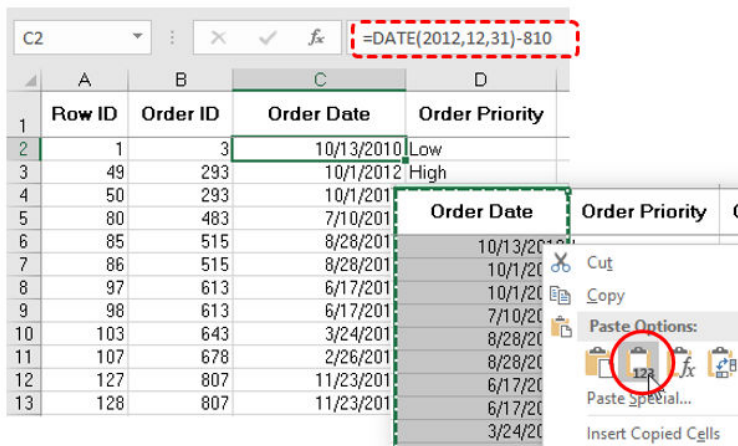
WebFOCUS BUE recognizes columns of data that have hierarchical relationships. This is useful because the field names are arranged more logically in InfoAssist+ and because it facilitates Auto-drill in the content. Auto-drill lets users drill up and down the field hierarchy automatically, making the content much more engaging and useful.

To help WebFOCUS BUE recognize hierarchical columns correctly, ensure that the column titles begin with a common word or words and are arranged left-to-right in the correct top-to-bottom direction, as shown in the image below. The Upload wizard allows you to define and edit dimension hierarchies prior to creating the synonym, but you can also do it before the upload.

	A	B	C	D	E	F	G
1	Store Name	Product Category	Product Subcategory	Sale Date	Revenue	Cost of Goods	Gross Profit
2	Des Moines	Video Production	Video Editing	9/5/2013	3487.45	2288	1199.45
3	Dayton	Video Production	Video Editing	9/1/2013	13371.56	9459	3912.56
4	Dayton	Video Production	Video Editing	12/1/2013	19097.49	13386	5711.49
5	South Salt Lake	Video Production	Video Editing	12/1/2013	10763.83	8138	2625.83
6	Brussels	Accessories	Universal Remote Controls	3/2/2010	10060.72	7822	2238.72
7	Brussels	Accessories	Universal Remote Controls	4/1/2010	12839.15	9456	3383.15
8	Brussels	Accessories	Universal Remote Controls	5/1/2010	9848.13	6992	2856.13
9	Brussels	Accessories	Universal Remote Controls	6/1/2010	10357.7	7584	2773.7
10	Brussels	Accessories	Universal Remote Controls	7/1/2010	10979.11	8283	2696.11

Removing Date Formulas

Spreadsheets may contain a date column where the values are computed by Excel using a formula. You need to convert these computed values into simple values before uploading a spreadsheet. To remove date formulas, select a column, right-click, and then click *Copy*. Then right-click the selected column again and click *Values*. This can be found under the Paste Options menu, as shown in the following image. The menu options may vary by Excel release. Now you can see that each cell contains a date value and WebFOCUS BUE can decompose your dates into useful components for use in InfoAssist+.



Uploading and Appending Spreadsheets

You can upload or append Excel spreadsheets or CSV files using the Upload wizard. Before uploading a new file, you should review [Preparing Data for Upload](#) on page 274, to ensure that your upload runs successfully.

After you upload a new file, you can append a file by adding new data to an existing synonym, without changing the structure. This option is useful when you are working with a complex or heavily edited synonym, by allowing you to retain all previous edits and data transformations.

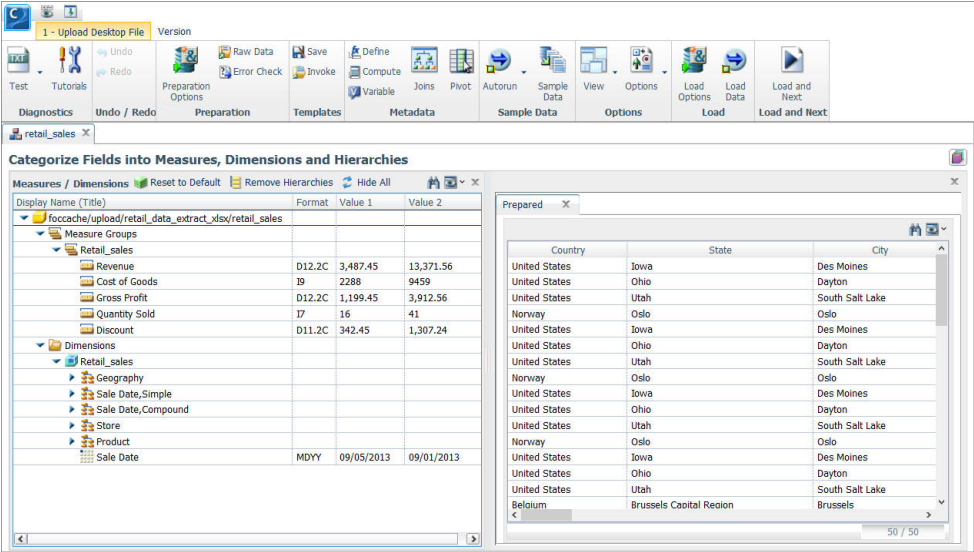
The procedures in this section provide step-by-step instructions for uploading and appending files using the default settings in the Upload wizard.

Procedure: How to Upload Excel Spreadsheets

1. Launch the Upload wizard in one of the following ways:
 - ☐ From the Resources tree, right-click a domain or folder, point to *Upload*, and then click *Data*.

- ❑ In the Open dialog box, in InfoAssist+, click *Upload Data*.
- 2. Drag the file that you want to upload into the Upload pane or click *Select File*, and navigate to the location of a file on your machine.
- 3. Click *Next*.

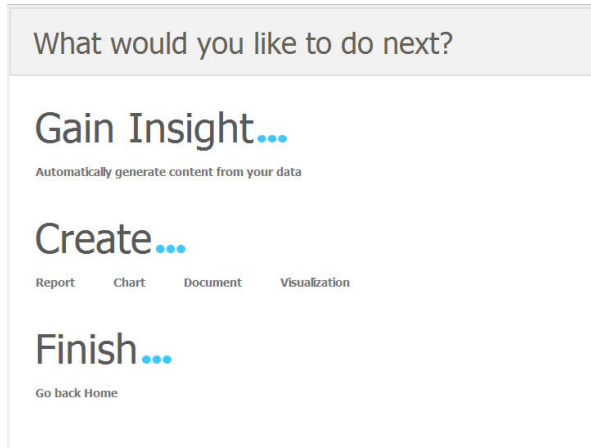
The Categorize Fields into Measures, Dimensions and Hierarchies screen opens, as shown in the following image. You can use the options on this screen to preview and change spreadsheets and delimited data files before you upload them to the target environment. This screen shows you the default breakdown of your data into measures, dimensions, and hierarchies.



For more information about the options that are available in the Categorize Fields into Measures, Dimensions and Hierarchies screen, see [Categorize Fields Into Measures, Dimensions and Hierarchies Screen Overview](#) on page 291.

- 4. Click *Load and Next*.
- The Load Options dialog box opens.
- 5. Click *Proceed to Load*.

If the load is successful, a window opens, and provides options to automatically generate content from your data, create content using the new Master File, or close the Upload wizard, as shown in the following image.



Note: If there are informational messages or if your upload is unsuccessful, the Status screen opens.

6. Select what you want to do next.

Related Information:

❏ [Categorize Fields Into Measures, Dimensions and Hierarchies Screen Overview](#) on page 291.


Procedure: How to Append New Data to an Existing Synonym

1. Launch the Upload wizard.
2. Drag the file containing the data that you want to append into the Upload pane or click *Select File*, and navigate to the location of a file on your machine.
3. Click *Next*.

The Categorize Fields into Measures, Dimensions and Hierarchies screen opens.

4. On the ribbon, in the Load group, click *Load Options*.

The Load Options dialog box opens.

5. From the Load Option drop-down list, click *Append Existing Data*.
6. Click the *Show File Picker* button  near the Select Target Synonym option, and select the synonym to which you want to append data.
7. Click *OK*.

8. On the ribbon, click *Load and Next*.

If the load is successful, a window opens, and provides options to automatically generate content from your data, create content using the new Master File, or close the Upload wizard.

Note: If there are informational messages or if your upload is unsuccessful, the Status screen opens.

9. Select what you want to do next.

Related Information:

❏ [Categorize Fields Into Measures, Dimensions and Hierarchies Screen Overview](#) on page 291.

Uploading Images and Documents

In WebFOCUS, you can upload other files, such as images and documents. These files can be used to enhance visualizations and shared with other users and groups.

Procedure: How to Upload an Image

1. From the Resources tree, right-click a domain or folder, point to *Upload* and then click *Image*.

The Image upload dialog box opens, as shown in the following image.



Note: Upload options may appear differently across browsers, because each browser utilizes its own upload dialog box.

2. Click *Browse* to select your image. You can upload the following formats: .jpg, .png, and .gif.
3. Select *Publish Image(s)* to provide other users with access to your images.

4. Click *Upload*.
A confirmation window opens.
5. Click *OK*.

Procedure: How to Upload a Document

1. From the Resources tree, right-click a domain or folder, point to *Upload*, and then click *Document*.

The Document upload dialog box opens, as shown in the following image.



Note: Upload options may appear differently across browsers, because each browser utilizes its own upload dialog box.

2. Click *Browse* to select your document.
3. Select *Publish Document(s)* to provide other users with access to your documents.
4. Click *Upload*.

The confirmation window opens.

5. Click *OK*.

Connecting to Data and Editing Data

Aside from uploading data, you can create synonyms by connecting to various data sources. The Connect to Data wizard leads you through this process and allows you to establish a connection to many native data sources. The data source being used determines the type of metadata that is required. For example:

- ❑ When the server accesses a relational data source, it needs to know how to interpret the data stored there. You must create a synonym that describes the structure of the data source and the server mapping of the data types.
- ❑ When the server invokes a transaction or procedure, it needs to know how to build the request, what parameters to pass, and how to format an answer set from the response. You must create a synonym that describes the layout of the request or response area.

Whatever your data source, the adapter you are using manages the synonym creation process for you, creating a synonym that meets your specific requirements.

Note: Although all synonym creation panes have the same look and feel, the parameters are specific to each adapter. To obtain detailed information for an adapter, click the question mark (?) next to a parameter.

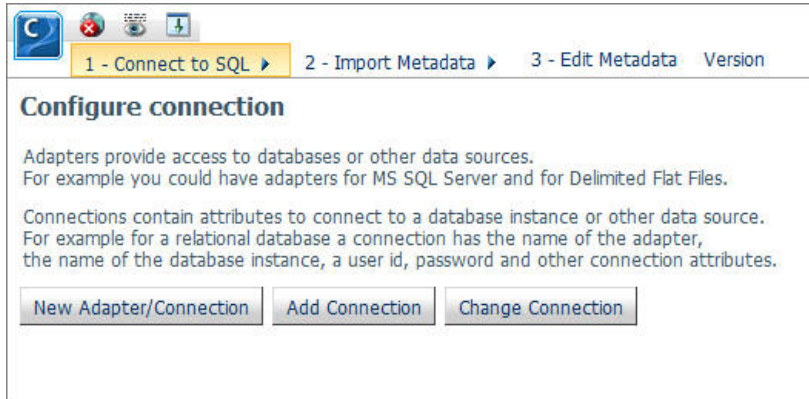
Creating a Synonym With the Connect to Data Wizard

The Connect to Data wizard provides a step-by-step process for configuring adapters that connect to data sources that you can use to build reports, charts, and visualizations in InfoAssist+.

Using the Connect to Data wizard, you can perform the following steps:

1. **Connect to SQL.** Configure an adapter, create a new connection, or change connection parameters.
2. **Import Metadata.** Create a synonym for one or more data sources.
3. **Edit Metadata.** Enhance metadata by adding measures, dimensions, and dimension hierarchies.

These steps are displayed in the navigation screen, as shown in the following image. As you complete the tasks for each step, the tool automatically takes you to the next step in the process. If you want to go directly to a specific step, click the step and the corresponding window opens.



You can launch the Connect to Data wizard in one of the following ways:

- ☐ From the Resources tree, right-click a domain or folder, point to *Metadata*, and then click *Connect to Data*.
- ☐ In the Open dialog box, in InfoAssist+, click *Connect to Data*.

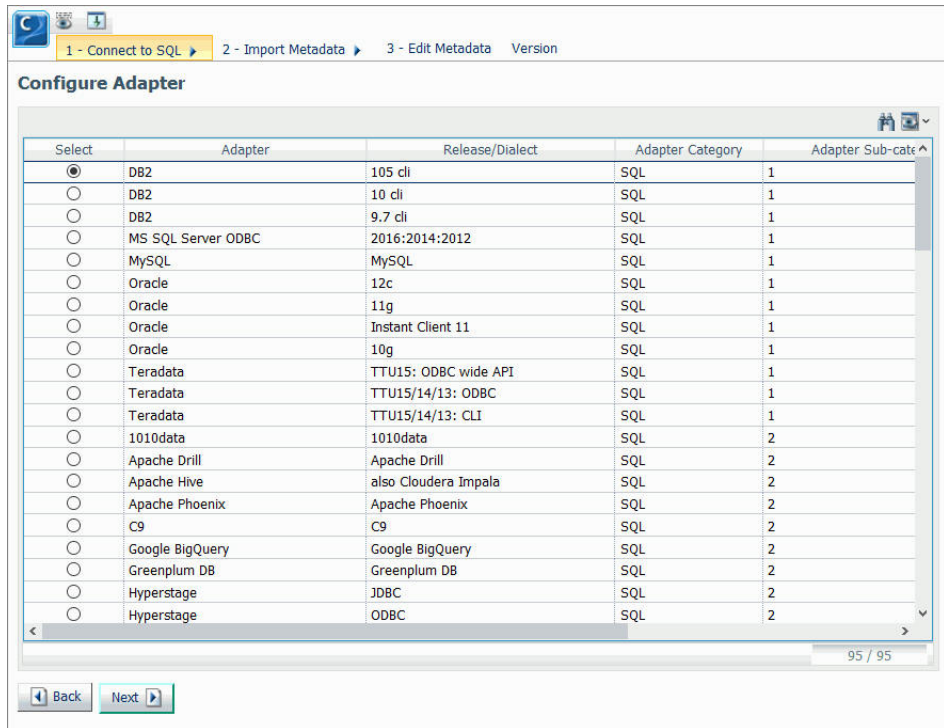
Step One: Connect to SQL

The first step in the Connect to Data wizard is creating a connection to your data, which will allow you to modify your data for use in WebFOCUS BUE. You can configure a new adapter or connection, add a connection to an existing adapter, or change adapter connections.

Procedure: How to Configure a New Adapter/Connection

1. From the Configure connection screen, click *New Adapter/Connection*.

The Configure Adapter screen opens, as shown in the following image.



2. Select the radio button for the adapter and click *Next*.

Note: You can click the *Adapter*, *Release/Dialect*, *Adapter Category*, or *Adapter Sub-category* columns to sort the available adapters.

The Add to Configuration screen opens.

3. Enter the parameters for the specific adapter and click *Configure*. For information on the parameters, see the online Help.

Procedure: How to Add a Connection

1. From the Configure connection screen, click *Add Connection*.

The Add Connections screen opens.

2. Select the radio button for the adapter and click *Next*.
3. Enter the parameters for the specific adapter and click *Configure*.

Procedure: How to Change a Connection

1. From the Configure connection screen, click *Change Connection*.

The Edit Connections screen opens.

2. Select the radio button for the connection and click *Next*.

Note: You can click the *Adapter*, *Connection*, *Connection string*, or *Profile* columns to sort the available connections.

3. Change the parameters for the specific connection and click *Configure*.

Step Two: Import Metadata

Once you configure a connection, you must select the metadata that you want to import. The first screen that you see is the Select connection to create synonyms screen. Here, you select the connection to your data.

Procedure: How to Import Metadata

1. Select the radio button for the adapter and click *Next*.

Note: You can click the *Adapter*, *Connection*, or *Connection string* columns to sort the available adapters.

The Select Synonym Candidates screen opens.

2. Select or enter the parameters for each screen and click *Next*.

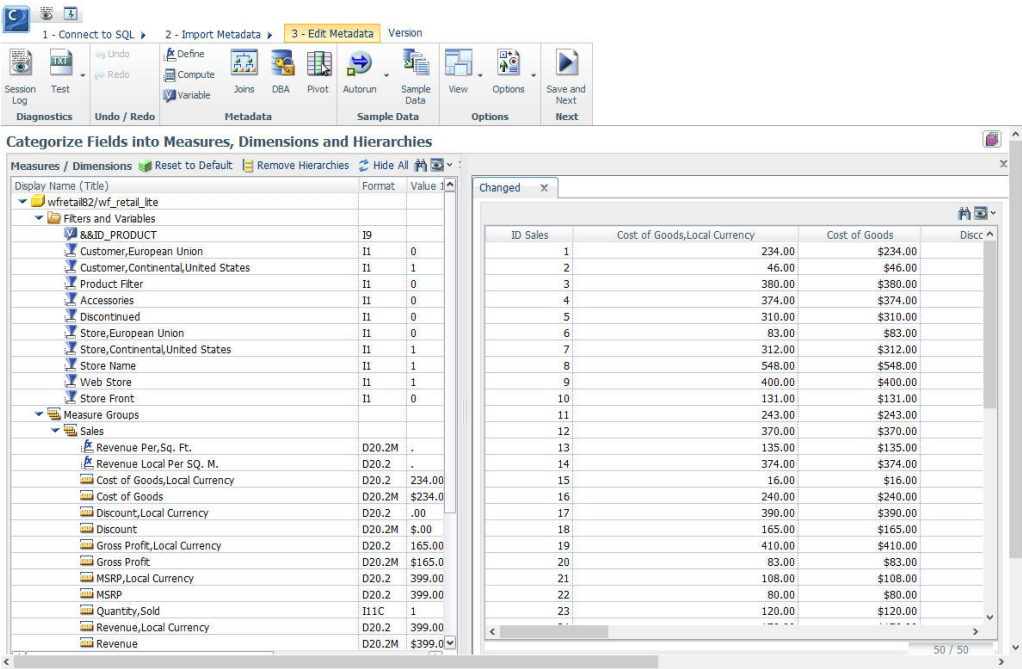
The Create Synonym screen opens.

3. Select one or more synonym check boxes.
4. Click *Next*.

The synonyms are created in the application directory.

Step Three: Edit Metadata

The first screen in this step is the Categorize Fields into Measures, Dimensions and Hierarchies screen. In this step, you can manage and enhance your metadata as required, to simplify and enrich future analytics. By default, you can see your fields interpreted as measures and dimensions, as shown in the following image. You can modify the synonym structure by using the ribbon functions and the shortcut menu options available in the Measures/Dimensions window.



For more information about the options that are available in the Categorize Fields into Measures, Dimensions and Hierarchies screen, see [Categorize Fields Into Measures, Dimensions and Hierarchies Screen Overview](#) on page 291.

Related Information:

- ☐ [Pivoting Repeating Columns Into Rows](#) on page 322.
- ☐ [Creating Hierarchies](#) on page 325
- ☐ [Creating Clusters](#) on page 321

Editing And Deleting Metadata

You are able to edit or delete previously created synonyms using the shortcut menu.

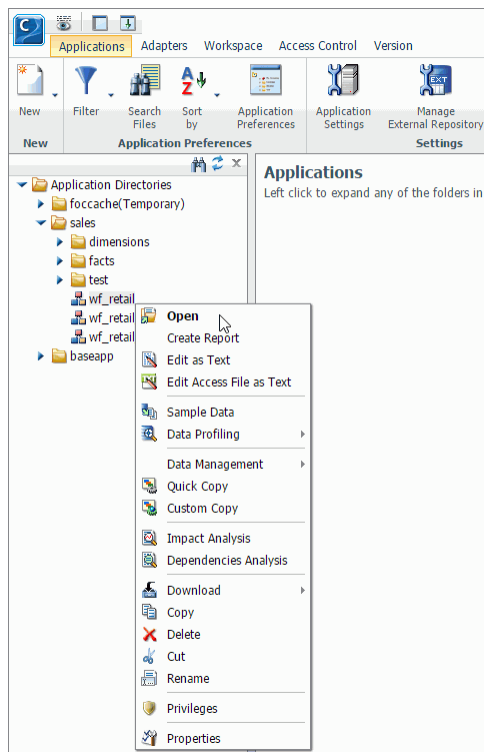
Procedure: How to Edit a Synonym

1. To edit an existing synonym, in the Resources tree, right-click a domain or folder, point to *Metadata*, and then click *Edit*.

The Reporting Server window opens. A list of synonyms associated with the folder and two other shared applications, foccache and baseapp, display in the Applications directory.

Note: Foccache is a temporary application directory for your current session, its contents are automatically deleted once you close the Reporting Server window. Baseapp is a permanent application directory that all users across all domains can see. You can use the baseapp application to store synonyms and images that need to be shared by all users. Only users with the manager user ID can populate the baseapp application.

2. Right-click a synonym and click *Open*, as shown in the following image.



The Categorize Fields into Measures, Dimensions and Hierarchies screen opens, where you can modify your synonym.

3. Make edits to the synonym, as required.
4. To save your synonym, click *File*, and then click *Save*.
5. Close the Reporting Server window.

***Procedure:* How to Delete Metadata**

1. To delete an existing synonym, right-click a domain or folder, point to *Metadata*, and then click *Edit*.

The Reporting Server window opens. A list of synonyms associated with the folder displays in the Applications directory.

2. Right-click the synonym that you want to Delete, point to *Data Management*, and then click *Delete All Data*.

A confirmation window opens, alerting you that you are deleting every row of data.

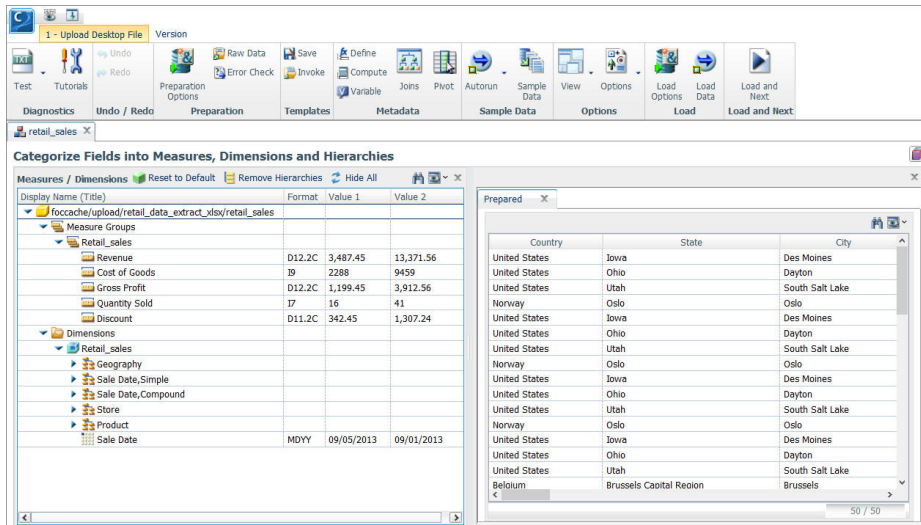
3. Click *OK* to proceed.
4. To remove the synonym file from the view, right-click this synonym again and then click *Delete*.
5. Close the Reporting server window.

Using Data Preparation Options

The Upload wizard and Connect to Data wizard offer a suite of data preparation options, which are designed to help you prepare your data for future analytics. With these data preparation tools, you can assess your data for validity and consistency, troubleshoot errors, and enhance accuracy and uniformity of your data. The following sections describe these options and explain how to use them.

Categorize Fields Into Measures, Dimensions and Hierarchies Screen Overview

The Categorize Fields into Measures, Dimensions and Hierarchies screen, displayed in the following image, allows you to preview and modify your synonym before you upload it to the target environment.



The Categorize Fields into Measures, Dimensions and Hierarchies screen consists of two sections:

- ☐ **Ribbon.** Provides access to data preparation functions that you can use to customize your synonym.
- ☐ **Windows.** Present areas of functionality that you can use to preview or modify the elements of the synonym.

The windows allow you to preview and modify the elements of the synonym.

Ribbon

The following image shows the ribbon of the Categorize Fields into Measures, Dimensions and Hierarchies screen.



The following sections describe the groups and functions of the ribbon.

Diagnostics

The Diagnostics group contains the Test and Tutorials buttons. You can enable the Test button to open the Session Log window, clear the session log, view the synonym being prepared in a text window, and view a list of prepared worksheets.

The options include:

Session Log

Opens the Session Log window. The session log contains all of the commands that have been processed for the current upload session. Using this window, you can filter the log file, clear the log file, download the log file to your PC, refresh the log, and manage your server agents.

Clear Session Log

Deletes all content from the log file.

View MFD

Opens the Master File being prepared in a text window. You can search for text within the Master File.

View AFD

Opens the Access File being prepared in a text window. You can search for text within the Access File.

Prepared Worksheets

Opens a window that lists the prepared Worksheets and the details of each, including the date on which it was prepared, the number of rows and columns, whether there is a header row, and the names of the target application, target synonym, target table, target adapter, and target connection.

The Tutorials button opens a high-level document that outlines the main functions of the Categorize Fields into Measures, Dimensions and Hierarchies screen, as well as the basic flow of common procedures.

Undo/Redo

The Undo / Redo group contains the Undo and Redo options. You can use these options to reverse the last action you performed, or to cancel your last Undo action. For example, if you renamed a field, but do not want to save it, you can click *Undo*.

Preparation

The Preparation group contains options that you can use to review your data and prepare it for upload. The options include:

Preparation Options

Opens the Preparation Options dialog box, where you can set the following options for Excel spreadsheets.

Data scan

Scans all records to determine data type for each column

Row scan limit

Indicates how many worksheet rows are scanned to determine the formats of the columns. Set this value to 0 if you want to scan all of the worksheet rows.

Extend character length

Extends character length by the percentage specified.

Number of header rows

Sets the number of rows used as a header row in the worksheet. Set the value to 0 if a header is not used.

Decompose Date fields into components

Decomposes Date fields into Year, Quarter, Month, and Day components.

Date Order

Allows you to select a format in which the Date fields appear.

Activate GEOGRAPHIC_ROLE assignment

Assigns geographic roles based on column name analysis.

Provides the following additional options that are only available for comma separated values (CSV) files:

Field Delimiter

Allows you to select a column separator from the number of available options, which include: none, comma, tab, pipe, space, semi-colon, colon, type-in delimiter. The default value is comma.

Non-printable Delimiter

Defines whether the string in the delimiter field is a printable string or a non-printable decimal character.

Field Enclosure

Allows you to select an enclosure characters for each field from the number of available options, which include: none, single quote, double quote, space, and type-in enclosure. The default value is double quote.

Header row

Defines if the header line be used as column names.

Preserve format

Indicates whether the original data layout, including empty records and linefeeds, will be preserved.

CODEPAGE

Specified the code page for the stored data that will be added to the Master File of the generated synonym.

CDN

Continental Decimal Notation (CDN) determines the punctuation used in numerical notation for input data.

Raw Data

Displays the raw data of the file to be uploaded in the Output window.

Error Check

Produces informational messages, if the uploaded file contains inconsistent data, based on the first row of data. The Error Check result is displayed in the Output window.

Templates

The Templates group contains options that you can use to create and apply a template to your selected file. The options include:

Save

Saves the existing synonym as a template.

Invoke

Invokes saved templates that you created earlier.

Metadata

The Metadata group contains several data preparation options that you can use to enhance your data. This includes defining virtual fields, creating Compute and Variable fields, joining data sources, and pivoting rows of data into columns. The options include:

Define

Opens the Expression builder, where you can create a define field.

Compute

Opens the Expression builder, where you can create a compute field.

Variable

Opens the properties window, which provides access to the following options:

Variable Name

Specifies the name of the variable.

Prompt Caption

A description of the variable that will appear in the prompt.

Default Value

Specifies the default value of the variable.

Prompt Values

Specifies optional prompt values for the variable.

Quoted

If selected, this option adds single quotation marks around the variable.

Format

Provides a drop-down menu of date formats for the variable. The options are Alpha fixed, Integer, and Decimal Packed.

Joins

Opens the Modeling View window, where you can join multiple worksheets to create a cluster.

Pivot

Opens the Pivot Prepared Data dialog box, where you can pivot any repeating columns or groups of columns into rows.

Sample Data

The Sample Data group contains options that you can use to toggle the Autorun and Data Preview options On and Off, stop the current request, and view the Sample Data in the Output Window to instantly visualize field values.

Autorun

Provides access to the following options:

Off

Disables autorun in the Output Window.

On

Enables autorun in the Output window.

Data Preview

Provides access to the following options:

Off

Disables the display of data values in the Measures/Dimensions Window.

On

Enables the display of data values in the Measures/Dimensions Window.

Sample Data

Runs the Sample Data request in the Output Window.

Options

The Options group consists contains the View and Options functions. You can use the View button to customize the view of your Categorize Fields into Measures, Dimensions and Hierarchies screen. You can use the Options button to specify options for generating and displaying the synonym and set upload limits.

Click View to access the following options to customize your screen:

Reset View

Reverts all your screen customizations to the default view.

Grid Details

Provides access to the following options:

View as Columns

Displays data in columns inside windows. This option is enabled by default.

View as Tooltip

Displays data in tooltips, as you hover over fields.

Table/Column Window

Provides access to the options that control placement of the Table/Column window on the screen, such as:

Tile Horizontal

Displays windows on the screen in the horizontal succession. This option is enabled by default.

Tile Vertical

Displays windows on the screen in the horizontal succession.

Show

Displays the Table/Column window on the screen.

Hide

Hides the Table/Column window from view. This option is enabled by default.

Measures/Dimensions Window

Provides access to the options that control placement of the Measures/Dimensions window on the screen, such as:

Show

Displays the Measures/Dimensions window on the screen. This option is enabled by default.

Hide

Hides the Measures/Dimensions window from view.

Filters/Groups Window

Provides access to the options that control placement of the Filters/Groups window on the screen, such as:

Show

Displays the Filters/Groups window on the screen.

Hide

Hides the Filters/Groups window from view. This option is enabled by default.

Output Window

Provides access to the options that control the placement of the Output window on the screen, such as:

Show Floating

Undocks the Output window, so it can be moved around the screen.

Show Docked

Docks the Output window on the right side of the screen. This option is enabled by default.

Show Maximized

Enlarges the Output Window to the size of the screen.

Hide

Hides the Output Window.

Multiple Output Windows

Provides access to the options that control the appearance of multiple Output windows on the screen, such as:

Show All Tabbed

If this option is selected, all output requests are shown in the Output window in tabs. This option is selected by default.

Show Only Latest

If this options is selected, the single-tab Output window is reloaded to display the latest output request.

Click *Options* and select *Advanced* to select the following synonym creation options:

Column Management

Provides access to the following options.

Name display strategy

You can select how the column names are displayed in the Tables/Columns and Measures/Dimensions windows from one of the following choices:

- ☐ **Name.** Assigns the FIELDNAME attribute from the synonym as the column name displayed in the Tables/Columns and Measures/Dimensions windows.
- ☐ **Title.** Assigns the TITLE attribute from the synonym as the column name displayed in the Tables/Columns and Measures/Dimensions windows. TITLE is the default value.
- ☐ **Description.** Assigns the DESCRIPTION attribute from the synonym as the column name displayed in the Tables/Columns and Measures/Dimensions windows.
- ☐ **Alias.** Assigns the ALIAS attribute from the synonym as the column name displayed in the Tables/Columns and Measures/Dimensions windows.

Function display strategy

Selects the functions information that displays in the Expression Builder, and on the context menu for a column, when you add a function to the synonym. Select one of the following values:

- ☐ **Syntax.** The function syntax (function name and parameters) is displayed. This is the default value.
- ☐ **Short Description.** A short description of what the function calculates is displayed.

Filter Options

Provides options for selections in check boxes.

Check boxes display strategy

Describes how selected values display when you select values in a filter. Select one of the following choices:

- ☐ **Show selected values on top.** Moves the selected (checked) values to the top of the list of values. This is the default.
- ☐ **Keep selected values in place.** Leaves the selected values in their original places in the list of values.

Check boxes: delay for server action (seconds)

Sets the time in seconds before the server acts on activation or deactivation of a check box. The default value is 1.5 seconds.

Language Generation Options

Provides options for generating field names and synonym references in the synonym being prepared.

Use segment to qualify field reference

Specifies when the segment name should be added to the field name in order to qualify the field name. Select one of the following values:

- ☐ **For duplicate fields.** Uses the segment name to qualify the field name only when multiple segments contain the same field name. This is the default value.
- ☐ **Always.** Uses the segment name to qualify the field name.

Use application name to qualify synonym reference

Specifies whether synonym references in the synonym being prepared will include the application name. Select one of the following values:

- ☐ **Yes.** Adds the application name to synonym references (appname/synonymname). This is the default value.
- ☐ **No.** Does not add the application name to synonym references.

Limits

Sets limits for working with the Upload or Connect to Data wizard.

Maximum number of rows for test

Sets the maximum number of rows of sample data displayed. The default value is 50.

Maximum number of columns for test

Sets the maximum number of columns of sample data displayed. The default is 999999.

Maximum number of identical error messages

Sets the maximum number of identical error messages that can be generated by the wizard.

Random Sampling limit (in %)

Defines the percentage for the random sample that is used for statistical analysis.

Undo/Redo Limit

Sets the maximum number of undo and redo actions supported. The default value is 50.

Load

The Load group contains the Load Options and Load Data items.

Click *Load Options* to select the following data loading options. These options may change depending on the target adapter selected.

Load Option

Provides access to the following options:

New

Uploads a new worksheet to create a synonym.

Append Existing Data

Adds new data to the existing synonym without changing the structure and parameters of the synonym.

Target Adapter

Allows you to select a database, in which the unloaded data is stored.

Target Connection

Specifies a connection for the selected adapter.

Target Synonym Application

Specifies the application directory for the synonym. Click the ellipsis button to change this directory.

Target Synonym

Specifies the name of the synonym, as appears in the repository. By default, the name is the same as the name of the source worksheet.

Target Table Name

Indicates the name of the target table in the database.

Bulk Load

Specifies the method of loading data.

Target Data File

Contains the name of the DFIX target data file in the database.

Field Delimiter

Allows you to select a column separator from the number of available options, which include: none, comma, tab, pipe, space, semi-colon, colon, type-in delimiter. The default value is comma.

Header row

Defines if the header line be used as column names.

Field Enclosure

Allows you to select an enclosure characters for each field from the number of available options, which include: none, single quote, double quote, space, and type-in enclosure. The default value is double quote.

Overwrite Existing Synonym

Overwrites the existing synonym with the same name.

Click *Load Data* to upload your synonym to the repository without leaving the Upload or Connect to Data wizard. The Load Data button also brings up the Load Options dialog box, allowing to review and modify your loading parameters.

Load and Next

The Load and Next group the Load and Next function, which uploads your synonym to the repository and brings you to the next screen, where you can select how to use your synonym.

Windows

The Categorize Fields into Measures, Dimensions and Hierarchies screen contains the following windows:

- ☐ Table/Column
- ☐ Measures/Dimensions
- ☐ Filters/Groups
- ☐ Output

The default view displays Measures/Dimensions and Output windows. You can customize the view of your screen by clicking *View* on the ribbon.

Table/Column Window

The Table/Column window displays the fields in the same order as they are displayed in the source file.

The Table/Column Window is shown in the following image.

Display Name (Title)	Format	Value 1	Value 2
▼ fccache/upload/retail_data_xlsx/retail_data			
▼ RETAIL_DATA			
Product Category	A19V	Video Production	Video Production
Product Subcategory	A30V	Video Editing	Video Editing
Store Type	A13V	Store Front	Store Front
Store Name	A19V	Des Moines	Dayton
Country	A16V	United States	United States
State	A27V	Iowa	Ohio
City	A19V	Des Moines	Dayton
Region	A15V	North America	North America
Sale Date	MDYY	09/05/2013	09/01/2013
Revenue	D12.2C	3,487.45	13,371.56
Cost of Goods	I9	2288	9459
Gross Profit	D12.2C	1,199.45	3,912.56
Quantity Sold	I7	16	41
Discount	D11.2C	342.45	1,307.24
Sale Date, Year	I4	2013	2013
Sale Date, Quarter	I1	3	3
Sale Date, Month	I2	9	9
Sale Date, Day	I2	5	1
Sale Date, Y	YYMDy	2013	2013
Sale Date, Y-Q	YYMDq	2013 Q3	2013 Q3
Sale Date, Y-M	YYMDm	2013/09	2013/09
Sale Date, M-D-Y	MDYY	09/05/2013	09/01/2013

The Table/Column window has the following options:

Display

Toggles between two display options, Columns and Tables.

Find

Opens the search field, where you can type the name of the field that you want to find and allows you to modify your search options.

View

Provides access to the following options:

Expand All

Expands all nodes, folders, and hierarchies.

Collapse All

Collapses all nodes, folders, and hierarchies.

Choose columns

Opens the Choose Columns dialog box, where you can select the columns that you want to display in the Measures/Dimensions window.








Reset to defaults

Reverts the window to its default view.

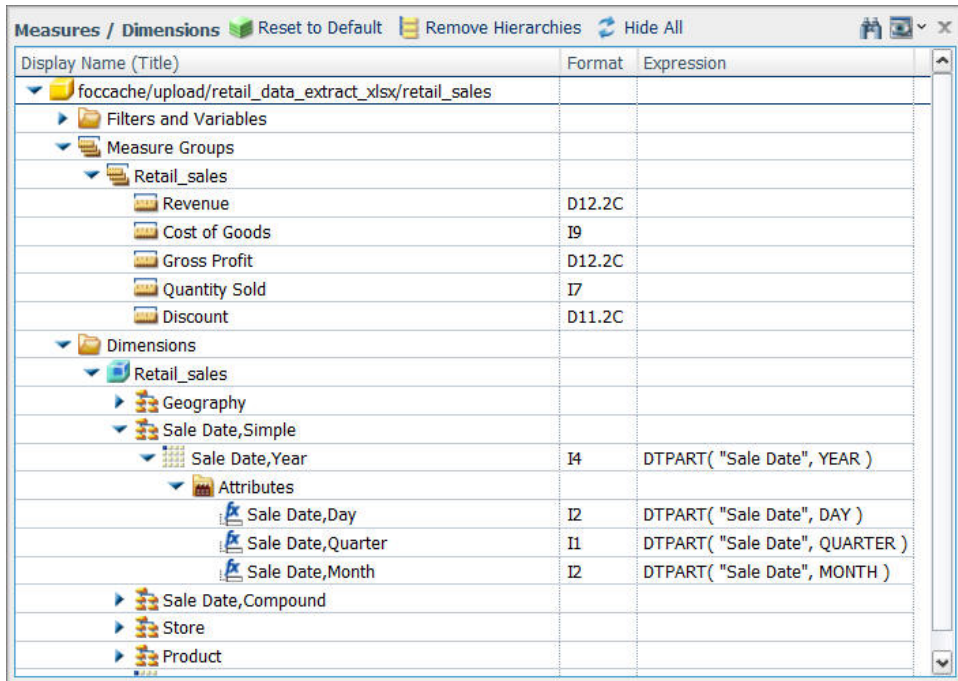
Measures/Dimensions Window

The Measures/Dimensions window shows you how the Upload and Connect to Data wizards interpret your data and designates the data type for each field. The numeric values, such as Gross Profit or Cost of Goods, are interpreted as measures. The alphanumeric values, such as Product or Geography, become dimensions. A dimension is a way to categorize data.

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 assign dimension attributes to any dimension field, whether or not it is in a hierarchy. When applied to a field, attributes provide supplementary information that can be used for analysis or display. For example, in a Geography hierarchy, which includes the Country, State, and City dimensions, Population can be assigned as an attribute of the City dimension. Each category of data is represented by an icon. The following table identifies these icons and describes what they represent.

Icon	Identifies	Description
	Measure group	Contains individual measures.
	Measure	Measure is a numeric field. It resides inside a measure group. Measures can be moved between measure groups and between measures and dimensions areas.
	Dimension folder	Contains dimensions, dimension hierarchies, and attributes.
	Dimension Hierarchy	Resides inside the dimension folder and contains individual dimensions and associated attributes.
	Dimension	Dimension is an alphanumeric field. It resides inside the dimension hierarchies. Dimensions can be moved between dimension folders, between and outside dimension hierarchies. You can move a dimension into a measure group and turn it into a measure. You can also move a dimension inside an attribute folder and turn it into an attribute.
	Attribute folder	Contains individual attributes associated with a dimension.
	Attribute	Resides inside an attribute folder. Attributes can be moved between attribute folders. If you move an attribute outside an attribute folder, it inherits the identity of a hierarchical level to which it is moved.

The Measures/Dimensions window is displayed in the following image.



The Measures/Dimensions window has the following options:

Reset to Default

Eliminates all changes and reverts the view back to the initial categorization of measures and dimensions.

Remove Hierarchies

Removes dimension hierarchies and displays individual dimensions under the Dimensions folder.

Hide All

Hides all fields from the view.

Note: When a field is hidden, it is moved to the Table/Column window.

Find

Opens the search field, where you can type the name of the field that you want to find and allows you to modify your search options. Any occurrence of the search string found in the window is highlighted.

View

Provides access to the following options:

Expand All

Expands all nodes, folders, and hierarchies.

Collapse All

Collapses all nodes, folders, and hierarchies.

Choose columns

Opens the Choose Columns dialog box, where you can select the columns that you want to display in the Measures/Dimensions window.

Reset to defaults

Reverts the window to its default view.

Each node in the Measures/Dimensions window opens a context-sensitive shortcut menu that can be used to insert and modify fields, view sample data, perform statistical analysis and data preparation techniques. You can access these options by right-clicking each level. These context menu options are conditional and depend on the type of an element and its place in the synonym structure. The following list identifies the context menu options for measure group folders, measure groups, measures, dimensions node, dimension folders, dimensions, hierarchies, and attributes.

Insert

Depending on the context, provides access to the following options:

New Measure Group

Creates a new measure group inside this Measure Group folder.

New Dimension Folder

Creates a new dimension folder inside the Dimensions node.

New Parent/Child Hierarchy

Opens the Create a Parent/Child Hierarchy dialog box, where you can create a new hierarchy.

New Levels Hierarchy

Creates a new level hierarchy inside this dimension folder.

Filter

Opens the Filters/Groups window, where you can create or modify a filter.

Character Function

Provides a choice of preconfigured character functions that open in the Function Assist for New Define dialog box.

Numeric Function

Provides a choice of preconfigured numeric functions that open in the Function Assist for New Define dialog box.

Advanced Function

Opens the Define dialog box, where you configure a new function.

Group Values

Opens the Filters/Groups window, where you can configure group values.

Group Numeric (Binning)

Opens the Binning wizard, where you can configure a numeric group.

Aggregation Function

Provides a choice of preconfigured aggregation functions that open in the Function Assist for New Compute dialog box.

Analytic Function

Provides a choice of preconfigured analytic functions that open in the Function Assist for New Compute dialog box.

Advanced Aggregation

Opens the Compute dialog box, where you can a compute field using advanced aggregation options.

Properties

Opens the Properties dialog box. For more information, see [Using the Properties Dialog Box](#) on page 312.

Pivot

Opens the Pivot Prepared Data dialog box, where you can pivot any repeating columns or groups of columns into rows.

Geographic Role

Opens the Set Geographic Role dialog box, where you can configure a geographic role for the field.

Create Hierarchy

Transforms the dimension into a hierarchy. This option is only available for standalone dimensions.

Remove Hierarchy

Removes a hierarchy and displays all the items inside this hierarchy as individual dimensions.

Remove from Hierarchy

Removes the field from a hierarchy. This option is only available for the dimensions that are part of a hierarchy.

Show/Hide Attribute

Toggles between displaying or hiding attributes for the field.

Sample Data

Displays the sample data for all the measure groups inside this measure group folder in the Output window.

Sample Map

Displays the data for this field in a map, which runs in the Output window.

Data Profiling

Provides access to the following options:

Statistics

Displays statistic analysis of the field in the Output window.

Values

Displays the values for this field for the selected number of rows in the Output window.

Duplicate Values

Displays the duplicate values for this field for the selected number of rows in the Output window.

Values (Bar Chart)

Displays the values for the field in a bar chart in the Output window. The chart is limited to 50 values.

Values (Pie Chart)

Displays the values for the field in a pie chart in the Output window.

Impact Analysis

Displays the impact analysis results in the Output window.

Rename

Opens the Rename dialog box where you can rename the field.

Hide

Moves the field to the Table/Column window.

Cut

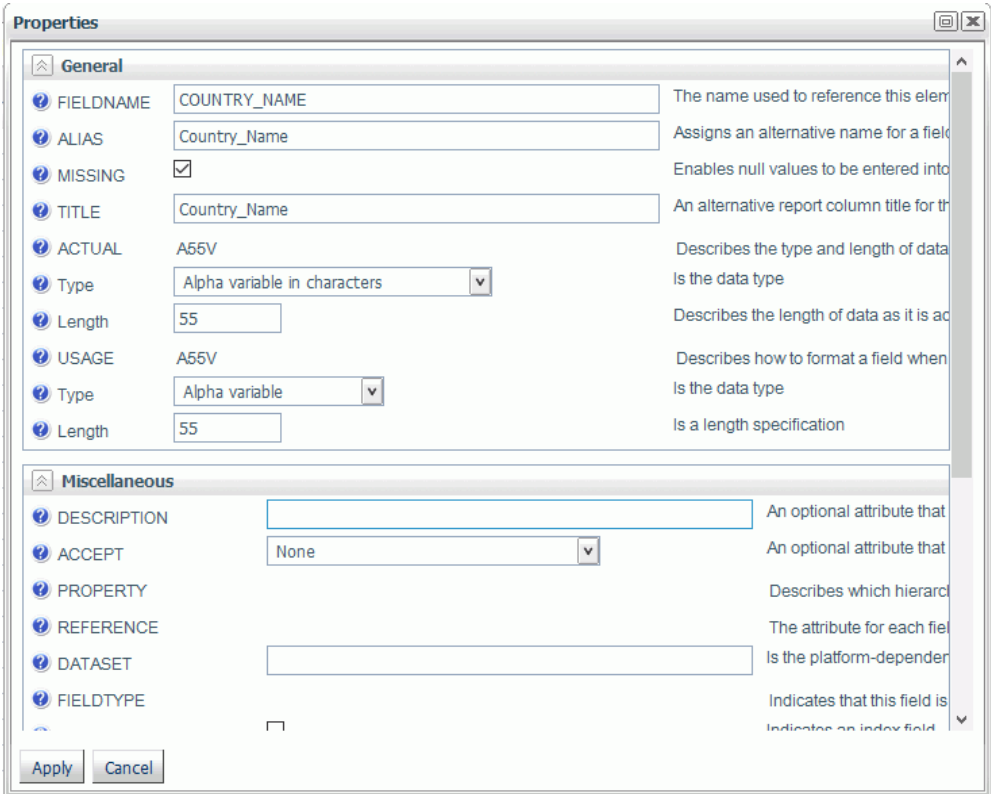
Cuts the field from its directory.

Paste

Pastes an item inside the directory.

Using the Properties Dialog Box

You can use the Properties dialog box to change settings for your data, such as appearance, formatting, data descriptions, and others. The options in this dialog box change depending on the type of data you are editing. An example of the Properties dialog box is shown in the following image.



The Properties dialog box is organized into various groups, as described below.

Codepage Group

Contains properties for a codepage.

CODEPAGE

Defines the code language used in the synonym.

Dimension Group

Contains properties for a dimension.

Caption

Displays the name of the dimension folder.

Number of Measures

Shows the number of measures that currently reside inside a measure group.

Number of Hierarchies

Displays the number of hierarchies that currently reside inside a dimension folder.

Number of UDAs

Displays the number of user defined attributes that currently reside inside a dimension folder.

General Group

Contains general properties that can be applied to various elements.

FIELDNAME

Indicates the name that is used to reference this data element in the request.

ALIAS

Specifies the name of the field, as it appears in the synonym.

MISSING

If selected, allows null values to be entered into and read from a field in data sources that support null data.

TITLE

Specifies an alternative report column title for the field.

ACTUAL

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

Type

Allows you to select a data type for this field, as it is stored in the data source.

Length

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

USAGE

Describes how to format a field when displaying it in a report or using it in a calculation.

Type

Allows you to select a data type for this field, as it is displayed in a report or used it in a calculation.

Length

Describes the length of data as it is displayed in a report or used it in a calculation.

Decimal

Specifies the number of digits that follow the decimal point.

Options

Displays options for this field.

Negative

Allows you to select a format in which negative numbers are displayed.

Comma

Allows you to choose whether large numbers are displayed with a comma.

L - Leading zeroes

If selected, adds leading zeroes to a number.

S - Print blank for zero

Specifies whether to display the blank field, if the data value is zero.

Percent sign

For the percentage values, toggles between displaying a percentage sign only, or calculating the percentage value and displaying a percentage sign.

E - Scientific notation

If selected, only significant digits are displayed.

Currency Symbol

Provides access to various currency formats in a drop-down list.

CURRENCY

Designates a value that represents a number of units in a specific type of currency.

SUFFIX

Identifies the type of data source associated with this description.

MRF_PROFILE

Defines the name of the focexec that will be executed before request containing MFD.

FDEFCENT

Defines the default century for handling cross-century dates.

FYRTHRESH

Defines the year threshold for handling cross-century dates.

REMARKS

Allows you to add notes and descriptive information about the data source.

DATASET

Specifies a path to the data source.

IOTYPE

Specifies how to read the data file in the absence of LRECL/RECFM information from a FILEDEF or allocation.

BYTEORDER

Specifies byte order.

Hierarchy Group

Contains properties for a hierarchy.

Caption

Displays the name of the dimension folder.

Hierarchy structure

Specifies the structure of the hierarchy. The options are: levels and parent/child.

Number of levels

Displays a number of levels inside a levels hierarchy or a number of elements inside a parent/child hierarchy.

Miscellaneous Group

Contains supplemental properties. All properties in the Miscellaneous group are disabled by default.

DESCRIPTION

Creates an attribute for comments and remarks within a field.

ACCEPT

Creates an attribute that can be used to populate an auto-prompt dialogue box or to validate data as it is entered into a field from a MODIFY procedure.

PROPERTY

Indicates the place of the field in the hierarchy of the Master File.

REFERENCE

Shows field attributes, such as physical parent/child relationships among the elements in the file.

DATASET

Specifies a physical name of the data source or alternate index.

FIELDTYPE

Specifies the type of the field.

I - Index

If selected, the field is displayed as an index field.

R - Readonly

If selected, the field is displayed as a read-only field.

ACCESS_PROPERTY

Specifies access options for the field's data.

INTERNAL

If selected, the field does not appear in the list of available fields or Sample Data.

NEED_VALUE

If selected, the field requires a value to access the data.

Select By

Allows you to set a selection parameter. The choices are: value, range, and multiple values.

HELPMESSAGE

Allows you to add a help message to the field.

GEOGRAPHIC_ROLE

Defines a geographic role of the field.

TEMPORAL_PROPERTY

Specifies a temporal property of the field.

USE_STYLE

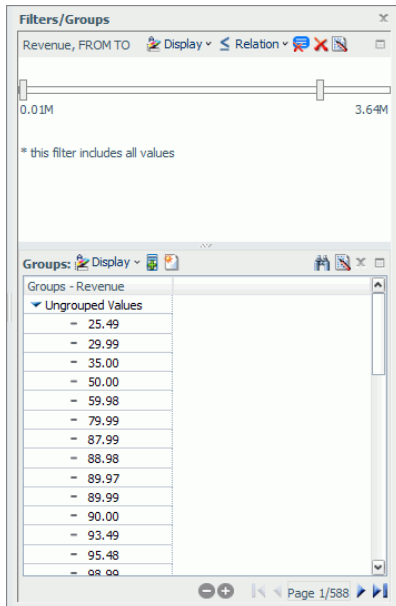
Allows you to select a style for the field.

DV_INCLUDE

When enabled, includes the base file DV information.

Filters/Groups Window

The Filters/Groups window automatically opens when you create a new filter or group. When you open two or more items at the same time, they display nested in the same window, as shown in the following image.



Depending on the field type, the Filters/Groups window displays the following options:

Find

Opens the search field, where you can type the name of the field that you want to find and allows you to modify your search options.

Expand

Maximizes the section of the window.

Drop-down arrow

Provides access to the following options for filters:

Display

Determines a display format for the filter. The options include: text box, double list, check-box list, pull-down list, and slider. The default option is slider for Numeric fields and check box for all other fields.

Relation

Creates a relational expression. The options include: EQ, NE, LT, GT, LE, GE, and Range. The default value is Range for Numeric and EQ for all other fields.

Values

Toggles between showing and hiding the NULL values for this field.

Hide Filter Card

Removes filter data from the Filters/Groups window, while leaving the window open.

Delete Filter

Deletes the filter from the synonym.

Properties

Opens the Filter properties dialog box, where you can change the Title for the filter, and view its Name and Base field name properties.

Move Up

Moves the filter up in the Filters/Groups window.

Move Down

Moves the filter down in the Filters/Groups window.

Provides access to the following options for groups:

Display

Determines a display format for the group. The options are: Double List and Single List. The default value is Single List.

Add all shown values to a new group

Selects all values and adds them to a new group.

Create a New Group

Creates a new empty group, to which you can add values manually.

Edit properties

Opens the Edit properties dialog box, where you can edit the Name, Title, and Default Value properties.

Hide Group Card

Removes the group data from the Filters/Groups window.

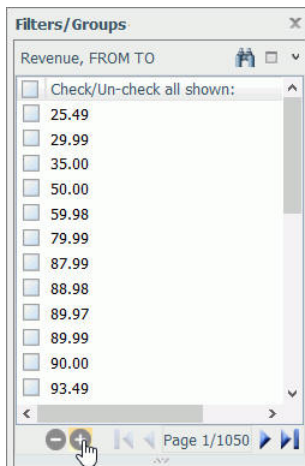
Move Up

Moves the group up in the Filters/Groups window.

Move Down

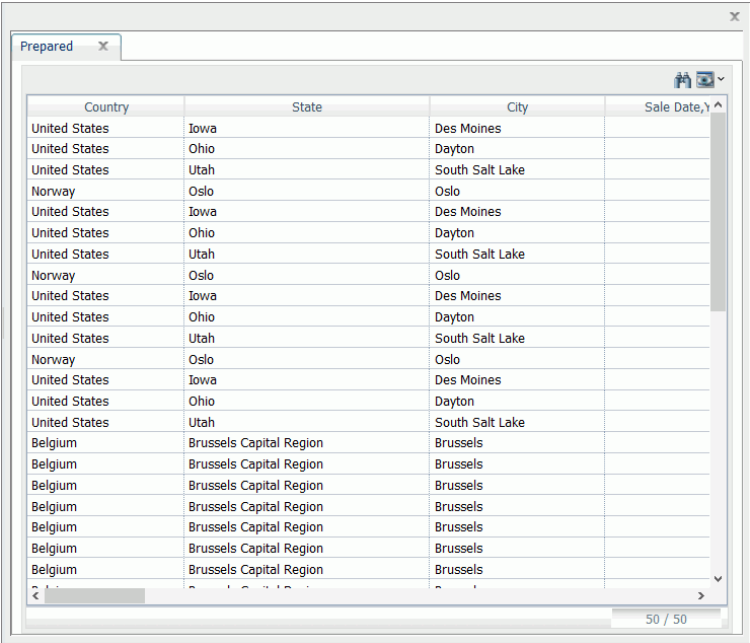
Moves the group down in the Filters/Groups window.

Additionally, if the list of values spans through multiple pages, you can increase or decrease the number of values displayed on each page by clicking the *Increase Page Size* or *Decrease Page Size* buttons, as shown in the following image. You can use the *First Page*, *Previous Page*, *Next Page*, and *Last Page* buttons to navigate between pages.



Output Window

The Output window displays data and status messages for various features. The Output window is shown in the following image.



The Output window has the following options:

Find

Opens the search field, where you can type the name of the field that you want to find and allows you to modify your search options.

View

Provides access to the following options:

Expand All

Expands all nodes, folders, and hierarchies.

Collapse All

Collapses all nodes, folders, and hierarchies.

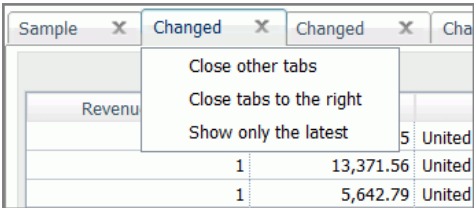
Choose columns

Opens the Choose Columns dialog box, where you can select the columns that you want to display in the Measures/Dimensions window.

Reset to defaults

Reverts the window to its default view.

If more than one tab is open, you can right-click a tab to access the tab menu, as shown in the following image. The menu allows you to close all other tabs, close tabs to the right, or show only the latest tab.



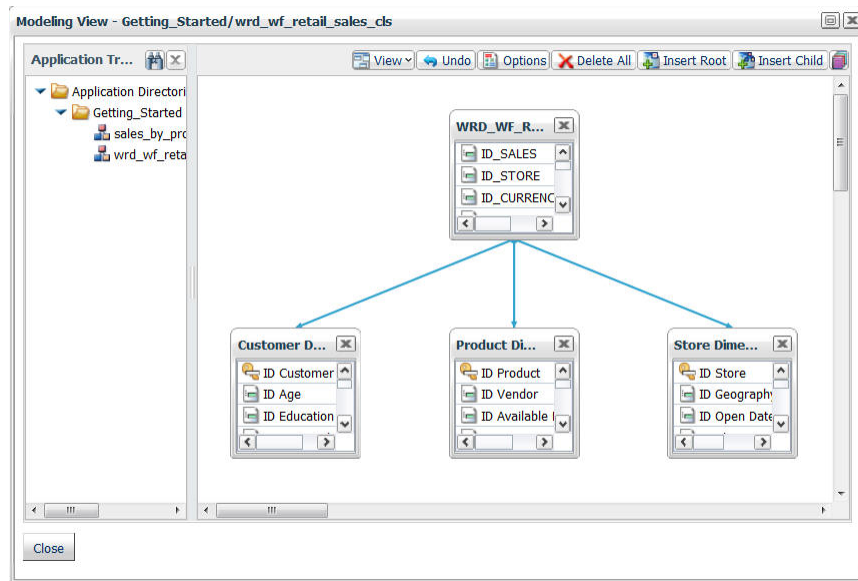
Creating Clusters

When you need to add more tables to your synonym, you can use the Join feature to create a cluster. This allows you to enhance the structure of your synonym by introducing more data.

Procedure: How to Create a Cluster

1. On the Categorize Fields into Measures, Dimensions and Hierarchies screen, on the ribbon, in the Metadata group, click *Joins*.
The Modeling View dialog box opens.
2. Click *Insert Child*.
The Insert Child dialog box opens.
3. Click a Master File from the list, and then click *OK*. You can select several Master Files by holding the Shift key.

The completed cluster displays in the Modeling View dialog box, as shown in the following image.



4. Click *Close*.

The new columns are added to the Table/Column window.

5. Drag columns to the Measures/Dimensions window to modify the synonym.

Related Information:

[Categorize Fields Into Measures, Dimensions and Hierarchies Screen Overview](#) on page 291.

Pivoting Repeating Columns Into Rows

Some Excel spreadsheets may contain repeating columns, such as sales figures for a series of years. There may even be repeating column groups, such as both budget and actual figures for a series of years. You can use the pivot option to transform these columns or groups of columns into rows.

Procedure: How to Pivot Columns Into Rows

1. On the Categorize Fields into Measures, Dimensions and Hierarchies screen, on the ribbon, in the Metadata group, click *Pivot*.

The Pivot Prepared Data dialog box opens.

2. Set the Pivot Type option to *Repeating column*.
3. In the First column drop-down box, select the first column in the range of repeating columns.
4. In the Last column drop-down box, select the last column in the range of repeating columns.
5. In the Column Title for Pivoted Data, type the new column title that reflects the numeric cell that you are describing.
6. In the Title for Pivoted Key field, type the new column title that represents the repeating columns that you are pivoting into rows.
7. Leave the Formula for Pivoted Key field value unedited. This value is automatically generated by the wizard, it should not be changed.

An example of the completed configuration for pivoting columns is shown in the following image.

8. Click *OK*.

The repeating columns now display as rows. The Pivot button turned into the Remove Pivot button, allowing you to quickly revert your pivoting changes.

Related Information:

- ❏ [Categorize Fields Into Measures, Dimensions and Hierarchies Screen Overview](#) on page 291.

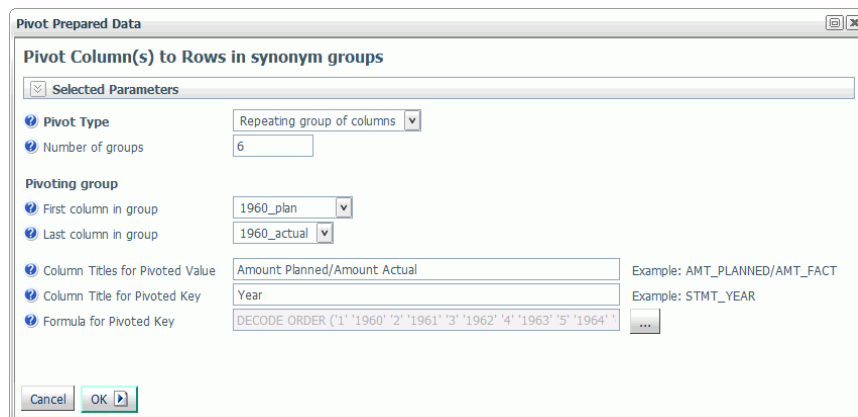
Procedure: How to Pivot Column Groups into Rows

1. On the Categorize Fields into Measures, Dimensions and Hierarchies screen, on the ribbon, in the Metadata group, click *Pivot*.

The Pivot Prepared Data dialog box opens.

2. Set the Pivot Type option to *Repeating group of columns*.
3. In the Number of groups field, specify the number of groups of columns that you are pivoting.
4. In the Column Title for Pivoted Data, type the new column title that will be used for all the columns across the repeating groups.
5. In the Title for Pivoted Key field, type the new column title that represents the repeating columns that you are pivoting into rows.
6. Edit the automatically generated formula in the Formula for Pivoted Key field by clicking the ellipsis button.

An example of the completed configuration for pivoting groups of columns is shown in the following image.



7. Click *OK*.

The repeating groups of columns now display as rows. The Pivot button turned into the Remove Pivot button, allowing you to quickly revert your pivoting changes.

Related Information:

-  [Categorize Fields Into Measures, Dimensions and Hierarchies Screen Overview](#) on page 291.

Creating Hierarchies

When you upload a data file, the wizard creates dimension hierarchies automatically, based on name pattern-matching and date-time analysis. Optionally, you can create additional hierarchies to organize individual columns, enable drill-downs in your data, and add more flexibility to your synonym.

Procedure: How to Create a Hierarchy

1. On the Categorize Fields into Measures, Dimensions and Hierarchies screen, in the Measures/Dimensions window, right-click a dimension folder, point to *Insert*, and then click *New Levels Hierarchy*.

The Hierarchy1 is added to the dimension folder.

2. Right-click the Hierarchy1, and then click *Rename*.

The Rename dialog box opens.

3. Type the name of your new hierarchy and click *OK*.

Note: You can also turn an existing dimension into a dimension hierarchy. To do so, right-click the dimension, and then click *Create Hierarchy*.

4. Drag individual columns into the new hierarchy in a logical order to populate it.
5. To remove a field from the hierarchy, right-click the field, and then click *Remove from Hierarchy*.
6. Once you have edited your metadata, click *Next*.

The Save As dialog box opens.

7. Enter a name for the synonym in the File Name field and click *OK*.

A pop-up window opens, providing access to the following options:

Create.

Allows you to create a report, chart, document, or visualization with your new Master File.

Finish.

Closes the Connect to Data wizard.

Related Information:

- ❏ [Categorize Fields Into Measures, Dimensions and Hierarchies Screen Overview](#) on page 291.

Joining and Blending Data

You can join two or more related data sources to create a larger integrated data structure from which you can report in a single request. The joined structure is virtual. It is a way of accessing multiple data sources as if they were a single data source. This can greatly increase the number of fields available for use in your content, giving you an expanded selection for your reporting and charting purposes.

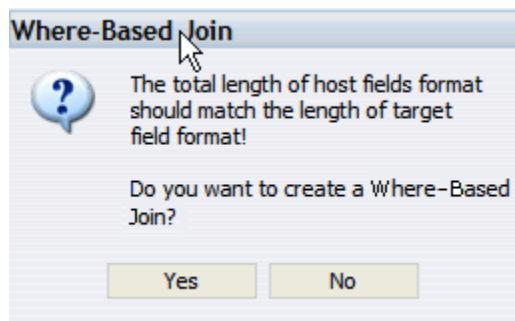
Joins

Using conditional joins, you can establish joins based on conditions other than equality between fields. In addition, the host and cross-referenced join fields do not have to contain matching formats, and the cross-referenced field does not have to be indexed.

Note: You can edit the description of a join by clicking *Edit* in the Join dialog box and typing in the Description section. You can only use letters, numbers, and underscores in your description. No special characters are allowed.

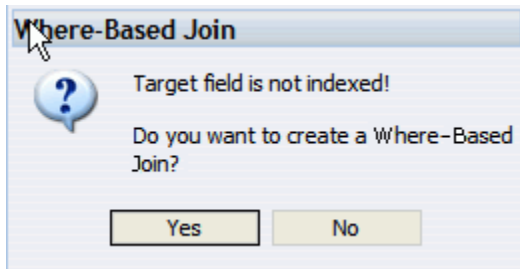
The conditional join is supported for FOCUS and all relational data adapters. Because each data source differs in its ability to handle complex conditional criteria, the optimization of the WHERE syntax differs depending on the specific data sources involved in the join and the complexity of the conditional criteria.

For FOCUS data sources, if the host and cross-referenced join fields do not have common matching formats, the following message appears.



Note: If you click Yes, the Filter dialog box opens, where you can create a Where-Based Join.

If the cross-referenced join field does not have an index, the following message appears.



Note: If you click Yes, the Filter dialog box opens, where you can create a where-based join.

Blending Data

The blend option allows you to explicitly select the data fields that you want to include. More specifically, you can join multi-fact data structures and combine pertinent external data into your current data source, creating a blended data resource. This can be from local or other system resources.

Blending is used to create customized data sources. For example, you may have some of the basic fields available in the current data source, but you can use the blend option to add relevant data fields from a different data source to the current data source in order to create a unique data set.

The blend option allows a new fact table to be added to a cluster master as a parent segment to an existing child segment. This option is available from the Join dialog box. You may want to blend data if you are reporting from two different fact tables that share a common dimension, such as a product dimension. For an example of this, you can view the sample retail database, wf_retail_lite. The wf_retail_lite Master File contains the WF_RETAIL_STORE_SALES segment and the WF_RETAIL_SALES segment. The WF_RETAIL_SALES segment is defined as the parent of the WF_RETAIL_STORE_SALES segment. You may want to add a second fact table to the report. The second fact table in this example will be an Excel spreadsheet that you upload to InfoAssist+ for use with a report or chart. The data in the spreadsheet file that you upload will be joined to the WF_RETAIL_LITE database using a common field.

Note: You can find supported, common fields in your selected database (for example, WF_RETAIL_LITE) using the search feature in the Data pane. If necessary, you may need to add a field in your spreadsheet to map your data to the structure of the database. For example, ID_CUSTOMER. In addition, the name of the primary sheet in Microsoft Excel becomes the name of your data source, so be sure to name the file accordingly.

The following list provides some general rules that apply to the Blend option:

1. The result of blending is that a single dimension is shared between two fact tables. The table must be based on a cluster with at least two segments. One segment is for Fact table 1, the second segment is for the dimension.
2. Two uploaded files cannot be blended because they result in single segment Master Files.
3. Do not use the fields from the blended table as a sort field, since these fields will not have common field when used with fields from other fact tables.

Procedure: How to Blend Data

This procedure describes how to blend data from an external data source into an existing data source. This example uses a Microsoft Excel spreadsheet file.

1. On the *Data* tab, in the *Join* group, click *Join*.

The Join dialog box displays.

2. Click *Add New*.

The Open dialog box displays.

Note: The options that display on the Open dialog box depend on your user privileges.

3. At the top of the Open dialog box, click *Upload Data*.

The Upload wizard opens.

4. Drag your Microsoft Excel spreadsheet file on to the Upload pane, or click *Select Upload File* to locate the file on your local drive.
5. Once the upload is confirmed, click *Next*.
6. On the Select Worksheets pane, confirm the sheet that you want to upload by selecting the check box adjacent to it, and click *Next*.

If your upload is successful, the Categorize Fields into Measures, Dimensions and Hierarchies pane opens.

Note: If there are informational messages or if your upload is unsuccessful, the Status pane opens.

7. Under Measures, drag a common field to Dimensions.

Note: A common field is a field used in both the current Master File and uploaded spreadsheet.

8. Click *Next*.

The data upload is complete and the data source is added to the Open dialog box.

9. In the Open dialog box, click the name of your uploaded data source and then click *Open*.
10. In the Join dialog box, click *Blend* and then click *OK*.

The blended data source fields are now available in the Data pane.

Creating Virtual Fields

A temporary field is a field whose value is not stored in the data source, but can be calculated from the data that is there, or assigned an absolute value. A temporary field takes up no storage space in the data source, and is created only when needed. DEFINE fields and COMPUTE fields are two different types of temporary fields.

When you create a temporary field, you determine its value by writing an expression. You can combine fields, constants, and operators in an expression to produce a single value. For example, if your data contains salary and deduction amounts, you can calculate the ratio of deductions to salaries using the following expression: `deduction / salary`.

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

Note: When creating a DEFINE or a COMPUTE field, the following characters are suppressed and cannot be entered in the Format text box.

space ! " # \$ % & ' () * + , / : ; < = > ? @ [\] ^ _ ` { | } ~

Selecting a Temporary Field

The following information is provided to help you choose the kind of temporary field that you need.

- ☐ **Choose a virtual field** when you want to:
 - ☐ Use the temporary field to select data for your report. You cannot use a calculated value, since it is evaluated after data selection takes place.
 - ☐ Use the temporary field to sort on data values. A calculated value is evaluated after the data is sorted. With the BY TOTAL phrase, you can sort on this type of field.
- ☐ **Choose a calculated value** when you want to:
 - ☐ Evaluate the temporary field using total values or prefix operators (which operate on total values). You cannot use a virtual field, since it is evaluated before any totaling takes place.

- ❑ Evaluate the temporary field using fields from different paths in the data structure. You cannot use a virtual field, since it is evaluated before the relationship between data in the different paths is established.

Detail (DEFINE)

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

The calculation that determines the value of a virtual field is performed on each retrieved record that passes any screening conditions on real fields.

You can define a virtual field in the following ways:

- ❑ **In a Master File.** These virtual fields are available whenever the data source is used for reporting. These fields cannot be cleared by JOIN or DEFINE FILE commands.
- ❑ **In a procedure.** A virtual field created in a procedure lasts only for that procedure.

A DEFINE field is an optional attribute used to create a virtual field for reporting. You can derive the virtual field value from information already in the data source (that is, from permanent fields).


You may define fields simultaneously (in addition to fields defined in the Master File) for as many data sources as desired. The total length of all virtual fields and real fields cannot exceed 32,000 characters.

The Detail Field (DEFINE) dialog box allows you to create a defined field, type a name for the field, and enter a format.

Summary (COMPUTE)

A calculated value (COMPUTE) is evaluated after all of the data that meets the selection criteria is retrieved, sorted, and summed. Therefore, the calculation is performed using the aggregated values of the fields. Calculated values are available only for the specified report request. You specify the COMPUTE command in the body of the report request, following the display command and optionally, introduced by AND. You can compute more than one field with a single COMPUTE command.

The Summary Field (COMPUTE) dialog box allows you to create a computed field, type a name for the field, and enter a format.

The Field List provides similar functionality, including options to display data source fields in a Logical, List, or Structured view. You can also view a complete set of functions, instead of data source fields, by clicking the *Functions* button .

Creating Temporary Fields Independent of a Master File

The temporary fields that you create with the `DEFINE` and `COMPUTE` commands are tied to a specific Master File, and in the case of values calculated with the `COMPUTE` command, to a specific request. However, you can create temporary fields that are independent of either a Master File or a request using the `DEFINE FUNCTION` command.

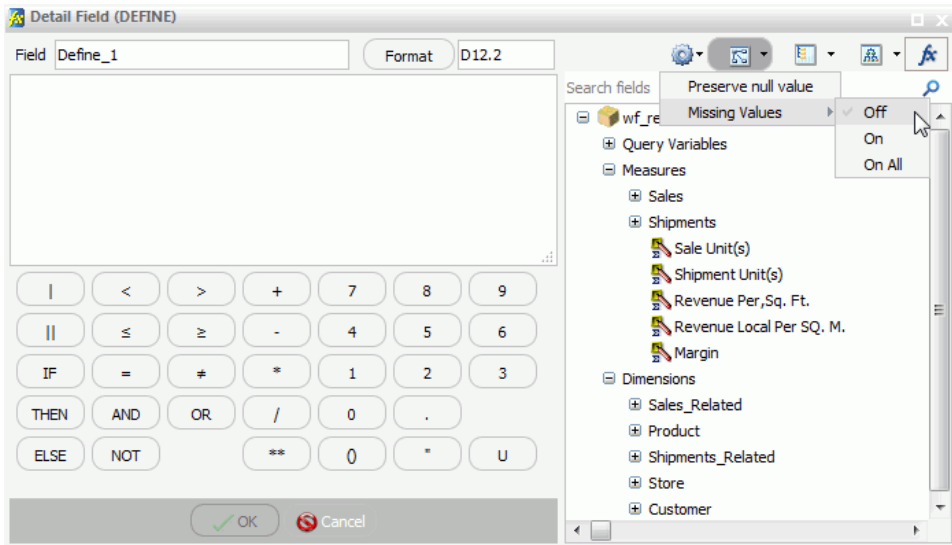
A `DEFINE` function is a named group of calculations that use any number of input values and produce a return value. When calling a `DEFINE` function, you must first define the function.

A `DEFINE` function can be called in most of the same situations that are valid for Information Builders-supplied functions. Data types are defined with each argument. When substituting values for these arguments, the format must match the defined format. Alphanumeric arguments shorter than the specified format are padded with blanks, while longer alphanumeric arguments are truncated.

All calculations within the function are done in double precision. Format conversions occur only across equal signs (=) in the assignments that define temporary fields.

Enabling the Display of Missing Values for a DEFINE or COMPUTE

When working with DEFINES and COMPUTES, you can use the Missing Values option to enable or disable the display of missing values for a DEFINE or COMPUTE field. This allows you to accurately display missing values in reports, charts, and visualizations. The Missing Values option, which is accessible through the Additional Options button, is shown in the following image.



The following descriptions explain each option on the Missing Values drop-down list:

- ☐ **Off.** When selected, MISSING syntax is removed from the DEFINE or COMPUTE field definition. This is the default selection. MISSING treats missing values for numeric fields as zeros and missing values for alphanumeric fields as blanks.
- ☐ **On.** When selected, MISSING ON is added after the format in the DEFINE or COMPUTE field definition. MISSING ON interprets the temporary field as missing.
- ☐ **On All.** When selected, MISSING ON ALL is added after the format in the DEFINE or COMPUTE field definition. MISSING ON ALL indicates that if all fields in the expression have values, then the temporary field has a value. If at least one field in the expression has a missing value, the temporary field also has a missing value.

Creating Content

Create reports, documents, charts, and interactive visualizations using the available features and functionality.

In this chapter:

- | | |
|--|---|
| <input type="checkbox"/> Generating Sample Content | <input type="checkbox"/> Using Navigation Options for Reports |
| <input type="checkbox"/> Creating Reports | <input type="checkbox"/> Creating Maps to Illustrate Trends |
| <input type="checkbox"/> Creating Charts | <input type="checkbox"/> Building InfoMini Applications |
| <input type="checkbox"/> Creating Your Own Chart Types | <input type="checkbox"/> Viewing Data Behind Visuals |
| <input type="checkbox"/> Building a Document | <input type="checkbox"/> Creating HOLD Files |
| <input type="checkbox"/> Creating Multi-Page Documents | <input type="checkbox"/> Creating Shortcuts and URLs |
| <input type="checkbox"/> Building Visualizations | <input type="checkbox"/> Working With the WebFOCUS BUE Portal |
| <input type="checkbox"/> Creating Matrix Charts | <input type="checkbox"/> Creating Report Queries With InfoAssist+ |
| <input type="checkbox"/> Using Active Technologies | <input type="checkbox"/> Ribbon Command Reference |
-

Generating Sample Content

This topic describes how to generate sample content from uploaded or existing data using the Samples Generator. It also provides information about how to interpret and analyze the sample content.

Generating Sample Content Overview

The Samples Generator automatically generates a suite of analytic content based on uploaded Excel workbooks, CSV files, or existing single-segment Master Files in your repository. The Samples Generator will populate your directory with sample charts, reports, and dashboards. Additionally, the suite will contain charts and reports that provide quick and easy navigation through the levels of each available hierarchy. You can view, edit, or delete this content, or use it as a starting point to create new dashboards.

The Samples Generator feature is useful to the novice user, because it introduces all major analytic content types in the InfoAssist+ toolset. Depending on the structure of your source worksheet or synonym, the Samples Generator automatically creates the following items:

☐ Active dashboards:

- ☐ Initial Dashboard
- ☐ Initial Dashboard by Year

Note: This item is only generated if your source dataset includes an eligible date, which contains data for minimum three unique months.

☐ Reports:

- ☐ Auto-drill report starting at the top of each identified hierarchy
- ☐ Overview accordion report
- ☐ Overview active report

☐ Charts:

- ☐ Pie chart
- ☐ Bar chart
- ☐ Line chart

Note: Line charts are only created if your source dataset includes an eligible date, which contains data for minimum three unique months.

The dashboards, auto-drill reports, overview accordion report, and overview active report are stored in the Analytics folder. All other items are placed in the folders that are named after the measures for which they are created. The content may vary depending on the structure and size of your synonym.

To see the most complete suite of analytical content, it is recommended that you use a data source that contains at least four measures, six dimensions, and data for a minimum of three unique months. Additionally, a record count called Trans appears in the sample content, if there are not sufficient measures available for analysis.

The following image shows an example of a synonym and the suite of sample content generated from this synonym and displayed in the Resources tree.

Measure Groups			
Retail_sales			
Revenue	D12.2C	3,487.45	13,371.56
Cost of Goods	I9	2288	9459
Gross Profit	D12.2C	1,199.45	3,912.56
Quantity Sold	I7	16	41
Discount	D11.2C	342.45	1,307.24
Dimensions			
Retail_sales			
Geography			
Country	A16V	United States	United States
State	A27V	Iowa	Ohio
City	A19V	Des Moines	Dayton
Sale Date, Simple			
Sale Date, Year	I4	2013	2013
Sale Date, Quarter	I1	3	3
Sale Date, Month	I2	9	9
Sale Date, Day	I2	5	1
Sale Date, Compound			
Sale Date, Y	YYMDy	2013	2013
Sale Date, Y-Q	YYMDq	2013 Q3	2013 Q3
Sale Date, Y-M	YYMDm	2013/09	2013/09
Sale Date, M-D-Y	MDYY	09/05/2013	09/01/2013
Store			
Store Type	A13V	Store Front	Store Front
Store Region	A15V	North America	North America
Store Name	A19V	Des Moines	Dayton
Product			
Product Category	A19V	Video Production	Video Production
Product Subcategory	A30V	Video Editing	Video Editing
Sale Date	MDYY	09/05/2013	09/01/2013

Resources	
Filter	
Domains	
Public	
retail_sales	
Analytics	
Country Auto Drill Report	
Initial Dashboard	
Initial Dashboard by Year	
Overview Accordion Report	
Overview Active Report	
Overview Auto Drill Report	
Product Category Auto Drill Report	
Sale Date, Year Auto Drill Report	
Cost of Goods Reports	
City Cost of Goods Top 10 Pie Chart	
Country Cost of Goods Top 10 Pie Chart	
Product Category Cost of Goods Top 10 Pie Chart	
Sale Date, Quarter Cost of Goods Bar Chart	
Sale Date, Year Cost of Goods Bar Chart	
Sale Date, Year Cost of Goods Line Chart	
State Cost of Goods Bar Chart	
Gross Profit Reports	
Quantity Sold Reports	
Revenue Reports	

Note: If you generate sample content more than once from the same source worksheet or synonym in the same location, the Samples Generator creates a new folder with an underscore and number appended to the title. For example, if a retail_sample folder already exists in a domain or folder, a new folder with the new set of content is created and titled retail_sample_1. The number is incremented each time you repeat this action.

You can generate sample content from one of the following ways:

- ☐ By navigating to the Sample Content option from the New option in the Resources tree for access to existing data.
- ☐ By uploading an Excel spreadsheet or CSV file and selecting the Gain Insight option at the end of the upload procedure.

Procedure: How to Generate Sample Content From the Resources Tree

1. In the Resources tree, right-click a domain or folder, point to *New*, and then click *Sample Content*.

The Open dialog box opens.

Note: Only single-segment Master Files are displayed for selection.

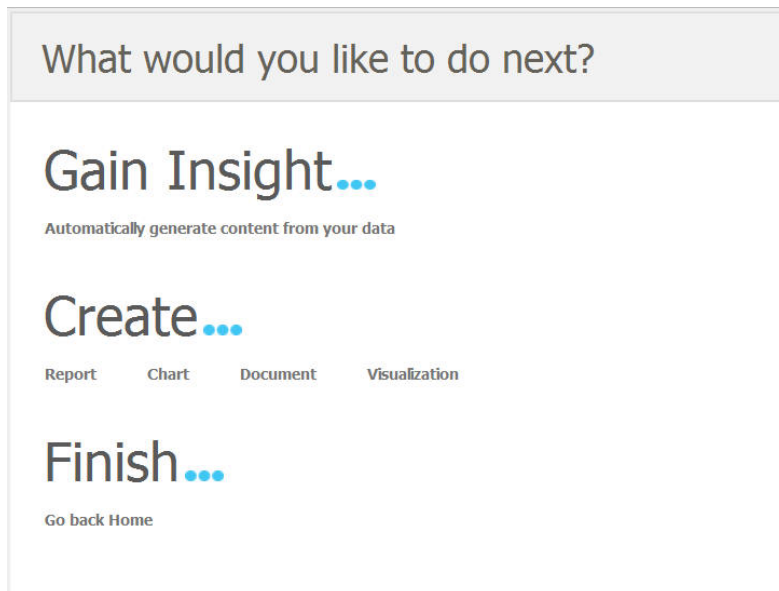
2. Locate a single-segment Master File that you want to use for generating sample content and click *Open*.

The new folder with the same name as your Master File appears in your chosen directory. It contains sample charts, reports, and dashboards.

Procedure: How to Generate Sample Content From the Upload Wizard

1. Upload a spreadsheet.

After the Upload procedure is complete, a selection screen opens, as shown in the following image.



2. Click *Automatically generate content from your data*.

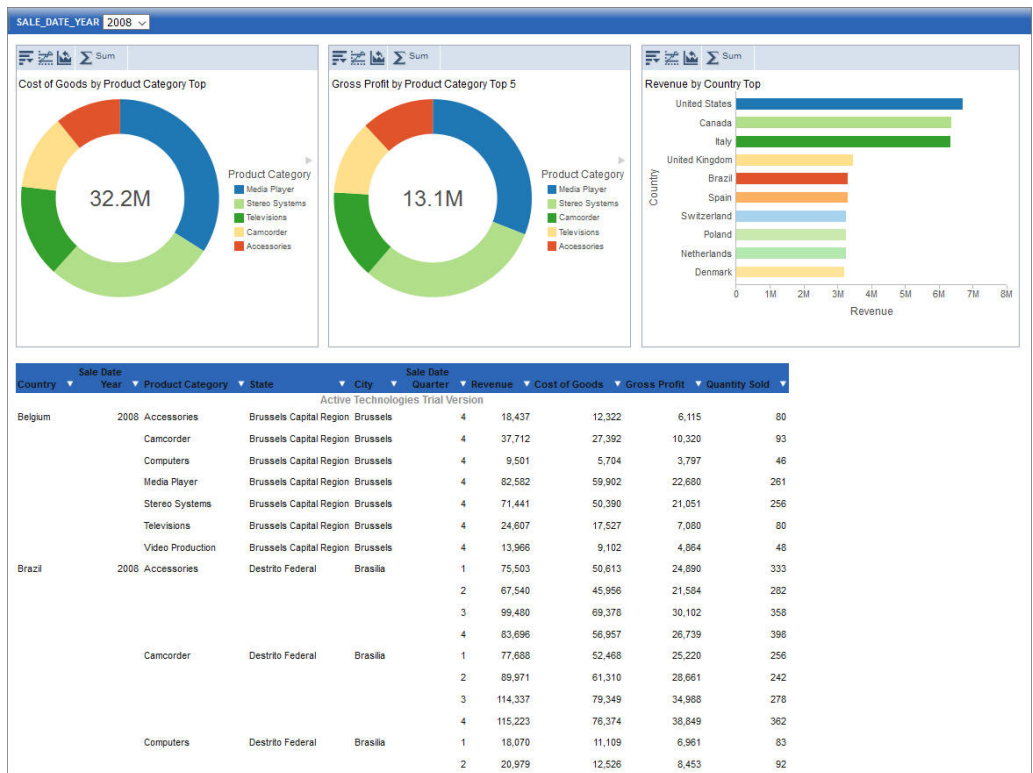
The sample content is generated in the same folder that you selected for your upload.

Analyzing Sample Content

The following section provides a closer look at each content type, and explains how this content is generated. These examples have been generated using the retail_data_extract.xlsx spreadsheet that is provided with your WebFOCUS installation.

Initial Dashboard by Year

The Initial Dashboard gives you a quick overview of your data. It is comprised of three charts and one report, as shown in the following image. Additionally, it displays a drop-down menu, which allows you to narrow down your results to a specific year. The Initial Dashboard by Year is only generated if your source worksheet or synonym includes an eligible date, which contains data for a minimum of three unique months.



The bar chart is generated from the first measure and first dimension in the source spreadsheet or synonym. It shows the Top 10 values.

The first pie chart is generated from the second measure and second dimension. It shows the Top 5 values.

The second pie chart is generated from the third measure and third dimension. It shows the Top 5 values.

The active report shows an overview of all measures and dimensions in the source worksheet or synonym.

Initial Dashboard

The Initial Dashboard is identical to the Initial Dashboard by Year, with the exception that the year drop-down menu is not displayed.

Auto-drill reports

The auto-drill reports are generated for each of the identified hierarchies in the synonym. For example, if your data set contains three hierarchies, the Samples Generator automatically creates three auto-drill reports and one overview auto-drill report. In each auto-drill report, the first four measures in the data set will be summed up by the dimension values in the hierarchy. The report also features data bars and grand totals for each dimension in the hierarchy. The name of the hierarchy is reflected in the title of the report. For example: Dimension_Auto_Drill_Report.fex where Dimension is the top level of the hierarchy. The following image shows an example of an Overview Auto-Drill Report.

Country	Sale Date Year	Product Category	Revenue	Cost of Goods	Gross Profit	Quantity Sold
Australia	2012	Accessories	3,967	2,653	1,314	15
		Camcorder	4,057	2,503	1,554	16
		Computers	4,315	2,917	1,398	17
		Media Player	11,656	9,053	2,603	35
		Stereo Systems	13,152	9,431	3,721	51
		Video Production	1,698	1,205	493	7
	2013	Accessories	149,408	102,379	47,029	577
		Camcorder	151,166	96,371	54,795	538
		Computers	164,166	114,529	49,637	509
		Media Player	293,022	225,836	67,186	949
		Stereo Systems	323,902	229,324	94,578	1,214
		Televisions	81,027	61,989	19,038	94
		Video Production	58,771	40,559	18,212	220

Click one of the links to drill-down to the lower level, as shown in the following image. You can use the breadcrumbs to navigate back to the original report.

Home → Australia

State	Sale Date Year	Product Category	Revenue	Cost of Goods	Gross Profit	Quantity Sold
New South Wales	2012	Accessories	3,967	2,653	1,314	15
		Camcorder	4,057	2,503	1,554	16
		Computers	4,315	2,917	1,398	17
		Media Player	11,656	9,053	2,603	35
		Stereo Systems	13,152	9,431	3,721	51
		Video Production	1,698	1,205	493	7
	2013	Accessories	149,408	102,379	47,029	577
		Camcorder	151,166	96,371	54,795	538
		Computers	164,166	114,529	49,637	509
		Media Player	293,022	225,836	67,186	949
		Stereo Systems	323,902	229,324	94,578	1,214
		Televisions	81,027	61,989	19,038	94
		Video Production	58,771	40,559	18,212	220
TOTAL			1,260,307	898,749	361,558	4,242

Overview Accordion Report

The Overview Accordion Report provides a way to control the amount of sorted data that appears on a page with the expandable views of data for each vertical sort field. It contains up to three dimensions and the first four measures that are found in your source synonym. The report displays total values for each measure. The following image shows an example of an Overview Accordion Report.

	Revenue	Cost of Goods	Gross Profit	Quantity Sold
[-] Australia	1,260,307	898,749	361,558	4,242
[-] 2012	38,845	27,762	11,083	141
Accessories	3,967	2,653	1,314	15
Camcorder	4,057	2,503	1,554	16
Computers	4,315	2,917	1,398	17
Media Player	11,656	9,053	2,603	35
Stereo Systems	13,152	9,431	3,721	51
Video Production	1,698	1,205	493	7
[+] 2013	1,221,462	870,987	350,475	4,101
[+] Belgium	20,148,423	14,449,034	5,699,389	66,721
[+] Brazil	25,974,012	18,615,007	7,359,005	86,606
[+] Canada	51,147,788	36,606,113	14,541,675	171,492

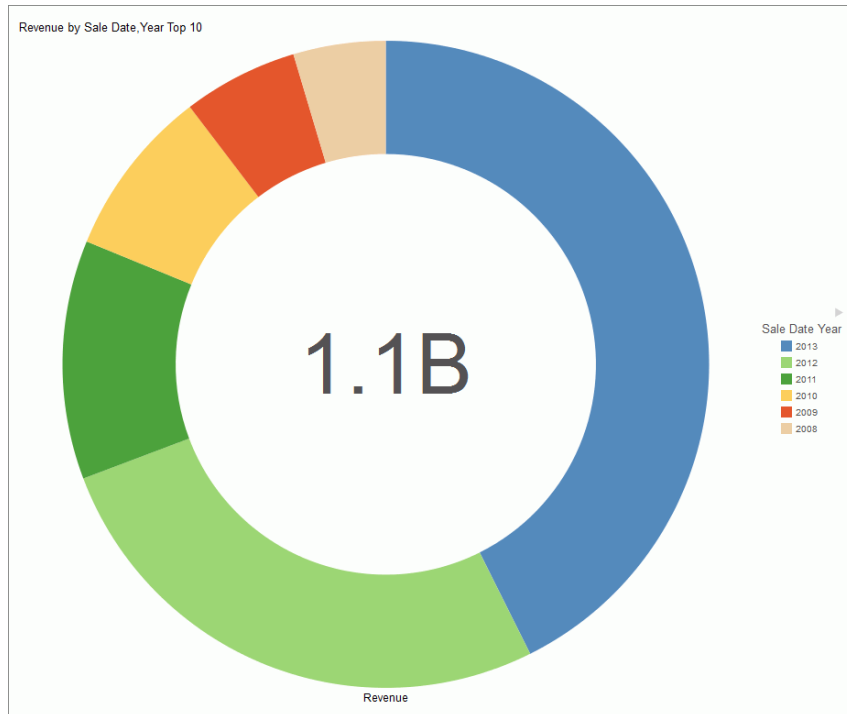
Overview Active Report

The Overview Active Report contains one BY field for each dimension. The report displays maximum of six dimensions and four measures. The total values are displayed for each measure. Each field features a drop-down menu, where you can access active report options. The Overview Active Report is always generated and resides in the Analytics folder. The following image shows an example of an Overview Active Report.

Country	Sale Date	Product Category	State	City	Sale Date	Revenue	Cost of Goods	Gross Profit	Quantity Sold
	Year				Quarter				
	Sort Ascending								
	Sort Descending								
Australia	Filter	ssories	New South Wales	Sydney	4	3,967	2,653	1,314	15
	Calculate	corder	New South Wales	Sydney	4	4,057	2,503	1,554	16
	Chart	puters	New South Wales	Sydney	4	4,315	2,917	1,398	17
	Rollup	a Player	New South Wales	Sydney	4	11,656	9,053	2,603	35
	Pivot (Cross Tab)	o Systems	New South Wales	Sydney	4	13,152	9,431	3,721	51
	Hide Column	y Production	New South Wales	Sydney	4	1,698	1,205	493	7
	Grid Tool	ssories	New South Wales	Sydney	1	47,881	32,973	14,908	171
	Chart/Rollup Tool				2	30,560	21,101	9,459	113
	Pivot Tool				3	29,284	19,379	9,905	135
	Show Records				4	41,682	28,926	12,756	158
	Comments				1	38,320	23,789	14,531	145
	Export	corder	New South Wales	Sydney	2	34,841	22,611	12,230	116
	Print				3	33,260	20,795	12,465	129
	Window				4	44,744	29,176	15,568	148
	Restore Original	Computers	New South Wales	Sydney	1	35,542	25,210	10,332	104

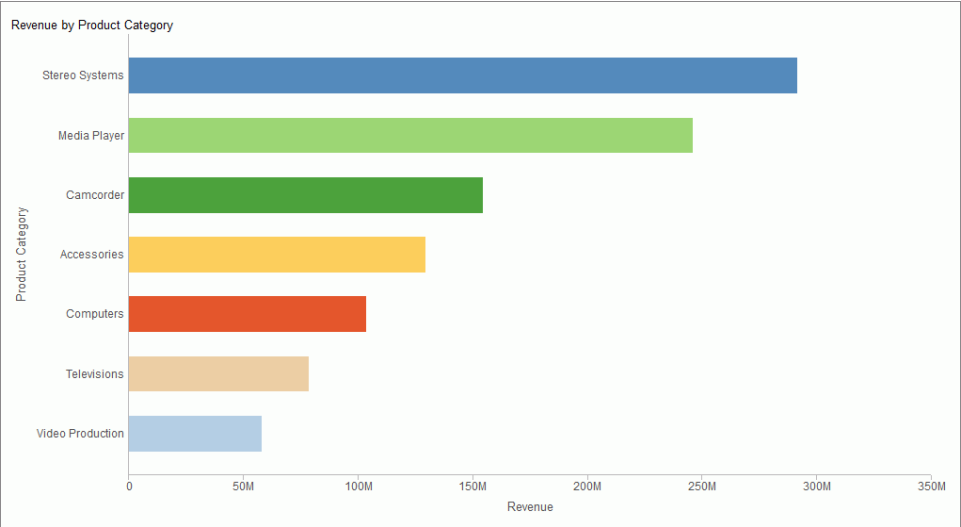
Pie Charts

The Samples Generator uses the first four measures to create Pie charts, by pairing these measures with up to three different dimensions found in the source synonym. Each pie chart shows the Top 10 values. The number of pie charts may vary depending on your synonym structure. The following image shows an example of a pie chart created by the Samples Generator.



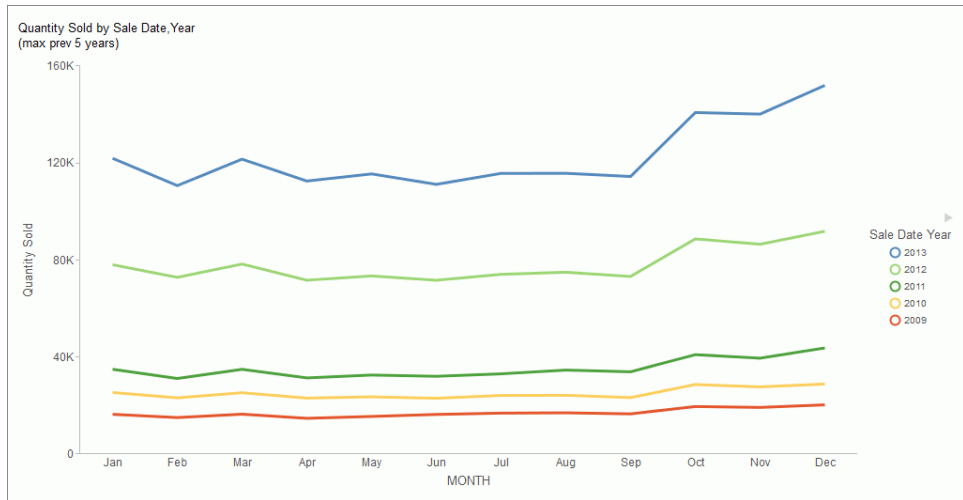
Bar Charts

The Sample Generator creates bar charts for each measure paired with up to three different dimensions. Each bar chart shows all values in the dataset for that dimension. If there are more values than can be displayed on the screen, a scrollbar appears. The number of bar charts may vary depending on the structure of your synonym. The following image shows an example of an automatically generated bar chart.



Line Charts

Line charts are only generated, if your source dataset includes an eligible date, which contains data for minimum three unique months. If this data is present in the dataset, a line chart is created for each measure. The following image shows an example of a line chart.



Creating Reports

You can use the following procedures to create a basic report.

Procedure: How to Create a Report From the BUE Portal

After you have signed in to the WebFOCUS Business User Edition, you can work with an existing folder, or create a new folder in the Resources tree to store your reports.

1. Right-click the folder that you want to use, point to *New*, and then click *Report*.

Or

Select a folder and on the Home page, on the Quick Links tab, under *Create*, click *Report*.

InfoAssist+ opens.

2. From the Open dialog box, select the data source that you want to use, and click *OK*.
The data source that you selected appears in the Data pane.
3. Drag fields onto the canvas or into the Query pane to begin building your report.

Procedure: How to Create a Report From the Application Main Menu

1. In the upper-left corner of the InfoAssist+ interface, click the *IA* button to open the Application Main Menu.
2. From the Application Main Menu, click *New*.
The InfoAssist+ splash screen opens.
3. On the InfoAssist+ splash screen, select *Build a Report*.
The Open dialog box opens.
4. From the Open dialog box, select the data source that you want to use, and click *Open*.
The data source that you selected appears in the Data pane of the Resources panel.
5. Drag fields onto the canvas or into the Query pane to begin building your report.

Procedure: How to Create a Report From the Quick Access Toolbar

1. On the Quick Access Toolbar, click the *New* icon.
The InfoAssist+ splash screen opens.
2. On the InfoAssist+ splash screen, select *Build a Report*.
The Open dialog box opens.
3. From the Open dialog box, select the data source that you want to use, and click *Open*.
The data source that you selected appears in the Data pane of the Resources panel.
4. Drag fields onto the canvas or into the Query pane to begin building your report.

Procedure: How to Create a Report From an Existing Chart

1. Open the chart that contains the data that you want to present in a report.
2. On the *Home* tab, in the *Format* group, click *Report*.
The data is presented as a report.

Choosing a Report Output

The following output types are available for reports:

- ☐ HTML
- ☐ active report
- ☐ PDF

- ☐ Excel
- ☐ PowerPoint

Note: When you create a report in Document view, you have access to Excel only.

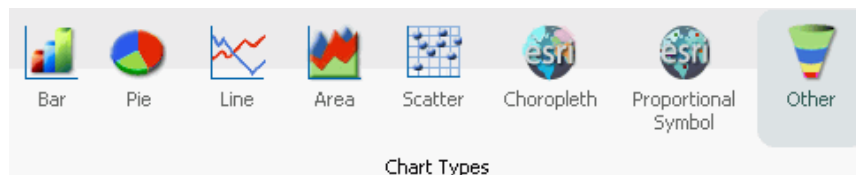
When you create a report in Live Preview or Query Design view, you have access to the following Excel output types:

- ☐ **Excel (xlsx).** Outputs the report in Excel 2007 (and higher) format.
- ☐ **Excel.** Outputs the report in Excel format.
- ☐ **Excel Formula (xlsx).** Outputs the report, using Excel formulas that calculate and display the results of any type of summed information, such as column totals, row totals, and subtotals. This format is for Excel 2007 (and higher).
- ☐ **Excel Formula.** Outputs the report, using native Excel formulas for totals and computed values.

Creating Charts

On the *Format* tab, the *Chart Types* group provides buttons for each of the five most commonly-used chart types. These include Bar (default), Pie, Line, Area, and Scatter. You also have access to Esri Choropleth and Proportional Symbol maps. A button labeled Other gives you access to the complete chart library of advanced charts.

The Chart Types group is shown in the following image.



When switching between chart formats that use a different syntax, you are prompted with a message that allows you to proceed with the change, or cancel your request. This message is shown in the following image.



On the Chart Attributes Modified dialog box, you can click Yes to accept the changes or click No to save the changes to a different file and preserve the originating procedure (.fex).

Procedure: How to Create a Basic Chart

You can run this procedure in Query Design view or Live Preview.

1. On the *Format* tab, in the *Chart Types* group, click the button of the chart that you want to create. Bar chart is the default.

The chart appears on the canvas.

2. Populate the chart with your data in one of the following ways:
 - ☐ Drag the dimension fields and measure fields onto the chart.
 - ☐ Drag the dimension fields and measure fields into the appropriate Query field containers in the Query pane.

Procedure: How to Create an Advanced Chart

You can run this procedure in Query Design view or Live Preview.

1. On the *Format* tab, in the *Chart Types* group, click *Other*.

The Select a chart dialog box opens. The chart types, depicted by icons, display on the left side of the dialog box.

2. To display the name of a chart type, hover over the chart type with the mouse.

From top to bottom, the chart type categories are Bar, Line, Area, Pie, XY Plots, 3D, Stock, Special, HTML5, and Map.

Note: For streamgraphs, which are in the HTML5 category, the tooltip has been enhanced to display specific information, by data point, depending on the underlying data source.

3. Click a chart type.

All supported variations of the chart type appear as thumbnail images in the dialog box.

4. Click an image to display a detailed description of that chart type.

If you are not familiar with a chart type, be sure to read the description carefully before finalizing your selection. Some chart types require a certain number of data values, or a certain type of data values. If your data does not satisfy the requirements, the chart will not accurately represent the data.

You can also hover over an image with your mouse to display the chart type name.

5. In the Select a chart dialog box, click *OK* to finalize your selection and close the dialog box.
6. Populate the chart with your data in one of the following ways:
 - ☐ Drag the dimension fields and measure fields onto the chart.
 - ☐ Drag the dimension fields and measure fields into the appropriate Query field containers in the Query pane.

Procedure: How to Create a Combination Chart

You can run this procedure in Query Design view or Live Preview.

1. On the *Format* tab, in the *Chart Types* group, click the button of the chart that you want to create. Bar chart is the default.

The chart appears on the canvas.

2. Populate the chart with your data in one of the following ways:
 - ☐ Drag the dimension fields and measure fields onto the chart.
 - ☐ Drag the dimension fields and measure fields into the appropriate Query field containers in the Query pane.
3. Change a series type in one of the following ways:
 - ☐ **Ribbon:** On the *Series* tab, in the *Select* group, select the series that you want to display in a different chart type. Then, in the *Properties* group, from the *Type* drop-down menu, select the chart type.
 - ☐ **Shortcut Menu:** Right-click the series that you want to display in a different chart type, point to *Series Type*, and click the chart type.

The series appears in the new chart type.

Procedure: How to Create a Dual-Axis Chart

When you create a dual-axis chart, you assign one data series to the Y1 axis and another data series to the Y2 axis.

1. Create a chart.
2. On the *Format* tab, in the *Chart Types* group, click *Other*.
The Select a chart dialog box opens.
3. Select a dual-axis chart, such as dual-axis bar, and then click *OK*.
4. Drag one field onto the Y1 field, and then drag another field onto the Y2 field.

Procedure: How to Create a Multi-Axis Chart

When you create a multi-axis chart, you assign one data series to the Y1 axis and another data series to the Y2, Y3, Y4, and Y5 axes (as needed).

1. Create a chart.
2. On the *Format* tab, in the *Chart Types* group, click *Other*.
The Select a chart dialog box opens.
3. Select a multi-axis chart, and then click *OK*.
4. Drag fields onto the Y1, Y2, Y3, Y4, and Y5 fields, as needed.

Chart Outputs

You can create charts using one of the following output formats:

- ☐ HTML
- ☐ HTML5 (default)
- ☐ active report
- ☐ PDF
- ☐ Excel
- ☐ PowerPoint

The HTML5 output format allows you to render a chart in the browser using a built-in JavaScript engine. Charts with this output format utilize the very latest capabilities of the HTML5 Web standard, including animation, high-quality vector output, and attractive alpha-channel and gradient effects.

Note: Not every chart type can be output in every format listed here. To make sure that the chart that you are creating can be output in the format that you want, please see the topic for that particular chart type.

Creating Your Own Chart Types

WebFOCUS BUE supports the ability to add new, custom chart types to its list of built-in charts. These custom chart types are called *extensions* or *plug-ins*. An extension is a block of code that accesses resources external to WebFOCUS BUE. This topic describes the structure of an extension and the steps necessary to create your own and add it to the chart library.

Introducing Chart Extensions

Chart extensions are written in JavaScript. The visual part of a visualization can be drawn with HTML, Canvas, or SVG. Extensions can include external CSS and JS libraries (such as d3), which can be used to build almost any visualization. The WebFOCUS Extension API is limited to new, complete chart types only. It is not possible to add features to existing chart types, and it is not possible to modify or extend parts of WebFOCUS BUE outside of the chart area allocated to your extension.

This topic summarizes the process of writing, configuring, and installing a chart extension. Detailed instructions can be found on the Information Builders GitHub site:

<https://github.com/ibi/wf-extensions-chart>

WebFOCUS BUE extensions must be placed in the extensions folder under the web_resource folder of your WebFOCUS BUE installation. By default, this is the following location:

```
c:\ibi\install_dir\config\web_resource\extensions
```

where:

```
install_dir
```

Is your WebFOCUS BUE installation directory.

Several sample chart extensions have already been installed in the extensions folder so that you can see their code, their structure, and how they are accessed in the WebFOCUS BUE tools.

Note: The user installing the extension must know how to write JavaScript code for what the chart extension needs to generate. The GitHub site documents how to make the extension conform to the WebFOCUS API and how to install the extension in the WebFOCUS BUE chart library. It does not describe how to write JavaScript code.

Creating a Chart Extension

This section summarizes the build cycle for creating an extension and the structure and components of an extension.

Reference: Build Cycle for Writing an Extension

Creating an extension often involves cycles of writing, running, and then debugging code.

When you make changes to the `properties.js` file for your extension, you need to clear the WebFOCUS BUE cache in order for those changes to be recognized. Clear the cache using the *Clear cache* link in the Administration Console.

If you change the `.js` code for your extension (for example, `com.ibi.simple_bar.js`), you do not need to make any changes to WebFOCUS BUE. You only need to clear your own browser cache, to ensure that the new JavaScript file is downloaded. The same is true if you change any additional `.js` files included by your extension.

Reference: Extension Structure

The Simple Bar extension example demonstrates the required and optional files in an extension, and how those files are typically laid out.

You can open `com.ibi.simple_bar` and `com.ibi.simple_bar.js` in a text editor to see exactly how an extension is written.

The extension ID (`ext_id`) is a string in the form `com.your_company.extension_name`. The `ext_id` must be all lowercase, and can include only letters, numbers, underscores and dots. The entire extension lives in a folder named `ext_id`. The core of the extension lives in a file named `ext_id.js`. This file includes code to render the extension as a new chart type within WebFOCUS BUE.

The `properties.json` file configures your extension to run in WebFOCUS BUE. This file includes all the metadata needed to include your extension in the WebFOCUS BUE user interface, as well as a list of all properties you wish to expose to end users, so they can customize the behavior of your extension.

The extension folder can also include optional additional folders for external css and lib resources. If your extension uses any additional CSS or JavaScript library files, you can keep those resources organized in dedicated folders, such as css and lib, as you choose. External resources are configured and loaded inside the base `ext_id.js` file of your extension.

Using the Chart Extension API

To see examples of everything that the chart extension API provides, look at *com.ibi.simple_bar.js*. It is divided into two main parts, chart rendering and extension configuration.

Rendering Charts

The extension API provides three entry points that you can use as needed by defining your own JavaScript callback functions. They are passed a set of properties in a config object. Some properties are available during the entire rendering process, and some are only available during render callback.

Reference: Chart Rendering Callback Functions

You can define the following three JavaScript callback functions. Only the `renderCallback` function is always required.

- ❑ **initCallback(successCallback, config)** This optional function is invoked by the engine exactly once during library load time, providing a way to implement `document.onload` initialization code. This function is passed a `successCallback`, which you must invoke with *true* if your initialization code succeeded or *false* if was not successful. If you call `successCallback(false)`, no further interaction with your extension will occur, and your extension will render as an empty page.
- ❑ **preRenderCallback(config)** This optional function is invoked each time your extension is to be rendered, as the very first step in the overall rendering process. This is a good place to examine and tweak or override any internal chart properties that will affect the subsequent rendering.
- ❑ **renderCallback(config)** This required function must contain all of the code that will actually draw your chart. The *config* object will contain the properties described in the following sections.

Each of the three entry point callbacks is passed a config object, which contains a set of useful properties.

Example: Sample renderCallback Function

The following sample renderCallback code renders the Simple Bar extension.

```
function renderCallback(renderConfig) {
  var chart = renderConfig.moonbeamInstance;
  var props = renderConfig.properties;
  var container = d3.select(renderConfig.container)
    .attr('class', 'com_ibi_chart');
  var data = renderConfig.data;
  if (renderConfig.dataBuckets.depth === 1) {
    data = [data];
  }

  var seriesCount = data[0].length;
  var seriesLabels = data[0].map(function(el){return el.labels;});
  data = d3.transpose(data).map(function(el, idx) {
    el = el[0];
    var v = Array.isArray(el.value) ? el.value : [el.value];
    var y0 = 0;
    return v.map(function(d, s) {
      return chart.mergeObjects(d, {y0: y0, y1: y0 += d, seriesID: s,
value: d, labels: seriesLabels[idx]});
    });
  });

  var w = renderConfig.width;
  var h = renderConfig.height;
  var x =
d3.scale.ordinal().domain(pv.range(seriesCount)).rangeRoundBands([0,
w], 0.2);
  var ymax = d3.max([].concat.apply([], data), function(d){return
d.y1;});
  var y = d3.scale.linear().domain([0, ymax]).range([25, h]);
  var svg = container.selectAll("g")
    .data(data)
    .enter().append('g')
    .attr('transform', function(d, i){return 'translate(' + x(i) + ',
0)';});

  svg.selectAll("rect")
    .data(function(d){return d;})
    .enter().append('rect')
    .attr("width", x.rangeBand())
    .attr("y", function(d) {return h - y(d.y1);})
    .attr("height", function(d){return y(d.y1) - y(d.y0);})
    .attr('tdgtitle', function(d, s, g) {
```

```

//To support tooltips, each chart object that should draw a tooltip
//must set its 'tdgttitle' attribute to the tooltip's content string.

    // Retrieve the chart engine's user-defined tooltip content with
    getToolTipContent():
    // 's' and 'g' are the series and group IDs for the riser in
    question.
    // 'd' is this riser's individual datum, and seriesData is the
    array of data for this riser's series.
    var seriesData = chart.data[s];
    var tooltip = renderConfig.modules.tooltip.getToolTipContent(s, g,
    d, seriesData);
    // getToolTipContent() return values:
    // - undefined: do not add any content to this riser's tooltip
    // - the string 'auto': you must define some 'nice' automatic
    tooltip content for this riser
    // - anything else: use this directly as the tooltip content
    if (tooltip === 'auto') {
        if (d.hasOwnProperty('color')) {
            return 'Bar Size: ' + d.value + '<br />Bar Color: ' + d.color;
        }
        return 'Bar Size: ' + d.value;
    }
    return tooltip;
})
.attr('class', function(d, s, g) {
    // To support data selection and tooltips, each riser must include
    a class name with the appropriate seriesID and groupID
    // Use chart.buildClassName to create an appropriate class name.
    // 1st argument must be 'riser', 2nd is seriesID, 3rd is groupID,
    4th is an optional extra string which can be used to identify the
    risers in your extension.
    return chart.buildClassName('riser', s, g, 'bar');
})
.attr('fill', function(d) {
    // getSeriesAndGroupProperty(seriesID, groupID, property) is a
    handy function
    // to easily look up any series dependent property. 'property' can
    be in
    // dot notation (eg: 'marker.border.width').
    return chart.getSeriesAndGroupProperty(d.seriesID, null, 'color');
});

svg.append('text')
    .attr('transform', function(d) {return 'translate(' +
    (x.rangeBand() / 2) + ', ' + (h - 5) + ')';})
    .text(function(d, i){return seriesLabels[i];})

renderConfig.modules.tooltip.updateToolTips(); // Tell the chart
engine your chart is ready for tooltips to be added
renderConfig.modules.dataSelection.activateSelection(); // Tell the
chart engine your chart is ready for data selection to be enabled
}

```

Reference: Properties That Are Always Available

The following properties are always available.

Property Name	Description
moonbeamInstance	The chart instance currently being rendered.
data	The data set being rendered.
properties	The block of properties for your extension, as set by the user.

Reference: Properties Available Only During Render Callback

The following properties are available only during render callback, and are used by your chart rendering code (renderCallback).

Property Name	Description
width	Width of the container your extension renders into, in pixels.
height	Height of the container your extension renders into, in pixels.
containerIDPrefix	The ID of the DOM container your extension renders into. Prepend this to <i>all</i> IDs your extension generates, to ensure multiple copies of your extension work on one page.
container	DOM node for your extension to render into, either an HTML DIV element or an SVG G element, depending on your chosen containerType extension configuration
rootContainer	DOM node containing the specific chart engine instance being rendered.

Configuring Your Chart Extension

Extension configuration consists of two parts.

- ❑ Chart Engine Configuration configures the extension to interact with the chart engine and chart canvas in WebFOCUS BUE. This part of the extension configuration is defined in the config object that is passed to the chart renderer functions.
- ❑ Chart Interface Configuration interacts with the chart type picker in the user interface and the chart attribute categories. This part of the extension configuration is defined in the properties.json file.

Creating a config Object for Chart Engine Configuration

To configure your extension, create a *config* object with all the information unique to your extension, then register your extension with the extension API.

Reference: Creating a config Object for Your Extension

Required and optional properties in your config object are described in the following table.

Property Name	Description
id	Is the extension ID described in Extension Structure on page 350.
name	Is the name for the chart type to be displayed in the user interface.
description	Is a description for the chart type to be displayed in the user interface.
containerType	Is either 'html' or 'svg' (the default).
initCallback	Optional. References your initCallback function, described in Rendering Charts on page 351.
preRenderCallback	Optional. References your preRenderCallback function, described in Rendering Charts on page 351.
renderCallback	Required. References your renderCallback function, described in Rendering Charts on page 351.

Property Name	Description
resources	Optional. Are additional external resources (CSS and JS) required by this extension.

Example: Sample config Object

The following code is a sample of the config object used with the Simple Bar extension.

```
var config = {
  id: 'com.ibi.simple_bar',      // string that uniquely identifies
  this extension
  containerType: 'svg', // either 'html' or 'svg' (default)
  initCallback: initCallback, // Refers to your init callback fn
  (optional)
  preRenderCallback: preRenderCallback, // Refers to your preRender
  callback fn (optional)
  renderCallback: renderCallback, // Refers to your render fn
  (required)
  resources: { // Additional external resources (CSS & JS) required
  by this extension (optional)
    script: ['lib/d3.min.js'],
    css: ['css/extension.css']
  },
}
```

Reference: Registering Your Extension

To register your extension with the WebFOCUS extension API, call:

```
tdgchart.extensionManager.register(config);
```

Reference: Tips for Building Your Extension

The easiest way to build your own extension is to clone the Simple Bar example, then tweak it. Assume the ID of the new extension is com.foo.bar:

1. Rename root folder to com.foo.bar. Rename com.ibi.simple_bar.js to com.foo.bar.js.
2. In com.foo.bar.js, delete the inner content of the three callback functions.
3. In com.foo.bar.js, change the entries for each property in config to match the requirements of your extension.
4. Add any external resources you need to *lib* and *css*, and load them by setting config.resources in com.foo.bar.js.
5. Implement renderCallback in com.foo.bar.js to draw your extension.

Configuring the Chart Interface

Each extension must include a `properties.json` file, which defines the information needed by WebFOCUS BUE when drawing its user interface.

The `properties.json` file consists of the following blocks.

- ❑ **info.** This block defines several general purpose configuration options.
- ❑ **properties.** This block defines any properties of your extension that the end user may want to change. The user can change these properties in the `GRAPH_JS` blocks in a WebFOCUS BUE chart procedure.
- ❑ **propertyAnnotations.** This block validates the content of the properties block. Everything in properties must appear in propertyAnnotations. The possible types of any non-object (leaf) property in properties must be notated as one of "str", "bool", or "number".
- ❑ **dataBuckets.** This block defines the set of chart attribute categories that appear in the Query pane in the WebFOCUS BUE user interface when creating a chart. Each member in the dataBuckets collection is a bucket.

There are two types of buckets, built-in and custom. Built-in buckets provide an easy way to reuse the existing WebFOCUS BUE data bucket logic. There are currently two built-in buckets, `tooltip`, and `series_break`. Use any of these buckets by setting the associated dataBuckets property to `true`.

- ❑ **bucket.** Each bucket block defines one custom chart attribute category. Each custom bucket requires the following properties:
 - ❑ **id.** This property corresponds exactly to the `dataArrayMap` and `data` properties that will be received by the render function for your chart.
 - ❑ **type.** This property defines the type of data field this bucket accepts, "measure", "dimension", or "both".
 - ❑ **count.** Consists of `count.min` and `count.max`, which define the minimum and maximum number of fields this bucket can accept. A minimum of 0 means this bucket is optional.
 - ❑ **translations.** Defines translations in different languages for every label to be drawn in the WebFOCUS BUE interface. The translation object has one property for each language the extension supports, keyed by ISO-639 two letter locale strings.

Example: Sample properties.json File

The following properties.json file is from the Simple Bar extension.

```
{
  // Define some general extension configuration options
  "info": {
    "version": "1.0", // version number of your extension.
    "implements_api_version": "1.0", // version number of the
    WebFocus API used by your extension.
    "author": "Information Builders",
    "copyright": "Information Builders Inc.",
    "url": "https://github.com/ibi/wf-extensions-chart/tree/master/
    simple_bar%20example",
    "icons": {
      "medium": "icons/medium.png" // Reference to an image in
    the extension, used in the WF chart picker
    }
  },
  // Define any properties of your extension that end user may want
  to change.
  "properties": {
    "exampleProperty": 50
  },
  // Define the possible values for each property in 'properties'.
  "propertyAnnotations": {
    "exampleProperty": "number"
  },
  // Define the available data buckets drawn in WF's 'Query' data
  bucket tree.
  "dataBuckets": {
    // Choose whether or not to reuse existing WF data buckets.
    All optional.
    "tooltip": false,
    "series_break": true,
```

```

// Define your own custom data buckets.  Optional
"buckets": [
  {
    "id": "value",
    "type": "measure",
    "count": {"min": 1, "max": 5}
  },
  {
    "id": "labels",
    "type": "dimension",
    "count": {"min": 1, "max": 5}
  }
],
// Define the set of labels used in the WF interface for buckets
and chart type picker.
"translations": {
  "en": {
    "name": "My Simple Bar Chart",
    "description": "This chart is just a simple bar chart,
nothing to see here.",
    "icon_tooltip": "This extension does ...",
    "value_name": "Value Bucket",
    "value_tooltip": "Drop a measure here",
    "labels_name": "Label Bucket",
    "labels_tooltip": "Drop a dimension here"
  },
  "fr": {
    "name": "Un Bar Chart tres simple",
    "description": "C'est un Bar Chart vraiment simple",
    "icon_tooltip": "This extension does ...",
    "value_name": "Value Bucket",
    "value_tooltip": "Drop a measure here",
    "labels_name": "Label Bucket",
    "labels_tooltip": "Drop a dimension here"
  }
}
}

```

Accessing Data for Your Extension

Each time an extension is rendered, the render callback for the extension is passed the current data set using the `renderConfig.data` argument. The overall structure of the data set is defined by the set of buckets listed in the `properties.json` file, while the specific content of the data is defined by the data fields the user has added to each bucket.

Defining and Using Buckets in an Extension

The data set is passed into an extension using the `data` property of the first argument of the `render` callback, typically named `renderConfig`. Additional information about the current set of fields in each bucket is in `renderConfig.dataBuckets`.

A data set is represented in JavaScript as arrays of objects. If an extension defines only custom buckets, the data set will be a flat array of objects. If an extension uses some built-in buckets, the data set may contain deeply nested arrays of arrays. The `renderConfig.dataBuckets.depth` property will be set to the number of array dimensions in the current data set.

Custom Buckets

Each innermost object within the arrays of data (called a *datum*) will have one property for each data bucket that contains a field. Each property will be the id of a custom bucket, as defined in the `dataBuckets.buckets` section of `properties.json`. The type of values of these properties depend on the bucket type. Dimension buckets have string values, while measure buckets have numeric values. If a bucket contains more than one field, the associated property for each innermost object will be an array of string or number values.

Built-in Buckets

An extension can use buckets that are built-in and predefined by WebFOCUS BUE. These buckets will affect more than just the data set. Each bucket will also set specific chart engine properties, to pass in additional information related to that bucket.

Each built in WebFOCUS BUE bucket is either a *standard* bucket or a *break* bucket.

- ❑ Standard buckets behave exactly like custom buckets. The data set remains a single array, and each datum object will include an additional property named after the bucket.
- ❑ Break buckets divide the data set into additional arrays of data. For each break bucket used, each datum object will be transformed into a full array of datum objects. The number of datum objects in each array will remain unchanged, but the number of arrays or datum arrays will correspond to the number of entries in the break field.

Types of Break Buckets

Break buckets can be of two types:

- ❑ A series-break bucket breaks the data set into one array for each entry in the series break field chosen by the user. A series-break bucket uses series-dependent properties defined in the chart engine, and the data names are now listed in those series-dependent properties. Each entry in the series-break field will generate a corresponding series property object in the chart engine, retrievable with `renderConfig.moonbeamInstance.getSeries(x)`, where `x` is an integer for the series to be retrieved. `getSeries` returns an object with properties such as `color` and `label`, which are unique to the chosen series.
- ❑ A matrix-break bucket is used for the sort fields that define the columns and rows in a matrix chart. A matrix-break bucket also adds more array dimensions to the data set. A matrix-break bucket is broken into *column* and *row* sub-buckets. If either the row or column bucket contains any fields, the data set will contain two additional dimensions of data, even if one of the matrix buckets is empty. That is, the data set will either contain neither row nor column data, or both row and column data, never just one or the other. `bucket.depth` will always be at least three.

The Tooltip Bucket

The tooltip bucket is not a break bucket, and does not add any additional array dimensions to the data set. Instead, tooltip behaves like a custom bucket. Each inner datum object will contain a property named *tooltip*, with a value of type *string* for dimensions, *number* for measures, and an array of values for multiple fields in the bucket.

The usefulness of this bucket is that in addition to including tooltip-specific data in the data set, WebFOCUS BUE also generates meaningful tooltip content for each series. This tooltip content is the same content used for all of the built in WebFOCUS BUE chart types. Using the tooltip bucket means the extension does not have to figure out what ought to go into each tooltip.

Example: Sample Series-Break Bucket Definition

This example uses the following sample data.

Car	Country	Seats
BMW	Germany	5
Audi	Germany	4

Car	Country	Seats
Peugeot	France	5
Alfa Romeo	Italy	4
Maserati	Italy	2
Toyota	Japan	4

The following code defines a series-break bucket.

```
dataBuckets:
  series_break: true,
  buckets: [
    {id: "label", type: "dimension"},
    {id: "value", type: "measure"}
  ]
```

Consider the following fields assigned to each of the buckets:

- ☐ "Country" assigned to the "series_break" bucket.
- ☐ "Car" assigned to the "label" bucket.
- ☐ "Seats" assigned to the "value" bucket.

In the renderConfig function, the renderConfig.data object will be similar to the following, in which the Country values are no longer part of the data array. However, a new array starts for each change in the Country value:

```
[{labels: "PEUGEOT", value: 5}],
  [{labels: "ALFA ROMEO", value: 4}, {labels: "MASERATI", value:
2},
  [{labels: "TOYOTA", value: 4}],
  [{labels: "AUDI", value: 4}, {labels: "BMW", value: 5}]
```

The renderConfig.dataBuckets object will be defined as follows:

```
renderConfig.dataBuckets = {
  depth: 2,
  series_break: {title: "Country"},
  buckets: {
    label: {title: "Car"},
    value: {title: "Seats"}
  }
}
```

Handling Partial and Null Data in an Extension

In many cases, the end user working with an extension cannot populate all of the extension buckets immediately. An extension must correctly handle these partial data cases, and cannot crash if one or more buckets are empty. It is important to check `renderConfig.dataBuckets` to see which buckets have been populated, and act accordingly.

In addition, data sets are often incomplete, missing some values for a given combination of dimensions and measures. These missing values may show up in the data set as null entries within an array (instead of datum objects), or they may show up as entirely empty arrays. It is important to detect and handle these missing data cases, and render a visualization appropriate for such missing data.

Most extensions require some minimum number of populated buckets before anything can be rendered. Use the `count.min` properties of each `dataBuckets.bucket` entry in `properties.json` to define these minimum requirements. If the fields in all buckets do not meet the minimum counts, then the `renderCallback` for the extension will not be called. Instead, the `noDataPreRenderCallback` for the extension is called. This allows the extension to render in a special *no data* mode. In this mode, the extension should render in grey scale, using `renderCallback.baseColor` as the main color. This should be a very simplified, sample rendering of the extension.

Example: Sample `noDataPreRenderCallback` Function

The following `noDataPreRenderCallback` function is from the Simple Bar sample extension.

```
function noDataRenderCallback(renderConfig) {
  var grey = renderConfig.baseColor;
  renderConfig.data = [{value: [3, 3]}, {value: [4, 4]}, {value: [5,
5]}, {value: [6, 6]}, {value: [7, 7]}};
  renderConfig.moonbeamInstance.getSeries(0).color = grey;
  renderConfig.moonbeamInstance.getSeries(1).color =
pv.color(grey).lighter(0.18).color;
  renderCallback(renderConfig);
}
```

Installing a Chart Extension

1. Find the extensions folder for your local WebFOCUS BUE installation. This is typically the following folder.

```
C:\ibi\install_dir\config\web_resource\extensions
```

where:

```
install_dir
```

Is your WebFOCUS BUE installation directory.

Note: The WebFOCUS Extension section of the Information Builders GitHub page maintains a list of publicly available and supported extensions. To install one of those, click the extension you want to install, then right click the zip file for that extension, for example *com.ibi.xyz.zip*, and choose *Save link as...*

2. Unzip the downloaded zip file into the WebFOCUS BUE extensions folder. For example, for the *com.ibi.xyz.zip* zip file, this should create the following folder.

```
C:\ibi\install_dir\config\web_resource\extensions\com.ibi.xyz
```

If you are installing your own extension from your own environment, copy or download it to the WebFOCUS BUE extensions folder, using the same naming conventions for the folder and the extension ID as described for the sample extensions.

3. Edit *C:\ibi\install_dir\config\web_resource\extensions\html5chart_extensions.json*. Create a new line for the new extension in the form:

```
"com.ibi.abc": {"enabled": true},
```

where:

abc

Is the name of the extension.

4. In the Administration Console, click *Clear cache*. This will force WebFOCUS to reload all extensions.

Following is a sample *html5chart_extensions.json*.

```
{
  "com.ibi.simple_bar": {enabled: true},
  "com.ibi.liquid_gauge": {enabled: false},
  "com.ibi.sankey": {enabled: true}
}
```

Reference: Preserving Custom Chart Types When Reinstalling the WebFOCUS Client

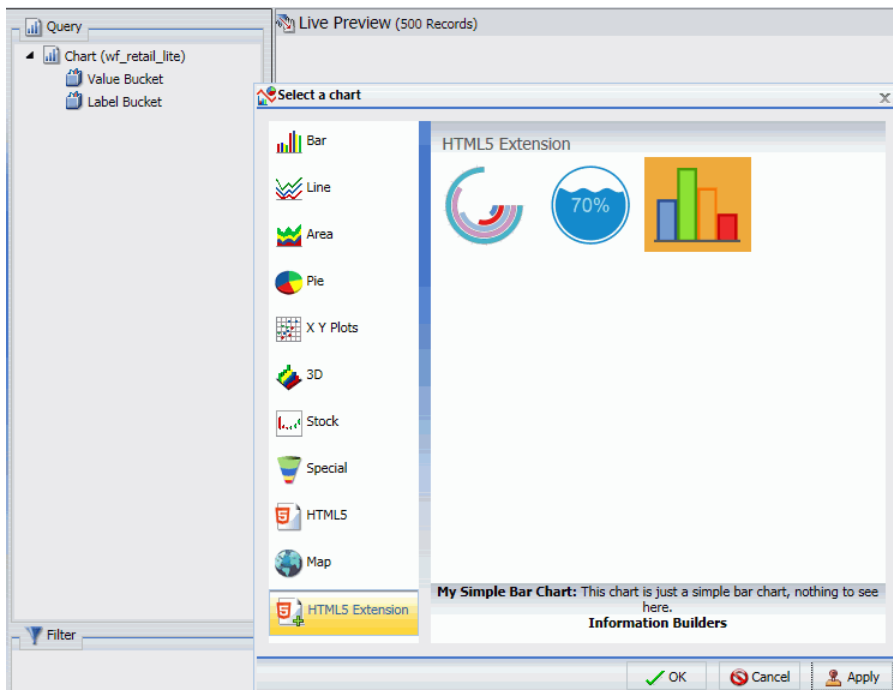
If you reinstall the WebFOCUS Client, your extensions folder will be overwritten. Therefore, if you have installed any custom chart extensions, you should preserve them by copying them to another location prior to reinstalling the WebFOCUS Client and copying them back to the extensions folder after reinstalling the WebFOCUS Client.

You will also have to copy the entries for your custom extensions into the new *html5chart_extensions.json* file installed with the new version of the WebFOCUS Client.

Note: The extensions that are delivered as part of WebFOCUS BUE will be reinstalled automatically, so you should not preserve those extensions. In that way, if any enhancements have been made to those extensions, you will automatically have access to the enhanced versions when you reinstall the WebFOCUS Client.

Using Your Extension in a WebFOCUS Request

If you have installed and configured your extension as described, your extension will be available for use in the WebFOCUS BUE tools as a chart type in the *Other* format category under *HTML5 Extension*, as shown in the following image.



The attribute categories you defined in the dataBuckets object of your extension are available in the query pane.

In the FOCEXEC:

- ☐ The LOOKGRAPH value is EXTENSION.
- ☐ The actual extension to use is identified in the chartType property of the *GRAPH_JS block in the StyleSheet. For example:

```
*GRAPH_JS
chartType: "com.ibi.simple_bar",
}
```

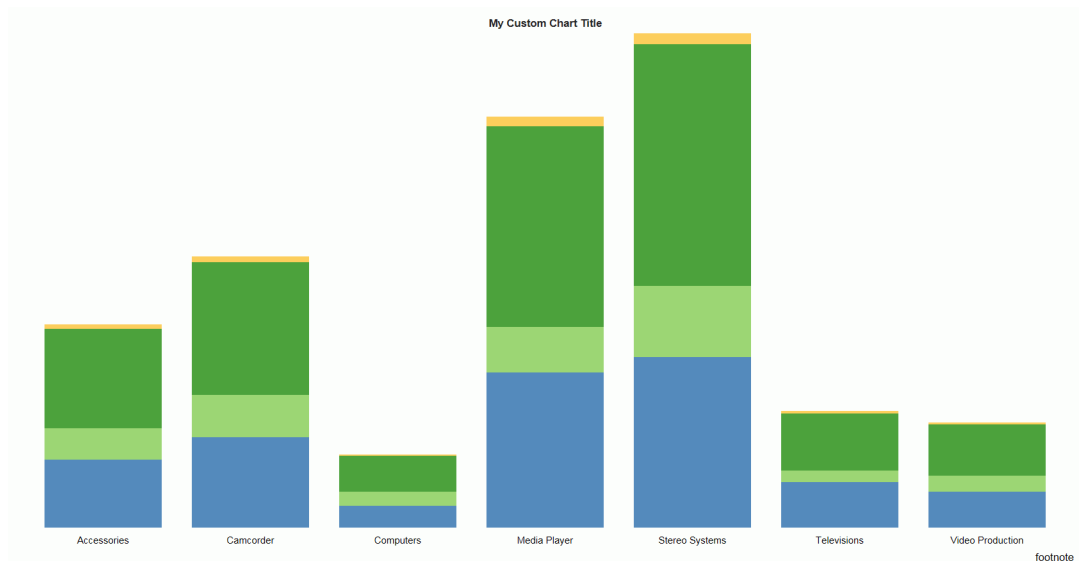
- ❑ Each custom attribute category name is prepended with a greater-than character (>). For example:

```
TYPE=DATA, COLUMN=N1, BUCKET= >labels, $
TYPE=DATA, COLUMN=N2, BUCKET= >value, $
TYPE=DATA, COLUMN=N3, BUCKET= >value, $
TYPE=DATA, COLUMN=N4, BUCKET= >value, $
TYPE=DATA, COLUMN=N5, BUCKET= >value, $
```

The following is a sample request using the Simple Bar extension.

```
GRAPH FILE WF_RETAIL_LITE
SUM COGS_US
GROSS_PROFIT_US
REVENUE_US
DISCOUNT_US
BY PRODUCT_CATEGORY
ON GRAPH PCHOLD FORMAT JSCHART
ON GRAPH SET LOOKGRAPH EXTENSION
ON GRAPH SET AUTOFIT ON
ON GRAPH SET STYLE *
INCLUDE=IBFS:/FILE/IBI_HTML_DIR/javaassist/intl/EN/combine_templates/
ENWarm.sty,$
TYPE=DATA, COLUMN=PRODUCT_CATEGORY, BUCKET= >labels, $
TYPE=DATA, COLUMN=COGS_US, BUCKET= >value, $
TYPE=DATA, COLUMN=GROSS_PROFIT_US, BUCKET= >value, $
TYPE=DATA, COLUMN=REVENUE_US, BUCKET= >value, $
TYPE=DATA, COLUMN=DISCOUNT_US, BUCKET= >value, $
*GRAPH_JS
chartType: "com.ibi.simple_bar",
*END
ENDSTYLE
END
```

Run the chart. The output is shown in the following image.



Building a Document

Document view allows you to build multiple reports and charts on the same canvas. The styling, design, and report building functionality of Live Preview and Query Design view is available in Document view.

In addition, there are many other features that simplify building documents. You can build and insert multiple reports in the form of reports and charts into documents. You can also insert images and text for presentation and organizational purposes.

Inserting Reports From Multiple Data Sources

With InfoAssist+ opened in Document view, you can insert multiple charts and reports onto the canvas. These reports can be from different data sources. With documents, you have the option to add additional data sources to the document.

In order to insert reports from different data sources, the document must have multiple data sources loaded. For more information on adding and switching between data sources, see [Data Tab](#) on page 78.

Procedure: How to Insert Two Reports From Two Different Data Sources

A document can display multiple reports from multiple data sources in the same document.

1. With InfoAssist+ open in Document view, click the Insert tab and select chart or report.

2. If your document has only one data source, insert additional data sources.
For more detailed instructions on inserting multiple data sources, see [Data Tab](#) on page 78.
3. Switch to a data source different than the one used in step 1.
For more detailed instructions on switching to a different data source, see [Data Tab](#) on page 78.
4. Insert a chart or report using this new data source, following the instructions given in [Inserting a New Report](#) on page 368.
Your document is now populated with reports that have data from different data sources. You can add as many data sources as you need.

Inserting a New Report

With InfoAssist+ opened in Document view, you can insert multiple charts and reports onto the canvas. The procedures in the following sections detail how to insert reports into documents.

In Document view, you can insert a report in the following ways.

- ☐ Use the Insert tab.
- ☐ Double-click a data source field.
- ☐ Right-click a data source field.
- ☐ Drag a data source field onto the canvas.

Note:

- ☐ When you use the Insert tab, double-click a data source field, or right-click a data source field, a report placeholder is added to the canvas.
- ☐ When you insert an existing report, which has already been created and is referenced via-`INCLUDE` syntax, and then select it on the Document canvas, the data fields do not display in the Query pane.

Dragging a data source field onto the canvas inserts the place holder at the location you dropped it.

The following procedures describe how to insert new reports. For more information on how to edit existing reports, see [How to Style and Customize a Report](#) on page 376.

Procedure: How to Insert a Report

Do one of the following, while in Document mode:

- ☐ On the *Insert* tab, in the *Reports* group, click *Report*. Add fields to the placeholder report.
- ☐ On the *Home* tab, in the *Format* group, click *Report*. Double-click a data source to automatically create a report with that data.
- ☐ On the *Home* tab, in the *Format* group, click *Report*. Drag a field to the canvas to create a report.

Inserting a New Chart

With InfoAssist+ opened in Document view, you can bring multiple charts and reports onto the canvas. The procedures in the following sections describe how to insert charts into documents.

In Document view, you can insert a chart in the following ways:

- ☐ Use the Insert tab.
- ☐ Double-click a data source field.
- ☐ Right-click a data source field.
- ☐ Drag a data source field from the Resources panel onto the canvas.

Note: When you use the Insert tab, double-click a data source field, or right-click a data source field, a chart placeholder is added to the canvas.

Dragging a data source field onto the canvas inserts the place holder at the location you dropped it.

The following procedures describe how to insert new charts. For more information on how to edit existing charts, see [How to Style and Customize a Chart](#) on page 376.

Procedure: How to Insert a Chart

1. Do one of the following, while in Document mode:

- ☐ On the *Insert* tab, in the *Reports* group, click *Chart*. Add fields to the placeholder chart.
- ☐ On the *Home* tab, in the *Format* group, click *Chart*. Double-click a data source to automatically create a chart with that data.

- ☐ On the *Home* tab, in the *Format* group, click *Chart*. Drag a field to the canvas to create a chart.
2. Optionally, change the chart format using the options on the *Format* tab, in the *Chart Types* group.

Inserting an Existing Report

With InfoAssist+ opened in Document view, you can insert existing charts and reports onto the canvas from the Insert tab. The following procedure describes how you can insert reports into new documents and documents that are already populated with reports, text, and images.

Note: When working in Document view, you cannot insert an existing report that has a HOLD.

Procedure: How to Insert an Existing Report With the Insert Tab

You can create a document in the Custom Reports section of the domain and use Standard Reports items as Existing Report components. You cannot use other Custom Report items as components.

1. With InfoAssist+ open in Document view, click the *Insert* tab.
2. In the *Reports* group, click *Existing Report*.

An Open dialog box appears.

3. Browse to the report that you want to insert and click *Open*.

The report placeholder is added to the canvas.

Note: You cannot edit an existing report that is inserted into a document.

Creating a Document From a Single Report

You can take a single report created in Live Preview or Query Design view and convert it into a document, displaying it in Document view.

When you convert a single report into a document, the original report is preserved and a copy of that report is opened as a document in Document view. You can then add additional reports, charts, images, and text.

Inserting Text and Images

With InfoAssist+ opened in Document view, you can bring text and images onto the canvas. The following procedures describe how you can insert text and images into new documents and documents that are already populated with reports, text, and images.

Note: You can only do this in Document view.

The following procedures detail how to insert text and images. For more information on how to edit existing text and images, see [Editing Components in a Document](#) on page 371.

Procedure: How to Insert Text

1. With InfoAssist+ open in Document view, click the *Insert* tab.
 2. In the *Objects* group, click *Text Box*.
- A text component is added to the canvas, containing default text.
3. Double-click, or right-click, the text component to edit the text.

For more information on editing and styling the text, see [How to Edit Text](#) on page 377.

Procedure: How to Insert an Image

1. With InfoAssist+ open in Document view, click the *Insert* tab.
2. In the *Objects* group, click *Image*.

An Open dialog box appears.

Note: By default, the Open dialog box displays image files in the current WebFOCUS Content folder.

3. Browse to the desired image and click *OK*.

The selected image is added to the canvas.

Editing Components in a Document

The reports, controls, and text in a document can be edited, moved, resized, and deleted. Each of these components has a context menu which can be accessed by right-clicking the component.

Images can be moved, resized, and deleted, but they have no context menu and cannot be edited. Right-clicking a image brings up the option to delete it.

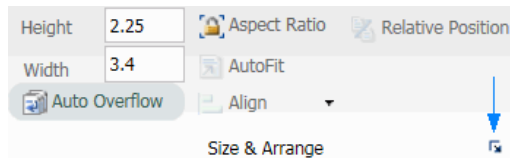
Procedure: How to Resize a Component

You can resize a component in the following ways:

- ☐ Using the component sizing handles.
- ☐ Changing the height and width on the *Layout* tab, in the *Size & Arrange* group.
- ☐ Accessing the options on the *Size* tab in the *Size and Position* dialog box.

The resize feature is available for all components that can be added to a document.

1. Open or create a document with at least one report, text component, control, or image.
2. Resize the component in one of the following ways:
 - ☐ **Sizing Handles:** Select the component and drag the sizing handles that appear around it. As you manually increase the height and width of the component, the new values appear in the corresponding text boxes in the Size & Arrange group of the Layout tab.
 - ☐ **Ribbon:** You can use the ribbon in one of the following ways:
 - ☐ Select the component in the document. On the *Layout* tab, in the *Size & Arrange* group, enter values in the Height and Width fields.
 - ☐ On the *Layout* tab, in the *Size & Arrange* group, click the dialog box launcher to open the Size and Position dialog box, as shown in the following image.



- ☐ **Shortcut Menu:** Right-click the component and select *Size and Position*. The Size and Position dialog box opens.

Note: You must right-click the corner of the component. For charts, if you click on any other point in the chart, the Size and Position option will not display.

From the Size and Position dialog box, open the Size tab.

Use the Height and Width options to change the position of the selected component. You can adjust the pixel size of the object with the Size options or the scale percentage of the object with the Scale options.

You can lock the aspect ratio using the Aspect Ratio button, which is available when working with charts, images, and text boxes while working in Document view. With the aspect ratio locked, changing the width automatically changes the height to keep the component to scale, and changing the height automatically changes the width.

Note: The Auto Overflow option is only available while working with reports in Document view through the Size & Arrange group. With Auto Overflow set, you cannot manually set the height and width of a report. The area of the report expands automatically to show all data.

When two objects are selected, the *Relative Position* button sets the bottom-left corner of the component that is higher on the page to the upper-left corner of the one that is lower. Once a relationship is created, arrows appear to show that relationship while both items are still selected.

Procedure: How to Move a Component

You can move a component by clicking it, or by accessing the Position section of the Size and Position dialog box. This feature is available for all components that can be added to a document.

You can also align components with each other so that their horizontal or vertical position matches. For more information, see [How to Align Components](#) on page 373.

1. Open or create a document with at least one report, text component, control, or image.
2. Select the component and move it by using one of the following methods:
 - ☐ Drag the component anywhere on the canvas.
 - or
 - ☐ Right-click the component and click *Size and Position*.
 - ☐ On the Size and Position dialog box, click the *Position* tab.
 - ☐ Use the Horizontal and Vertical options to change the position of the selected component.

Procedure: How to Align Components

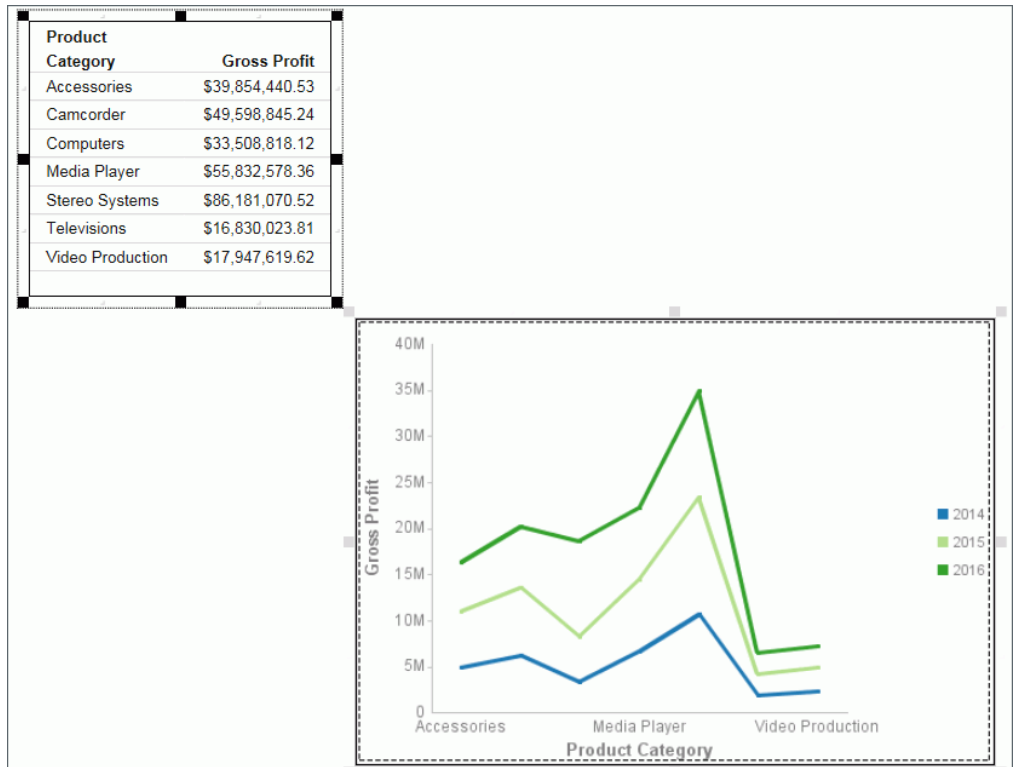
You can align components with each other so that their horizontal or vertical positions match. You must have multiple components selected to use the align options.

The alignment is anchored by the component that is in the farthest position of the selected alignment. For example, if you select two components and click *Align Left*, the components align horizontally with the component farthest to the left.

1. Open or create a document with at least two components.
2. Select a component.
3. Select a second component by holding the Ctrl key and clicking a component.

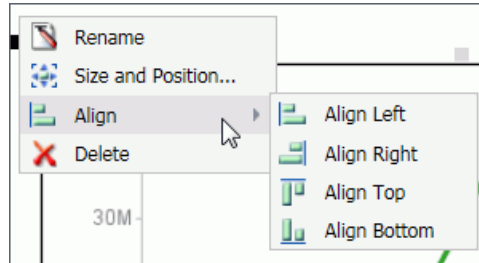
Note: You can select multiple components simultaneously by holding the Ctrl key and with the left mouse dragging a selection box around the components. When you release the mouse, sizing handles appear around each component that you selected. If the components display with light-grey coloring, this indicates that the components are selected.

Sizing handles appear around the components, as shown in the following image.



4. Align the components using one of the following methods:

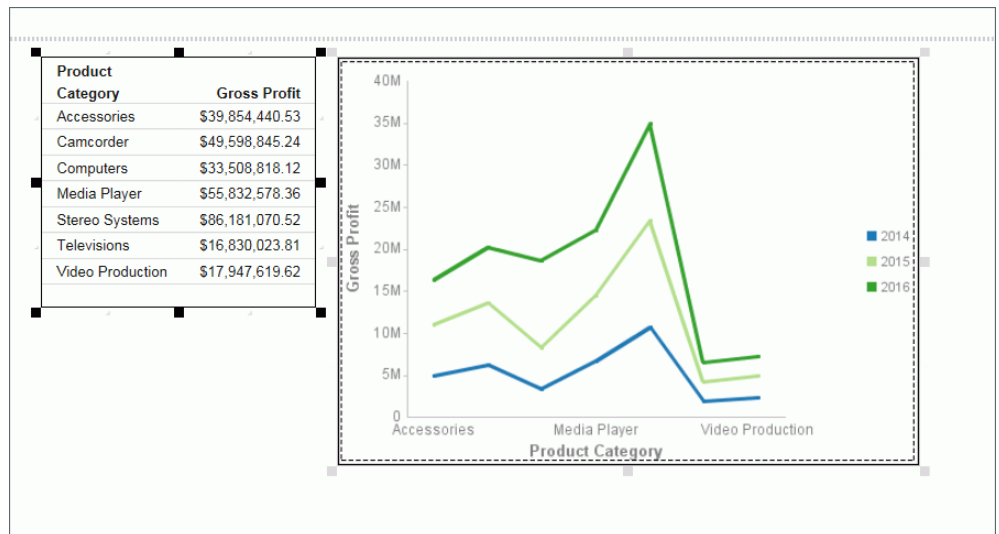
- ☐ Right-click one of the selected components and select an alignment option from the Align drop-down menu, as shown in the following image.



or

- ☐ Access the alignment options from the Align drop-down menu. The menu is available on the *Layout* tab, in the *Size & Arrange* group.

The selected components align, as shown in the following image.



5. Click anywhere in the canvas to deselect the components.

Procedure: How to Style and Customize a Report

When you select a component, you can perform various functions on the component, such as moving and resizing it, as explained in [How to Move a Component](#) on page 373 and [How to Resize a Component](#) on page 371. After clicking a component, you can use the ribbon to affect all settings of the selected component, except for fields. You can right-click a component to select individual fields to edit through the context menu.

In addition to reports, you can style and customize charts and text. For more information on charts, see [How to Style and Customize a Chart](#) on page 376. For more information on text, see [How to Edit Text](#) on page 377.

Note: Images cannot be edited.

1. Open or create a document with at least one report.
2. Click the report.

The Query pane becomes active and you can now select fields within the report. Select a field by clicking it in the canvas or in the Query pane.

Procedure: How to Style and Customize a Chart

When you select a component, you can perform various functions on the component, such as moving and resizing it, as explained in [How to Move a Component](#) on page 373 and [How to Resize a Component](#) on page 371. After clicking a component, you can use the ribbon to affect all settings of the selected component, except for fields. You can double-click or right-click a component to select individual fields to edit through the context menu or Field tab.

In addition to charts, you can style and customize reports and text. For more information on reports, see [How to Style and Customize a Report](#) on page 376. For more information on text, see [How to Edit Text](#) on page 377.

Note: Images cannot be edited.

1. Open or create a document with at least one chart.
2. Click the chart.

The Query pane becomes active and you can now select fields within the chart. Select a field by clicking it in the canvas or in the Query pane.

You can now edit the selected chart using commands available through the context menu or the ribbon.

Procedure: How to Edit Text

When you select a component, you can perform various functions on the component, such as moving and resizing it, as explained in [How to Move a Component](#) on page 373 and [How to Resize a Component](#) on page 371. After clicking a component, you can use the ribbon to affect all settings of the selected component, except for fields. You can right-click a component to select individual fields to edit through the context menu.

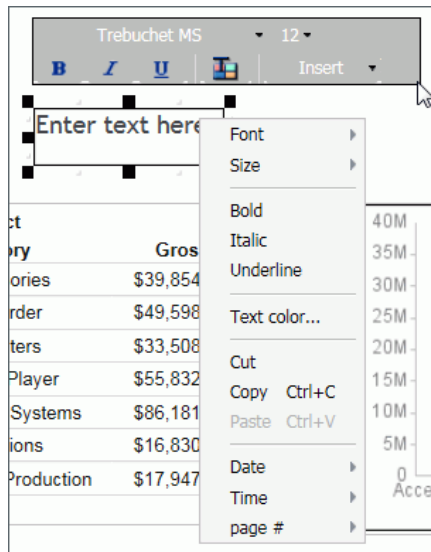
In addition to editing text, you can style and customize reports and charts. For more information on reports, see [How to Style and Customize a Report](#) on page 376. For more information on charts, see [How to Style and Customize a Chart](#) on page 376.

Note: Images cannot be edited.

1. Open or create a document with at least one text component.
2. Click the text box.

Sizing handles appear around the border and the text box toolbar becomes active.

3. Click anywhere in the text box and begin entering text.
4. Highlight the text you would like to edit, and right-click it. A menu of options appears, as shown in the following image.



5. Using the menu options, you can style the text and insert quick text.

The text component menu options are as follows:

- ☐ **Font.** Opens a list of available fonts for the selected text.

- ☐ **Size.** Opens a list of available text sizes for the selected text.
- ☐ **Bold.** Formats the selected text in bold.
- ☐ **Italic.** Formats the selected text in italics.
- ☐ **Underline.** Underlines the selected text.
- ☐ **Text color.** Opens the Color dialog box where you can select a color for the selected text.
- ☐ **Cut.** Cuts the selected text.
- ☐ **Copy.** Copies the selected text to the clipboard.
- ☐ **Paste.** Pastes the selected text from the clipboard.
- ☐ **Date.** Opens a list of date quick text in various formats.
- ☐ **Time.** Opens a list of time quick text in various formats.
- ☐ **page #.** Opens a list of page number quick text in various formats.

Procedure: How to Delete a Component

The following procedure applies to all components in Document view.

1. Open or create a document with at least one component.
2. Right-click the component and click *Delete*.

The component is deleted from the canvas.

Note: You can also delete a component by clicking it and pressing the Delete key.

Creating Multi-Page Documents

In Document mode, content can be created on multiple pages. The available output formats are: HTML, active report, PDF, Excel (xlsx), and PowerPoint (pptx). Additional Excel formats are available, as well.

Note: When working in Document mode using the active report format, you can create a rich, multi-object document that integrates various reports and charts, closely resembling a dashboard.

Microsoft Excel 2007 and Microsoft PowerPoint 2007 are enabled by default in the Administration Console. To enable or disable formats, you must do so in the Administration Console.

- ☐ The active report output format combines multiple reports and charts into one document, resulting in a tabbed active dashboard.
- ☐ Excel combines multiple reports as different sheets in a workbook.
- ☐ PowerPoint combines multiple reports and charts in a single slide.

Creating Multi-page Documents

You can create multi-page documents, allowing you to display an array of information across different pages.

Procedure: How to Create a Multi-page Document

1. Create a new document.

Page 1 appears on the canvas title bar.

2. Add content, such as a new or existing report, chart, text, and images to Page 1.
3. To add another page, do one of the following:

- ☐ On the Insert tab, in the Pages group, click *Page*.

- ☐ On the canvas title bar, click the page icon. From the Page menu that opens, select *New Page*.

A new page, for example, Page 2, is inserted after the current page, and appears on the canvas.

Each new page that you add is named Page *n*, where *n* is a unique number increasing by an increment of 1.

4. Add content to Page 2.
5. Repeat steps 3 - 4 until your document is complete.

To navigate between pages, open the Page menu by clicking the Page icon at the top of the canvas.

Creating a Multi-page Active Technologies Dashboard

You can create a multi-page active technologies dashboard using InfoAssist+.

***Procedure:* How to Create a Multi-page Active Technologies Dashboard**

1. Create a new active dashboard by setting the output format to active report.
Page 1 appears on the canvas title bar.
2. Add content, such as a new or existing report, chart, text, images, and active dashboard prompts to Page 1.
3. To add another page, do one of the following:

- ☐ On the Insert tab, in the Pages group, click *Page*.

- ☐ On the canvas title bar, click the page icon. From the Page menu that opens, select *New Page*.

A new page, for example, Page 2, is inserted after the current page, and appears on the canvas.

Each new page that you add is named Page *n*, where *n* is a unique number increasing by an increment of 1.

4. Add content to Page 2.
5. Repeat steps 3 - 4 until your dashboard is complete.

To navigate between pages, open the Page menu by clicking the Page icon at the top of the canvas.

6. Run the active dashboard.

The tabs appear at the top of the canvas.

Navigating the Page Menu

You can access the Page menu by clicking the Page icon in Design mode.

The Page menu lists the pages in the order in which you created them. You can rearrange the pages using drag-and-drop functionality. You can also select multiple pages and delete them.

In addition, the Page menu contains the *New Page* option to add a new page to the document. The *Duplicate* option creates a duplicate page.

The Page menu also contains *Page Options* which you can click to launch a dialog box of the following options:

- ☐ Rename Page
- ☐ Move Page Up

☐ Move Page Down

☐ Delete

When you select a page, the Rename, Move Up, Move Down, and Delete options become active in the menu bar at the top of the dialog box. Also, when you right-click a page, a context menu of these same options opens.

The position of the page that you have selected determines which directional options are available. For example, *Move Up* would not be an option for Page 1. *Move Down* would not be an option for the last page.

To close the dialog box, click *OK*.

Using the Active Cache Option

Because all post-retrieval processing is performed in the memory of the web browser, an active report has a processing limit of approximately 5,000 records or 100 pages of output. The active cache option enables you to send only the first page of active report output to the browser and retrieve subsequent pages from a temporary cache on the Reporting Server.

Tip: It is recommended that you set the number of rows retrieved five times greater than the number of lines retrieved per page (as indicated in SET LINES). The minimum number of rows retrieved is 100.

Enabling Active Cache Through InfoAssist+

Active cache is enabled when you select *active report* as the output type and the *Pages on Demand* button (Format tab, Navigation group) is enabled.

The Advanced tab on the active report options dialog box contains the Rows Retrieved drop-down list. Use this setting to establish the increments in which the cached data stored in a binary file is returned to the output window. The default is 100.

Note: In a multi-page document, active cache must be enabled per component. It is not globally set. Therefore, when creating a document in AHTML format, you must select each component separately to enable active cache. When you do so, the Pages on Demand button is activated.

Building 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.

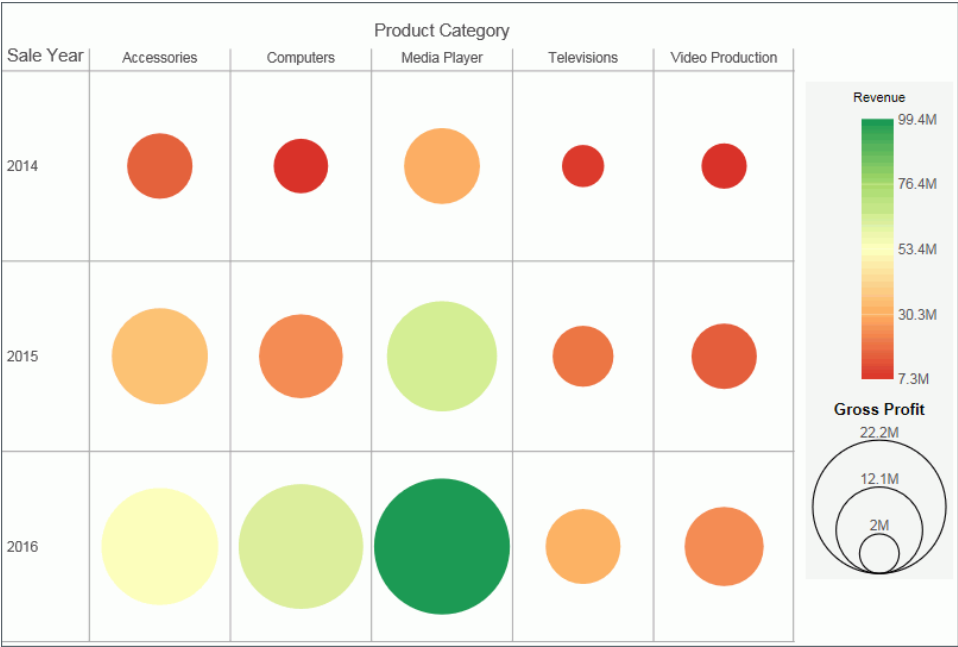
Visualizations allow you to monitor changes in data. They also serve to provide information in real-time, based on changes in underlying data or other components. A visualization can be updated, changed, or revised at any time to account for shifts in data needs.

Creating a Visual

You can create charts, maps, and grids to visually represent your data. You can add multiple visuals to the canvas to create a complete visualization.

The default visual is a bar stacked chart. You can use the Change option in the Visual group on the Home tab to change the visual type.

The following visual is a matrix marker chart that shows sales data for a range of electronic products.



Procedure: How to Create a Visualization From InfoAssist+

You can have multiple file types opened at once. To create a visualization:

1. On the Quick Access toolbar, click *New*.

or

Click the Application Main Menu button, and click *New*.

The InfoAssist+ splash screen displays.

2. Click *Build a Visualization*.
3. In the Open dialog box, select a data source and click *Open*.

InfoAssist+ switches to visualization mode.

Changing the Visual Type

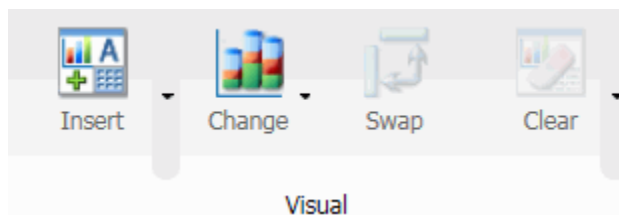
You can create a visual using the default chart type, which is a stacked bar chart. You can add your data to this chart and then change the chart type, or you can change the chart type prior to making your data selections.

Once you have started exploring your data, you can switch between the different types to obtain the graphical image that you wish to display.

You change the visual type from the Home tab.

Procedure: How to Change the Visual Type

1. On the *Home* tab, in the *Visual* group, click *Change*, as shown in the following image.



Note: The *Change* icon updates depending on the chart, map, or grid that you select from the Select a Visual menu. By default, the *Change* icon displays a stacked bar chart.

The Select a Visual menu displays.

2. On the Select a Visual menu, click the type of visual that you want to use.








Your canvas refreshes and displays the visual that you selected.






Note: Depending on the type of visual that you select, you may need to select additional or different data fields.





Selecting a Visual

It is important that you select a chart, grid, or map that appropriately displays a meaningful view of your data. InfoAssist+ provides a library of visuals.

You can select a visual type from the Select a Visual menu, on the *Home* tab, in the *Visual* group. The following table describes the types of charts available.

Icon	Visual Type	Description
	Grid	Grids provide a tabular view of data. They allow you to review data in a row and column format, similar to a printed report.
	Bar chart	Bar charts plot numerical data by displaying rectangular blocks against a scale (numbers or variable measure fields that appear along the axis).
	Stacked bar chart	A stacked bar chart is the default visual.
	Absolute line chart	Line charts allow you to trace the evolution of a data point by working backwards or interpolating. Highs and lows, rapid or slow movement, or a tendency towards stability are all types of trends well suited for a line chart.
	Area chart	Area charts analyze trends over time and look for differences in values.
	Stacked area chart	Stacked area charts allow you to stack data on top of each other.
	Pie chart	Pie charts are circular charts that represent parts of a whole. A pie chart emphasizes where your data fits, in relation to the other components in the pie.

Icon	Visual Type	Description
	Ring pie chart	Ring pie charts are useful when you want to review the value of each segment, which represents the measure value for the selected dimension, as it relates to the total for the selected measure.
	Scatter Plot	Scatter charts enable you to plot data using variable scales on both axes. When you use a scatter chart, the data is plotted with a hollow marker, so that you can visualize the density of individual data values around particular points, or discern patterns in the data.
	Bubble chart	Bubble charts can have two column fields representing X and Y data values, or have three column fields representing X, Y, and Z data values. The third variable (Z) represents size. The size of each bubble is used to show the relative importance of the data.
	Matrix Marker chart	Matrix marker charts are useful for analyzing one or two measures against a crosstab of two categorical dimensions. The result is a color-scaled matrix chart that shows categorized trends.
	Treemap	Treemaps are used to display large amounts of hierarchically structured data. Using a set of nested rectangles to illustrate data relationships, sections of a treemap represent branches of a tree.

Icon	Visual Type	Description
	Gauge	Gauges are used to display the value of a measure. In particular, circular gauges are used to represent a single data value within a given spectrum. You can create a single circular gauge for a measure or a matrix circular gauge, which shows the value of the selected measure across different dimensions, such as product category or yearly sales.
	Choropleth map	A geographically-based heat map. It is useful for visualizing location-based data, trends, and distributions across a geographic area.
	Proportional symbol map	A technique that uses symbols of different sizes to represent data associated with different areas or locations within the map.
	Heatmap	A heatmap is a graphical representation of data where the individual values that comprise a matrix are represented as colors. Using radiant hues, you can track the intensity of a data relationship using the colors defined in the legend.

Note: When new data is added to a bar, line, area, pie, scatter, bubble, gauge, or treemap chart, the chart will morph and rebuild, revealing the new values in a smooth transition.

Use the topics in this section to select and create your visuals.

Grids

Grids provide a tabular view of data. They allow you to review data in a row and column format, similar to a printed report.

In the following example, we review the Sale Year and Product Category data for the following measure fields:

☐ Revenue

☐ Gross Profit

Sale Year	Product Category	Revenue	Gross Profit
2014	Accessories	\$16,060,415.69	\$4,945,779.69
	Computers	\$7,857,928.55	\$3,376,380.55
	Media Player	\$30,105,200.05	\$6,730,870.05
	Televisions	\$9,295,726.31	\$1,964,927.31
	Video Production	\$7,313,170.38	\$2,286,521.38
2015	Accessories	\$35,619,872.81	\$10,953,840.81
	Computers	\$24,176,475.33	\$8,277,897.33
	Media Player	\$65,002,426.97	\$14,480,370.97
	Televisions	\$20,042,855.67	\$4,262,155.67
	Video Production	\$15,959,696.26	\$4,938,902.26
2016	Accessories	\$53,208,007.57	\$16,362,313.57
	Computers	\$63,190,001.88	\$18,677,664.88
	Media Player	\$99,448,235.40	\$22,237,625.40
	Televisions	\$30,964,700.29	\$6,560,087.29
	Video Production	\$23,810,094.17	\$7,330,486.17

Procedure: How to Create a Grid

1. Change the visual to a grid, or insert a new grid.
2. Drag data fields to the canvas or to the Query field containers to add them to your visual. The following Query field containers must be populated for this visual:

☐ Rows or Columns - one or more data fields

☐ Measure - one or more data fields

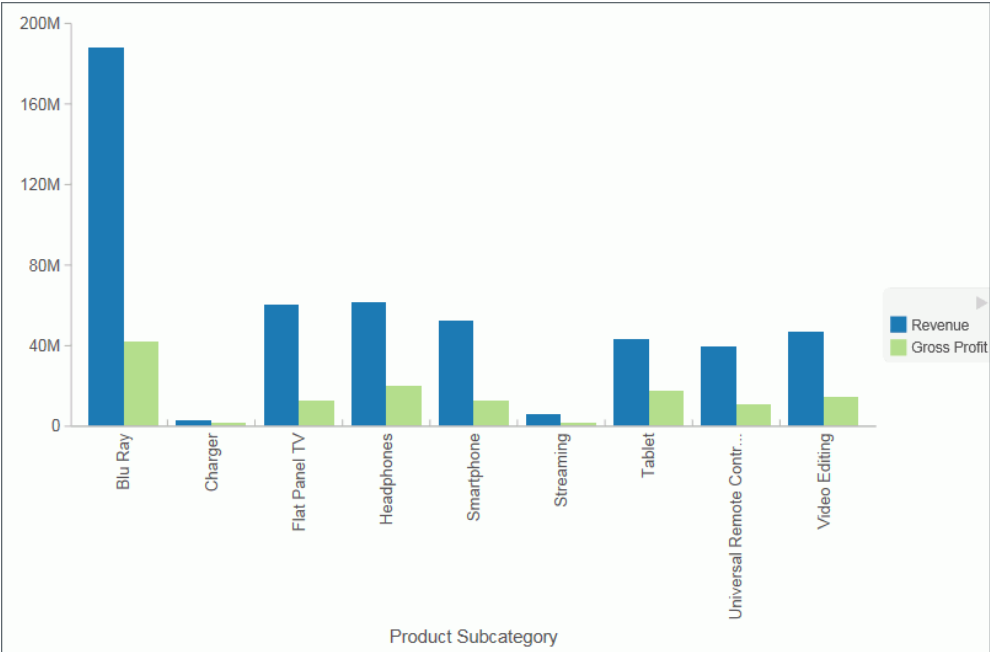
As you add, edit, or rearrange the fields in your Query field containers, your canvas refreshes.

Bar Charts

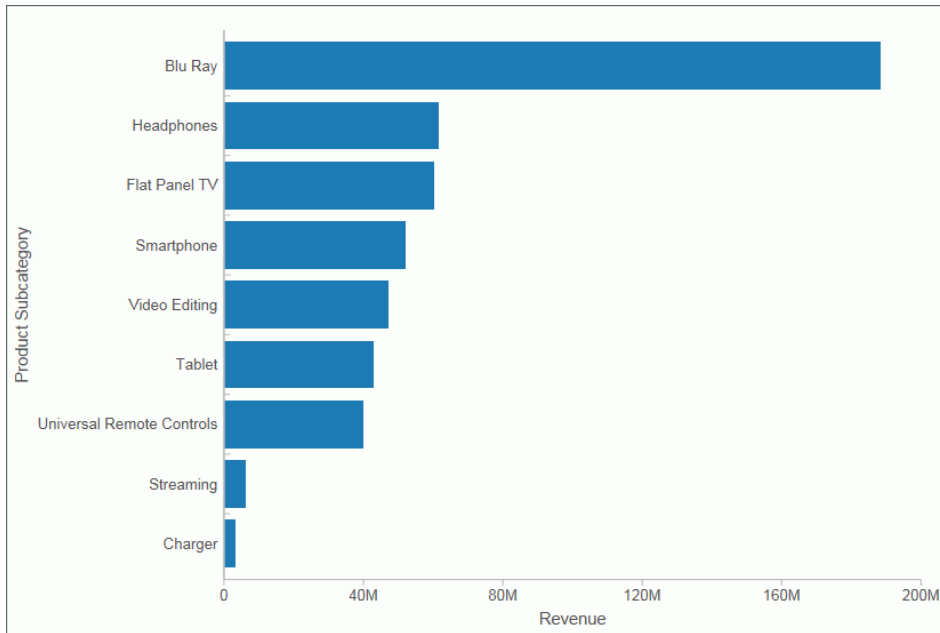
Bar charts plot numerical data by displaying rectangular blocks against a scale (numbers or variable measure fields that appear along the axis). The length of a bar corresponds to a value or amount. You can clearly compare data series (fields) by the relative heights of the bars. Use a bar chart to display the distribution of numerical data. You can create horizontal and vertical bar charts.

Note: If you are working with a large dataset, a scroll bar displays under your chart, enabling you to easily scroll through your data from left to right. In visualization mode, scroll bars are automatically enabled, but if you want to disable or re-enable scroll bars, click the Format tab and then click *Interactive Options*. In the Interactive Options dialog box, select the *Auto Enable X-Axis Scrolling* check box. If you are working in any other mode, you must enable this functionality.

Use a bar chart when individual values are important. For example, the following image is a basic vertical bar chart that compares the individual products sold to the total amount in sales for each product. A retailer would find it important to know which pieces of inventory are selling and how much revenue each item is generating for the company.



A horizontal bar chart becomes useful when you want to emphasize a ranking relationship in descending order, or the X-axis labels are too long to fit legibly side-by-side. For example, the following image is a basic horizontal bar chart that ranks which products are generating the most revenue for the retailer.



Note: You can swap the orientation of your data in a bar chart. To do so, on the *Home* tab, in the *Visual* group, click *Swap*.

Procedure: How to Insert a New Bar Chart

1. Change the visual to a bar chart or insert a new bar chart.
2. Drag data fields to the canvas or to the Query field containers to add them to your visual. The following Query field containers must be populated for this visual:

- ☐ Vertical Axis - one or more data fields
- ☐ Horizontal Axis - one data field

Note: You can also double-click a data field to add it to your Query field containers.

The bar chart displays on the canvas. You can add additional data fields for comparative purposes. You can also view underlying data by hovering over any particular point on the bar chart.

Procedure: How to Create a Stacked Bar Chart

The bar stacked visual is the default visual.

1. Change the visual to a stacked bar chart or insert a new stacked bar chart.
2. Drag data fields to the canvas or to the Query field containers to add them to your visual. The following Query field containers must be populated for this visual:

- ☐ Vertical Axis - one or more data fields
- ☐ Horizontal Axis - one or more data fields
- ☐ Color - one data field

Note: You can also double-click a data field to add it to your Query field containers.

The stacked bar chart displays on the canvas. You can add additional data fields for comparative purposes. You can also view underlying data by hovering over any particular point on the stacked bar chart.

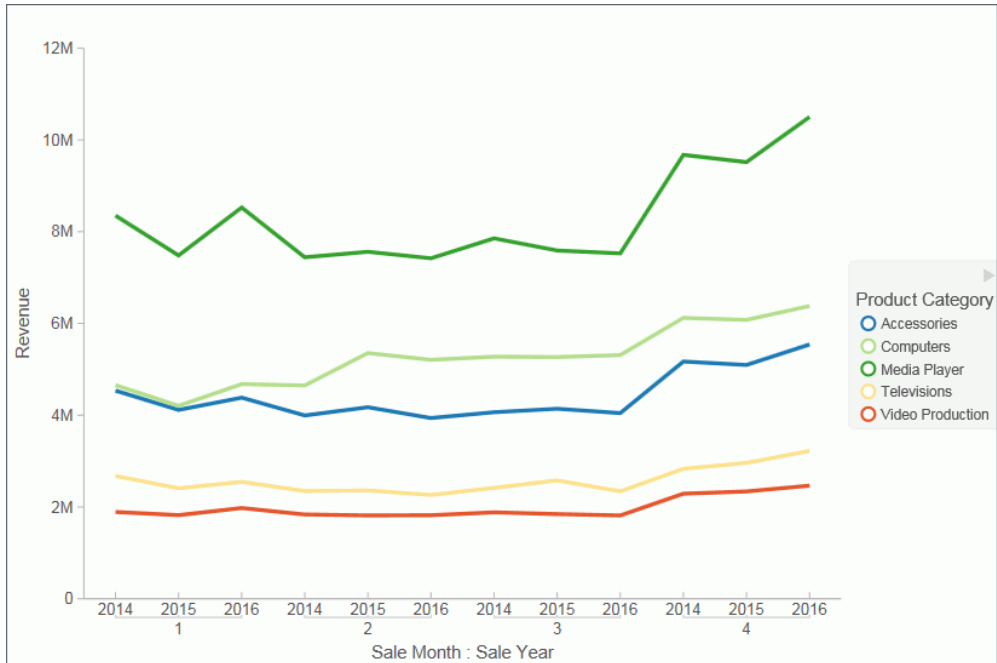
Line Charts

Line charts allow you to trace the evolution of a data point by working backwards or interpolating. Highs and lows, rapid or slow movement, or a tendency towards stability are all types of trends well suited for a line chart.

You can also plot line charts with two or more scales to present a comparison of the same value, or set of values, in different time periods.

Note: If you are working with a large dataset, a scroll bar displays under your chart, enabling you to easily scroll through your data from left to right. In visualization mode, scroll bars are automatically enabled, but if you want to disable or re-enable scroll bars, click the Format tab and then click *Interactive Options*. In the Interactive Options dialog box, select the *Auto Enable X-Axis Scrolling* check box. If you are working in any other mode, you must enable this functionality.

Use a line chart when you want to trend data over time, for example, monthly changes in employment figures, or yearly sales of an item in your inventory. The following image is a line visual that shows the gross profit in monthly sales for products.



Procedure: How to Create a Line Chart

1. Change the visual type to a line chart or insert a new line chart.
2. Drag data fields to the canvas or to the Query field containers to add them to your visual. The following Query field containers must be populated for this visual:

☐ Vertical Axis - one or more data fields

☐ Horizontal Axis - one data field

☐ Color - one data field (optional)

Note: You can also double-click a data field to add it to your Query field containers.

To add insight, you can drag a data field to the color Query field container. This displays the values for this field using color.

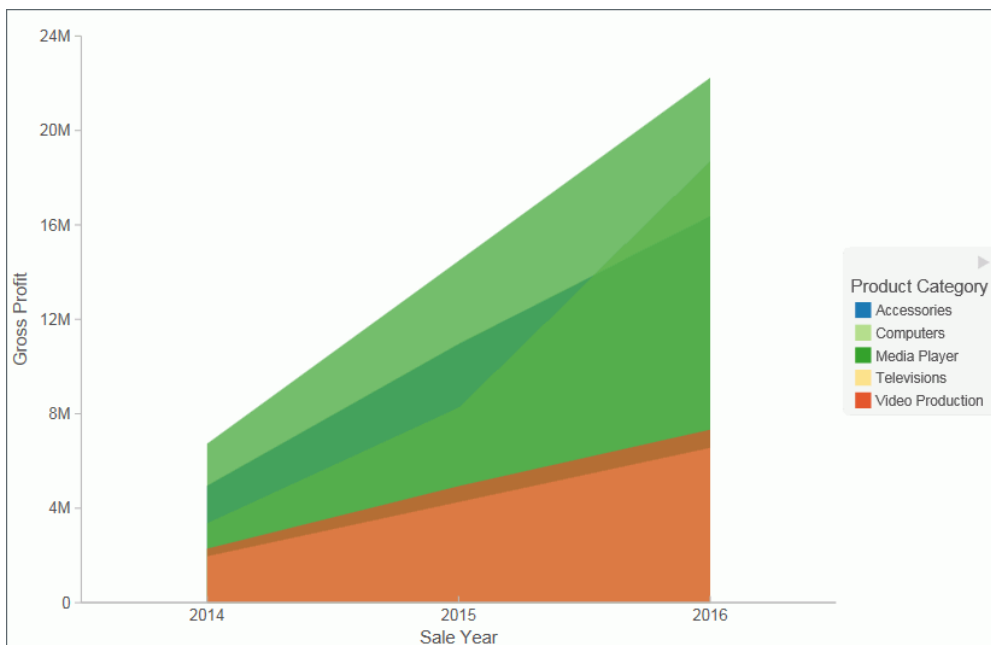
The line chart displays on the canvas. You can add additional data fields for comparative purposes. You can also view underlying data by hovering over any particular point on the line chart.

Area Charts

Area charts analyze trends over time and look for differences in values by using the *see-thru* nature of the area fills. Stacked area charts allow you to stack data on top of each other. Stacking allows you to highlight the relationship between data series, showing how some data series approach a second series.

Note: If you are working with a large dataset, a scroll bar displays under your chart, enabling you to easily scroll through your data from left to right. In visualization mode, scroll bars are automatically enabled, but if you want to disable or re-enable scroll bars, click the Format tab and then click *Interactive Options*. In the Interactive Options dialog box, select the *Auto Enable X-Axis Scrolling* check box. If you are working in any other mode, you must enable this functionality.

Use an area chart when you want to distinguish the data more dramatically by highlighting volume with color. For example, the following image is a basic area chart that depicts the yearly gross profit for various electronic products.



Procedure: How to Create an Area Chart

1. Change the visual type to an area chart or insert a new area chart.
2. Drag data fields to the canvas or to the Query field containers to add them to your visual. The following Query field containers must be populated for this visual:

- ☐ Vertical Axis - one or more data fields
- ☐ Horizontal Axis - one data field
- ☐ Color - one data field (optional)

Note: You can also double-click a data field to add it to your Query field containers.

The area chart displays on the canvas. You can add additional data fields for comparative purposes. You can also view underlying data by hovering over any particular point on the area chart.

Procedure: How to Create a Stacked Area Chart

1. Change the visual type to a stacked area chart or insert a new stacked area chart.
2. Drag data fields to the canvas or to the Query field containers to add them to your visual. The following Query field containers must be populated for this visual:

- ☐ Vertical Axis - one or more data fields
- ☐ Horizontal Axis - one data field
- ☐ Color - one data field (optional)

Note: You can also double-click a data field to add it to your Query field containers.

The stacked area chart displays on the canvas. You can add additional data fields for comparative purposes. You can also view underlying data by hovering over any particular point on the stacked area chart.

Pie Charts

Pie charts are circular charts that represent parts of a whole. A pie chart emphasizes where your data fits, in relation to the other components in the pie. Pie charts work best when there are a limited number of slices (for example, less than 10) and the slices are all of a sufficient value as to reveal their fill color inside their wedge.

Use a pie chart when you have segments of data that you want to display as a whole. For example, the following image is a pie chart that shows the proportions of various electronic products based on the quarterly revenue.



You can add one or more measures to the Measure field container. Each measure will be used to create a separate, unique pie chart, to which you can add a measure or dimension to the Color field container to add color to your chart.

Note: When working with pie charts, you can add one measure field to the Color field container. This adds the measure as a By field, and determines how the pie chart is colored. Depending on your measure data, this may result in a large number of pie segments.

Procedure: How to Create a Pie Chart

1. Change the visual type to a pie chart or insert a new pie chart.
2. Drag data fields to the canvas or to the Query field containers to add them to your visual. The following Query field containers must be populated for this visual:
 - ☐ Measure - one data field. Data in this category is used to indicate the size of the pie slice for the relevant category.
 - ☐ Color - one data field. Data in this category indicates the colors in your pie chart.

Note: You can also double-click a data field to add it to your Query field containers.

The pie chart displays on the canvas. You can add additional data fields for comparative purposes, or to create another pie chart on the same canvas. You can also view underlying data by hovering over any particular point on the pie chart.

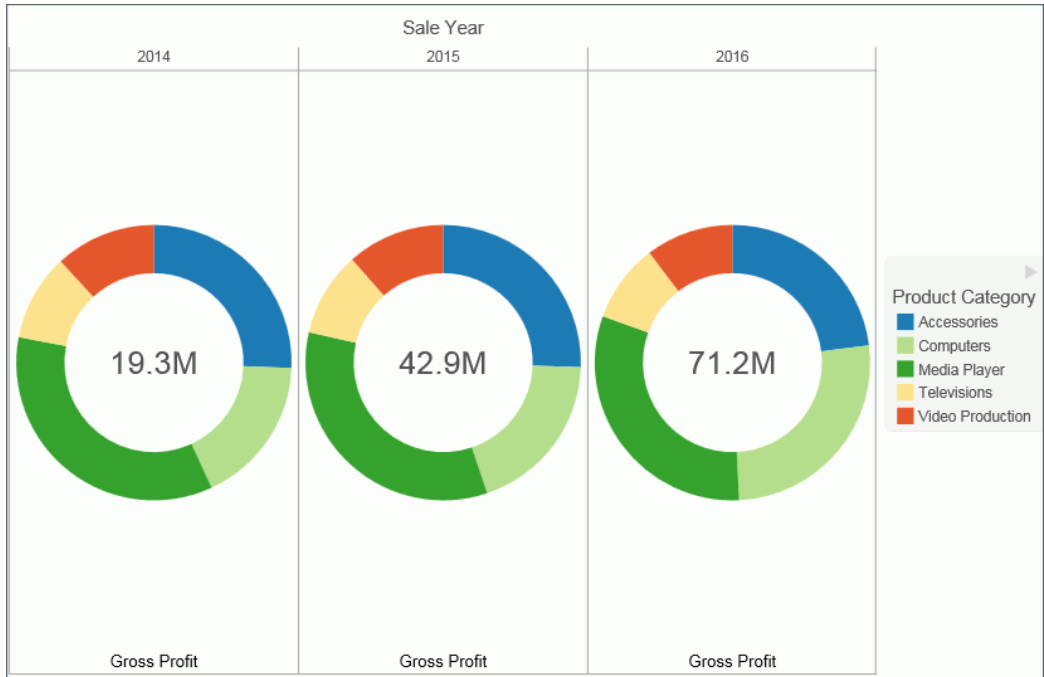
Ring Pie Charts

Ring pie charts are circular charts that display the total for the selected measure, as well as the individual segments that comprise the ring pie chart. You can hover over each segment to review the underlying data values. This is useful when comparing the measure value for an individual segment against the total for the measure, which displays in the center of the ring pie.

You can add one or more measures to the Measure field container. Each measure will be used to create a separate, unique ring pie chart, to which you can add a measure or dimension to the Color field container to add color to your chart.

Note: The font size of the value label in the middle of the ring is automatically set by the chart engine.

Use a ring pie chart when you want to review the value of each segment, which represents the measure value for the selected dimension, as it relates to the total for the selected measure. The following image is an example of a ring pie chart.



Procedure: How to Create a Ring Pie Chart

1. Change the visual type to a ring pie chart or insert a new ring pie chart.
2. Drag data fields to the canvas or to the Query field containers to add them to your visual. The following Query field containers must be populated for this visual:

- ☐ Measure - one data field. Data in this category is used to indicate the size of the ring pie segment for the relevant category.
- ☐ Color - one data field. Data in this category indicates the colors in your ring pie chart.

Note: You can also double-click a data field to add it to your Query field containers.

The ring pie chart displays on the canvas. The total for the selected measure displays in the center of the ring pie chart. You can view underlying data by hovering over any of the ring pie chart segments.

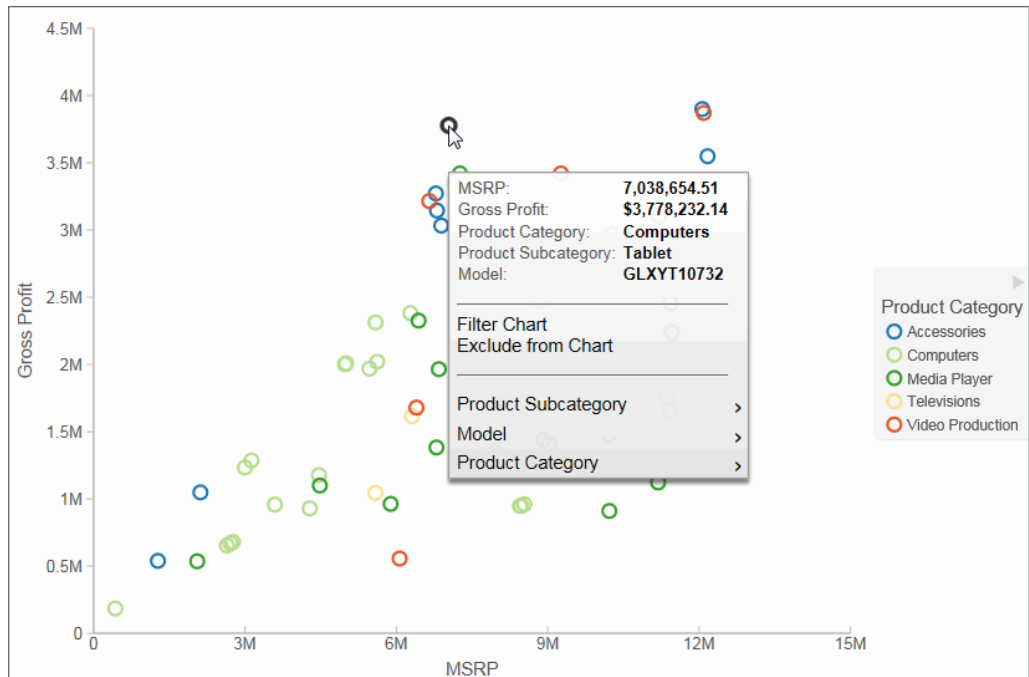
Scatter Charts

Scatter charts enable you to plot data using variable scales on both axes. When you use a scatter chart, the data is plotted with a hollow marker, so that you can visualize the density of individual data values around particular points, or discern patterns in the data. A numeric X axis, or sort field, always yields a scatter chart, by default.

Note: You can specify a non-measure (dimension) data field on the horizontal or vertical axis, or both.

If your chart reveals clouds of points, there is a strong relationship between X and Y values. If data points are scattered, there is a weak relationship, or no relationship.

Adding data fields to the Detail Query field container creates additional BY fields on the scatter chart. For example, the following image shows the results when adding the Product, SubCategory and Model dimension fields to Detail Query field container in a scatter chart which showed gross profit and MSRP data.



Procedure: How to Create a Scatter Chart

1. Change the visual type to a scatter chart or insert a new scatter chart.

2. Drag data fields to the canvas or to the Query field containers to add them to your visual. The following Query field containers must be populated for this visual:

- ☐ Vertical Axis - one data field
- ☐ Horizontal Axis - one data field
- ☐ Detail - one or more data fields
- ☐ Color - one data field

Note: You can also double-click a data field to add it to your Query field containers.

The scatter chart displays on the canvas. You can also view underlying data by hovering over any particular point on the scatter chart.

Bubble Charts

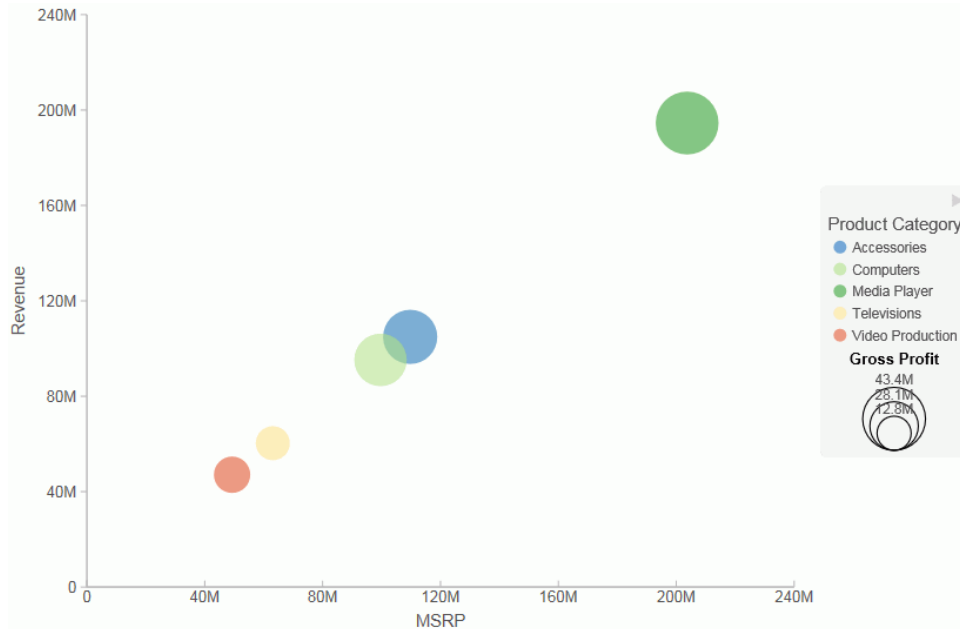
A bubble chart is a chart in which the data points are represented by bubbles. Bubble charts can have two column fields representing X and Y data values, or have three column fields representing X, Y, and Z data values, in that order. The Z variable represents size. The size of each bubble is used to show the relative importance of the data.

When you add a data field to the Size field container, this value is represented as the Z Axis Title in the legend. It displays as an empty Z Axis Title when a size data field is not specified. If you choose to indicate a Z, or size, data value, the data label displays in the legend. A Size Legend also displays, showing the estimated data value for a range of circle sizes. This allows you to estimate the value of the data based on the size of the circle.

Note:

- ☐ You can hover over the circles in the visual to obtain exact data values for any given point.
- ☐ You can specify a non-measure (dimension) data field on the horizontal or vertical axis, or both.
- ☐ In Visualization mode and for HTML5 charts, if you select the No fill option for your Series style when creating a bubble chart, the series displays in shades of black. For active charts, you must enable the Show Border Color option in order to view the bubbles in your chart at run time, otherwise the bubbles are invisible.

In the following image, a bubble chart is used to show the Manufacturers Suggested Retail Price (MSRP) plotted against Revenue for a variety of electronics products. It also shows the values for Gross Profit, which was specified in the Size field container in the Query pane.



Procedure: How to Create a Bubble Chart

1. Change the visual to a bubble chart or insert a new bubble chart.
2. Drag data fields to the canvas or to the Query field containers to add them to your visual. The following Query field containers must be populated for this visual:

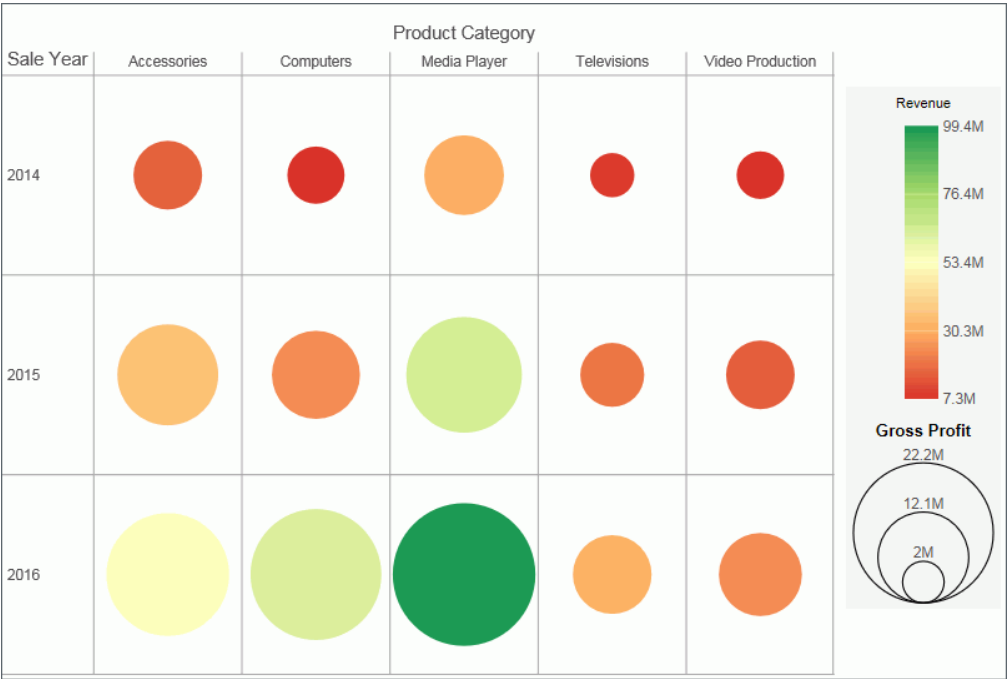
- ☐ Vertical Axis - one data field
- ☐ Horizontal Axis - one data field
- ☐ Detail - one or more data fields
- ☐ Size - one data field
- ☐ Color - one data field (optional). Labels for the values in this data field will comprise the legend.

Note: You can also double-click a data field to add it to your Query field containers.

The bubble chart displays on the canvas. You can also view underlying data by hovering over any particular point on the bar chart.

Matrix Marker

Matrix marker charts are useful for analyzing one or two measures against a crosstab of two categorical dimensions. You can use the Size Query field container for one measure and the Color Query field container for a second measure. The result is a color-scaled matrix chart that shows categorized trends, as shown in the following image.



Procedure: How to Create a Matrix Marker Chart

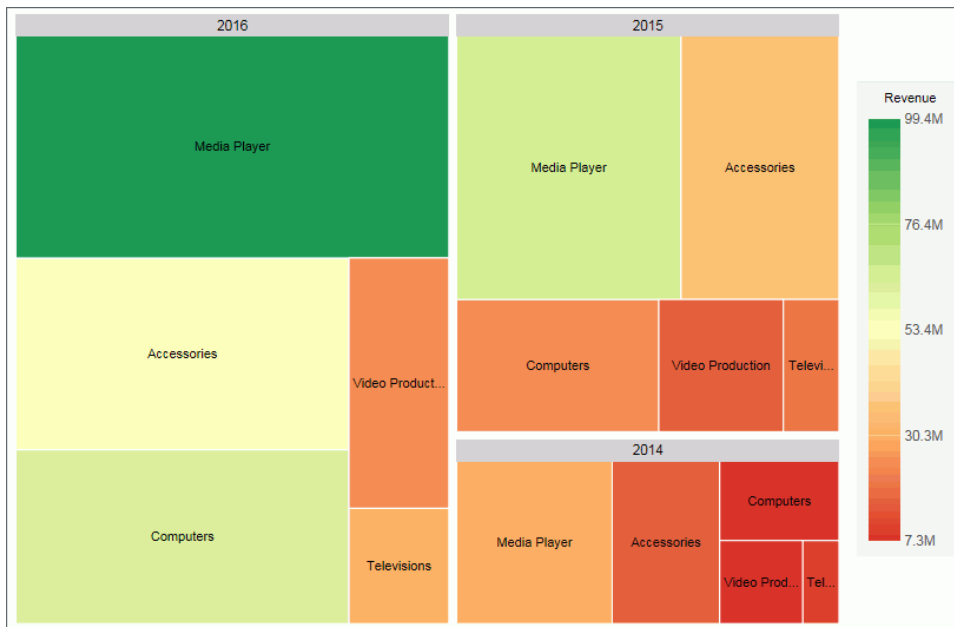
1. Change the visual to a matrix marker chart or insert a new matrix marker chart.
2. Drag data fields to the canvas or to the Query field containers to add them to your visual. The following Query field containers must be populated for this visual:
 - ☐ Matrix Rows - one data field
 - ☐ Matrix Column - one data field
 - ☐ Size - one data field. The data for this field determines the size of the marker.
 - ☐ Color - one data field. The data in this field determines the color of the marker.

The matrix marker chart displays.

Treemaps

Treemaps are used to display large amounts of hierarchically structured data. Using a set of nested rectangles to illustrate data relationships, sections of a treemap represent branches of a tree. Each branch is given a rectangle, to which any number of smaller sub-branches can be assigned. The size of each branch is proportional to the summed values of the elements inside the branch.

The following treemap shows the categories of the selected dimension fields, using two data fields to determine the size and color of the treemap segments.



Procedure: How to Create a Treemap

1. Change the visual to a treemap or insert a new treemap.
2. Drag data fields to the canvas or to the Query field containers to add them to your visual. The following Query field containers must be populated for this visual:
 - ☐ Grouping - one or more data fields, which establishes the hierarchy of the Treemap grouping.
 - ☐ Size - one data field. This data controls the size of the branches that display.
 - ☐ Color - one data field. This data controls the colors that display based on the accompanying gradient.

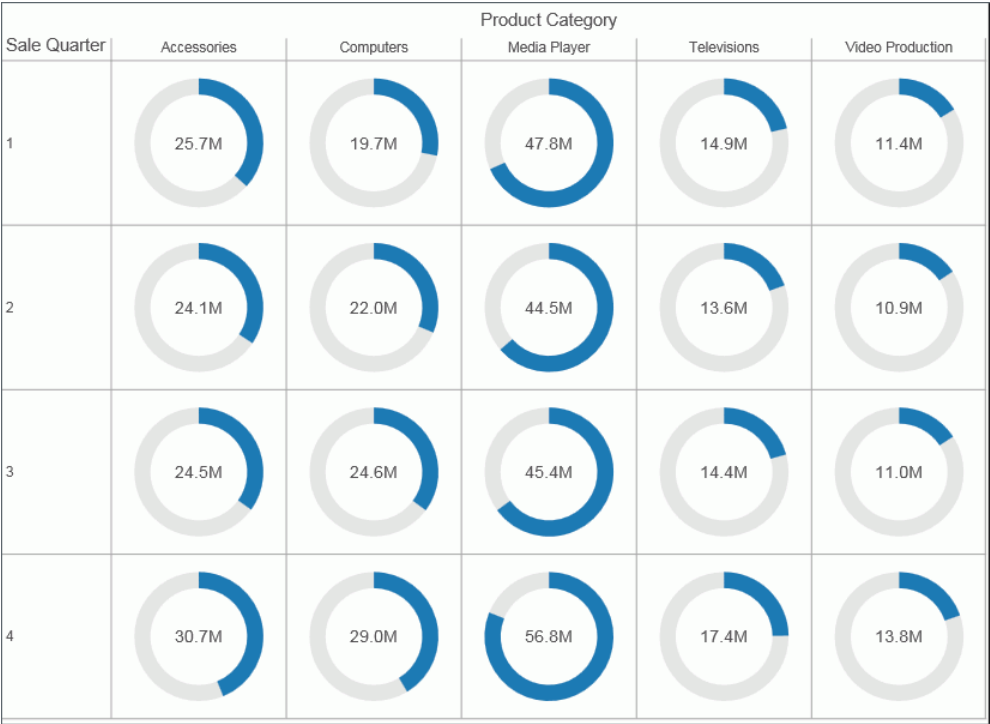
The treemap displays.

Gauges

Gauges are used to display the value of a measure. In particular, circular gauges are used to represent a single data value within a given spectrum. These gauges have a circular shape. You can create a single circular gauge for a measure or a matrix circular gauge, which shows the value of the selected measure across different dimensions, such as product category or yearly sales. The value of the measure that displays in a circular gauge is determined by the underlying data stored for that measure in the database.

The circular gauge functionality uses only one measure in its presentation. The legend reflects the color of the measure within the circular gauge.

In the following example, we review revenue data for each product category by quarterly sales in a matrix circular gauge chart.



Procedure: How to Create a Circular Gauge

- 1. Change the visual type to a gauge or insert a new gauge.

2. Drag data fields to the canvas or to the Query field containers to add them to your visual. The following query field containers must be populated for this visual:

- ☐ Measure - one data field. Data in this category is used to indicate the value of the selected measure, which displays within the gauge.
- ☐ Tooltip - one or more data fields. The fields that you add provide you with the ability to review additional related, underlying data for different measures. Tooltips are optional.

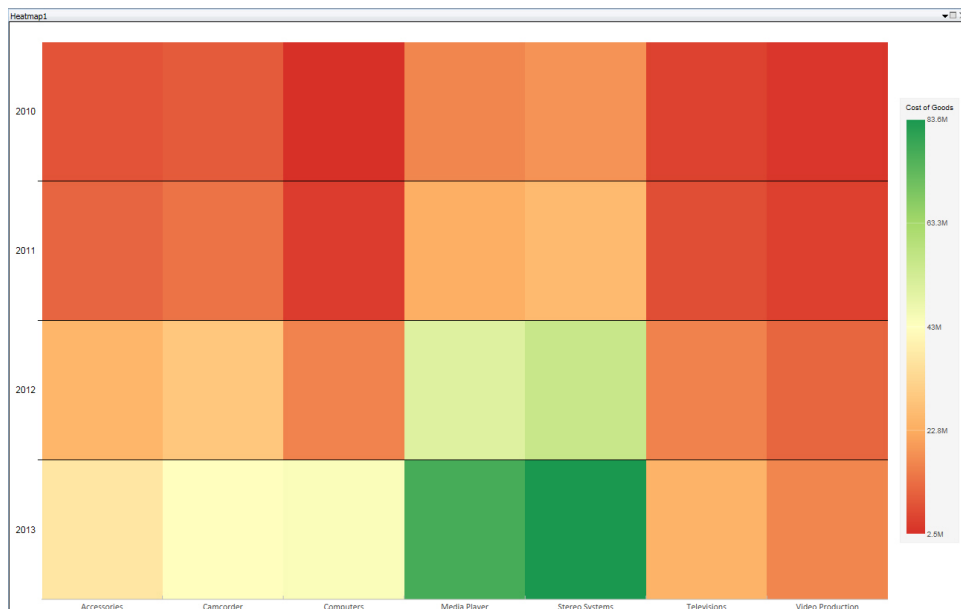
Note: You can also double-click a data field to add it to your Query field containers.

The circular gauge displays on the canvas. You can select additional measure fields for which to include in the tooltip.

Heatmaps

A heatmap is a graphical representation of data where the individual values that comprise a matrix are represented as colors. Using radiant hues, you can track the intensity of a data relationship using the colors defined in the legend.

Heatmaps are useful when you are looking for hot spots in your data, or areas of focus or interest, as shown in the following image.



Procedure: How to Create a Heatmap

- 1. Change the visual to a heatmap or insert a new heatmap.
- 2. Drag data fields to the canvas or to the Query field containers to add them to your visual. The following Query field containers must be populated for this visual:

- ☐ Color - one data field. This data controls the colors that display based on the accompanying gradient.
- ☐ Horizontal field container - one data field.
- ☐ Vertical field container - one data field

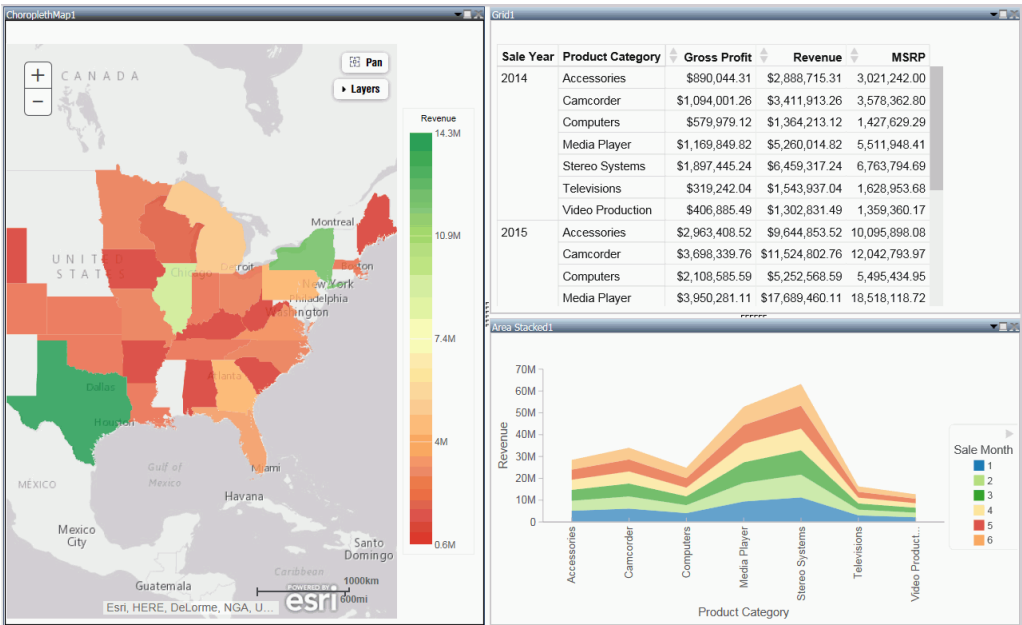
Note: You can optionally populate the Matrix Row and Column fields to increase the segmentation of your heatmap.

The heatmap displays.

Interacting With Visualizations

A visualization is comprised of one or more visuals, such as charts, maps, or grids and text. You can create different views of your data in a single visualization, and share that visualization with others in your enterprise.

The following image shows a sample visualization. This visualization includes a map, a matrix grid, and a stacked area chart.



This section summarizes the tasks that are available to you when working with visuals. It provides centralized instructional information on performing each task and offers links to the most common topics when working with visuals.

Task	How To
Change visual	<p>On the <i>Home</i> tab, in the <i>Visual</i> group, click <i>Change</i>.</p> <p>Note: The <i>Change</i> icon updates depending on the chart, map, or grid that you select from the Select a Visual menu. By default, the <i>Change</i> icon displays a stacked bar chart.</p> <p>Select a chart, map, or grid from the Select a Visual menu.</p>
Insert new visual	<p>On the <i>Home</i> tab, in the <i>Visual</i> group, click <i>Insert</i>. Use the default stacked bar chart or click <i>Change</i> to select a different chart, map, or grid from the Select a Visual menu.</p> <p>Note: You can also add additional charts, maps, or grids to a visualization by dragging a data field onto the canvas and placing it using the handles that are available.</p>
Rearrange visuals	<p>Drag a visual on top of another visual to activate a shaded area that contains handles, which can be used to indicate placement.</p>
Copying a visual	<p>On the canvas, select a visual. On the <i>Home</i> tab, in the <i>Clipboard</i> group, click <i>Copy</i>.</p> <p>Note: You can also press CTRL+C to copy a selected visual.</p>
Pasting a visual	<p>Copy a visual. On the <i>Home</i> tab, in the <i>Clipboard</i> group, click <i>Paste</i>.</p> <p>Note: You can also press CTRL+V to paste a copied visual on the canvas.</p>

Task	How To
Duplicating a visual	On the canvas, select a visual. On the <i>Home</i> tab in the <i>Clipboard</i> group, click <i>Duplicate</i> . A duplicate visual is created and a sequential number is assigned based on the type of visual.
Delete visual	Select a visual. On the <i>Home</i> tab, in the <i>Clipboard</i> group, click <i>Cut</i> . You can click the <i>Close</i> button in the upper-right corner of the current visual. From the Query pane, right-click a visual and click <i>Delete</i> . You can also press the Delete key when a visual is selected.
Apply Filter	Drag a dimension field or measure field into the Filter pane to access the filter options that are available. To add filter options for a field that is already in the Query pane, select the field and on the <i>Field</i> tab, in the <i>Filter</i> group, click <i>Filter</i> .
Add visuals to the storyboard	Create a visual. On the <i>Home</i> tab, in the <i>Storyboard</i> group, click <i>Add</i> .

Procedure: How to Insert a New Visual

- On the *Home* tab, in the *Visual* group, click the down arrow next to *Insert*.
- On the menu, click one of the following options:
 - ☐ **Chart.** Inserts a stacked bar chart visual.
 - ☐ **Grid.** Inserts a grid visual.
 - ☐ **Text.** Inserts a blank text cell.
- Populate your visual with data or add text to the text cell.

Note:

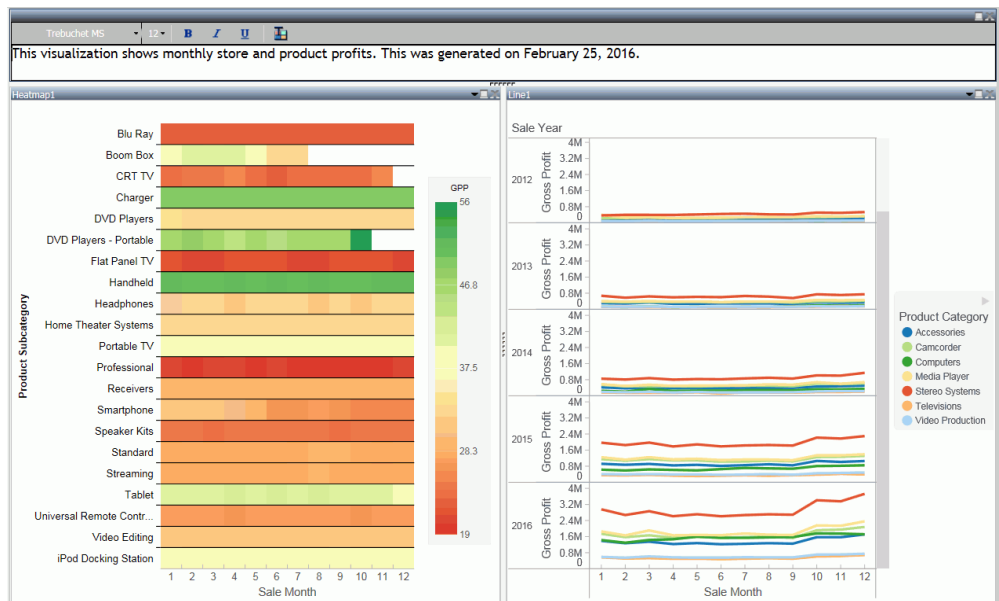
- ☐ By default, when you click *Insert*, a stacked bar chart visual is inserted.

- You can also drag a data field from the Data pane to the canvas to insert a new visual. This inserts the default visual, a stacked bar chart. You can use the placement handles to position your new visual on the canvas, for example, above an existing visual or to the side of an existing visual.

Procedure: How to Add Text to Your Visualization

- On the *Home* Tab, in the *Visual* group, click the down arrow on next to *Insert*.
- On the menu, click *Text*.
A text cell opens on the canvas.
- Add text to your visualization.

Note: You can resize the text cell and use the text formatting options to customize the display of any text that you add, as shown in the following image.

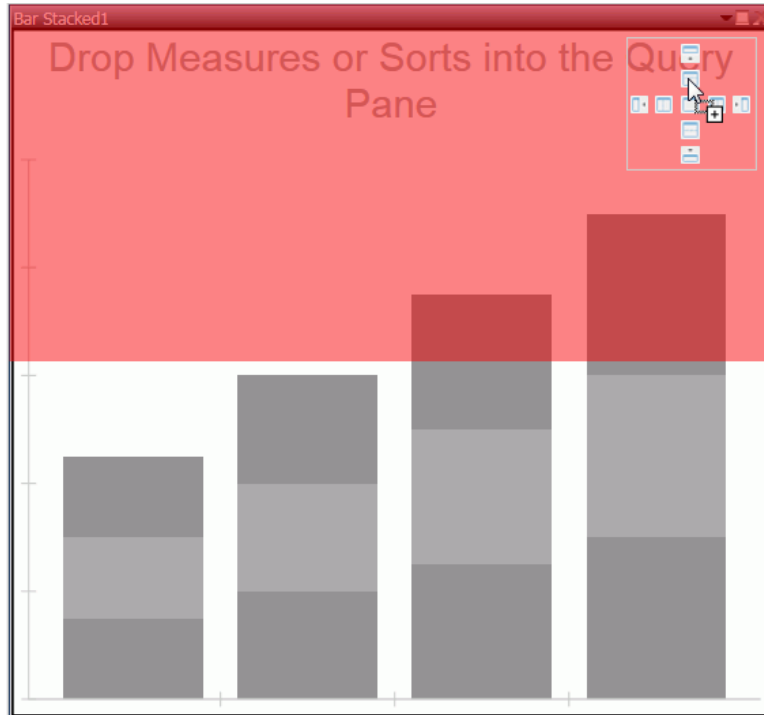


You can also position the text cell in your visualization by dragging the text cell on top of a visual. Use the placement handles to indicate placement of the text cell.

Procedure: How to Create a Visualization

- Begin with the default canvas, which consists of a stacked bar chart template.

2. Insert a new visual in one of the following ways:
 - a. Drag a data field from the Data pane onto the canvas. Handles display, which allow you to select the location for the new visual, for example, top (above) or left of the current visual, as shown in the following image.



- b. On the *Home* tab, in the *Visual* group, click *Insert*.
Note: You can optionally click the down arrow on the Insert button to specify the addition of a chart, grid, or text.
3. Add another visual.
Now, three visual cells display side-by-side.
4. Click a visual to select it.
Note: You can click a visual to activate it, or double-click on the visual number or name in the Query pane.
5. Reorganize your visuals using the handles.

6. Once you have organized the placement of your visuals, select one and specify the visual type.
 - a. On the *Home* tab, in the *Visual* group, click *Change*.

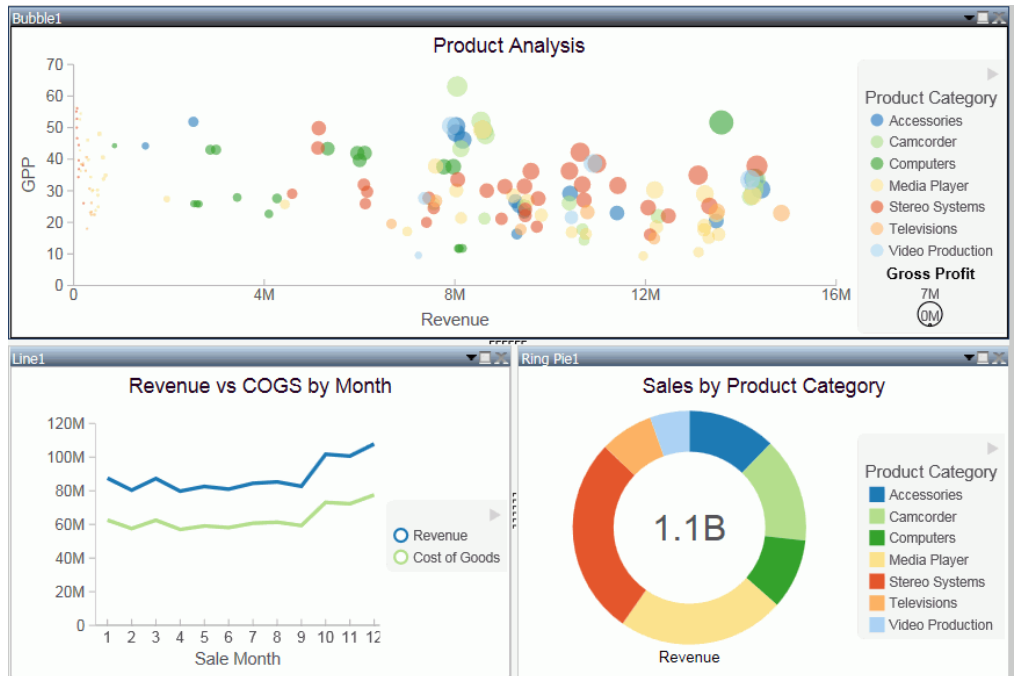
Note: The *Change* icon updates depending on the chart, map, or grid that you select from the Select a Visual menu. By default, the *Change* icon displays a stacked bar chart.
 - b. In the Select a Visual menu, click the type of visual you want to use. For example, Line, Area, or Map.
 - c. Repeat these steps for all three visuals on your canvas.
7. Populate each visual with your data.

You can change the type of visual that you previously selected at any time. You can also resize or reorganize the position of each visual as you add data.

For example, move the lower-left visual to the top of the visualization.



The bubble chart now runs across the top of the visualization.



8. Click Save to save your visualization.

Minimizing or Maximizing a Visual

When working on a visualization with more than one chart, map, or grid, you can maximize and minimize individual visuals. This allows you to focus on one visual at a time, and then minimize it to view it alongside the other visuals.

The maximize and minimize icons are located in the top-right corner of each visual, next to the Close button. When you click the *Maximize* icon, the current visual moves to the foreground and is the only visual that displays on the canvas. You can work on this visual, and then minimize it to view the other visuals.

Note: You can view other visuals in the maximized mode by selecting a different visual in the Query pane.

Procedure: How to Minimize or Maximize a Visual

1. Create a visualization with two or more visuals.

2. Perform the following actions to minimize or maximize your visual:

- ☐ Click the maximize icon or double-click on the Title bar to maximize your visual.
- ☐ Click the minimize icon or double-click on the Title bar to minimize your visual.

You can maximize one visual at a time, and you can switch between visuals in this mode by double-clicking a different visual in the Query pane.

Procedure: How to Delete a Visual

1. In your visualization, select the chart, map, or grid that you want to delete.
2. Perform one of the following tasks to delete the visual:

- ☐ Press the Delete key.
- ☐ On the *Home* tab, in the *Clipboard* group, click *Cut*.
- ☐ Click the *Close* button in the upper-right corner of the current visual.
- ☐ From the Query pane, right-click on a visual, and click *Delete*.

Note: You can use the Undo and Redo options on the Quick Access Toolbar to reverse or redo any prior actions.

Renaming a Visual

You can rename a visual on the canvas or within your visualization. You may want to do this for presentation and organizational purposes, as each visual has a default label (for example, Bar 1, Bar 2, and Bar 3). You can change these labels by renaming the visual in the Query pane.

Once new labels are in place, it is easier to recognize which visual you want to select at any given time.

Using the shortcut menu for a visual in the Query pane, you can also rename your visual.

Procedure: How to Rename a Visual

1. Create a visualization with one or more chart, map, or grid.
2. In the Query pane, right-click the visual number for which you want to modify the title.
3. Click *Rename*.
4. In the Edit Title dialog box, enter a new name for the visual.
5. Click *OK*.

The visual is renamed in the Query pane and the new title is reflected at the top of the selected visual.

Creating Matrix Charts

Matrix charts are powerful, comparative tools. They provide enough detail to show a trend and they organize information in a categorical fashion.

Matrix charts display data in a grid, showing the comparative values on either axis. They provide you with a quick glance at trends over time, giving you a succinct synopsis of a situation (for example, sales or investment trends).

You can use various formats in your matrix chart (for example, pie or line chart).

In the following example, we review quarterly revenue data, by product category, for a range of years (2014 - 2016, specifically). Using a bar chart for the matrix, we are able to review how gross profit for each product category shifts over time.



You can plot one value on the X axis and one value on the Y axis. For example, sales against region. You can also plot just one value for the rows or columns in the matrix chart.

Procedure: How to Create a Matrix Bar Chart

1. Launch InfoAssist+ in Chart or Visualization mode.
 - ☐ In Visualization mode, on the *Format* tab, in the *Chart Types* group, click *Bar chart*.
 - ☐ In Visualization mode, on the *Home* tab, in the *Visual* group, click *Change* and click *Bar chart*.
2. Drag data fields to the canvas or to the Query field containers to add them to your chart. The following Query field containers must be populated for this chart:
 - ☐ Vertical Axis - one or more data fields
 - ☐ Horizontal Axis - one data field
 - ☐ Matrix Rows - one data field
 - ☐ Matrix Columns - one data field

Note: You can also double-click a data field to add it to your Query field containers.

The matrix bar chart displays on the canvas. You can add additional fields for comparative purposes. In Visualization mode, you can also view underlying data by hovering over any particular point on the matrix bar chart.

Procedure: How to Create a Matrix Line Chart

1. Launch InfoAssist+ in Chart or Visualization mode.
 - ☐ In Chart mode, on the *Format* tab, in the *Chart Types* group, click *Line chart*.
 - ☐ In Visualization mode, on the *Home* tab, in the *Visual* group, click *Change* and click *Line chart*.
2. Drag data fields to the canvas or to the Query field containers to add them to your chart. The following Query field containers must be populated for this chart:
 - ☐ Vertical Axis - one or more data fields
 - ☐ Horizontal Axis - one data field
 - ☐ Matrix Rows - one data field
 - ☐ Matrix Columns - one data field

Note: You can also double-click a data field to add it to your Query field containers.

The matrix line chart displays on the canvas. You can add additional fields for comparative purposes. In visualizations mode, you can also view underlying data by hovering over any particular point on the matrix line chart.

Related Information:

- ☐ Working with Visualizations

Procedure: How to Create a Matrix Area Chart

1. Launch InfoAssist+ in Chart or Visualization mode.
 - ☐ In Chart mode, on the *Format* tab, in the *Chart Types* group, click *Area chart*.
 - ☐ In Visualization mode, on the *Home* tab, in the *Visual* group, click *Change* and click *Area chart*.
2. Drag data fields to the canvas or to the Query field containers to add them to your visual. The following Query field containers must be populated for this visual:
 - ☐ Vertical Axis - one or more data fields
 - ☐ Horizontal Axis - one data field
 - ☐ Matrix Rows - one data field
 - ☐ Matrix Columns - one data field

Note: You can also double-click a data field to add it to your Query field containers.

A matrix area chart displays on the canvas. You can add additional data fields for comparative purposes. In Visualization mode, you can also view underlying data by hovering over any particular point on the matrix area chart.

Procedure: How to Create a Matrix Pie Chart

1. Launch InfoAssist+ in Chart or Visualization mode.
 - ☐ In Chart mode, on the *Format* tab, in the *Chart Types* group, click *Pie chart*.
 - ☐ In Visualization mode, on the *Home* tab, in the *Visual* group, click *Change* and click *Pie chart*.
2. Drag data fields to the canvas or to the Query field containers to add them to your chart. The following Query field containers must be populated for this chart:
 - ☐ Measure - one data field

Note: Each unique measure field is represented by a separate pie chart.

- ☐ Color - one data field
- ☐ Matrix Rows - one data field
- ☐ Matrix Columns - one data field

Note: You can also double-click a data field to add it to your Query field containers.

The matrix pie chart displays on the canvas. You can add additional data fields for comparative purposes, or to create another pie chart unique to the additional measure fields you select. In Visualization mode, you can also view underlying data by hovering over any particular point on the matrix pie chart.

Procedure: How to Create a Matrix Ring Pie Chart

1. Launch InfoAssist+ in Chart or Visualization mode.
 - ☐ In Chart mode, on the *Format* tab, in the *Chart Types* group, click *Other*. In the Select a chart dialog box, click *Pie*, then click *Ring Pie*.
 - ☐ In Visualization mode, on the *Home* tab, in the *Visual* group, click *Change* and click *Line chart*.
2. Drag data fields to the canvas or to the Query field containers to add them to your chart. The following Query field containers must be populated for this chart:
 - ☐ Measure - one data field
 - ☐ Color - one data field
 - ☐ Matrix Rows - one data field
 - ☐ Matrix Columns - one data field

Note: You can also double-click a data field to add it to your Query field containers.

The matrix ring pie chart displays on the canvas. You can add additional fields for comparative purposes, or to create another pie chart unique to the additional measure fields you select. In Visualization mode, you can also view underlying data by hovering over any particular point on the matrix ring pie chart.

Procedure: How to Create a Matrix Circular Gauge

1. Launch InfoAssist+ in Chart or Visualization mode.
 - ☐ In Chart mode, on the *Format* tab, in the *Chart Types* group, click *Other*. In the Select a chart dialog box, click *Special*, then click *Gauge*.
 - ☐ In Visualization mode, on the *Home* tab, in the *Visual* group, click *Change* and click *Gauge*.
2. Drag data fields to the canvas or to the Query field containers to add them to your chart. The following query field containers must be populated for this chart:
 - ☐ Measure - one data field. Data in this category is used to indicate the value of the selected measure, which displays within the gauge.

Note: Since the gauge relies on a constant (measure field), each intersection of the matrix chart is calculated using that measure along with the various matrix rows and columns in the matrix chart.
 - ☐ Matrix Rows - one data field.
 - ☐ Matrix Columns - one data field.
 - ☐ Tooltip - one or more data fields. The fields that you add provide you with the ability to review additional related, underlying data for different measures. Tooltips are optional.

Note: You can also double-click a data field to add it to your Query field containers.

The matrix circular gauge displays on the canvas. You can select additional measure fields for which to include in the tooltip.

Using Active Technologies

This topic provides an overview of Active Technologies and discusses security and active cache processing. It includes additional information about the features of the product that will help you use it. It is intended for administrators and developers who are responsible for creating active reports, charts, and dashboards.

This topic also describes the features of an Active Technologies report, which is a report that is enabled to use the full capabilities of Active Technologies. An Active Technologies report is also called an active report.

Active Technologies Report Overview

An active report is a report that is designed for offline analysis. When using an active report, you can:

- ☐ Interact with the data, using analysis options similar to those found in an Excel[®] workbook, without any connection to a server. Analysis options include filtering, sorting, charting, and much more.
- ☐ Work offline without any additional plug-ins or programs. An active report is a self-contained report, meaning that it contains all the data and JavaScript[®] within the HTML output file. Packaging the data and the interactive functions in the HTML file also makes the output highly compressible for email and transparent to security systems.
- ☐ Save the report on a local machine with active report functionality. Since no connection to a server is required to view the data or use the analysis options, you can save and use the report anywhere.

Performance may vary across browsers due to browser-specific memory limitations. For very large reports, Internet Explorer[®] may produce an error. For more information, refer to the Microsoft[®] website.

When working with an active report, you can:

- ☐ Filter or highlight data.
- ☐ Sort data within any column in ascending or descending order.
- ☐ Apply calculations to columns and choose the location at which to display results.
- ☐ Control the display of data by hiding columns, freezing columns, limiting the number of rows per page, and using graphic visualization to compare column values.
- ☐ Create a variety of simple or advanced charts (pie, line, bar, or scatter) and Rollup Tables.
- ☐ Apply a global filter to multiple reports within the same HTML page.
- ☐ Export report data and chart data.
- ☐ Restore original report settings.
- ☐ Run active reports on your mobile device with the Opera browser (Version 8.60 U2 or higher) installed. See the Opera website for a list of supported devices.
- ☐ Run active reports on your iPhone[®] mobile device. For the best performance results, it is recommended that you set a maximum of 500 records for a mobile report.

Some active report functionality is drag and drop based, and thus not supported with iPhone.

The following image shows an HTML active report. The pop-up menu is open for the Sale Unit(s) column, with the Calculate Avg operator selected.

Store Business Region	Store Business Sub Region	Revenue	Sale Unit(s)
EMEA	Africa	\$	Sort Ascending 31
	Asia	\$26,4	Sort Descending 07
	Europe	\$396,2	Filter 76
North America	Canada	\$51,1	Calculate
	East	\$39,0	Chart
	Mexico	\$12,9	Rollup
	Midwest	\$60,9	Pivot (Cross Tab
	Northeast	\$1,5	Visualize
	South	\$77,4	Sum
	Southeast	\$4,4	Avg
Oceania	West	\$362,3	Min
	Australia-New Zealand	\$1,2	Max
South America	SA-Port	\$25,9	Count
	SA-Span	\$1,2	Distinct
14 of 14 records, Page 1 of 1			% of Total

The following image shows the options that are available at the cell level for a report in active report format. Included are the options that reflect Auto Drill, Multi Drill, and Auto Linking functionality. For more information, see *Customizing Content*.

Product Category ▼	Cost of Goods ▼	Quantity Sold ▼
Accessories	\$89,753,898.00	511,667
Camcorder	\$104,666,857.00	455,244
Drill down to Product Subcategory		351,777
		771,934
Summary Sales Report Information Center Website		1,114,332
Auto Links ▶		105,188
		199,749
Comments		
Highlight Value		
Highlight Row		
Unhighlight All		
Filter Cell		

Security Features

You can password protect an active report. This feature restricts users from viewing the report by requiring them to enter a password before opening the report. The data is encrypted using the 256-bit Advanced Encryption Standard (AES) specification. The password is used as the key for decrypting and encrypting the data. Therefore, the password is not stored in the report, and you do not need a connection to go back to the server for password verification.

The HTML page that you receive contains both the JavaScript and the data for the report so that you can interact with the data in a disconnected mode. Internet Explorer detects the JavaScript and issues a warning. If you look at the Internet Explorer warning, it mentions explicitly the detection of active content, which is the JavaScript. The same warning appears when pop-ups are blocked in the browser.

Handling a Large Amount of Data

Because all post-retrieval processing is performed in the memory of the web browser, an active report has a processing limit of approximately 5,000 records or 100 pages of output.

Usage Notes for Active Technologies

The following applies to browser support.

ActiveX, a technology from Microsoft, is not supported in Microsoft Edge. Any Active Technologies feature that requires the use of ActiveX controls is therefore not available in Microsoft Edge. These features include the following, which are accessible in other browsers at run time, using the column drop-down menu on an active report:

- ☐ Send as E-mail (supported only in Internet Explorer)
- ☐ Save Changes (supported only in Internet Explorer)
- ☐ Export to XML (Excel) when active cache is disabled

Creating an Active Technologies Report

An active report is a self-contained report that is designed for offline analysis.

Procedure: How to Create an Active Technologies Report

With InfoAssist+ in Report mode, you can perform this procedure in Query Design view or Live Preview.

1. On the status bar, in the Output Types menu, click *active report*.
2. Populate the report with your data in one of the following ways:
 - ☐ Drag the dimension fields and measure fields onto canvas.
 - ☐ Drag the dimension fields and measure fields into the appropriate field containers in the Query pane.

Active Technologies Report Menu Options

Menu options for an active report are described in the following table.

Note: The following options described in the table require the use of ActiveX controls. Since Microsoft Edge does not support ActiveX technology, these options are not available in that browser:

- ☐ Send as E-mail
- ☐ Save Changes
- ☐ Export to XML (Excel) when active cache is disabled

Option	Definition
Sort Ascending	Sorts the column in ascending order.
Sort Descending	Sorts the column in descending order.
Filter	<p>Filters the data. Options are:</p> <ul style="list-style-type: none"><input type="checkbox"/> Equals<input type="checkbox"/> Not equal<input type="checkbox"/> Greater than<input type="checkbox"/> Greater than or equal to<input type="checkbox"/> Less than<input type="checkbox"/> Less than or equal to<input type="checkbox"/> Between<input type="checkbox"/> Not Between<input type="checkbox"/> Contains<input type="checkbox"/> Contains (match case)<input type="checkbox"/> Omits<input type="checkbox"/> Omits (match case)

Option	Definition
Calculate	<p>Calculation types that you can apply to the column:</p> <ul style="list-style-type: none"> <input type="checkbox"/> Clear <input type="checkbox"/> Clear All <input type="checkbox"/> Count <input type="checkbox"/> Distinct, which counts the number of distinct values within a field. <p>For numeric fields, you can also apply:</p> <ul style="list-style-type: none"> <input type="checkbox"/> Sum <input type="checkbox"/> Avg <input type="checkbox"/> Min <input type="checkbox"/> Max <input type="checkbox"/> % of Total
Chart	Creates an active chart from the report. Options are Pie, Line, Column, and Scatter.
Rollup	Lists the fields available to create a table.
Pivot (Cross Tab)	Lists the fields available to create a Pivot table.
Visualize	Adds visualization bars to, or removes them from, the selected column. The Visualize option is available for numeric data columns.
Hide Column	Suppresses the display of the selected column in the report.
Show Columns	<p>Lists the names of the columns that are hidden in the report, allowing you to individually restore a column.</p> <p>Select the name of a specific column in the hidden columns list to restore that column to the report.</p>

Option	Definition
Freeze Column	<p>Freezes the report at a particular point so that columns to the left of the freeze point remain in view while the user scrolls through the other report columns.</p> <p>Note: If the report can be fully viewed in the browser window, freeze is not applied. The Freeze column option is not available for expandable report (Accordion) views.</p>
Unfreeze All	Unfreezes the columns.
Grid Tool	Opens the Grid Tool, which you can use to change the column order, select multiple columns to sort ascending or descending, hide and show columns, add a calculation result to a column, and add subtotals to the active report.
Chart/Rollup Tool	Opens the Chart/Rollup Tool, which you can use to select multiple group fields to generate the chart or rollup table. The Chart/Rollup Tool contains a list of columns available in the active report to add to Group By and Measure fields. Drag the columns into the field that you want.
Pivot Tool	Opens the Pivot Tool, which you can use to select multiple group fields to generate the chart or pivot table. The Pivot Tool contains a list of columns available in the active report to add to Group By, Across, and Measure fields. Drag the columns into the field that you want.
Show Records	Opens the Show Records menu option to list the number of records available for display per page in the report. Select a number (for example, 10) to display, per page. Default displays the number of records (lines) per page that is specified in the WebFOCUS report procedure.
Comments	Options to display comments under cells or hide indicators for comments in the active report output.

Option	Definition
Send as E-mail	<p>Enables you to save the current state of the active report and send the report as email.</p> <p>Note: To use this feature, you must have ActiveX enabled in your browser security settings.</p> <p>This feature is only supported in Internet Explorer.</p>
Save Changes	<p>Saves the current state of the active report.</p> <p>When you save an active report using the browser Save as option, the report is saved in its original default state. In the browser Save as dialog box, it is recommended that you select the <i>Webpage, HTML Only</i> save option to ensure that the page is saved properly.</p> <p>Note: To use this feature, you must have ActiveX enabled in your browser security settings.</p> <p>This feature is only supported in Internet Explorer.</p>
Export	<p>When active cache is enabled, exports all records or filtered only records to HTML, CSV, Excel, or PDF.</p> <p>Note: active cache is enabled on an active report when you click <i>Pages On Demand</i> on the Format tab, in the Navigation group.</p> <p>When active cache is disabled, exports all records or filtered only records to HTML, CSV, or XML (Excel). To use this feature, you must enable ActiveX in your browser security settings.</p>
Print	Prints all records or filtered only records.
Window	Displays reports in a cascade or separate tabs.
Restore Original	Restores the active report to the default state specified in the report procedure.

Active Technologies Cell Menu Options

When you are working in active report format, the following data cell options display.

Option	Definition
Drill down	Enables you to drill down one level in the hierarchy of your data source. This option displays for reports that have Auto Drill enabled.
Drill up	Enables you to drill up one level in the hierarchy of your data source. This option displays for reports that have Auto Drill enabled.
Restore Original	Restores the active report to the default state specified in the report procedure.
Auto Links	Displays a list of target reports that are linked to the Auto Link enabled report. This option displays for reports that have Auto Linking enabled.
Comments	Enables you to add comments about data in your report. The result is an annotation that displays when you hover over it run time.
Highlight Value	Enables you to highlight a particular value in your report.
Highlight Row	Applies highlighting to the selected row in your report.
Unhighlight All	Removes any applied highlighting from values or rows in your report.
Filter Cell	Enables you to filter the output, showing only a selected row of data.
Remove Cell Filter	Removes any applied cell filters.

Configuring Active Technologies Report Options

You can configure active report options, including menu options, based on user role, through the active report options dialog box.

You can access the dialog box on the Format tab, in the Features group, by clicking the *active report options* button. The button is available when active report is selected as the output type.

The active report options dialog box contains the following tabs:

- ☐ General
- ☐ Menu Options
- ☐ Colors
- ☐ Advanced

General Tab

Use the General tab to set common properties specific to active reports.

The General tab contains the following options:

Display. This area contains options to set the window to cascade or tabs, and options to freeze columns.

☐ **Window.** Select the window setting. The options are Cascade and Tabs.

☐ **Freeze Columns.** Select the columns you would like to freeze. You can also select None.

Page Options. This area contains options to set the number of records per page, enable the display of page information, edit the alignment, and set the location of the page information.

☐ **Records Per Page.** Select or type the number of records that you would like to display per page. The default value is 57.

☐ **Display Page Information.** Select this option to display page navigation information. Clear this option to disable the display of page navigation information.

☐ **Alignment.** Click the appropriate button to set the alignment of the page navigation information. Options are Left, Center, and Right.

☐ **Location.** Select the location for the page navigation information. The options are Top Row and Bottom Row.

Menu Options Tab

Use the Menu Options tab to select a user type and which options to display in the menu.

The Menu Options tab contains the following options:

☐ **User Type.** The options are Power, Analyst, Basic, and Custom.

☐ **Power.** This is the default user type. It enables all functionality.

- ☐ **Analyst.** This user type has the following functionality: Show Records, Freeze, Hide/Unhide, Export, Sorting, Pivot, Filter, Calculations, Chart, Visualize, Restore Original, Save Changes, and Accordion.
- ☐ **Basic.** This user type has the following functionality: Show Records, Freeze, Hide/Unhide, Sorting, Filter, Calculations, Visualize, and Restore Original.
- ☐ **Custom.** If you select a combination of options that does not match one of the existing user types (Power, Analyst, Basic), the User level name that appears in the User Type field is Custom. This is not a default user type or a selectable user type. It indicates that options for this user do not match any of the existing user types.

The options available according to user type include the following:

- ☐ **Show Records.** Shows all records or specific numbers of records.
- ☐ **Freeze.** Freezes and unfreezes columns.
- ☐ **Hide/Unhide.** Hides and shows columns.
- ☐ **Export.** Exports data to HTML, CSV, Excel, or PDF if active cache is enabled, or to HTML, CSV, or XML (Excel) if active cache is disabled.
- ☐ **Sorting.** Sorts data in ascending or descending order.
- ☐ **Pivot.** Lists the fields available to create a Pivot table.
- ☐ **Window Type.** Shows windows as cascade or tabs.
- ☐ **Send as Email.** Enables you to save the current changes and send a report as email.
- ☐ **Print.** Prints all records or filtered-only records.
- ☐ **Advanced Tools.** Accesses the Chart/Rollup, Pivot, and Grid Tools.
- ☐ **Filter.** Opens the Filter Selection dialog box.
- ☐ **Calculations.** Performs the following calculations: Sum, Avg, Min, Max, Count, Distinct, % of Total.
- ☐ **Chart.** Converts a report to a pie, line, bar, or scatter chart.
- ☐ **Visualize.** Adds data visualization bars to a report.
- ☐ **Rollup.** Performs rollup on data.
- ☐ **Comments.** Adds comments.

- ☐ **Restore Original.** Restores the active report to the default state specified in the report procedure.
- ☐ **Save Changes.** Saves the current changes.
- ☐ **Accordion.** Produces accordion reports.
- ☐ **Grid Tool.** Opens the Grid Tool dialog box.

Colors Tab

Use the Colors tab to select colors for various objects on the report.

The Colors tab contains the following options:

Page. This area contains options to set the colors for the font and background of the page text.

- ☐ **Font.** Opens the Color dialog box, where you can select the font color.
- ☐ **Background.** Opens the Color dialog box, where you can select the background color for the page text.

Row Selection. This area contains options to set the colors that appear when you point to or select a row on the report.

- ☐ **Hover.** Opens the Color dialog box, where you can select the color that the row becomes when you hold the mouse over the row.
- ☐ **Selected.** Opens the Color dialog box, where you can select the highlight color that the row becomes when you use the highlight option.

Visual. This area contains options to set the colors for the data visualization bars.

- ☐ **Positive.** Opens the Color dialog box, where you can select the color for a data visualization bar that represents a positive number.
- ☐ **Negative.** Opens the Color dialog box, where you can select the color for a data visualization bar that represents a negative number.

Calculations. This area contains options to set the colors for values in a calculation.

- ☐ **Font.** Opens the Color dialog box, where you can select the font color for the calculation.
- ☐ **Background.** Opens the Color dialog box, where you can select the background color for the calculation.

Menu. This area contains options to change the color of the menu.

☐ **Normal**

- ☐ **Font.** Opens the Color dialog box, where you can select the color for the text of the options on the column menus.
- ☐ **Background.** Opens the Color dialog box, where you can select the background color for the column menus.
- ☐ **Border.** Opens the Color dialog box, where you can select the color for the border of the column menus.

☐ **Hover**

- ☐ **Font.** Opens the Color dialog box, where you can select the color for the text of the options on the column menus when you point to them.
- ☐ **Background.** Opens the Color dialog box, where you can select the background color that appears behind options on the column menus when you point to them.

Advanced Tab

Use the Advanced tab to control the number of rows retrieved from active cache and to add security settings.

Note: active cache is enabled when you select *active report* as the output type, and click *Pages On Demand* on the Format tab, in the Navigation group.

The Advanced tab contains the following options:

active cache. Enables a report to cache the data in a binary file and return the data to the output window in pre-set increments.

- ☐ **Rows Retrieved.** Select the number of rows retrieved in the output. The default value is 100.

Security. This area allows you to set a password to access the report and enable expiration by date or by days.

Note: When setting security options for active reports, be aware that security options can be set for each individual component on the canvas, but only one password can be set for the entire document.

Creating an Active Technologies Chart

An active chart is a chart that is designed for offline analysis. For more information, see [Active Technologies Report Overview](#) on page 417.

Procedure: **How to Create an Active Technologies Chart**

With InfoAssist+ in Chart mode, you can perform this procedure in Query Design view or Live Preview.

- 1. On the status bar, in the Output Types menu, click *active report*.
- 2. On the Format tab, in the *Chart Types* group, click the button of the chart that you want to create. Bar chart is the default.

The chart appears on the canvas.

- 3. Populate the chart with your data in one of the following ways:
 - ☐ Drag the dimension fields and measure fields onto the chart.
 - ☐ Drag the dimension fields and measure fields into the appropriate field containers in the Query pane.

Active Technologies Options for Charts

Active chart toolbar options are shown in the following image.



Menu options for an active chart are described in the following table.

Option	Definition
More Options	New. Creates a new instance of the chart. This option is available only when the chart is created from a column menu on a tabular report.
	Group By (X). Changes groups by the horizontal sort field.
	Add (Y). Adds a vertical sort field.
	X-Axis. Specifies a measure or dimension sort field. Applies to scatter charts.
	Y-Axis. Specifies a measure. Applies to scatter charts.
	Arrange By. Specifies the marker color. The marker color depends on the field assigned to the color attribute. If no field is assigned to this category, then all of the markers will be the same color. Applies to scatter charts.
	Export to. Exports to Excel, Word, or PowerPoint.
	Stacked. Stacks the risers on top of each other, with the length of each riser representing the data value. Applies to column charts.
	Top. Displays the top values. Options are Top 3, Top 5, Top 10, and Clear Top. Applies to pie charts.
	Trend. Draws a trendline and equation label for an individual series. Applies to scatter charts.
Column	Chart/Rollup Tool. Opens the Chart/Rollup Tool, which you can use to select multiple group fields in the chart or rollup table generated. The Chart/Rollup Tool contains a list of columns available in the active report and Group By and Measure sort fields. Drag the columns into the desired sort field.
	Restore Original. Restores the active report to the default state specified in the report procedure.
Pie	Displays data as a pie chart.

Option	Definition
Line	Displays data as a line chart.
Scatter	Displays data as a scatter chart.
Rollup	Displays the chart as a rollup table.
Advanced Chart	Opens the Chart/Rollup Tool.
Original Chart	Restores the active chart to the chart type specified in the report procedure.
Lock/Unlock	<p>Freezes the chart or rollup table. You can link or unlink a chart or rollup table to the filters that you have applied in your report using the Freeze Chart or Freeze Rollup icon. The icon indicates whether the report is linked to the filter (Freeze Chart or Freeze Rollup) or not (Unfreeze Chart or Unfreeze Rollup).</p> <p>This option is available only when the chart is created from a column menu on a tabular report.</p>
Aggregation	Applies the following options to a Measure field: Sum, Avg, Min, Max, Count, and Distinct. The default value is Sum.
Remove Filter	Removes a filter from a chart. You can apply a filter by pointing to or lassoing an area of the chart and then clicking the Filter Chart or Exclude from Chart option from the chart tooltip.

Creating an Active Technologies Dashboard

You can create an active dashboard by inserting multiple content types, such as reports, charts, images, and text, into a document. An active dashboard will run any report or chart using the active output format, even if the report or chart itself is not in active output format.

You can also insert active dashboard prompts into a document to act as filters for the reports and charts on the dashboard. You can cascade (chain) prompts to populate them based on the selections from the previous prompts.

The output format of the active dashboard must be active report in order to add active dashboard prompts.

Active Technologies Dashboard Prompts

The Active Dashboard Prompts group contains buttons that insert active dashboard prompts into your dashboard. This group is only visible when the output format of the dashboard is set to active report. You can access the active dashboard prompts on the Insert tab, in the active dashboard prompts group.

The following are the types of active dashboard prompts that you can use to apply filters to an active dashboard:

- ☐ **Drop Down.** Inserts a drop down prompt placeholder in the upper-left corner of the canvas.
- ☐ **List.** Inserts a list prompt placeholder in the upper-left corner of the canvas.
- ☐ **Checkbox.** Inserts a check box prompt placeholder in the upper-left corner of the canvas.
- ☐ **Radio Button.** Inserts a radio button prompt placeholder in the upper-left corner of the canvas.
- ☐ **Text.** Inserts a text area prompt placeholder in the upper-left corner of the canvas.

Note: The display of values populated in active dashboard prompts is dependent on the data setting. For example, if sample data is turned on, then active dashboard prompts will show sample data, such as:

WF_RETAIL1
WF_RETAIL2
WF_RETAIL3

Target Reports

When you bind a field to an active dashboard prompt, the default target report is the report from which you dragged the field. You can add or remove target reports from an active dashboard prompt through the active dashboard properties dialog box. For more information on using the active dashboard properties dialog box, see [Using Multiple Reports as Targets and Sources](#) on page 434.

A report must meet one of the following requirements to be a target report:

- ☐ The report must contain a field with the same name as the source field (actual field name or AS name).
- ☐ The Master File of the report must contain a field with the same name as the source field.

If a report is eligible to be a target report because the field has the same user-supplied title and the title is changed, the report is automatically removed as a target.

Procedure: How to Add an Active Technologies Dashboard Prompt to a Dashboard

This procedure describes how to begin to create a dashboard by creating one report and binding a single prompt to one of the fields of the report.

With InfoAssist+ in Document mode:

1. On the status bar, in the Output Types menu, accept the default output type (active report).
2. On the Insert tab, in the Reports group, click *Report*.
A placeholder appears on the canvas.
3. Drag fields onto the canvas, or into the Query pane, to create the report and start building the dashboard.
4. On the Insert tab, in the *active dashboard prompts* group, select a dashboard prompt to insert into the document.

An active dashboard prompt appears in the upper-left corner of the canvas. If the report is located in the upper-left corner of the canvas, you will have to drag the prompt off the report.

5. Select the report and bind one of its data source fields to the prompt in one of the following ways:
 - ☐ **Query pane:** Select the report. From the Query pane, drag the field that you want to bind onto the prompt.
 - ☐ **Report on the canvas:** Click the report on the canvas. You can now edit it. Highlight the column that contains the data that you want and drag it onto the prompt.

Once you have bound the field to the prompt, the values of the field appear in the prompt.

Note: Once an active dashboard prompt is added to the canvas, the document is locked in an active output format. You cannot change the active report format if there are prompts present on the canvas. To switch to a non-active output format, you must remove all prompts.

Using Multiple Reports as Targets and Sources

You can add multiple reports and charts to an active dashboard. Each report can have multiple prompts associated with it.

Procedure: How to Build a Dashboard With Multiple Reports as Targets and Sources

The following procedure describes how to set up active dashboard prompts for two reports on a dashboard. In the example that is used, the first report contains information about the categories of electronics products sold in various regions. The Product,Category field will be bound to a group of radio buttons. Each radio button will represent a particular product category of electronics. When you select a radio button for a product category, for example, Accessories, the report will be filtered by your selection.

The second report contains information about the gender and geographic location of electronics consumers. The Gender field will be bound to a drop-down list. The list will display the values, F (female) and M (male). When you select a gender from the drop-down list, the report will be filtered by your selection.

1. Open InfoAssist+ in Document mode using the wf_retail_lite Master File.
2. Create an active dashboard by adding two reports with the following components, respectively:

Report 1:

- ☐ Product,Category
- ☐ Store,Business,Region
- ☐ Discount
- ☐ Gross Profit

Report 2:

- ☐ Gender
- ☐ Customer, Continent
- ☐ Product, Category

3. On the Insert tab, in the Active Dashboard Prompts group, add the following active dashboard prompts to the dashboard, positioning them relative to each respective report.

☐ **Radio Button:** This prompt will be used for Report1.

☐ **Drop Down:** This prompt will be used for Report2.

For more information on working with active dashboard prompts, see [How to Add an Active Technologies Dashboard Prompt to a Dashboard](#) on page 434.

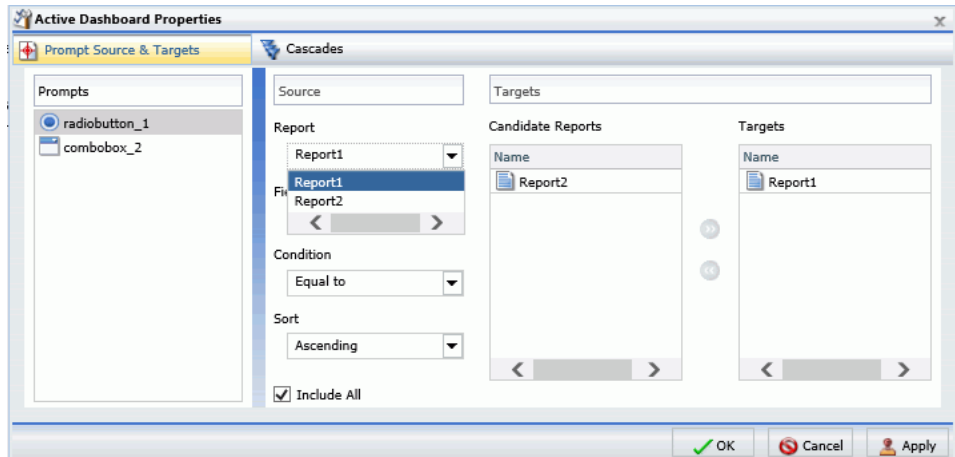
4. Right-click the radio button active dashboard prompt for which you want to bind a field to and click *Properties*.

The active dashboard properties dialog box opens.

The Prompts list displays the two prompts (for example, radiobutton_1 and radiobutton_2) that were added to the dashboard in step 3.

5. From the Report drop-down menu, select the report that contains the field to which you want to bind an active dashboard prompt.

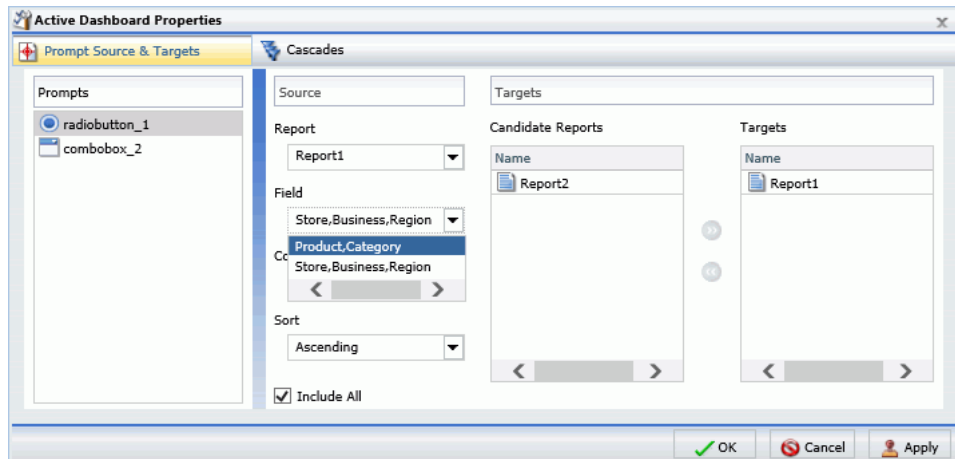
In this example, the radio button list (radiobutton_1) has been selected as the prompt for the region report (Report1), as shown in the following image.



The next step describes how to bind the Product,Category field from the region report to the radio button list to filter that report.

6. From the Field drop-down menu, select the field to which you want to bind the active dashboard prompt.

In this example, the Product,Category field has been selected for the radio buttons list (radiobutton_1), as shown in the following image.



Note: You can optionally specify an ascending or descending sort order for the current scenario.

7. Click *OK*.

The prompt is now bound to the field on the dashboard.

In the following image, the radio buttons list is bound to the Product,Category field. It displays all product categories by which a user can filter the report.

<input checked="" type="radio"/> [All]	Product Category	Store Business Region	Discount	Gross Profit
<input type="radio"/> Accessories	Accessories	EMEA	\$2,437,956.41	\$15,898,776.98
<input type="radio"/> Camcorder		North America	\$3,416,809.19	\$22,879,859.37
<input type="radio"/> Computers		South America	\$153,257.58	\$1,027,460.88
<input type="radio"/> Media Player	Camcorder	EMEA	\$2,840,963.46	\$19,928,603.81
<input type="radio"/> Stereo Systems		North America	\$4,075,160.76	\$28,304,171.49
		South America	\$185,793.84	\$1,309,721.34
	Computers	EMEA	\$1,700,675.30	\$12,147,362.23
		North America	\$2,999,148.76	\$20,556,914.70
		South America	\$100,862.80	\$753,505.60
	Media Player	EMEA	\$4,678,461.09	\$22,897,933.25
		North America	\$6,522,133.78	\$31,364,927.54
		South America	\$304,865.09	\$1,499,928.75
	Stereo Systems	EMEA	\$5,428,981.57	\$34,639,882.85
		North America	\$7,744,502.52	\$49,228,757.29
		South America	\$341,022.82	\$2,214,130.64
	Televisions	EMEA	\$1,518,099.74	\$6,990,404.98
		North America	\$2,013,911.63	\$9,358,461.12
		South America	\$102,362.82	\$462,120.21
	Video Production	EMEA	\$1,066,117.73	\$7,195,930.82
		North America	\$1,552,040.11	\$10,272,127.15
		South America	\$73,964.80	\$460,856.91

21 of 21 records, Page 1 of 1

The following steps describe how to bind the Gender field in the gender report (Report2) to the drop-down list prompt.

- Right-click the Drop Down active dashboard prompt for which you want to bind a field and click *Properties*.

The active dashboard properties dialog box opens again.

Notice that combobox_2, the prompt selected on the dashboard, is selected in the Prompts list.

9. From the Report drop-down menu, select the report (Report2) that contains the field to which you want to bind an active dashboard prompt.

The next step describes how to bind the Gender field from the gender report to the drop-down list to filter that report.

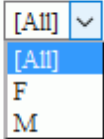
10. From the Field drop-down menu, select the field (Gender) to which you want to bind the active dashboard prompt.

Once the Gender field has been selected, Report2 (gender report) appears in the Targets list and Report1 (region report) appears in the Candidate Reports list.

Note: To move a report from the Candidate Reports list box to the Targets list box, select it and click the *Add to List* arrow. To remove a report from the Targets list box, select it and click the *Remove from List* arrow. You can select multiple reports by holding down the Ctrl key and clicking each one.

11. Click OK.

The prompt is now bound to the field on the dashboard. You can now filter the gender report by female or male, as shown in the following image.



Gender ▼	Customer Continent ▼	Product Category ▼
F	Africa	Accessories
		Camcorder
		Computers
		Media Player
		Stereo Systems
		Televisions
		Video Production
	Asia	Accessories
		Camcorder
		Computers
		Media Player
		Stereo Systems
		Televisions
		Video Production

The final active dashboard displays, as shown in the following image.

The screenshot shows a Tableau active dashboard. On the left, there is a list of filters: [All], Accessories, Camcorder, Computers, Media Player, Stereo Systems, and Televisions. The main table displays sales data with columns: Product Category, Store Business Region, Discount, and Gross Profit. The data is filtered by Gender (F, M) and Customer Continent (Africa, Asia). The Product Category filter is set to 'All'. The table shows 28 records, with the first 10 rows displayed. The bottom of the table indicates '28 of 28 records, Page 1 of 1'.

Product Category	Store Business Region	Discount	Gross Profit
Accessories	EMEA	\$2,437,956.41	\$15,898,776.98
Accessories	North America	\$3,416,809.19	\$22,879,859.37
Accessories	Oceania	\$6,822.34	\$48,343.30
Accessories	South America	\$153,257.58	\$1,027,460.88
Camcorder	EMEA	\$2,840,963.46	\$19,928,603.81
Camcorder	North America	\$4,075,160.76	\$28,304,171.49
Camcorder	Oceania	\$6,490.21	\$56,348.60
Camcorder	South America	\$185,793.84	\$1,309,721.34
Computers	EMEA	\$1,700,675.30	\$12,147,362.23
Computers	North America	\$2,999,148.76	\$20,556,914.70
Computers	Oceania	\$8,223.24	\$51,035.59
Computers	South America	\$100,862.80	\$753,505.60
Media Player	EMEA	\$4,678,461.09	\$22,897,933.25
Media Player	North America	\$6,522,133.78	\$31,364,927.54
Media Player	Oceania	\$13,682.45	\$69,788.82
Media Player	South America	\$304,865.09	\$1,499,928.75
Stereo Systems	EMEA	\$5,428,981.57	\$34,639,882.85
Stereo Systems	North America	\$7,744,502.52	\$49,228,757.29
Stereo Systems	Oceania	\$16,366.66	\$98,299.74
Stereo Systems	South America	\$341,022.82	\$2,214,130.64
Televisions	EMEA	\$1,518,099.74	\$6,990,404.98
Televisions	North America	\$2,013,911.63	\$9,358,461.12
Televisions	Oceania	\$1,319.19	\$19,037.50
Televisions	South America	\$102,362.82	\$462,120.21
Video Production	EMEA	\$1,066,117.73	\$7,195,930.82
Video Production	North America	\$1,552,040.11	\$10,272,127.15
Video Production	Oceania	\$3,768.12	\$18,704.74
Video Production	South America	\$73,964.80	\$460,856.91

Procedure: How to Change the Field

You can change the field to which the active prompt is bound.

1. Create an active dashboard in Document mode, or open an existing dashboard, and bind an active prompt to a field.
2. Right-click the active dashboard prompt that you want to configure, and click *Properties*.

The active dashboard properties dialog box opens.

3. From the Field menu, select a different field.

A warning message alerts you that changing the source field for the prompt will remove the existing prompt and any dependent (child) prompts from the cascades.

4. Click *OK* to close the warning.
5. Click *OK* to close the active dashboard properties dialog box.

The active dashboard prompt is updated with the new source field.

Procedure: How to Change the Filter Condition

1. Create an active dashboard in Document mode, or open an existing dashboard, and bind an active dashboard prompt to a field, as described in [How to Add an Active Technologies Dashboard Prompt to a Dashboard](#) on page 434.
2. Right-click the active dashboard prompt that you want to work with, and from the shortcut menu, select *Properties*.

The active dashboard properties dialog box opens.

3. From the Condition drop-down menu, select the filter condition for the active dashboard prompt. The options are Equal to, Not equal to, Less than, Less than or equal to, Greater than, and Greater than or equal to.
4. Click *OK*.

The filter condition is applied to the active dashboard prompt.

Procedure: How to Add Multiple Prompts to a Dashboard

1. Create an active dashboard in Document mode, or open an existing dashboard containing at least one report, and add at least two active dashboard prompts, as described in [How to Add an Active Technologies Dashboard Prompt to a Dashboard](#) on page 434.
2. Bind the fields to prompts that you have added, as described in [How to Add an Active Technologies Dashboard Prompt to a Dashboard](#) on page 434.

Procedure: How to Cascade Prompts

When you have more than one prompt on the canvas, you can cascade prompts to populate them based on the selections of the previous prompts. Cascading prompts have a parent-child relationship, in which the parent filters the available options of the child.


An active prompt can be the parent of more than one other prompt, but cannot be a child of more than one prompt.

1. Create an active dashboard in Document mode, or open an existing dashboard, and bind at least two active prompts to fields.
2. Right-click the active dashboard prompt that you want to configure, and click *Properties*.

The active dashboard properties dialog box opens.

3. Click *Cascades*.

By default, a cascade named Cascade1 appears in the Cascades section of the active dashboard properties dialog box.

☐ You can click the *Create a new cascade*  button to create a new cascade.

☐ You can click the *Delete selected cascade*  button to delete the selected cascade.

4. Select the cascade to which you want to add prompts.
5. From the Available Prompts list box, select the prompt that you want to add.
6. Click the *Add to List* arrow to move the selected prompt to the Selected Prompts list box.

Note: You can remove prompts from the Selected Prompts list box by selecting them and clicking the *Remove from List* arrow.

7. Add any additional prompts you want to be part of the cascade by repeating steps 5 and 6.

By default, the hierarchy of the prompts is determined by the order in which they are added to the Selected Prompts list. The cascade of the prompts is from top to bottom. The prompts that come first in the Selected Prompts list are the parents of the lower prompts.

8. You can change the hierarchy of the prompts by selecting a prompt in the Selected Prompt list box and clicking the Move Up and Move Down arrows.
9. Click *OK*.

The cascade is created.

10. Run the report.

Note: If you set up more than one cascade, the cascade that you interact with last is the one that filters the report.

Using Navigation Options for Reports

When working with reports, you can use the following options to customize output display and navigation.

- ☐ **Table.** Generates standard browser output. This is the default.
- ☐ **Table of Contents.** Generates output by displaying a table of contents icon in the upper-left corner where report output typically appears. Clicking *Table of Contents* opens a menu that enables you to select (view) individual values of the first Sort By (By) field, one value at a time.

You can also select options to view the entire report or remove the table of contents.

Note:

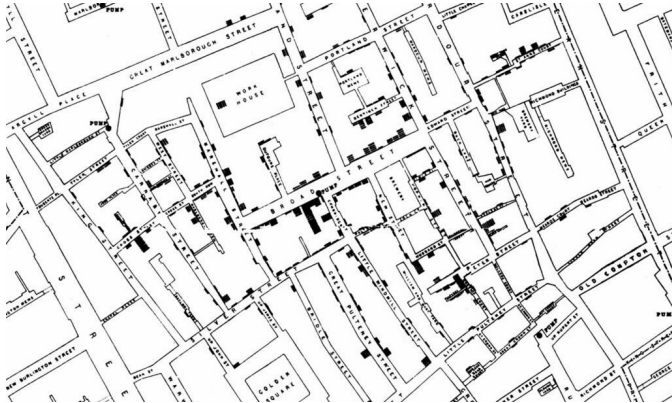
- ☐ The Table of Contents option is activated only when HTML, active report, Excel, or PowerPoint output format is selected.
- ☐ You cannot use the Table of Contents option with the Accordion feature.
- ☐ **Freeze.** Generates output with column titles that freeze (remain in view) when you scroll through pages of the report output.
- ☐ **Pages On Demand.** Provides access to two distinct features depending upon the output type that you have selected.
 - ☐ **HTML.** If you select this output type, and click *Pages on Demand*, then the report opens in the WebFOCUS Viewer.
 - ☐ **active report.** If you select this output type, and click *Pages on Demand*, then active cache is enabled. For more information on active cache, see [Using the Active Cache Option](#) on page 381.
- ☐ **Auto Drill.** Provides access to Auto Drill functionality, which enables you to navigate the hierarchy of your data at run-time. For more information, see [Using Auto Drill](#) on page 648.

Note: Auto Drill functionality is only available for the HTML and active report output formats.

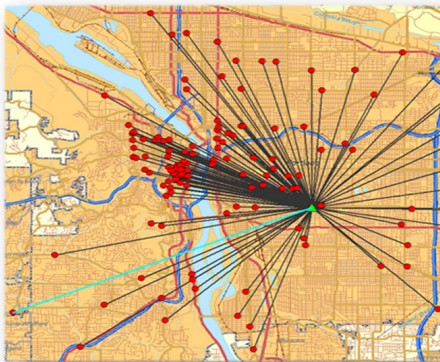
Creating Maps to Illustrate Trends

Using InfoAssist+, you can create maps to identify patterns or trends in your data. By converting data into values that can be displayed on a map, you are able to visualize scenarios, illustrate hot spots, and identify potential problem areas. For example, a law enforcement agency may use mapping functionality to identify areas of higher crime within the locations they cover. You can also use maps to determine how places are related, understand where things are located, and identify the best actions to take. By illustrating trends on a map, a decision maker can identify patterns easily, and reach conclusions sooner.

An early example of how maps can be used to illustrate trends is the case of Dr. John Snow, an epidemiologist who was one of the first to use data to map occurrences of cholera to find the cause of infection. By plotting the cholera data on a map of a town, Dr. Snow was able to visualize a trend that showed higher incidences of cholera closest to water pumps. This example is shown in the following image.



Maps also allow you to measure size, shape, and distribution to detect and quantify patterns, and even perform predictive analytics. An example of how maps can help detect and quantify patterns is the scenario in which a state agency used a WebFOCUS mapping application to solve a problem with their food stamp system. Using this application, odd food stamp redemptions, such as rounded numbers transactions, were discovered. By plotting those transactions on a map, the agency discovered that the redemptions appeared in the same geographic location. Upon further investigation, the agency identified that individuals were selling their food stamps at reduced prices, \$50 worth of food stamps for \$40 in cash, to others instead using them as intended. This map example is shown in the following image.



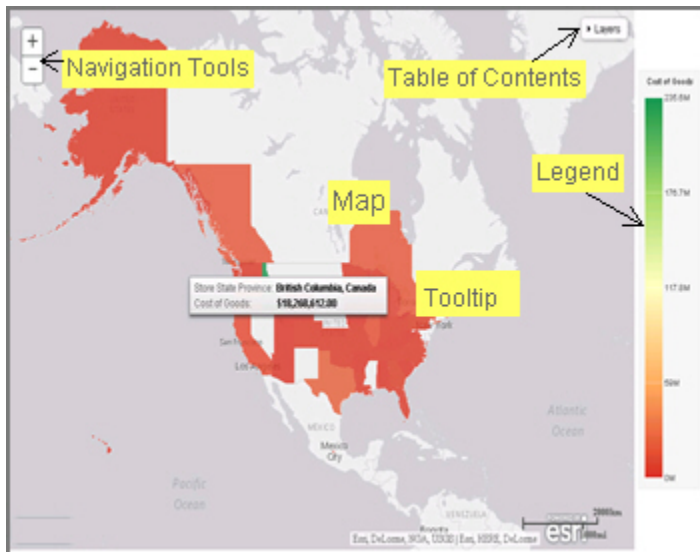
Citizen	Date	Amount
Jane Smith	01/03/2015	35.26
Jane Smith	01/05/2015	44.12
Jane Smith	01/10/2015	12.42
John Wilson	01/02/2015	100.00
John Wilson	01/08/2015	50.00
John Wilson	01/11/2015	75.00
Mary Johnson	01/03/2015	23.24
Mary Johnson	01/06/2015	14.32
Mary Johnson	01/11/2015	34.88

When working with maps, the concepts of location intelligence and business intelligence are important to understand. A Geographic Information System (GIS) captures, stores, analyzes, manages, and presents data linked to a location, while Business Intelligence (BI) relies on the conversion of raw data into meaningful information. Location intelligence is the process of analyzing data to make better business decisions. It combines GIS and BI/Analytics to allow the recognition of patterns in your data, including the visualization and discovery of geospatial outliers, which would not be easily discovered if you use the technology independently and separately.

More specifically, maps use non-intrusive GIS workflows with existing data. You can view symbol layers for data bound to a geo-location, such as state, country, and ZIP code, in an integrated map viewer. Using metrics from your data, you can also visualize geographic roles or dimensions. Geographic roles, or dimensions, can be built directly into your Metadata or assigned to a data field when you create a map.

InfoAssist+ and Esri Integration

Using WebFOCUS InfoAssist+ with the Esri integration, you can create maps that help you illustrate or identify trends, so that you can take action quickly. WebFOCUS architecture provides the framework in which this system operates. Using a Javascript map viewer, you can navigate the interface easily, as shown in the following image.



In addition, this integration utilizes the capabilities of Esri by leveraging the ArcGIS Javascript API and content. Specifically, you can integrate data into maps with published content in ArcGIS Online platform. For more information, see <http://www.esri.com/software/arcgis/arcgisonline>. Additionally, by using this integration, you can include information about demographics, spending habits, crime, and lifestyle to maps that contain your data. These maps include layers with extensive demographic or reference detail and topography and allow you to view information about people, businesses, climate, and much more.

You can create the following maps in InfoAssist+:

- ❑ **Choropleth.** A common thematic map that uses geographical measures (for example, states and countries), representing the values aerially while employing a varying intensity of colors. It is useful for visualizing location-based data, trends, and distributions across a geographic area. The color hues for Choropleth maps are dictated by the legend, based on the selected measure, enabling you to determine data concentration across your map.
- ❑ **Proportional Symbol (Bubble).** A map that represents coordinates, such as an address or intersection, using symbols of different sizes to represent any measure. These maps focus on specific areas, for which data concentrations may vary. When the data concentration is larger, the bubble will be bigger.

Both maps can be created in Chart or Visualization mode. Built-in zooming capabilities allow you to drill down to a specific geographic area of focus easily. This allows you to get a closer look at regional or local data, draw inferences, and make recommendations, without changing the initial view of your data.

In Chart mode, you can also use the Auto Drill and Auto Linking features that are available when you create charts or reports in InfoAssist+. In Visualization mode, you can also drill up and down within different levels in a data hierarchy in a map. Auto Drill allows you to navigate through the geographical hierarchy of your map data at run time. You can use this information to visualize the same measure at different geographical hierarchies, such as Countries to States and States to Cities. Auto Linking allows you to connect to related charts or reports in your environment that share similar data parameters.

Using the Esri integration in InfoAssist+, you can also add the following layers to your map:

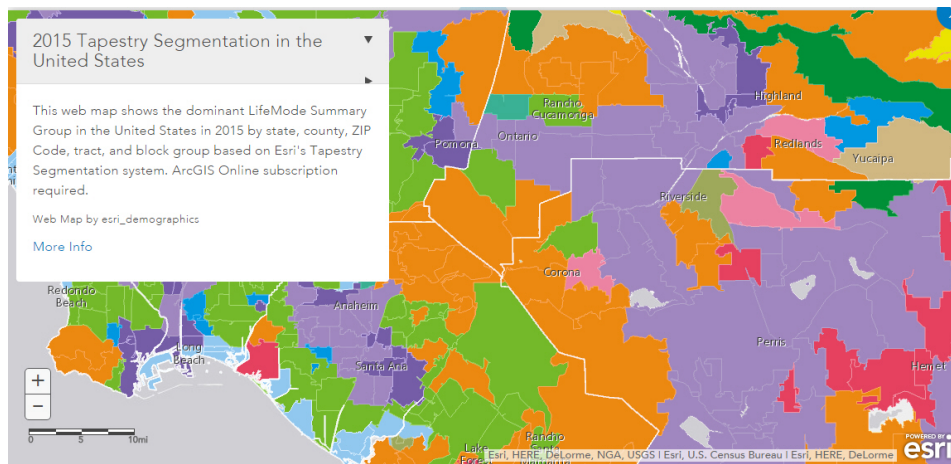
- ❑ **Backgrounds.** Display a layer that positions data as it is located, in context to other geographical features, such as streets, terrain, and imagery. Some standard Background options may combine road, aerial, and topographic data using a variety of symbols. Hosted on ArcGIS, you can change your background at any time, to review your data in a different context.

When you apply a Background to your map, its appearance changes. You can then adjust the view of your data, showing different terrain or geographical views. Backgrounds provide at least 17 levels of zoom. For more information, see <https://developers.arcgis.com/javascript/jsapi/esri.basemaps-amd.html>.

- ☐ **Reference Layers.** Display a layer of boundaries and locations that range from a continental scale to country, state, and even local neighborhood. For example, if you are viewing World data on electricity usage, you may want to add a Reference Layer that displays the borders and concentration of your data within each country.
- ☐ **Demographic Layers.** Display a layer of information about people and businesses in a specific demographic area. This includes the United States and 120 other countries. Demographic Layers are thematic maps that provide additional information about the location, such as spending habits, population, and lifestyles. You can add Demographic Layers to a map about sales data, to identify new locations for stores, based on the spending habits for a specific area.

Both mapextent and the Layers menu functionality are applied to your map when you select a Background, Reference Layer, or Demographic Layer. Mapextent is an automatic view of the map. Layers is a menu that appears on the map and provides access to options that allow you to adjust the information that is being displayed.

The map example in the following image shows the use of layers.




Note: Backgrounds, Demographic Layers, and Reference Layers can be accessed from the Format tab for maps in both Chart and Visualization mode. These layers are static, standard options that Esri provides for use with InfoAssist+, and do not change based on the data source that you select.

Creating and Customizing Maps in InfoAssist+

The following procedures provide step-by-step instructions on how to create and customize maps.

As you create your maps, you can use the following built-in map viewer features:

- ☐ You can use the plus (+) and minus (-) symbols, , within the map to zoom in and out of different areas of the map. You can also click your left mouse button to zoom in to a specific location.
- ☐ Like all HTML5 visualizations, the highlighted markers and regions on a map support drill, multi-drill, auto-linking, and informational tooltip features.
- ☐ When working with maps in Chart mode, you can use the Pan / Selection button to alternate between the Pan and Selection controls. This option is in the upper-right corner of the map.
- ☐ When working with maps in Visualization mode, you can toggle the Pan or Selection button to make a selection. The Pan control allows you to click, hold, and move the map with your mouse. The Selection control allows you to lasso a specific area of the map and select data in the map.

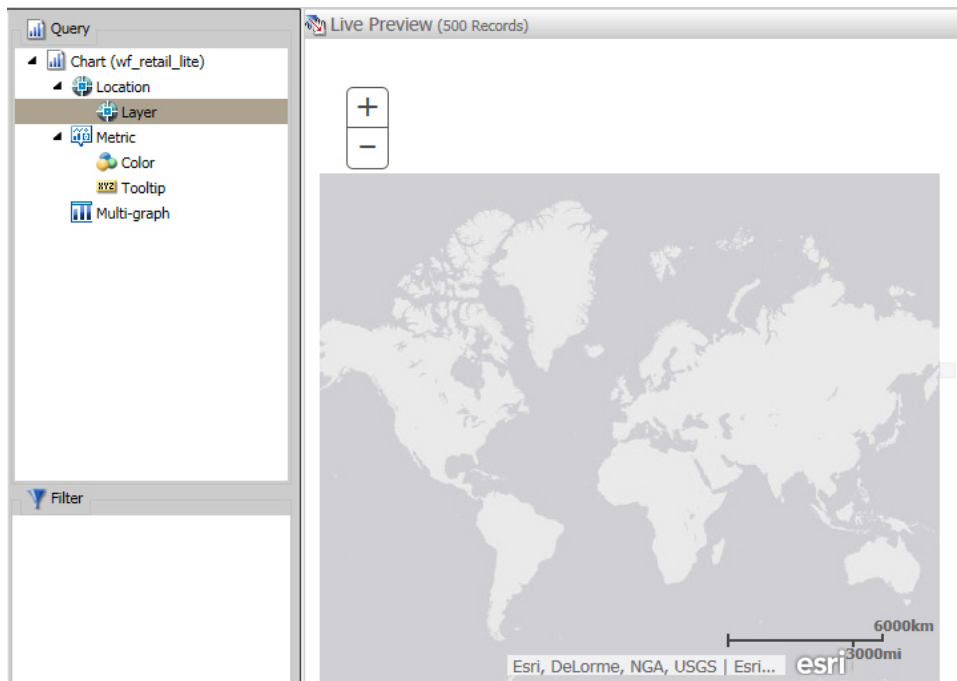
Procedure: How to Create an Esri Choropleth Map

Note: The default option of creating a map utilizes the ArcGIS Javascript API that Esri provides.


1. Launch InfoAssist+ in Chart or Visualization mode.

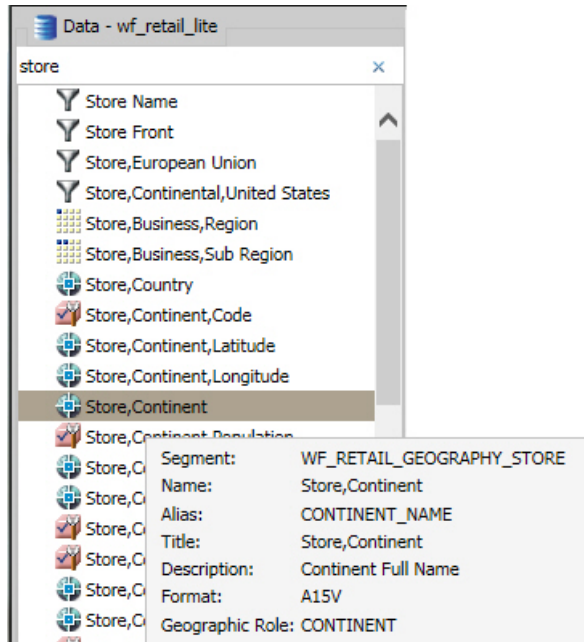
- ☐ In Chart mode, on the *Format* tab, in the *Chart Types* group, click *Choropleth*.
- ☐ In Visualization mode, on the *Home* tab, in the *Visual* group, click *Change* and click *Choropleth*.

A blank map displays and the Layer field container is enabled, as shown in the following image.



2. Add a Geolocation field to the Layer field container.

This field, which already has a geographic role assigned, is denoted with a Layer icon, , in the Data panel, as shown in the following image. You can also hover over a data field to view the geographic role assignment.



For more information, see [Geographic Roles](#) on page 464.

The canvas refreshes, and your map displays.

3. Before saving your map, to add insight, you can also do following:

- ☐ Click *Run*, to preview your map.
- ☐ Add a measure or dimension to the Color field container, to color your chart by that underlying data value. When you add a measure or dimension field to the Color field container, a legend displays for that data value. If you specify a dimension in the Color field container, the label changes to Color BY.
- ☐ Add a dimension or measure to the Tooltip field container, which will display tooltip information when you place your mouse over an area of the map.
- ☐ Add a Background, Demographic Layer, or Reference Layer.


4. Click Save to save your map.

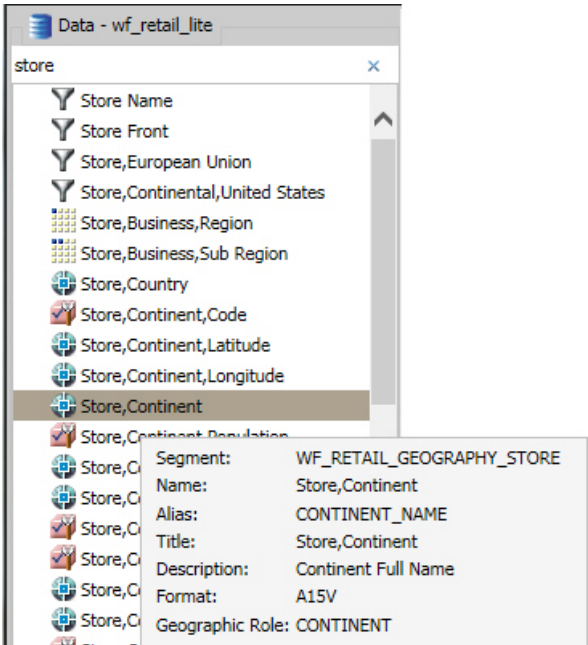
Procedure: How to Create an Esri Proportional Symbol (Bubble) Map

- 1. Launch InfoAssist+ in Chart or Visualization mode.
 - ❑ In Chart mode, on the *Format* tab, in the *Chart Types* group, click *Proportional Symbol*.
 - ❑ In Visualization mode, on the *Home* tab, in the *Visual* group, click *Change* and select *Proportional Symbol*.

A blank map displays and the Layer field container is enabled.

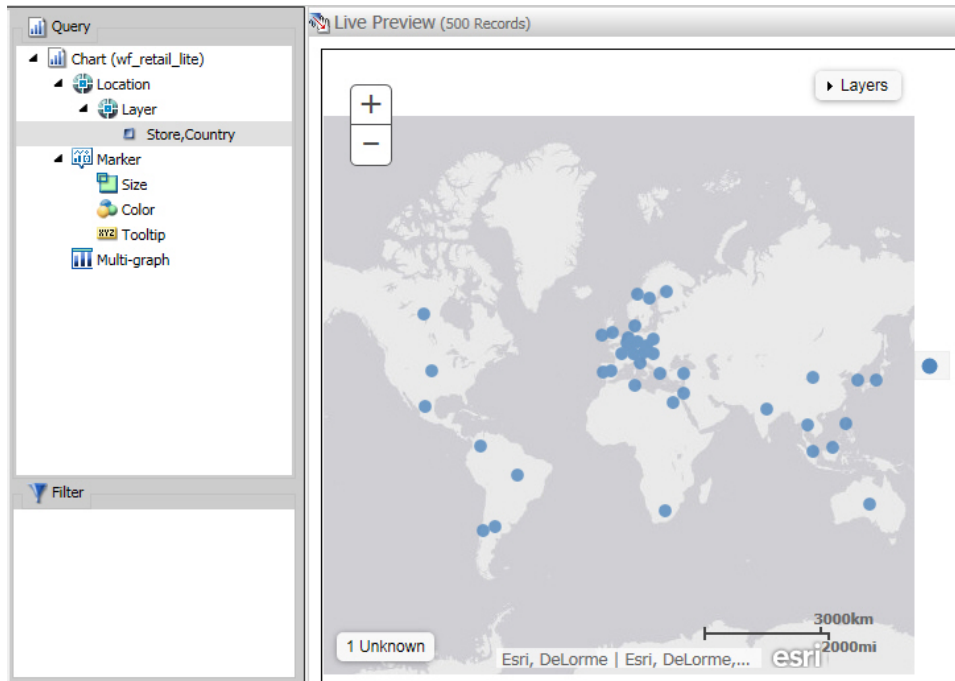
- 2. Place a data field with a defined geographic role in the Layer field container.

This field, which already has a geographic role assigned, is denoted with a Layer icon, , in the Data panel, as shown in the following image. You can also hover over a data field to view the geographic role assignment.



For more information, see [Geographic Roles](#) on page 464.

A basic bubble map displays, as shown in the following image.

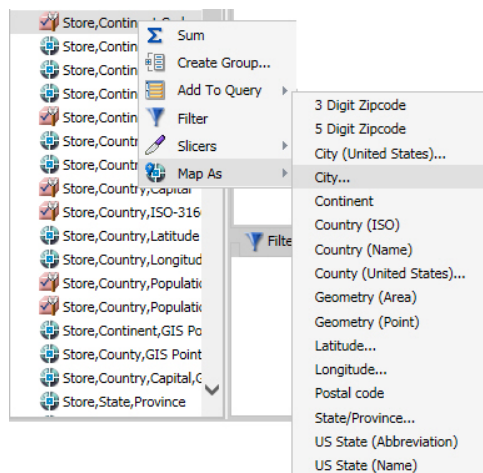


3. Before saving your map, to add insight, you can also do following:
 - ☐ Click *Run*, to preview your map.
 - ☐ Add a measure or dimension to the Color field container, to color your chart by that underlying data value.
 - ☐ Add a measure to the Size field container, to control the size of the bubbles on your map.
 - ☐ Add a measure to the Tooltip field container, to display tooltip information when you place your mouse over an area of the map at run time.
 - ☐ Add a Background, Demographic Layer, or Reference Layer.
4. Click Save to save your map.

Procedure: How to Assign a Geographic Role to a Data Field

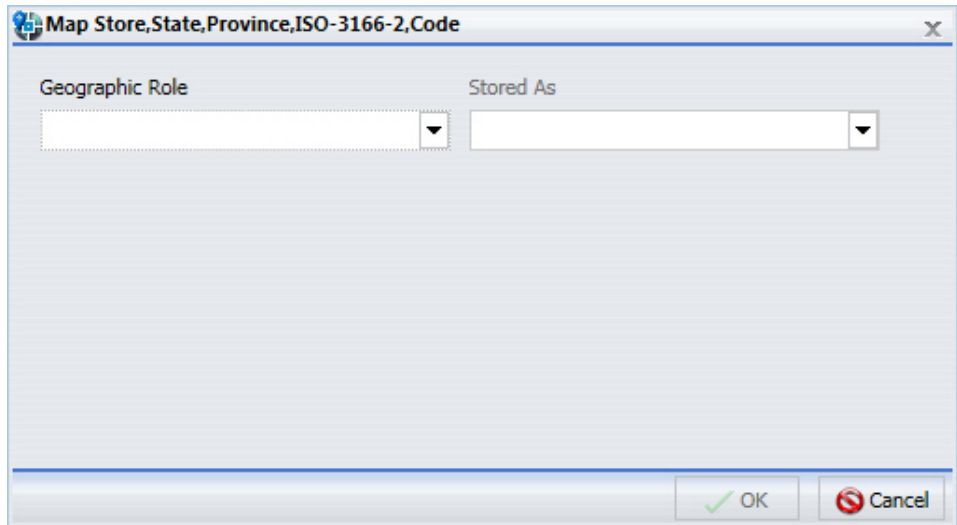
1. Launch InfoAssist+ in Chart or Visualization mode.
 - ☐ In Chart mode, on the *Format* tab, in the *Chart Types* group, click *Choropleth*.

- ❑ In Visualization mode, on the *Home* tab, in the *Visual* group, click *Change* and click *Choropleth*.
- 2. In the Data pane, select a data field without a geolocation assignment.
- 3. Perform one of the following tasks to open the Map dialog box and assign a geographic role:
 - ❑ Right-click the desired data field, click *Map As* and select a geographic role, as shown in the following image.



- ❑ Drag the desired data field into the Layer field container.

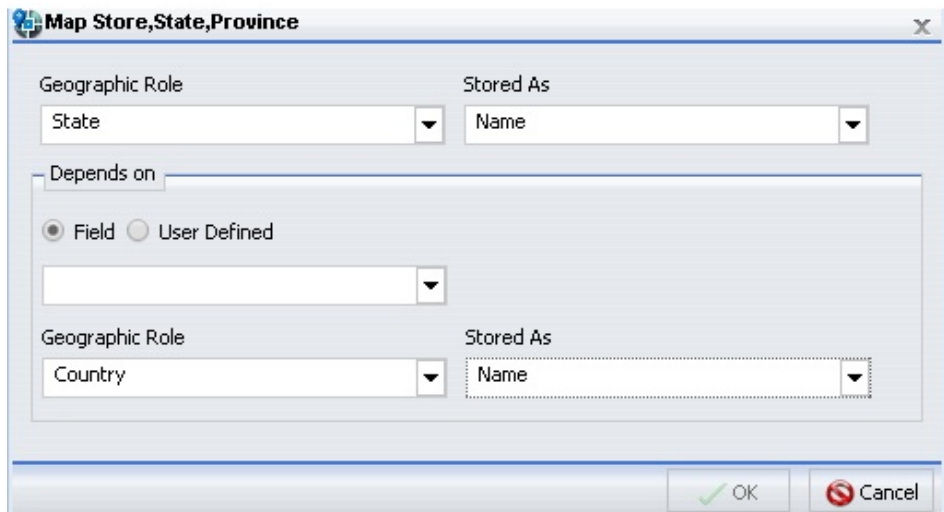
The Map dialog box displays, as shown in the following image.



4. In the Map dialog box, select a geographic role. For example, State.

Note: When specifying a geographic role, you can use Name or an ISO-2 value for countries. The ISO-2 codes are recognized worldwide, as published in http://www.iso.org/iso/country_codes

The Map dialog box refreshes and shows the Depends on section, as shown in the following image.



Note: If you used the Map As option, the Depends on section automatically displays, since a geographic role was selected at that time.

5. In the Depends on section, choose from the following options:

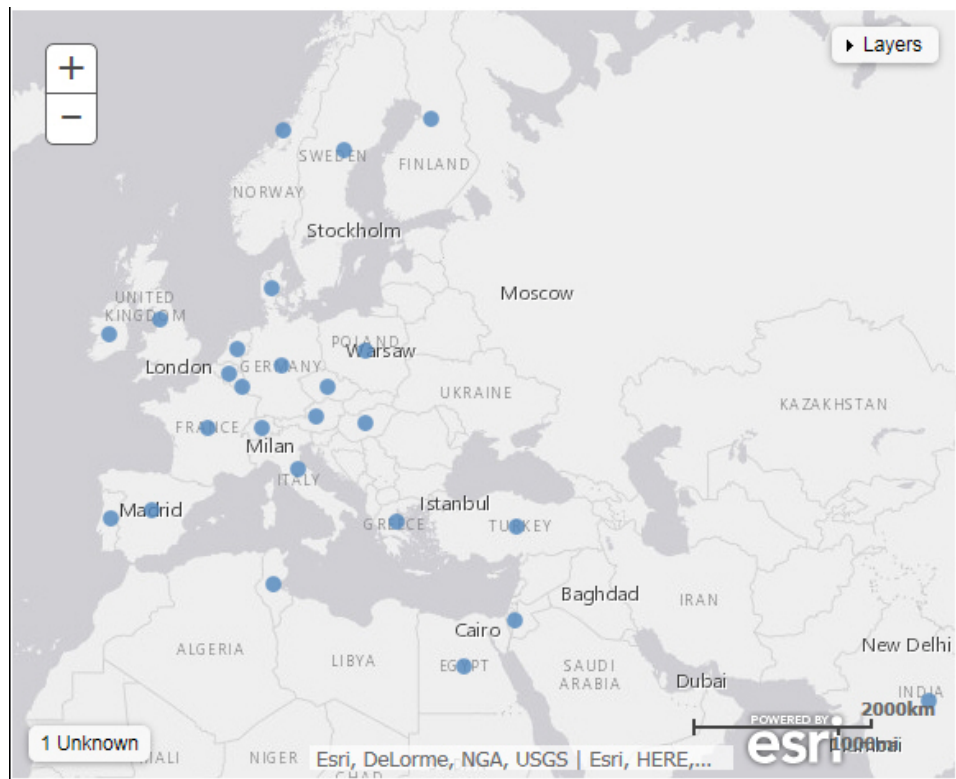
- ☐ **Field.** Identifies a specific field on which the geographic role depends. For example, you can select Country or Continent.
- ☐ **User Defined.** Enables the definition of a specific value from the data source. Selections can be as simple as a specific country. For example, you can select US.

The Geographic Role field automatically populates based on the hierarchy of your data source. For example, if your primary geographic role was State, and in your metadata hierarchy, State depends on Country, this option displays.

6. Click OK.

If you used the Map As option, you must place the data field with the defined geographic role in the Layer field container. If you placed a data field in the Layer field container and defined a geographic role, the field is automatically added to the Layer field container.

A basic map displays, as shown in the following image.



7. Before saving your map, to add insight, you can also do following:
 - ☐ Click *Run*, to preview your map.
 - ☐ Add a measure or dimension to the Color field container, to color your chart by that underlying data value.
 - ☐ Add a measure to the Size field container, to control the size of the bubbles on your map.
 - ☐ Add a measure to the Tooltip field container, to display tooltip information when you place your mouse over an area of the map at run time.
 - ☐ Add a Background, Demographic Layer, or Reference Layer.
8. Click *Save* to save your map.

Procedure: How to Change the Geographic Role of a Geolocation Field

You can change the geographic role assignment of any geolocation field using the following steps.

1. Launch InfoAssist+ in Chart or Visualization mode.
 - ☐ In Chart mode, on the *Format* tab, in the *Chart Types* group, click *Choropleth*.
 - ☐ In Visualization mode, on the *Home* tab, in the *Visual* group, click *Change* and click *Choropleth*.

2. From the Data pane, right-click a geolocation field and click *Map As*.
3. Select a geographic role.

The Map dialog box displays using the selected Geographic Role.

4. In the Map dialog box, optionally select a geographic role from the drop-down list. For example, Country.

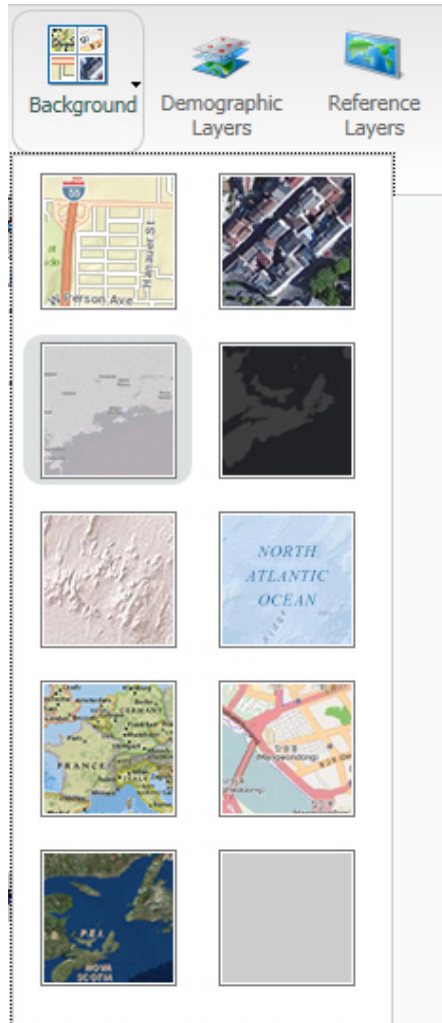
Note: This changes the selection that you made on the Map As list.

5. Accept the default value for Stored As, or choose a new value from the drop-down list, for example, ISO code. Stored As indicates how the data values are represented in the table.
6. Click *OK*.

The geographic role changes for the selected Geolocation field in the Data pane, and the map refreshes using the new geolocation that you specified.

Procedure: How to Change the Default Background of a Map

1. Create a new map or open an existing map in InfoAssist+.
2. On the *Format* tab, expand the Map group and click *Background*, as shown in the following image.



3. Select one of the following options:
 - ☐ World Street Map
 - ☐ Terrain with Labels

- ☐ Oceans Basemap
- ☐ OpenStreetMap
- ☐ World Imagery
- ☐ Light Gray Canvas
- ☐ National Geographic World Map
- ☐ Dark Gray Canvas
- ☐ None

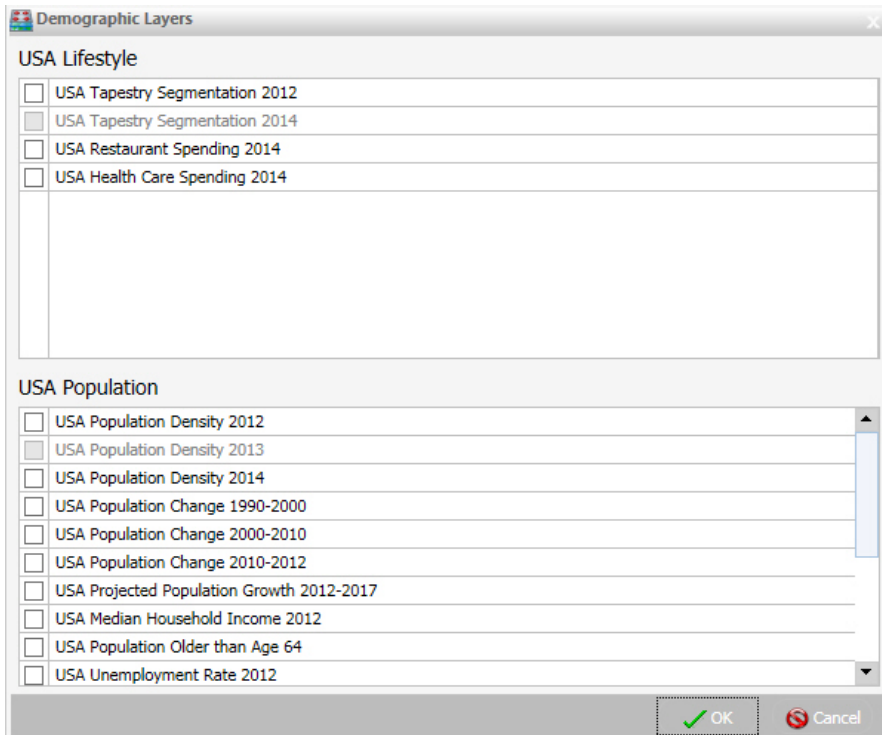
Note: The Imagery with Labels Background provides the terrain for your map, ranging from land contours to city streets.

Once you make a selection, the background of the map refreshes. You can continue to change your background until it displays the desired information.

Procedure: How to Add Demographic Layers to a Map

1. Create a new map or open an existing map in InfoAssist+.
2. On the *Format* tab, expand the Map group and click *Demographic Layers*.

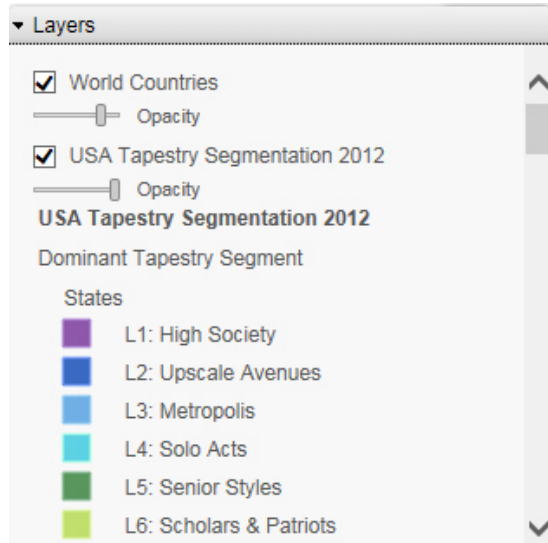
3. Select from various population and lifestyle groups, as shown in the following image.



Note: These are pre-defined demographic profiles, provided by ArcGIS. You can select multiple options in either category to gain additional insight into your data. Specifically, each Demographic Layer has its own profile and provides a layering option, when comparing values across different layers or profiles.

4. Click *OK*.

The Demographic Layers that you select are applied to your map. The map engine displays the different groups with unique hues and coloring. You can use the Table of Contents or Layers option, to toggle between the different layers that you have specified. The Layers option is shown in the following image.

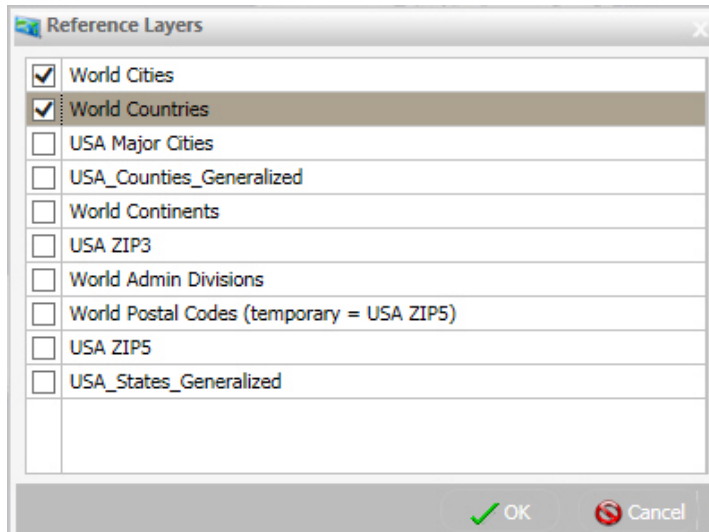


Note: You can select and clear the check boxes to enable the display of one or more Demographic Layers to compare and contrast the different demographic scenarios.

Procedure: How to Add Reference Layers

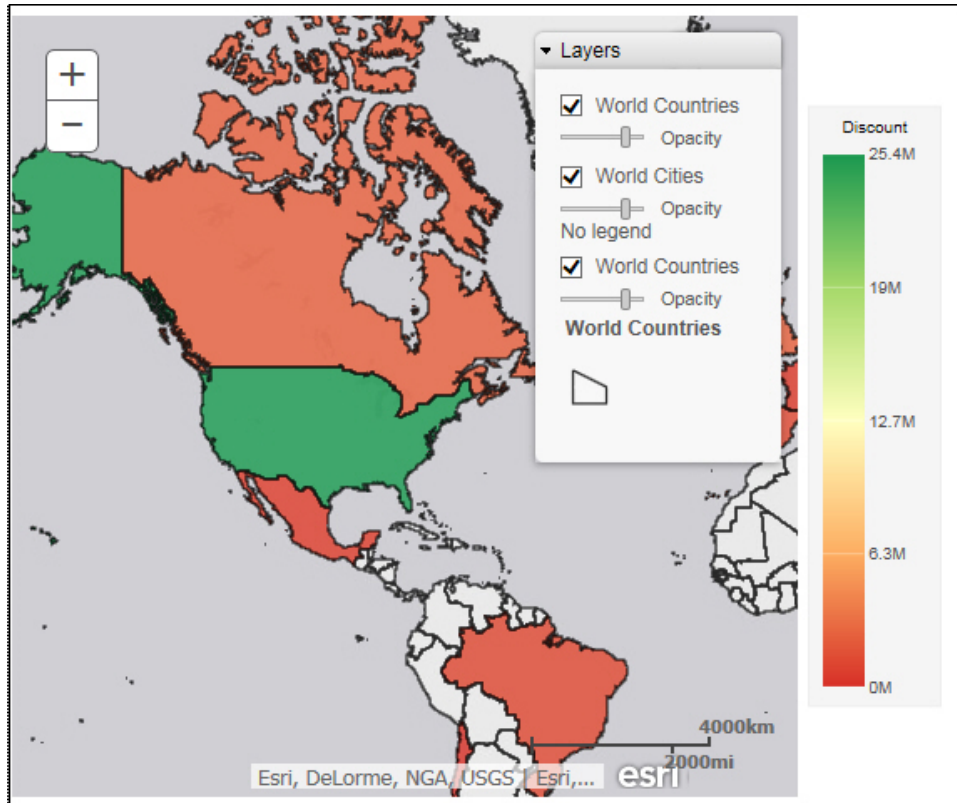
1. Create a new map or open an existing map in InfoAssist+.
2. On the *Format* tab, expand the Map group and click *Reference Layers*.

The Reference Layers dialog box displays, as shown in the following image.



3. Select one or more Reference Layers, such as World Countries, to add to your map, and then click *OK*.

Your map refreshes, and the definitions and borders of the References Layers display on the canvas. You can use the Table of Contents or Layers option, to toggle different Reference Layers in your map. These options are shown in the following image.



Reference: Query Field Containers by Map Type

This section presents the Query field containers that display for both charts and visualizations, by map type.

Query field container	Chart mode		Visualization mode	
Layer. One data field, specifically a field containing location data (for example, State).	✓	✓	✓	✓
Color. One data field.	✓	✓	✓	✓

Query field container	Chart mode		Visualization mode	
Tooltip. Up to one data field (not required).	✓	✓	✓	✓
Multi-graph. Up to one data field (not required).	✓	✓	✗	✗
Size. One data field.	✗	✓	✗	✓

Reference: Geographic Roles

This section contains information on the geographic roles that are supported for Esri maps in InfoAssist+.

Geographic Role	Description	Maps Supported
POINT OF INTEREST	Points of Interest	Proportional Symbol
CITY	World Cities	Proportional Symbol
CONTINENT	World Continents	Choropleth, Proportional Symbol
COUNTRY	World Countries	Choropleth, Proportional Symbol
COUNTRY_ISO2	World Countries (ISO2 Code)	Choropleth, Proportional Symbol
COUNTRY_ISO3	World Countries (ISO3 Code)	Choropleth, Proportional Symbol
STATE	World Admin Divisions	Choropleth, Proportional Symbol
STATE_ISO_SUB	World Admin Divisions (by ISO_SUB)	Choropleth, Proportional Symbol
USCITY	USA Major Cities	Proportional Symbol
USCITY_FIPS	USA Major Cities_FIPS	Proportional Symbol

Geographic Role	Description	Maps Supported
USCOUNTY	USA_Counties	Choropleth, Proportional Symbol
USCOUNTY_FIPS	USA_Counties_FIPS	Choropleth, Proportional Symbol
USSTATE	USA_States	Choropleth, Proportional Symbol
USSTATE_FIPS	USA_States_FIPS	Choropleth, Proportional Symbol
USSTATE_ABBR	USA_States (by abbreviation)	Choropleth, Proportional Symbol
ZIP3	USA ZIP3	Choropleth, Proportional Symbol
ZIP5	USA ZIP5	Choropleth, Proportional Symbol

The following table summarizes additional geographic role information.

Note: All of the following roles are geographic roles, with the exception of Latitude and Longitude, which are coordinates.

Role Name	Role Format	Geographic Role
Address	Full	ADDRESS_FULL
	Line	ADDRESS_LINE
City	Name	CITY
Continent	ISO-3166 code	CONTINENT_ISO2
	Name	CONTINENT

Role Name	Role Format	Geographic Role
Country	FIPS code	COUNTRY_FIPS
	ISO-3166-2 code	COUNTRY_ISO2
	ISO-3166-3 code	COUNTRY_ISO3
	Name	COUNTRY
Country (NUTS level 0)	NUTS code	NUTS0_CC
	Name	NUTS0
District (NUTS level 3)	NUTS code	NUTS3_CC
	Name	NUTS3
Geometry area		GEOMETRY_AREA
Geometry line		GEOMETRY_LINE
Geometry point		GEOMETRY_POINT
Latitude		LATITUDE
Longitude		LONGITUDE
Postal code		POSTAL-CODE
Province (NUTS level 2)	NUTS code	NUTS2_CC
	Name	NUTS2
Region (NUTS level 1)	NUTS code	NUTS1_CC
	Name	NUTS1
State	FIPS code	STATE_FIPS
	Name	STATE
US County FIPS	FIPS code	USCOUNTY_FIPS

Role Name	Role Format	Geographic Role
US city	FIPS code	USCITY_FIPS
	Name	USCITY
US county	Name	USCOUNTY
US Postal code	3 digits	ZIP3
	5 digits	ZIP5
US state	Abbreviation	USSTATE_ABBR
	FIPS code	USSTATE_FIPS
	Name	USSTATE
	US ISO subdivision code	STATE_ISO_SUB

The following table illustrates the geographic roles and their dependencies. Level 1 indicates the highest level of hierarchy and level 5 is the lowest level of hierarchy.

Region	Hierarchy Level	Geographic Role
United States	1	COUNTRY, COUNTRY_ISO_CC
	2	USSTATE, USSTATE_ABBR, USSTATE_FIPS
	3	USCOUNTY, USCOUNTY_FIPS
	4	USCITY, USCITY_FIPS
	5	ZIP3, ZIP5

Region	Hierarchy Level	Geographic Role
World	1	CONTINENT, CONTINENT_ISO_CC
	2	COUNTRY, COUNTRY_FIPS, COUNTRY_ISO_CC, COUNTRY_ISO2, COUNTRY_ISO3
	3	STATE, STATE_ISO_SUB
	4	CITY
	5	POSTAL CODE

Adding a Custom Geographic Role

Customers with Enterprise Data often have map layers that represent their territories, events, or logistical information. These are published as Map Services to either a subscription based in the Esri Cloud (ArcGIS.com) or on an internal portal. This portal is available with ArcGIS Server 10.3 installations. More information can be found at <http://server.arcgis.com/en/portal/>.

The WebFOCUS Reporting Server comes with a configuration file (geo_services.xml) that contains elements that describe all of the geographic roles, geographic hierarchies, URLs to the map services, and base maps available to the Esri map viewer. This file is located in the catalog directory under the server home directory:

```
c:\ibi\srvnn\home\catalog
```

where nn is the release of your WebFOCUS Reporting Server. For example, 82 for version 8.2.

The geographic role selections that you can make while using the InfoAssist+ Map As option are built dynamically using this configuration file. Each role definition in the configuration file, when selected in InfoAssist+, generates Metadata and a request that is sent to Esri in order to download the appropriate map and place the markers or polygons on the map.

A geographic role can be part of a hierarchy. For example, the World geographic role is at the top of a hierarchy that contains continents, countries, states, and cities. These hierarchies are also described in the geo_services.xml file. The default location is:

```
C:\ibi\srvnn\home\catalog\geo_services.xml
```

where nn is the release of your WebFOCUS Reporting Server. For example, 82 for version 8.2.

To add a custom geographic role, you must add the necessary parameters for the geography to this file.

Reference: Geographic Role Definitions

A geographic role is stored as `geo_role` element in the `geo_roles` object of the `geo_services.xml` file. A geographic role must be defined with:

- ☐ An ID that will identify the role in the configuration file.
- ☐ A format and length for the data to be returned.
- ☐ A role name.
- ☐ A display title for the role name (to appear as a selection when using the InfoAssist+ Map As option).
- ☐ An optional role format (if the role can have multiple formats, such as a name and an abbreviation).
- ☐ A display title for the format.
- ☐ A role type (GEOGRAPHY for polygons or GEOMETRY for points).
- ☐ An optional vocabulary rule element containing vocabulary elements for associating the role with a field in the metadata.

The following attributes define a geographic role.

id

Is an alphanumeric uppercase value, up to 50 characters, used to identify the geographic role.

type

Is the data type for the ID. Can be one of the following.

- ☐ **"alpha"**. For alphanumeric data, formats An or In.
- ☐ **"integer"**. For integer numeric data, format In.
- ☐ **"numeric"**. For fractional numeric data, formats Pn.m, Dn.m, or Fn.m.
- "text"**. For text data, format TXn.

value_size

Is the optional number of characters in USAGE format length (any, if not set).

role_name

Is the name of the geographic role.

role_name_title

Is the title of the geographic role, which displays when you hover over a data field with a geographic role assigned. You can also assign a geographic role, by title, using the Map As option.

role_format

Is an optional format for the geographic role. This is useful when the role can be referenced using multiple formats, such as a name, an ISO code, and an abbreviation.

role_format_title

Is an optional title for the format of the geographic role. It will be shown in parentheses along with the role title when you use the Map As option. For example, US State (abbreviation).

geo_type

Is one of the following predefined role types.

- ☐ **"geography"**. For geographic objects, such as country or city.
- ☐ **"geometry"**. For geometry objects, such as latitude, longitude, or geometry point and area.

vocabulary_rules

Is an element that consists of a group of vocabulary elements that explicitly describe column names for the geographic role. These rules will be used to select the best geographic data for the role.

Elements in a rule are connected by the Boolean logic operation OR (only one needs to be satisfied). Each vocabulary element contains words enclosed with special characters. Words in the rule element are connected by the Boolean logic operation AND (all need to be satisfied).

A word may be prefixed and/or suffixed with the percent character (%), which is a place holder for any sequence of characters. If an element contains more than one word, each word has to be prefixed by the character plus (+) or minus (-). Plus indicates that the word must be found in the column name. Minus indicates that the word must not be found in the column name.

Example: Sample Geographic Role Definitions

The following defines the State Abbreviation geographic role. The role ID is USSTATE_ABBR. The role name is USSTATE with a role format of ABBR. The titles that show when using the Map As option are US State (Abbreviation). The format is A2, and the vocabulary rules specify that the characters *state* must be present, but the characters *iso*, *capital*, and *population* must not be present. The geo type is geography, indicating that the returned data will be a geographic area.

```

<geo_role
  id="USSTATE_ABBR"
  value_size="2"
  type="alpha"
  role_name="USSTATE"
  role_name_title="US state"
  role_format="ABBR"
  role_format_title="Abbreviation"
  geo_type="geography">
<vocabulary_rules>
<vocabulary>+%state%-iso%-capital%-population%</vocabulary>
</vocabulary_rules>
</geo_role>

```

The following is a role definition for latitude values. The role ID is LATITUDE. The role name is also LATITUDE. Its format is numeric. The title that displays when you hover over a field with a geographic role assigned (or when using the Map As option) is Latitude. The geo type is geometry, indicating that the returned data will be points or areas described using points. The vocabulary rules specify that the characters *latitude* must be present.

```

<geo_role
  id="LATITUDE"
  type="numeric"
  role_name="LATITUDE"
  role_name_title="Latitude"
  geo_type="coordinate">
  <vocabulary_rules>
    <vocabulary>%latitude%</vocabulary>
  </vocabulary_rules>
</geo_role>

```

The following is the definition for the city role. The ID is CITY. The role name is also CITY. Its format is NAME. The title that displays when you hover over a field with a geographic role assigned (or when using the Map As option) is City (Name). The definition has a set of vocabulary elements. Only one of the elements in the list must be true. Therefore, the characters *city*, or *town*, or *country* plus *capital*, or *state* plus *capital* must be present.

```
<geo_role
  id="CITY"
  type="alpha"
  role_name="CITY"
  role_name_title="City"
  role_format="NAME"
  role_format_title="Name"
  geo_type="geography">
  <vocabulary_rules>
    <vocabulary>+%city%-%population%</vocabulary>
    <vocabulary>+%town%-%population%</vocabulary>
    <vocabulary>+%country%+%capital%-%population%</vocabulary>
    <vocabulary>+%state%+%capital%-%population%</vocabulary>
  </vocabulary_rules>
</geo_role>
```

Reference: Geographic Hierarchy Definitions

Some geographic roles exist as part of a hierarchy, and the data for the hierarchical roles are stored at the same map services endpoint (URL). Hierarchical role relationships are stored as hier elements in the geo_services.xml file.

Hierarchy definitions provide WebFOCUS with the information needed to use Auto Drill from a geographic role at a higher level of the geographic hierarchy to a geographic role at a lower level of the geographic hierarchy.

Each hierarchy has the name of hierarchy (attribute ID) and a group of LEV elements with the attributes level, geo_role, and, optionally, value. Not all defined roles can be used in hierarchies. The same role can be included in more than one hierarchy and may be on different hierarchical levels in each. However, the same role cannot be used more than once in the same hierarchy. Multiple geographic roles can be assigned to the same hierarchical level in a hierarchy.

Geographic hierarchies are defined with the following attributes:

id

Is a name of up to 50 alphanumeric characters used to identify the hierarchy.

level

Is a natural number (integer starting with 1 for the top level) that specifies the level of the role within the hierarchy.

geo_role

Is the ID attribute of a geographic role (geo_role element).

value

Is an alphanumeric value, up to 50 characters, predefined for this geo role in this hierarchy.

Example: Sample Geographic Hierarchy Definition

The following element defines the world hierarchy. The top level is CONTINENT, both the Name role and the ISO code role. Level 2 has four COUNTRY geographic roles, corresponding to four different country formats. Level 3 contains three state formats, level 4 contains the city name, and level 5 contains two address formats and the postal code.

```
<hier id="World">
<lev level="1" geo_role="CONTINENT"/>
<lev level="1" geo_role="CONTINENT_ISO2"/>
<lev level="2" geo_role="COUNTRY"/>
<lev level="2" geo_role="COUNTRY_FIPS"/>
<lev level="2" geo_role="COUNTRY_ISO2"/>
<lev level="2" geo_role="COUNTRY_ISO3"/>
<lev level="3" geo_role="STATE"/>
<lev level="3" geo_role="STATE_ISO_SUB"/>
<lev level="3" geo_role="STATE_FIPS"/>
<lev level="4" geo_role="CITY"/>
<lev level="5" geo_role="ADDRESS_FULL"/>
<lev level="5" geo_role="ADDRESS_LINE"/>
<lev level="5" geo_role="POSTAL_CODE"/>
</hier>
```

Reference: Adding the Federal Reserve Districts Geographic Role

These steps describe how to add the Federal Reserve Districts geographic role to the geo_services.xml file.

1. Open the geo_services.xml file. The default location is:

```
C:\ibi\sravn\home\catalog\geo_services.xml
```

where *nn* is the release of your WebFOCUS Reporting Server. For example, 82 for version 8.2.

2. Add the role to the end of the GEO_ROLES object:

```
<geo_role id="FED-DIST" value_size="50" type="alpha"
role_name="FEDDIST"
role_name_title="FED District" role_format="FR_Distric"
role_format_title="FED District Name" geo_type="geography">
  <vocabulary_rules>
    <vocabulary>+%FR_Distric%</vocabulary>
  </vocabulary_rules>
</geo_role>
```

3. The ID is FED-DIST. The role name is also FED-DIST. Its format is FR_Distric. The title that displays when you hover over a field with a geographic role assigned (or when using the Map As option) is FED District. The definition has a vocabulary rule. The characters *FR_Distric* must be present.

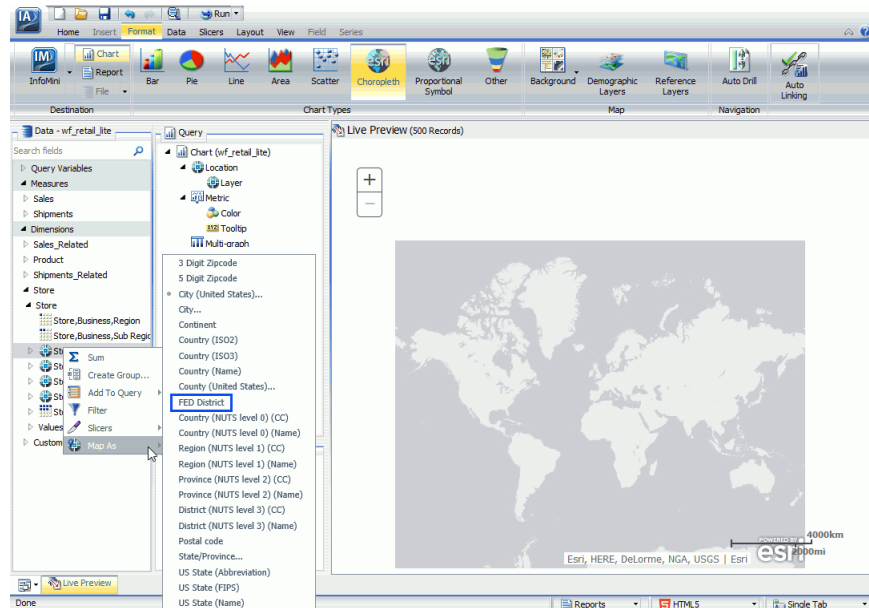
4. Add this role to the US Hierarchy:

```
<hier id="US">
  <lev level="1" value="United States" geo_role="COUNTRY"/>
  <lev level="1" value="US" geo_role="COUNTRY_ISO2"/>
  <lev level="1" value="USA" geo_role="COUNTRY_ISO3"/>
  <lev level="2" geo_role="USSTATE"/>
  <lev level="2" geo_role="USSTATE_ABBR"/>
  <lev level="2" geo_role="USSTATE_FIPS"/>
  <lev level="3" geo_role="USCOUNTY"/>
  <lev level="3" geo_role="USCOUNTY_FIPS"/>
  <lev level="4" geo_role="USCITY"/>
  <lev level="4" geo_role="USCITY_FIPS"/>
  <lev level="5" geo_role="ADDRESS_FULL"/>
  <lev level="5" geo_role="ADDRESS_LINE"/>
  <lev level="5" geo_role="ZIP3"/>
  <lev level="5" geo_role="ZIP5"/>
  <lev level="6" geo_role="FED-DIST"/>
</hier>
```

5. Add the URI to the map server layer for this role at the end of the URIS object:

```
<uri description="FedReserve Districts">
  <returned_geometry>GEOMETRY_AREA </returned_geometry>
  <returned_georole>FED-DIST</returned_georole>
  <url type="esri" authorization="none" synonym="">
http://services7.arcgis.com/L95Wwv90jRQ0tjAs/arcgis/rest/services/FRDISTRICTS/FeatureServer/0</url>
  <parameters>
    <parm order="1" parm_name="FR_Distric" parm_georole="FED-DIST"/>
  </parameters>
</uri>
```

You will now be able to select this role when using the Map As option the next time you start InfoAssist+, as shown in the following image.



Building InfoMini Applications

InfoMini applications are built from an InfoAssist+ report and contain a subset of InfoAssist+ functionality available at run time.

You can build an InfoMini application and provide the run-time user with the option to interact with and edit the report.

Understanding InfoMini Applications

When you create a report in InfoAssist+, you have the option to activate InfoMini. You can run a report with InfoMini activated, which creates an InfoMini application. An InfoMini application contains a subset of the functionality available in the full version of the report or chart. You can limit or expand the functionality that is available to the user at run time when you build the report in InfoAssist+.

An InfoMini application opens in its own browser window when it is run from within InfoAssist+ to test. An InfoMini application does not open in its own browser window in the BUE Portal or in any other application that you build yourself.

An InfoMini application has many of the components an InfoAssist+ report has, with the following exceptions:

- ☐ The main menu is not accessible.
- ☐ The New, Open, and View code buttons on the Quick Access Toolbar are not available.
- ☐ Certain tabs and groups are unavailable or limited.
- ☐ The status bar is not accessible.
- ☐ The navigation taskbar is not accessible.
- ☐ InfoMini does not support referencing existing procedures.

For more information on the available components and their functionality, see [InfoAssist+ Application Window](#) on page 69.

Using the InfoMini Button

The InfoMini button can be found on the Format tab, in the Destination group. You can click the *InfoMini* button to activate the InfoMini option. With the InfoMini button active, you can run a report to open the InfoMini application.

To deactivate the InfoMini option, click the *InfoMini* button again. There must be at least one option selected from the InfoMini button menu for InfoMini to be activated.

You can set the options available to the user at run time from the menu on the InfoMini button. If you select an option from the menu when the InfoMini button is inactive, the InfoMini option is activated. The options are:

- ☐ Format Tab
- ☐ Slicers Tab
- ☐ Run Immediately
- ☐ Run Deferred

When you select an option from the menu, a check mark appears next to the option. The check mark indicates the option is available for the user at run time within the InfoMini application. If you select a checked option to clear it, the check mark disappears, and the option is no longer available through the InfoMini application. If you clear all of the options from the menu, InfoMini is deactivated. All options are selected by default.

The Run Immediately option enables reports to run immediately when InfoMini first launches. You might want to clear this option so that the user can choose a format and pick slicers before running a report.

Understanding the InfoMini Layout

The Resources panel is not available in InfoMini. If no options are selected from the InfoMini button when the application runs, an error message displays, indicating that you must select at least one tab.

From the Format tab, you can access the Output Types group, with the following exceptions:

- ☐ The Destination and Features groups, which are on the Format tab in InfoAssist+, are not available in an InfoMini application.
- ☐ The Other button, which is on the Format tab, in the Chart Types group in InfoAssist+, is not available in an InfoMini application.
- ☐ The InfoMini button, which is on the Format tab, in the Destination group in InfoAssist+, is not available in an InfoMini application.

The Output Types group contains commands to create output in any of the supported formats.

For reports and charts, you also have access to the Auto Linking group. This group contains options for enabling the functions of Auto Linking, a feature that allows you to create a suite of referenceable reports and charts in your enterprise.

For more information on the functionality of the available groups on the Format tab, see [Format Tab](#) on page 77.

From the Slicers tab, you can access the Options, Record Limit, and Slicer Group groups, with the following exceptions:

- ☐ The New Group option, which is on the Slicers tab, in the Options group in InfoAssist+, is not available in an InfoMini application.
- ☐ The Update Preview option, which is on Slicers tab, in the Options group in InfoAssist+, is not available in an InfoMini application.
- ☐ The Preview list, which is on the Slicers tab, in the Record Limit group in InfoAssist+, is not available in an InfoMini application.

For more information on the functionality of the available groups on the Slicers tab, see [Working With Slicers](#) on page 670.

Creating an InfoMini Application

To create an InfoMini application in InfoAssist+, build a report as you normally would, then activate InfoMini and add the functionality you want the user to have available to them at run time. For more information on what functionality is available to InfoMini applications, see [Understanding InfoMini Applications](#) on page 475.

Procedure: How to Activate InfoMini

1. With a report or chart open, click the *Format* tab.
2. In the Destination group, click InfoMini.

Note: At least one option from the InfoMini menu must be selected in order to activate InfoMini. By default, the Format tab and Slicers tab are selected on the menu when you activate InfoMini in a new report. For more information on enabling InfoMini options, see [How to Enable and Disable InfoMini Application Options](#) on page 478.

The InfoMini button is highlighted and the InfoMini mode is activated. For more information about running an InfoMini application, see [How to Test an InfoMini Application](#) on page 478.

Procedure: How to Enable and Disable InfoMini Application Options

You can choose which options will be available at run time in an InfoMini application. By default, the Format tab and Slicers tab are selected on the menu when you activate InfoMini in a new report. For more information about the functionality of each option, see [Understanding InfoMini Applications](#) on page 475.

1. With a report or chart open, click the *Format* tab.
2. Click the arrow next to the *InfoMini* button. A menu of available tabs and options displays.

InfoMini does not have to be active for you to access the menu. When you select an option from the menu, InfoMini is activated.
3. From the menu, select any options you want to display in your InfoMini application.

Procedure: How to Test an InfoMini Application

1. With an InfoAssist+ report open, activate InfoMini as described in [How to Activate InfoMini](#) on page 478.
2. Enable the options that you want, as described in [How to Enable and Disable InfoMini Application Options](#) on page 478.
3. Run the report.

An InfoMini application opens in a new window.

Procedure: How to Interact With an InfoMini Application

With an InfoMini application open, you can edit the application using the functionality that was enabled in InfoAssist+. You have access to certain options, depending on which options were enabled.

You have the ability to alter the InfoMini application at run time. Changes to the application are not reflected on the canvas dynamically and you must run the report to see the updates.

1. Run a report with InfoMini activated, as described in [How to Test an InfoMini Application](#) on page 478.

An InfoMini application opens in a new window.

2. By default, the ribbon is hidden in an InfoMini application. To display the ribbon, do one of the following:

- ☐ Click one of the tabs (Format or Slicers).
- ☐ Click the down arrow next to the Help icon.

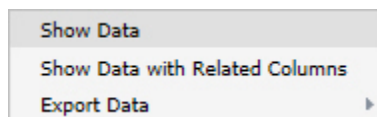
The available options on these tabs provide the same functionality as they do in InfoAssist+. You can use this embedded functionality to change the report at run time.

3. After making your changes, click *Run* to see an updated version of the report.

Viewing Data Behind Visuals

You use InfoAssist+ to analyze your data by creating or building interactive visualizations. As you develop these visualizations, you can create different views of the data, and find patterns or trends.

As you gain insight and spot patterns, you may want to share only the underlying data that comprises a specific visual with others in your enterprise. You can do this using the data options, as shown in the following image.



The Show data option provides options for you to view and share this data. The Export data option enables you to export the data specific to your visualization in a summary or detailed format. You can also review related data by using the Show Data with Related Columns option.

The Show Data with Related Columns functionality allows you to review more specific underlying data based on the fields that you have selected. Specifically, it displays data related to other fields that are part of the dimension hierarchy that you select. For example, if you selected the Product,Category dimension field, the related column data would include data from the Product,Subcategory and Model dimension fields, because these data fields are part of that dimension hierarchy.

Similar to the Show data option, the Show Data with Related Columns option provides more detailed data based on the data fields that you select. You can sort and review the data columns, which display in a separate browser window. Depending on the size and breadth of your dimension hierarchy, a multiple page report may be produced.

Note: You can export data in .xls or .csv format.

When you select the option to show data or show data with related columns, a report is generated in a separate browser window. This report is an active report, which you can review, sort, and modify using the drop-down menus that are available.

When you point to the Export Data option, you can:

- ☐ Export data in summary format, which includes totals for categories based on the data fields that you selected for your visual.
- ☐ Export more granular data based on the data values that you selected in your visual.

Note:

- ☐ When exporting data using the Summary or Data Detail options, you can save the resulting data file, which is in Microsoft® Excel® format, to your local machine for further analysis and sharing.
- ☐ The maximum number of records that can be exported is 100,000.

Procedure: How to Show the Data Behind Your Visual

1. Create a visual, such as a chart, map, or grid.
2. In the upper-right corner of the visual cell, click the down arrow.
3. In the menu that appears, click *Show Data*.

A new browser window opens. This window displays the data for your visual as a WebFOCUS active report. You can use this active report to create and share lightweight, browser-based data discovery analytics that are portable, and only require access to a browser.

Procedure: How to Show Data With Related Columns

1. Create a visual, such as a chart, map, or grid.
2. In the upper-right corner of the visual cell, click the down arrow.
3. In the menu that appears, click *Show Data with Related Columns*.

A new browser window opens. This window displays the data for your visual as a WebFOCUS active report. You can use this active report to sort and work with the underlying hierarchical data in your visual.

Procedure: How to Export the Data Behind Your Visual

1. Create a visual, such as a chart, map, or grid.
2. In the upper-right corner of the visual cell, click the down arrow.
3. In the menu that appears, point to *Export Data*, and then click one of the following:

- ☐ **Summary.** A prompt appears, asking you to open or save a Microsoft Excel file. This file is a summary of your data from a high level for a general analysis, as shown in the following image.

	A	B	C	D
1	Product Category	Product Name	Revenue	Gross Profit
2	Accessories	Headphones	\$9,341,397.65	\$2,354,397.65
3		Cycle Energy Quick with Refresh Charger	\$1,508,212.41	\$666,612.41
4		Denon AH-D5000 Over-Ear Headphones	\$9,272,133.77	\$2,477,303.77
5		Grado RS1i Reference Series Headphones	\$9,452,243.25	\$2,208,713.25
6		Universal Remote	\$14,419,020.31	\$4,401,161.31
7		Control	\$9,301,960.89	\$1,518,520.89
8		Anywhere! Kit with Tabletop Sensor	\$11,401,805.70	\$2,616,237.70
9		Anywhere! Kit for Home Theater	\$14,276,128.75	\$4,825,372.75
10		Headphones	\$8,028,218.25	\$4,049,178.25
11		Samsung OEM 2.0 Amp Travel Charger	\$2,514,622.50	\$1,303,511.50

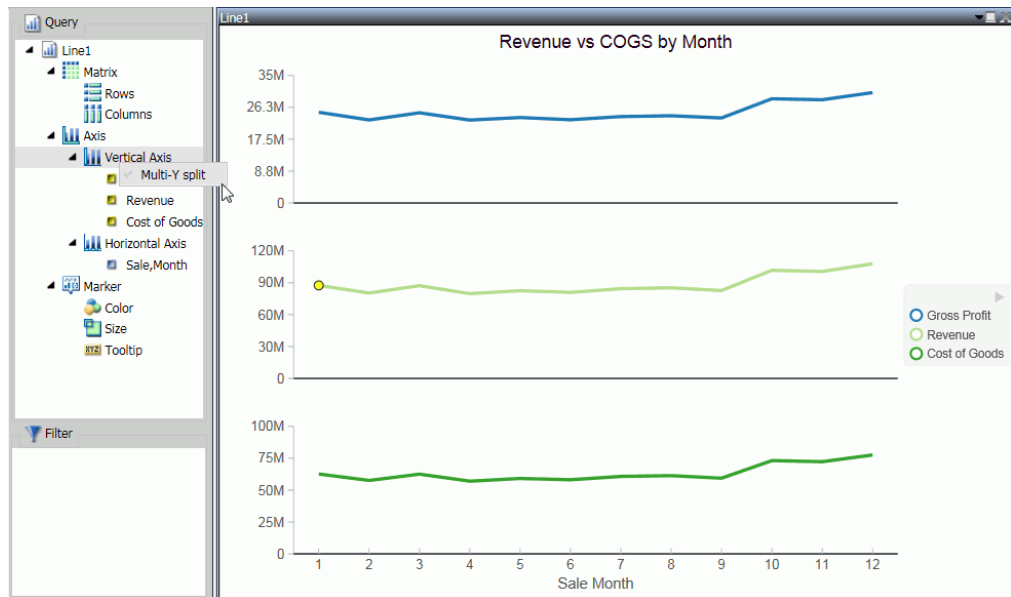
- ☐ **Data detail.** Generates a detailed report that provides specific data regarding your data analysis.

Creating Multi-Y Axis Comparative Visuals

When creating a chart with more than one measure (numeric) field, you can split the Y axis to create multiple charts, based on each unique measure field.

Note: Measure fields are selected from the Data pane and display under the Vertical Axis (Y) in the Query pane.

This functionality is useful when analyzing trends for multiple measure fields, as you can view data for each measure field separately within the same chart, as shown in the following image.



This functionality is available for Bar, Bar Stacked, Line, Area, and Area Stacked charts.

Procedure: How to Create a Multi-Y Axis Comparative Visual

1. Launch InfoAssist+ in visualization mode.
2. On the *Home* tab, in the *Visual* group, click *Change*.
3. Select one of the following visual types: Bar, Bar Stacked, Line, Area, or Area Stacked.

Note: Bar Stacked is the default visual type.

4. Add multiple measure fields to the visual. For example, Gross Profit and Revenue.
5. Add at least one dimension field to the visual. For example, Product Category.
6. In the Query pane, right-click *Vertical Axis*.
7. Click *Multi-Y split*.

The chart changes to display individual charts for each measure field.

Note: You can revert your chart to an integrated display by right-clicking *Vertical Axis* and then clicking *Multi-Y split*.

Creating HOLD Files

A HOLD file is the output of a report request stored in a file that you can use as input to another WebFOCUS procedure.

You can then create new report requests that extract data from the HOLD file, resulting in multi-step report.

HOLD files can be created to use in a report, chart, document, or visualization.

Valuable Applications of HOLD Files

A HOLD file is valuable when you want to do the following:

- ☐ Extract fields from a large data source for faster and more efficient retrieval in subsequent requests.
- ☐ Store virtual field values or summary values calculated in one request for further processing in another request.

Storing HOLD Files

HOLD files can be created for immediate use and saved temporarily or they can be stored for future and repeated use.

Output Formats for Reports

You can save a HOLD file for a report in the following formats:

- ☐ Binary (*.ftm)
- ☐ FOCUS (*.foc). For more information, see [FOCUS Format Index Fields](#) on page 487.
- ☐ Comma Delimited with Titles (*.csv)
- ☐ Plain Text (*.ftm)
- ☐ Tab Delimited (*.tab)
- ☐ Tab Delimited with Titles (*.tab)
- ☐ Database Table (*.sql)

Note: The Database Table output is only available when working against an SQL database.

- ☐ SQL Script (*.sql)
- ☐ Hyperstage (*.bht)

Note: The Hyperstage output is only available when the reporting server has a Hyperstage adapter configuration.

- ☐ XML (*.xml)
- ☐ Visual Discovery AE (*.txt)

Note: The Visual Discovery AE output option is only available with a Visual Discovery Analyst Edition license.

Creating Hold Files

This section contains examples of how you would use a HOLD file.

Note:

- ☐ Across fields are not allowed in HOLD files.
- ☐ Using the Change Title option in the Query pane, you can change the title of a field prior to creating a HOLD file. A blank space in the title will be replaced by an underscore. This functionality enables you to control the names used for the fields included in the HOLD file, and makes it easier to find the fields when creating report.
- ☐ When creating a HOLD file, the Temporary dialog box displays only those reporting server applications to which you have access.
- ☐ You can use Auto Linking when working with HOLD files. For more information, see [Using the Auto Linking Feature to Link Content](#) on page 650.

Procedure: How to Create a Tabular Report From a HOLD File

To create a tabular report from a HOLD file, begin by creating a report.

1. In the Open dialog box, select the wf_retail_lite Master File.
2. Add the following measure fields to the report:
 - ☐ Cost of Goods
 - ☐ Discount
 - ☐ Gross Profit
 - ☐ Quantity,Sold
 - ☐ Revenue

3. Add the following dimension fields to the report:

- ☐ Product Category
- ☐ Product,SubCategory
- ☐ Sale,Year

4. On the Home tab, in the Format group, click *File*.

The Temporary dialog box opens.

5. In the Temporary dialog box, name the HOLD file, keep the default file type, Binary (*.ftm), and click *Save*.
6. At the bottom of the canvas, click *Create Report*.

The custom database structure displays in the Resources panel. The canvas is returned to a default blank state, enabling the development of a new report using the HOLD file.

7. From the HOLD file, drag Quantity,Sold to the canvas.
8. On the canvas, select the Quantity,Sold column heading.
9. On the Field tab, in the Display group, click *Aggregation* and then click *First Value*.

Note: The heading changes to FST Quantity Sold.

10. Select the FST Quantity Sold heading.
11. On the Field tab, in the Display group, click *Hide Field* to hide Quantity,Sold, as it will be used in a subsequent calculation.
12. On the Data tab, in the Calculation group, click *Summary (Compute)*.
13. In the Summary Field (COMPUTE) dialog box, do the following:

- ☐ In the Format field, type D8.2%.
- ☐ Double-click the Quantity,Sold field to add it to the formula box.
- ☐ Add / 100 after the Quantity Sold field to calculate the percentage.

14. Click *OK* to close the dialog box.
15. Drag Product,Category to the By Query field container.
16. In the Query pane, select the Product,Category field.
17. On the Field tab, in the Break group, select *Subtotal* to create Subtotals on Product,Category.
18. Drag Product, Subcategory to the By Query field container.
19. Drag Sale,Year to Across.

20. On the *Home* tab, in the *Report* group, click *Row Totals*.

The final report displays.

Procedure: How to Rearrange HOLD File Components

The following procedure describes how to rearrange file components in a HOLD file.

Note: This procedure creates a binary HOLD file and a subquery to illustrate how to rearrange HOLD files. It also shows the result of this rearrangement.

1. Create a new document, using the *wf_retail_lite* Master File for the data source.
2. On the *Insert* tab, in the *Reports* group, click *Report*. Populate the report with the following fields from the *Data* pane:
 - ☐ Gross Profit
 - ☐ Quantity,Sold
 - ☐ Revenue
 - ☐ Product,Category
 - ☐ Product,Subcategory
3. On the *Home* tab, in the *Format* group, click *File* to create a HOLD file.

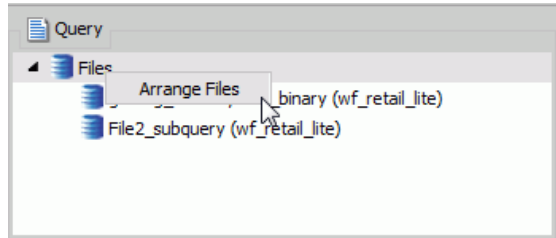
In the *Temporary* dialog box, enter a name for the file. For example, *File1_binary*.
4. Click *Save*.
5. Create a report using the HOLD file, specifying *Product,Category*, *Product,Subcategory*, and *Quantity,Sold*.
6. Next, using the following steps, add a subquery SQL script for use as a filter on the first report.
 - a. On the *Data* tab, in the *Data Source* group, click *Switch*.

Select the original master file (*wf_retail_lite.mas*).
 - b. Locate and double-click the *Product,Category* dimension field.

This creates a second report, which you can drag and resize as needed on the *Document* canvas.
 - c. Create a filter on *Product,Category*, where the product category is equal to *Televisions*.
7. With the new component selected, click the *Home* tab and in the *Format* group, click *File*.

In the *File name* field, enter *File2_subquery* and select the *SQL Script (*.sql)* format from the *file types* menu.

8. Click **Save**.
9. Rearrange the order of the HOLD files so that the `File2_subquery` is positioned above the `File1_binary` HOLD file using the following steps:
 - a. Right-click *Files* in the Query pane and click *Arrange Files*, as shown in the following image.



The Arrange Files dialog box opens.

- b. Using the Arrange Files dialog box that displays, select `File2_subquery` and click *Move Up* to move the file above `File1_binary`.
 - c. Click *OK*.
10. Edit the first report and create a filter using the subquery.
11. Click *OK* to exit the Create a filtering condition dialog box.

Your report is refreshed to reflect the filtering you have applied.

FOCUS Format Index Fields

FOCUS is the only format that supports an index field. The maximum number of fields to index is four. If the file format is FOCUS, then *Index* appears on the Query pane.

Creating a Subquery Filter Using a HOLD File

You can create a subquery using a HOLD file. A subquery is a nested query that is added to the *Where* clause of an SQL statement. A subquery is valuable because it is highly reusable.

Procedure: How to Create a Subquery Filter Using a HOLD File

This procedure describes how to create a subquery filter using a HOLD file created in the previous procedure.

1. Build a report.
2. On the **Data** tab, in the **Filter** group, click *Filter*.

The Filter dialog box opens.

3. In the Filter dialog box, from the Type drop-down menu, select *Subquery* as the filter type for the left-most part of the expression.
4. From the Subquery drop-down menu, select *In list* as the comparison operator.
5. From the list of subqueries, select the subquery that was created (in this example, File1) for the right-most part of the expression.
6. Click *OK*.

The report is filtered by the subquery that you created.

Note: To view the SQL statements generated by the request, go to the Quick Access Toolbar, open the Run drop-down menu, and select *SQL Trace*.

Creating Shortcuts and URLs

The use of shortcuts and URLs can simplify workflow and provide additional context to your analytics. The following sections explain how to use shortcuts and URLs and describe how to create these content items in your repository.

Creating Shortcuts

Authorized users can create shortcuts to repository resources and Master Files. Once a shortcut to a Master File is created, you can use it to build visualizations and reports. Once a shortcut to a repository resource is created, you can copy, delete, edit, and run this item. You can also publish and share your shortcuts to make them available to other users and groups.

Procedure: How to Create a Shortcut to a Repository Resource

1. In the Resources tree, right-click a domain or folder, point to *New*, point to *Shortcut*, and then click *To Repository Resource*.

The Create Shortcut dialog box opens.

2. Click *Browse*.

The Shortcut Target dialog box opens.

3. Navigate to the resource of your choice, such as folder, report, chart, dashboard, or visualization, and then click *OK*.

The Title field is automatically populated with the name of the original item and the word *Shortcut*.

4. Optionally, edit the Title and populate the Summary field.
5. Click *OK*.

A confirmation message indicates that your shortcut is created successfully.

6. Proceed in one of two ways:

- ☐ Click *Browse* again to create another shortcut.
- ☐ Click *Cancel* to close the Create Shortcut dialog box and begin using your new shortcut.

Procedure: How to Create a Shortcut To a Master File

1. In the Resources tree, right-click a domain or folder, point to *New*, point to *Shortcut*, and then click *To Master File*.

The Open dialog box opens.

2. Navigate to a Master File, previously generated through the Upload wizard or Connect to Data wizard, and then click *Open*.

The shortcut to the Master File appears in your selected directory in the Resources tree. You can use this shortcut to create reports and visualizations. You can also publish or share the shortcut to make it available to other users.

Creating URLs

Authorized users can create URLs to webpages and store them within the repository. These URLs can further enhance analytics, providing an additional interactive context to your data. You can run a URL in a new tab, use it in a dashboard, and add it to a personal page. You can also publish a URL to make it available to other users and groups.

Procedure: How to Create a URL

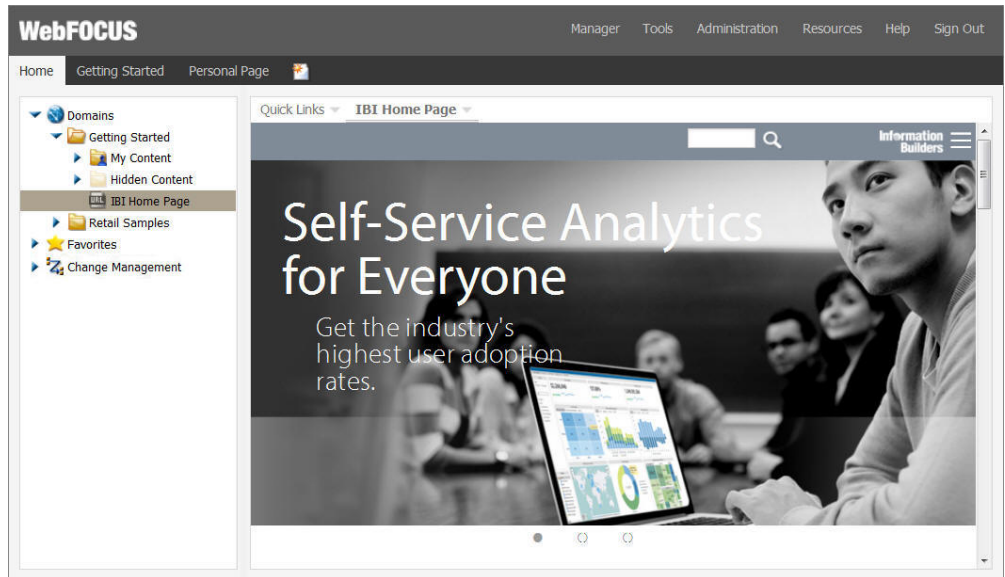
1. In the Resources tree, right-click a domain or folder, point to *New*, and then click *URL*.

The Create URL dialog box opens.

2. Populate the Title field, type the URL to your selected webpage in the URL field, and then click *OK*.

The URL appears in your selected location in the Resources tree.

The following image shows how a URL appears in the Resources tree and displays in a new tab inside a portal.

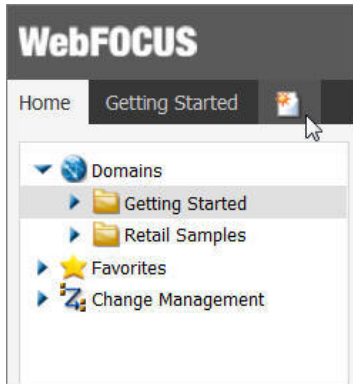


Working With the WebFOCUS BUE Portal

The first page that you see as a Manager, is the Getting Started page. If you sign in as any other user, the first page you see is the Home page. If you create a new page and sign out of the portal with the new page in focus, the new page opens by default the next time you sign in. You can create new pages in the WebFOCUS Business User Edition to display the content that is available to you.

Procedure: How to Add a New Page

Click the *New Page* icon, as shown in the following image.



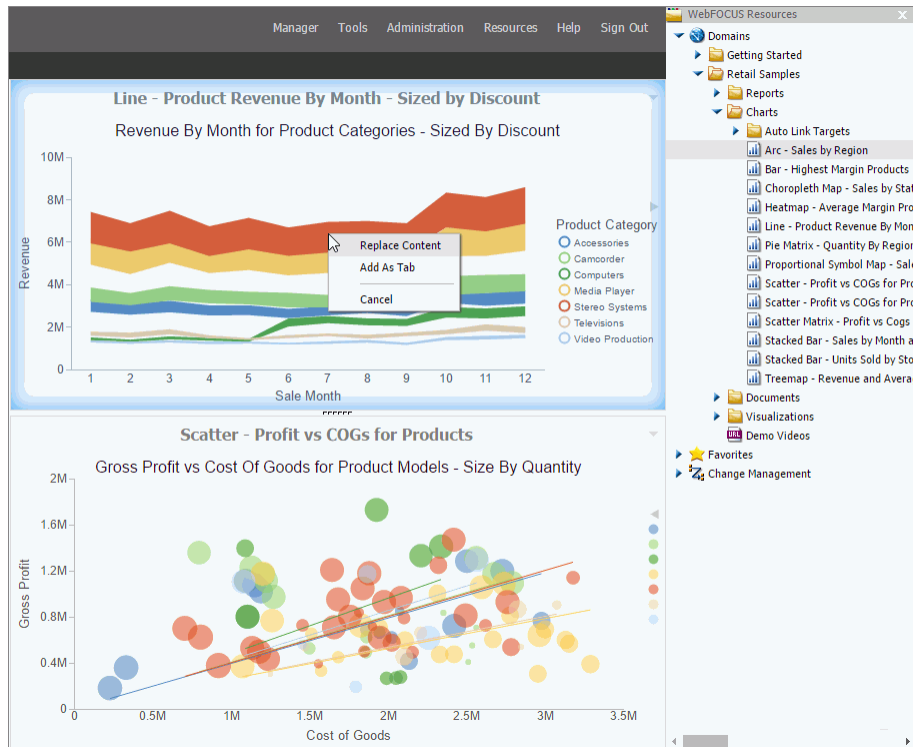
A blank page and the WebFOCUS Resources panel opens. You can drag content from the Resources tree panel to the page.

Procedure: How to Add Content to the BUE Portal

1. Navigate to the page that you want to populate with content.
2. On the Menu bar, click *Resources*.

The WebFOCUS Resources panel opens.

3. Drag an item on the page. Use the shaded area to position the item on the page, as shown in the following image.

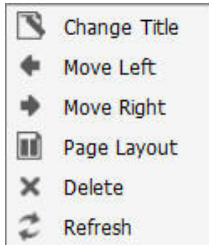


Note: When you drag one item on top of another and position your cursor in the center of the other item, you have a choice to either replace the existing item or add the new item as a new tab. If you choose the latter, a tabbed container will be automatically created. If you choose the latter, a tabbed container will be automatically created. If you multi-select several items and drag them to a page, they will display as tabs inside a single panel. You can refresh or delete each tab individually. You can rearrange content by moving it on the page with your pointer. Optionally, you can change the layout of the page by right-clicking the page title, and then clicking *Page Layout*. The default layout of the personal pages is Fluid Canvas.

4. After you finish adding content to the page, close the WebFOCUS Resources panel by clicking the *Close* icon in the upper-right corner of the panel.

Accessing Page Shortcut Menu Options

Right-click the page title to access page shortcut menu options. The shortcut menu opens, as shown in the following image.



Change Title

Renames the selected page.

Move Left

Moves the selected page to the left.

Move Right

Moves the selected page to the right.

Page Layout

Allows you to choose a page layout to align your content into the desired number of columns.

Delete

Deletes the selected page.

Note: The Welcome and Getting Started pages do not have shortcut menu options, since these are static pages for the BUE portal.

Creating Report Queries With InfoAssist+

You can create a new report query directly from Excel by accessing the WebFOCUS Quick Data Add-in. Specify connection attributes and the data source for your query, and then build your report using InfoAssist+. You can place multiple queries in the same worksheet, or spread them out over multiple worksheets in a workbook.

There are limitations with queries that overlap. However, there are data layout options available in the Query properties of Excel that can assist with overlapping queries. This behavior is governed by Excel, not WebFOCUS Quick Data.

Note: Quick Data is a WebFOCUS Business User Edition option, which requires a separate license and installation. For more information about licensing Quick Data, contact your Information Builders representative.

Procedure: How to Create a New Report Query in InfoAssist

1. Open an Excel file.
2. Select a cell in which to place the query results.
3. With the Add-Ins tab selected, click the *WebFOCUS* option in the Excel menu, and then click *Create Query*.

You can also right-click any cell and select *Create WebFOCUS Query*.

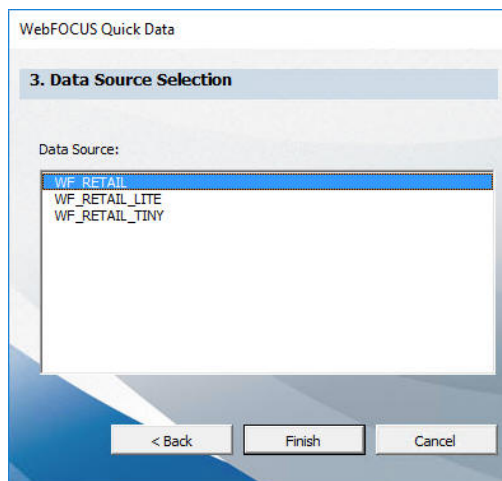
4. In the Web Server Connection dialog box, specify the desired Web Server URL and connection settings, and then click *Next*.

Note: Do not end the URL with the / sign.

5. If prompted for sign-in credentials, sign in with your WebFOCUS Business User Edition login and password.

The next screen opens, where you can select a folder to open a list of available data sources.

6. Select a folder and click *Next*.
7. In the Data Source Selection dialog box, select a Master File, as shown in the following image, and then click *Finish*.



InfoAssist+ opens, where you can build a query and run it to return the output data to Excel.

Example: Creating a New Report Query in InfoAssist

This example covers multiple aspects of creating a new report query using the WebFOCUS Quick Data tool from an Excel file.

1. Open an Excel file, select a cell in which to place the query results, select the **ADD-INS** tab, click the **WebFOCUS** option in the Excel menu, and select **Create Query**.

The Web Server Connection dialog box opens.

2. In the Web Server URL field, type the URL for a web server in your reporting environment, or select one from the drop-down list. An example of a Web Server URL is:

<http://localhost:26000>

Note: Do not end the URL with the / sign.

3. Click **Next**.
4. Select a folder to open a list of available data sources. In this example we use the **Retail Samples** folder. Click **Next**.
5. In the Data Source Selection dialog box that opens, select **WF_RETAIL_LITE** from the Data Source list, and click **Finish**.

InfoAssist+ opens.

6. Drag fields from the Data pane to the Query Pane to create a report.
7. On the **Home** tab, in the **Report** group, select **Column Totals**.

After you perform the steps up to this point, the InfoAssist window looks similar to the following.

Product	Category	Subcategory	Cost of Goods	Discount	Gross Profit
Accessories	Charger		\$2,052,711.00	\$187,485.88	\$1,971,225.12
Accessories	Headphones		\$51,663,564.00	\$3,515,913.33	\$24,522,146.67
Accessories	Universal Remote Controls		\$36,037,623.00	\$2,310,446.31	\$13,368,146.69
Camcorder	Handheld		\$20,576,916.00	\$1,959,624.28	\$21,398,140.28
Camcorder	Professional		\$35,218,308.00	\$1,933,997.25	\$8,832,140.75
Camcorder	Standard		\$49,071,633.00	\$3,214,786.74	\$19,368,146.26
Computers	Smartphone		\$44,035,774.00	\$2,790,775.51	\$15,832,146.49
Computers	Tablet		\$25,771,890.00	\$2,018,134.59	\$17,677,146.41
Media Player	Blu Ray		\$181,112,921.00	\$10,895,633.29	\$51,777,146.71
Media Player	DVD Players		\$3,756,254.00	\$258,593.13	\$1,857,146.87
Media Player	DVD Players - Portable		\$306,576.00	\$26,356.05	\$280,220.95
Media Player	Streaming		\$5,064,730.00	\$338,559.94	\$1,936,146.06
Stereo Systems	Boom Box		\$840,373.00	\$62,739.39	\$54,146.61
Stereo Systems	Home Theater Systems		\$56,428,589.00	\$3,943,920.23	\$27,932,146.77
Stereo Systems	Receivers		\$40,329,668.00	\$2,643,045.39	\$16,557,146.61
Stereo Systems	Speaker Kits		\$81,396,140.00	\$4,954,243.27	\$25,811,146.73

- 8. Click the Save button.

The report query data is transferred to the Excel file, as shown in the following image.

Product Category	Product Subcategory	Cost of Goods	Discount	Gross Profit	Quantity Sold	Revenue
Accessories	Charger	\$2,052,711.00	\$187,485.88	\$1,970,123.91	105,257	\$4,022,834.91
Accessories	Headphones	\$51,663,564.00	\$3,516,913.33	\$24,523,023.97	228,349	\$76,186,587.97
Accessories	Universal Remote Controls	\$36,037,623.00	\$2,310,446.31	\$13,361,292.65	178,061	\$49,398,915.65
Camcorder	Handheld	\$20,576,916.00	\$1,959,824.28	\$21,393,654.97	250,167	\$41,970,570.97
Camcorder	Professional	\$35,218,308.00	\$1,933,997.25	\$8,835,522.75	12,872	\$44,053,830.75
Camcorder	Standard	\$49,071,633.00	\$3,214,786.74	\$19,369,667.52	192,205	\$68,441,300.52
Computers	Smartphone	\$44,035,774.00	\$2,790,775.51	\$15,834,702.15	205,049	\$59,870,476.15
Computers	Tablet	\$25,771,890.00	\$2,018,134.59	\$17,674,115.97	146,728	\$43,446,005.97
Media Player	Blu Ray	\$181,112,921.00	\$10,895,633.29	\$51,771,195.13	679,495	\$232,884,116.13
Media Player	DVD Players	\$3,756,254.00	\$258,593.13	\$1,859,645.81	18,835	\$5,615,899.81
Media Player	DVD Players - Portable	\$306,576.00	\$26,356.05	\$265,150.77	5,694	\$571,726.77
Media Player	Streaming	\$5,064,730.00	\$338,559.94	\$1,936,586.65	67,910	\$7,001,316.65
Stereo Systems	Boom Box	\$840,373.00	\$62,739.39	\$546,423.99	9,370	\$1,386,796.99
Stereo Systems	Home Theater Systems	\$56,428,589.00	\$3,943,920.23	\$27,931,096.22	399,092	\$84,359,685.22
Stereo Systems	Receivers	\$40,329,668.00	\$2,643,045.39	\$16,555,835.56	150,568	\$56,885,503.56
Stereo Systems	Speaker Kits	\$81,396,140.00	\$4,954,243.27	\$25,819,241.69	244,199	\$107,215,381.69
Stereo Systems	iPod Docking Station	\$26,119,093.00	\$1,926,925.29	\$15,328,473.06	311,103	\$41,447,566.06
Televisions	CRT TV	\$1,928,416.00	\$118,654.68	\$602,419.65	4,638	\$2,530,835.65
Televisions	Flat Panel TV	\$59,077,345.00	\$3,478,828.52	\$15,885,498.71	92,501	\$74,962,843.71
Televisions	Portable TV	\$545,348.00	\$38,210.18	\$342,105.45	8,049	\$887,453.45
Video Production	Video Editing	\$40,105,657.00	\$2,695,890.76	\$17,947,619.62	199,749	\$58,053,276.62
TOTAL		\$761,439,529.00	\$49,313,764.01	\$299,753,396.20	3,509,891	\$1,061,192,925.20

- 9. Click the drop-down arrow to the right of the Name Box. You will see named ranges that are automatically added to the query, as shown in the following image.

Product	Subcategory	Cost of Goods
	Charger	\$2,052,711.00
	Headphones	\$51,663,564.00
	Universal Remote Controls	\$36,037,623.00
	Handheld	\$20,576,916.00
	Professional	\$35,218,308.00
	Standard	\$49,071,633.00
	Smartphone	\$44,035,774.00

Named ranges are added to the entire data table. The named range for the entire data table is QDATA1.

10. Select QDATA1 from the Name Box. The data in the table is automatically highlighted.

11. Save the Excel file so that it can be reused in the example on editing an existing report query.

Ribbon Command Reference

The ribbon is contextual and changes depending on the type of file that you are developing. This topic describes each of the available ribbons and commands for each InfoAssist+ tool.

Ribbon Commands for Reports

When creating and customizing reports in Report mode, you can use the following ribbons and commands to customize report functionality.

Home Tab

Command	Description
Format Group	
Output File Format	Displays a drop-down menu of all supported output formats.
Chart	Switches to Chart mode. Converts a report to a chart using the fields specified in the report.
Report	Indicates that you are in Report mode.
File	Creates a data file from a report.
Design Group	
Query (Design view)	Displays the Data, Query, and Filter panes across the entire canvas, eliminating Live Preview. This view provides a larger work area for creating the report.
Live Preview (Design view)	Displays the report on the canvas as you create the report. You can use the Live Preview to add, remove, and arrange fields, as well as style the report.

Command	Description
Document (Design view)	Converts a report into a document. Opens the document on canvas, which you can use to add text, images, lines, reports, and charts to create documents.
Data from Source	Uses the selected data source to display a live preview of the output on the canvas.
Use Sample Data	Displays sample data, which reduces processing time by eliminating the need to access the actual data source.
Records	Limits the number of rows retrieved from the data source when Live Preview is selected. This feature is useful in reducing response time if you are working with a large amount of data. Type the number of rows that you want directly in the Records field, or use the drop-down menu to select one of the preset record limits. The preset choices are All rows, 1, 10, 50, 100, 500, 1000, 2000, 5000, and 10000.
Filter Group	
Filter	Opens the Filter dialog box for creating filters. Filters enable you to select only the data that you want and to exclude unwanted data.
Exclude	Turns off a filter.
Include	Turns on a filter.
Report Group	
Theme	<p>Opens a dialog box where you can select a theme to style your report or chart. You can use the default style sheet by clicking the <i>Use Default Stylesheet</i> button.</p> <p>You can also select a document styling theme or an application theme to style all reports created. Use the Environment and Styling section of the Options window, which is accessible by clicking <i>Options</i> in the Application main menu.</p>

Command	Description
Style	Opens a Report Style dialog box for applying global styling to the entire report. For more information about styling reports, see Styling Reports on page 556.
Banded	Opens a Color dialog box for choosing a color that provides an alternating color scheme for the report. The report output displays alternating rows of data, using a white background for one row and a background of the selected color for the next row. This pattern continues throughout the report.
Header & Footer	Opens the Header & Footer dialog box, from which you can add and style headings and footings.
Column Totals	Adds a grand total row to the bottom of the report to sum numeric data in each column.
Row Totals	Adds a grand total column to the right side of the report to sum numeric data in each row.

Format Tab

Command	Description
Destination Group	
InfoMini	Enables the creation of an InfoMini application. For more information on using InfoMini, see Building InfoMini Applications on page 475.
Chart	Switches to Chart mode. Converts a report to a chart using the fields specified in the report.
Report	Indicates that you are in Report mode.
File	Creates a data file from a report.
Navigation Group	

Command	Description
Table	Generates standard browser output. This is the default.
Table of Contents	<p>Generates output by displaying a table of contents icon in the upper-left corner where report output typically appears. Clicking <i>Table of Contents</i> opens a menu that enables you to select (view) individual values of the first Sort By (By) field, one value at a time.</p> <p>You can also select options to view the entire report or remove the table of contents.</p>
Freeze	Generates output with column titles that freeze (remain in view) when you scroll through pages of the report output.
Pages On Demand	Provides access to two distinct features, depending upon the output type that you have selected.
Auto Drill	Enables you to navigate through different levels within the dimension hierarchy of your data source. Click <i>Auto Drill</i> to enable the functionality. For more information, see Using Auto Drill on page 648.
Features Group	
Title Popup	Displays pop-up titles when the mouse pointer hovers over a column title in the report output.
Accordion	Creates expandable views of data for each vertical sort field. This option displays data values only for the first vertical sort field when you first view the output. You can manually expand your view to expose the data values of lower-level sort fields.
Repeat Sort Value	Displays all repeated sort values instead of blanks in the output after the first instance of a new sort value, which is the default behavior.

Command	Description
Stack Measures	Displays all numeric measure field names in a column of the report output with the corresponding numeric data values.
active report Options	Opens the active report options dialog box where you can configure your active report options such as menu items, graph engine, and colors. For more information, see Using Active Technologies on page 416.
Accessibility	Allows a title to be added to a report, chart, or document that is Section 508-compliant.
Auto Linking Group	
Enable Auto Linking	Enables auto linking. For more information, see Using the Auto Linking Feature to Link Content on page 650.
Auto Link Target	Sets procedure as an available target for auto linking.

Data Tab

Command	Description
Calculation Group	
Detail (Define)	Opens the Detail Field (DEFINE) dialog box, where you can create a defined field, type a name for the field, and enter a format. A Define field is an optional attribute used to create a virtual field for reporting. You can derive the virtual field value from information already in the data source (that is, from permanent fields).
Summary (Compute)	Opens the Summary Field (COMPUTE) dialog box, where you can create a computed field, type a name for the field, and enter a format.
Join Group	

Command	Description
Join	Opens the Join dialog box, where you can create a new join, edit or delete existing joins, and add data sources to a join.
Filter Group	
Filter	Opens the Filter dialog box for creating filters. Filters enable you to select only the data that you want and to exclude unwanted data.
Display Group	
Missing Data	This option is disabled for reports.
Data Source Group	
Add	Opens the Open dialog box, where you can add additional data sources to a document, enabling you to insert reports from different data sources into the same document. This option is activated when you add a HOLD file. This option is unavailable, by default.
Switch	Opens a drop-down list of all the data sources that have been added. You can choose which data source is currently active and being used to create new reports. This option is activated when you add a HOLD file. This option is unavailable, by default.

Slicers Tab

Command	Description
Options Group	
New Group	Creates a new group of similar slicers.
Clear Slicers	Resets all slicers so that no filtering is done.
Update Preview	Applies slicers to preview.

Command	Description
Options	Opens the Edit Slicers dialog box to the General tab, where you can set general options for your slicers.
Record Limit Group	
Preview	Sets the number of records retrieved from the data source for preview.
Run Time	Sets the number of records retrieved at run time.
Record Limit	Opens the Edit Slicers dialog box to the Record Limit tab, where you can set record limits for your slicers.
Group Number Group	
Group <i>n</i>	Contains a group for each Slicer group that is added. Group 1 is the default slicer group to which you can drag fields to create slicers. To access slicer group options, click Group <i>n</i> to open the Edit Slicers dialog box where you can rename the slicer group and modify the order of the slicers in the group.

Layout Tab

Command	Description
Page Setup Group	
Margins	Enables you to set margin values by choosing Normal (1 inch all around), Narrow (.5 inch all around), Moderate (.5 inch left or right), Wide (1.5 inch left or right), or Custom. Choosing <i>Custom</i> opens the Margins dialog box, where you can set specific margins as needed.
Orientation	Enables you to set the orientation of your report to portrait or landscape.

Command	Description
Size	Enables you to select the size of the paper for printing output. You can choose A3, A4, A5, Letter, Tabloid, Legal, PowerPoint, or Large Size (34 x 44 Inches).
Units	Enables you to select the unit of measurement used for customizing the dimension fields of your report. You can choose Inches, Centimeters, or Points.
Page Numbers	<p>Enables you to select page numbering options. You can choose one of the following:</p> <ul style="list-style-type: none"><input type="checkbox"/> No Lead (no space for headers)<input type="checkbox"/> On (page numbers only in headers)<input type="checkbox"/> Off (space for headers, but no page numbering) <p>The Page Numbers value is overridden by header and footer text options.</p>
Report Group	
Cell Padding	Opens the Cell Padding dialog box, where you can set specific values to control the amount of space inserted between rows and columns in a report. For more information, see How to Use Cell Padding in a Report on page 565.
AutoFit	Limits the width of columns in a report to be no wider than the largest value in each column. AutoFit Column is selected, by default.

View Tab

Command	Description
Design Group	

Command	Description
Query (Design view)	Displays the Data, Query, and Filter panes across the entire canvas, eliminating Live Preview. This view provides a larger work area for creating the report.
Live Preview (Design view)	Displays the report on the canvas as you create the report. You can use the Live Preview to add, remove, and arrange fields, as well as style the report.
Document (Design view)	Converts a report into a document. Opens the document on canvas, which you can use to add text, images, lines, reports, and charts to create documents.
Show/Hide Group	
Resources	Minimizes the Resources panel and expands the size of the canvas to also occupy the area where the Resources panel typically appears. The canvas can display a preview of a report, output of a report, or the Query Design pane.
Data Panel Group	
Logical	Displays the data source fields by type. This is the default view. The Logical view options include Title, Description, Field, and Alias.
List	Displays the data source fields in a tabular list format. This list contains a header row. You can sort fields differently by clicking a column header. The List view options include Title, Description, Field, Alias, Format, Segment, Filename, and Reference.
Structured	Displays the hierarchical structure of the data source files. The Structured view options include Title, Description, Field, and Alias.
Query Panel Group	
Areas 2x2	Displays data in a two column by two-row grid.
Areas 1x4	Displays data in a one column by four-row grid.

Command	Description
Tree	Displays data in a tree. This is the default.
Output Window Group	
Arrange	Opens a drop-down menu where you can choose how to display multiple output windows. The options are Cascade, Tile Horizontally, and Tile Vertically.
Output Location	Opens a drop-down menu where you can choose how to direct new output. The options are Single tab (default), New Tab, Single Window, and New Window.
Switch Output	Opens a drop-down menu for choosing to view any active output window.
Report Group	
Switch Report	Lists any active reports, charts, documents, and visuals to which you can switch.

Field Tab

Command	Description
Filter Group	
Filter	Opens the Filter dialog box for creating filters. Filters enable you to select only the data that you want and to exclude unwanted data.
Exclude	Removes, but does not delete, the filter from the report.
Include	Restores a filter that was previously excluded from a report.

Command	Description
Prompt	<p>Opens the Create a filtering condition dialog box for creating an auto prompting parameter that you can select when you run a report. The Create a filtering condition dialog box is used to create both filters and auto prompting parameters. The following prompt options are available when Parameter is selected from the Type drop-down menu:</p> <ul style="list-style-type: none"> <input type="checkbox"/> Simple. This is used for prompts using Text Input. This is the default value. <input type="checkbox"/> Static. This is used for prompts using Selection. This option allows you to select multiple values at run time. <input type="checkbox"/> Dynamic. This is used for prompts using Data Values. This option allows you to select multiple values at run time. <input type="checkbox"/> Optional. This is used for prompts using Single or Multiselect parameters.
Sort Group	
Up	Sorts the selected field in ascending order. This option is activated when you click on a measure or dimension.
Down	Sorts the selected field in descending order. This option is activated when you click on a measure or dimension.
Rank	Inserts a rank column immediately to the left of the report if a Sort By field is selected. It also adds a rank column to the left of the Sort By field if a Measure field is selected. Ranking a Measure field results in two copies of the field, the original Measure field, and the Sort By field that is created during ranking. This option is activated when you click on a measure or dimension.

Command	Description
Group	Opens the Create a Group dialog box where you can create a group to combine values together. This option is activated when you click on a dimension.
Limit	Opens a drop-down menu that allows you to specify the number of unique values to display for a sort group that has been added. This option is activated when you click on a measure or dimension.
Break Group	
Page Break	Starts a new page when the primary sort field changes. Clicking the drop-down icon enables you to select <i>Reset Page Numbers</i> , which allows you to reset page numbers on a page break to start at 1. This option is activated when you click on a dimension.
Line Break	Inserts a line in the report output when the primary sort field changes. This option is activated when you click on a dimension.
Subtotal	Inserts a line, total text (TOTAL FIELD Value), and subtotals for all numeric fields when the primary sort field changes. This option is activated when you click on a dimension.
Sub Header	Opens a dialog box where you can type text to add a subheading just below the column titles in the report output when the primary sort field changes. This option is activated when you click on a dimension.
Sub Footer	Opens a dialog box where you can type text to add a subfooting at the end of the data on each page of the report output when the primary sort field changes. This option is activated when you click on a dimension.
Style Group	
Font	Opens the Font list, which you can use to change the font.

Command	Description
Font Size	Opens the Font Size list, which you can use to change the numeric value for the font size.
Font Color	Opens the Color dialog box, where you can select the font color.
Style Reset	Resets all settings to the default settings from the template.
Bold	Applies bold font formatting to the selected text.
Italic	Applies italic font formatting to the selected text.
Underline	Underlines the selected text.
Justify Left	Aligns the text to the left of the report.
Justify Center	Aligns the text to the center of the report.
Justify Right	Aligns the text to the right of the report.
Background Color	Opens the Color dialog box, where you can select the background color for the report.
Data Style	Styles only the data for the selected data source field.
Title Style	Styles only the column title for the selected data source field.
Data + Title	Styles both the data and the column title for the selected data source field.
Format Group	
Decimal	The default value for the data format of the selected measure is Decimal. Use the drop-down menu to select Alphanumeric, Integer, or More options to open the Field Format Options dialog box.
Change currency options	Changes the currency options for the selected field. This option is activated when you click on a measure.

Command	Description
Percent	Specifies the value of the field as a percentage. This option is activated when you click on a measure.
Comma	Specifies the use of commas for the selected field. This option is activated when you click on a measure.
Increase Decimal Places	Increases the number of decimal places that display for the selected field. This option is activated when you click on a measure.
Decrease Decimal Places	Decreases the number of decimal places that display for the selected field. This option is activated when you click on a measure.
Display Group	
Hide Field	Allows you to hide a selected field.
Hide Missing	Allows you to hide fields that have no value.
Aggregation	Enables you to apply an aggregation function to a field in a report. Opens a drop-down menu of the following options: None (default), Sum, Average, Count, Count Distinct, Percent of Count, First Value, Last Value, Maximum, Minimum, Total, Percent, Row Percent, Median, Average Square.
Traffic Lights	Opens the Traffic Light Condition dialog box. From this dialog box, you can add new conditional styling by applying traffic light (and other) colors to a selected field in the output when the field meets specified criteria, modify existing conditional styling, and enable conditional drill-down.
Data Bars	Adds a data visualization column to the right of a selected numeric field. The column displays values in each row using horizontal bars that extend from left to right in varying lengths, depending on the corresponding data values.

Command	Description
Within	Allows you to use specific aggregation tasks at different report levels. You can use the Within phrase to manipulate display field values as they are aggregated within a sort group rather than a report column.
Column(s)	This option is disabled for reports.
Links Group	
Drill Down	Opens the Drill Down dialog box, where you can configure a hyperlink or a drill-down procedure for the selected field. Clicking that field in the report output, at run time, redirects you to the URL you specified or executes the indicated procedure.

Ribbon Commands for Charts

When creating and customizing charts in Chart mode, you can use the following ribbons and commands to customize chart functionality.

Home Tab

Command	Description
Format Group	
Output File Format	Displays a drop-down menu of all supported output formats.
Chart	Indicates that you are in Chart mode.
Report	Switches to Report mode. Converts a chart to a report using the fields specified in the chart.
File	Creates an image file from a chart. This option is disabled by default and is only enabled for HTML format.
Design Group	

Command	Description
Query (Design view)	Displays the Data, Query, and Filter panes across the entire canvas, eliminating Live Preview. This view provides a larger work area for creating the chart.
Live Preview (Design view)	Displays the chart on the canvas as you create the chart. You can use the Live Preview to add, remove, and arrange fields, as well as style the chart.
Document (Design view)	Opens the document on canvas, which you can use to add text, images, lines, reports, and charts to create documents.
Data from Source	Uses the selected data source to display a live preview of the output on the canvas.
Use Sample Data	Displays sample data, which reduces processing time by eliminating the need to access the actual data source.
Records	Limits the number of rows retrieved from the data source when Live Preview is selected. This feature is useful in reducing response time if you are working with a large amount of data. Type the number of rows that you want directly in the Records field, or use the drop-down menu to select one of the preset record limits. The preset choices are All rows, 1, 10, 50, 100, 500, 1000, 2000, 5000, and 10000.
Filter Group	
Filter	Opens the Filter dialog box for creating filters. Filters enable you to select only the data that you want and to exclude unwanted data.
Exclude	Turns off a filter.
Include	Turns on a filter.
Report Group	

Command	Description
Theme	<p>Opens a dialog box where you can select a theme to style your report or chart. You can use the default style sheet by clicking the <i>Use Default Stylesheet</i> button.</p> <p>You can also select a document styling theme or an application theme to style all reports created. Use the Environment and Styling section of the Options window, which is accessible by clicking Options in the Application main menu.</p>
Style	This option is disabled for charts.
Banded	This option is disabled for charts.
Header & Footer	Opens the Header & Footer dialog box, from which you can add and style headings and footings.
Column Totals	This option is disabled for charts.
Row Totals	This option is disabled for charts.

Format Tab

Command	Description
Destination Group	
InfoMini	Enables the creation of an InfoMini application. For more information on using InfoMini, see Building InfoMini Applications on page 475.
Chart	Indicates that you are in Chart mode.
Report	Switches to Report mode. Converts a chart to a report using the fields specified in the chart.
File	Creates an image file from a chart. This option is disabled by default and is only enabled for HTML format.

Command	Description
Chart Types Group	
Bar	Changes the chart type to a bar chart.
Pie	Changes the chart type to a pie chart.
Line	Changes the chart type to a line chart.
Area	Changes the chart type to an area chart.
Scatter	Changes the chart type to a scatter chart.
Choropleth	Changes the chart type to a choropleth map.
Proportional Symbol	Changes the chart type to a proportional symbol (bubble) map.
Other	Opens the Select a chart dialog box. The chart types, depicted by icons, display on the left side of the dialog box. The first chart type represents the bar chart category, which is selected by default. When a chart type is selected, the dialog box displays thumbnail images of bar chart variations that are supported.
Map Group	
Background	Provides terrain options and various other geographical views. This option only displays when a map is selected as the chart type.
Demographic Layers	Allows you to apply one or more pre-defined demographic layers, which can narrow the scope of your data using these underlying layers of demographic categorization. This option only displays when a map is selected as the chart type.
Reference Layers	Enables you to define one or more reference layers, which creates borders based on your geographical selection. This option only displays when a map is selected as the chart type.

Command	Description
Features Group	
3D Effect	Sets the three-dimensional view to on or off. The 3D Effect feature is disabled for 3D, stock, gauge, gauge thermometer, Pareto, spectral map, and funnel chart types. This is the default. This option is not available for maps.
Rotate	Toggles between a vertical display or horizontal display of a chart. For more information, see How to Rotate a Chart on page 639. The Rotate feature is disabled for pie, scatter, 3D, stock, gauge, gauge thermometer, Pareto, spectral map, and funnel chart types. This option is not available for maps.
Reference	Opens a drop-down menu that provides the Add Reference Line to Y-Axis and Add Reference Line to X-Axis options. Selecting one of these options opens the appropriate Reference Line dialog box, where you can set the specific X-axis or Y-axis value, type the text that you want, and position the reference line on a chart. For more information, see How to Display a Static Reference Line on page 639. The Reference feature is disabled for pie, 3D, stock, gauge, gauge thermometer, Pareto, spectral map, and funnel chart types. This option is not available for maps.
Annotate	<p>Opens a drop-down menu that provides the Add an annotation option. Selecting this option opens the Annotation dialog box, where you can type the text that you want and position the annotation on a chart. For more information, see How to Display Annotations on page 642.</p> <p>The annotation option is not available in HTML5. This option is not available for maps.</p>

Command	Description
Grid	Opens a drop-down menu allowing you to expand options for Horizontal or Vertical Gridlines. Both selections allow you to enable or disable Major and Minor Gridlines. Clicking <i>More Options</i> opens the Format Grid Lines dialog box. This option is not available for maps. For more information, see Formatting Gridlines on page 598.
Frame & Background	Opens the Frame & Background dialog box where you can edit the background style and frames for charts. The dialog contains different options depending on the chart type selected. This option is not available for maps. For more information, see Formatting a Frame and a Background on page 617.
Gauges	Opens the Gauge dialog box where you can edit your gauge chart. This button is only available when a gauge chart type is selected. This option is not available for maps. For more information, see How to Style a Gauge Needle on page 634.
active report Options	Opens the active report options dialog box where you can configure your active report options, such as menu items, graph engine, and colors. This button is available when the output type is set to active report. This option is not available for maps.
Accessibility	<p>Allows a title to be added to a report, chart, or document that is Section 508 compliant. This option is only available for reports and charts when the output type is HTML or PDF. For documents, the output type must be set to PDF.</p> <p>The chart features are unavailable when designing a chart that will be output in active report format. This option is not available for maps.</p>
Labels Group	

Command	Description
Axes	Opens a drop-down menu, where you can enable and rotate horizontal and vertical axis labels, and stagger horizontal axis labels. You can also edit the axis labels by clicking <i>More Horizontal Axis Options</i> or <i>More Vertical Axis Options</i> . For more information, see Formatting Axis Labels on page 608. This option is not available for maps.
Legend	Opens a drop-down menu, where you can select the Show Legend option to display the legend on the chart, or clear your selection to hide the legend, change the default legend position, and change the default legend orientation. For more information, see Format Legend Dialog Box on page 594. This option is not available for maps.
Interactive Group	
Interactive Options	Opens the Interactive Options dialog box, which enables you to specify animation and mouse over effects in your chart. This option is only available for HTML5 and active outputs. This option is not available for maps.
Navigation Group	
Auto Drill	<p>Enables you to navigate through different levels within the dimension hierarchy of your data source. Click <i>Auto Drill</i> to enable the functionality.</p> <p>Note: Auto Drill functionality requires the specification of at least one dimension sort field in the request.</p> <p>For more information, see Using Auto Drill on page 648.</p>
Auto Linking Group	
Enable Auto Linking	Enables auto linking.

Command	Description
Auto Link Target	Sets procedure as an available target for auto linking.

Data Tab

Command	Description
Calculation Group	
Detail (Define)	Opens the Detail Field (DEFINE) dialog box, where you can create a defined field, type a name for the field, and enter a format. A Define field is an optional attribute used to create a virtual field for reporting. You can derive the virtual field value from information already in the data source (that is, from permanent fields).
Summary (Compute)	Opens the Summary Field (COMPUTE) dialog box, where you can create a computed field, type a name for the field, and enter a format.
Join Group	
Join	Opens the Join dialog box, where you can create a new join, edit or delete existing joins, and add data sources to a join.
Filter Group	
Filter	Opens the Filter dialog box, enabling you to set filtering options. Filter options include Where, Where Total, the And conjunction, and the Or conjunctions in a single expression.
Display Group	
Missing Data	Includes options for how to display missing values in charts.
Data Source Group	

Command	Description
Add	Opens the Open dialog box, where you can add additional data sources to a document, enabling you to insert reports from different data sources into the same document. This option is only enabled if the chart was created from a HOLD file.
Switch	Opens a drop-down list of all the data sources that have been added. You can choose which data source is currently active and being used to create new reports. This option is only enabled if the chart was created from a HOLD file.

Slicers Tab

Command	Description
Options Group	
New Group	Creates a new group of similar slicers.
Clear Slicers	Resets all slicers so that no filtering is done.
Update Preview	Applies slicers to preview.
Options	Opens the Edit Slicers dialog box to the General tab, where you can set general options for your slicers.
Record Limit Group	
Preview	Sets the number of records retrieved from the data source for preview.
Run Time	Sets the number of records retrieved at run time.
Record Limit	Opens the Edit Slicers dialog box to the Record Limit tab, where you can set record limits for your slicers.
Group Number Group	

Command	Description
Group <i>n</i>	Contains a group for each Slicer group that is added. Group 1 is the default slicer group to which you can drag fields to create slicers. To access slicer group options, click Group <i>n</i> to open the Edit Slicers dialog box where you can rename the slicer group and modify the order of the slicers in the group.

Layout Tab

Command	Description
Page Setup Group	
Margins	Enables you to set margin values by choosing Normal (1 inch all around), Narrow (.5 inch all around), Moderate (.5 inch left or right), Wide (1.5 inch left or right), or Custom. Choosing <i>Custom</i> opens the Margins dialog box, where you can set specific margins as needed.
Orientation	Enables you to set the orientation of your report to portrait or landscape.
Size	Enables you to select the size of the paper for printing output. You can choose A3, A4, A5, Letter, Tabloid, Legal, PowerPoint, or Large Size (34 x 44 Inches).
Units	Enables you to select the unit of measurement used for customizing the dimension fields of your report or chart. You can choose Inches, Centimeters, or Points.
Page Numbers	This option is disabled for charts.
Size & Arrange Group	
Height	Sets the height of the chart.
Width	Sets the width of the chart.

Command	Description
Auto Overflow	This option is disabled for charts.
Aspect Ratio	Lock the height and width aspect ratio. With the aspect ratio locked, changing the width automatically changes the height to keep the component to scale, and changing the height automatically changes the width.
AutoFit	Expands the chart, at design time, when additional fields are added. At run time, the chart is resized dynamically to fit into the container in which it is placed. AutoFit is enabled, by default.
Align	This option is available in Document mode only.
Relative Position	This option is available in Document mode only.

View Tab

Command	Description
Design Group	
Query (Design view)	Displays the Data, Query, and Filter panes across the entire canvas, eliminating Live Preview. This view provides a larger work area for creating the chart.
Live Preview (Design view)	Displays the chart on the canvas as you create the report. You can use the Live Preview to add, remove, and arrange fields, as well as style the chart.
Document (Design view)	Converts a chart into a document. Opens the document on canvas, which you can use to add text, images, lines, reports, and charts to create documents.
Show/Hide Group	

Command	Description
Resources	Minimizes the Resources panel and expands the size of the canvas to also occupy the area where the Resources panel typically appears. The canvas can display a preview of a report, output of a report, or the Query Design pane.
Data Panel Group	
Logical	Displays the data source fields by type. This is the default view. The Logical view options include Title, Description, Field, and Alias.
List	Displays the data source fields in a tabular list format. This list contains a header row. You can sort fields differently by clicking a column header. The List view options include Title, Description, Field, Alias, Format, Segment, Filename, and Reference.
Structured	Displays the hierarchical structure of the data source files. The Structured view options include Title, Description, Field, and Alias.
Query Panel Group	
Areas 2x2	Displays data in a two column by two-row grid. This option is disabled for charts that use the new field container syntax.
Areas 1x4	Displays data in a one column by four-row grid. This option is disabled for charts that use the new field container syntax.
Tree	Displays data in a tree. This is the default.
Output Window Group	
Arrange	Opens a drop-down menu where you can choose how to display multiple output windows. The options are Cascade, Tile Horizontally, and Tile Vertically.

Command	Description
Output Location	Opens a drop-down menu where you can choose how to direct new output. The options are Single tab (default), New Tab, Single Window, and New Window.
Switch Output	Opens a drop-down menu for choosing to view any active output window.
Report Group	
Switch Report	Lists any active report or chart to which you can switch.

Field Tab

Note: The Break, Style, and Format groups are disabled in Chart mode.

Command	Description
Filter Group	
Filter	Opens the Filter dialog box for creating filters. Filters enable you to select only the data that you want and to exclude unwanted data.
Exclude	Removes, but does not delete, the filter from the chart.
Include	Restores a filter that was previously excluded from a chart.

Command	Description
Prompt	<p>Opens the Create a filtering condition dialog box for creating an auto prompting parameter that you can select when you run a chart. The Create a filtering condition dialog box is used to create both filters and auto prompting parameters. The following prompt options are available when Parameter is selected from the Type drop-down menu:</p> <ul style="list-style-type: none"><input type="checkbox"/> Simple. This is used for prompts using Text Input. This is the default value.<input type="checkbox"/> Static. This is used for prompts using Selection. This option allows you to select multiple values at run time.<input type="checkbox"/> Dynamic. This is used for prompts using Data Values. This option allows you to select multiple values at run time.<input type="checkbox"/> Optional. This is used for prompts using Single or Multiselect parameters.
Sort Group	
Up	Sorts the selected field in ascending order.
Down	Sorts the selected field in descending order.
Rank	This option is disabled for charts.
Group	Opens the Create a Group dialog box where you can create a group to combine values together. This option is enabled for dimension fields only.
Limit	Opens a drop-down menu that allows you to specify the number of unique values to display for a sort group that has been added.
Break Group	
Note: These options are disabled for charts.	

Command	Description
Style Group	
Note: These options are disabled for charts.	
Format Group	
Note: These options are disabled for charts.	
Display Group	
Hide Field	Allows you to hide a selected field.
Hide Missing	Allows you to hide fields that have no value. This option is disabled for charts.
Aggregation	Opens a drop-down menu of the following options: None (default), Sum, Average, Count, Count Distinct, Percent of Count, First Value, Last Value, Maximum, Minimum, Total, Percent, Row Percent, Median, Average Square. This option is only available for measure fields or dimensions (alpha field only) that are in a numeric field container. Otherwise, aggregations will not display.
Traffic Lights	Opens the Traffic Light Condition dialog box. From this dialog box, you can add new conditional styling by applying traffic light (and other) colors to a selected field in the output when the field meets specified criteria, modify existing conditional styling, and enable conditional drill-down. This option is only available for measure fields.
Data Bars	This option is disabled for charts.
Within	Allows you to use specific aggregation tasks at different report levels. You can use the Within phrase to manipulate display field values as they are aggregated within a sort group rather than a report column. This option is disabled for charts.

Command	Description
Column(s)	Allows you to indicate the number of columns in which you wish to display multiple graphs. The value can be between 1 and 512. The default is 1. This option is also available from the Query Design pane shortcut menu for a Multi-graph component. This option is only enabled when the multi-graph field container is populated.

Links Group

Drill Down	Opens the Drill Down dialog box, where you can configure a hyperlink or a drill-down procedure for the selected field. Clicking that field in the report output, at run time, redirects you to the URL you specified or executes the indicated procedure. This option is available for measure fields only. If you are working in PDF format, this option is disabled.
------------	--

Series Tab

Command	Description
Select Group	
Series drop-down list	Lists the available series in the current chart.
Style Group	
Style	Opens the Format Series dialog box, where you can edit the styling options for the selected series. You can also open this dialog box by right-clicking a series, and then clicking <i>More Style Options</i> .
Properties Group	

Command	Description
Data Labels	<p>Adds data labels to the chart. The drop-down menu contains the following data position options for selecting where to display data values as labels on a chart: Above (default), On top edge, Below top edge, Center, and Base. If you are working with a Pie chart, the options are: On Slice, Outside Slice, and Outside with feeler lines.</p> <p>Clicking <i>More Data Label Options</i> opens the Format Labels dialog box, where you can further edit your data labels.</p>
Type	<p>Opens a drop-down menu with the following options for selecting different chart types: None (default), Bar, Line, and Area.</p> <p>Note: When you make a change to the chart type using the Type button on the Series tab, changes to the chart type on the Format tab are overwritten.</p>
Trendline	<p>Opens a drop-down menu that provides options for adding a trendline to a chart.</p>
Equation	<p>Displays the associated mathematical equation for the selected trendline on the chart.</p> <p>The equation is not available in HTML5.</p>
Line Group	
Smooth Line	<p>Draws the chart using smooth lines.</p>
Connect Lines	<p>Controls the display of connecting lines between markers on a line or scatter chart. By default, lines are connected on a line chart and disconnected on a scatter chart.</p>
Marker	<p>Opens a drop-down menu from which you can select options to change the display of the default data and legend markers on line and scatter chart types. For more information, see How to Change the Appearance of a Marker on page 581.</p>

Command	Description
Pie Group	
Note: The following options are only enabled when you are working with a pie chart.	
Expand	Expands pie slices.
Hide	Hides pie slices.

Ribbon Commands for Documents

When creating and customizing documents in Document mode, you can use the following ribbons and commands to customize document functionality.

Home Tab

Command	Description
Format Group	
Output File Format	Displays a drop-down menu of all supported output formats.
Chart	Determines whether chart-specific functionality is available in the InfoAssist+ tool. The default name Chart (data source) is given for each new chart created in a given InfoAssist+ session, where data source is the name of the underlying data source you are using. You can rename the chart by right-clicking <i>Chart</i> in the Query pane and clicking <i>Rename</i> .
Report	Determines whether report-specific functionality is available in the InfoAssist+ tool. The default name Report (data source) is given for each new report created in a given InfoAssist+ session, where data source is the name of the underlying data source you are using. You can rename the report by right-clicking <i>Report</i> in the Query pane and clicking <i>Rename</i> .

Command	Description
File	Creates a data file from a report component of a document.
Design Group	
Query (Design view)	This option is disabled in Document mode.
Live Preview (Design view)	This option is disabled in Document mode.
Document (Design view)	Once you are in Document mode, it is selected by default in the Design group. The document displays on the canvas, to which you can add text, images, lines, reports, and charts.
Data from Source	Uses the selected data source to display a live preview of the output on the canvas.
Use Sample Data	Displays sample data, which reduces processing time by eliminating the need to access the actual data source.
Records	Limits the number of rows retrieved from the data source when Live Preview is selected. This feature is useful in reducing response time if you are working with a large amount of data. Type the number of rows that you want directly in the Records field, or use the drop-down menu to select one of the preset record limits. The preset choices are All rows, 1, 10, 50, 100, 500, 1000, 2000, 5000, and 10000.
Filter Group	
Filter	Opens the Filter dialog box for creating filters. Filters enable you to select only the data that you want and to exclude unwanted data.
Exclude	Turns off a filter.
Include	Turns on a filter.
Clipboard Group	

Command	Description
Paste	Enables you to paste a text, report, or chart object that you have copied to or placed on the clipboard.
Cut	Enables you to cut a text, report, or chart object from your document, placing it on the clipboard.
Copy	Enables you to copy a text, report, or chart object to the clipboard.
Duplicate	Enables you to duplicate a text, report, or chart object in your document, placing it on the clipboard.
Report Group	
Theme	<p>Opens a dialog box where you can select a theme to style your report or chart. You can use the default style sheet by clicking the <i>Use Default Stylesheet</i> button.</p> <p>You can also select a document styling theme or an application theme to style all reports created. Use the Environment and Styling section of the Options window, which is accessible by clicking <i>Options</i> in the Application main menu.</p>
Style	Opens a Report Style dialog box for applying global styling to the entire report. This option is disabled for charts in Document mode. For more information about styling reports, see Styling Reports on page 556.
Banded	Opens a Color dialog box for choosing a color that provides an alternating color scheme for the report. The report output displays alternating rows of data, using a white background for one row and a background of the selected color for the next row. This pattern continues throughout the report. This option is disabled for charts in Document mode.
Header & Footer	Opens the Header & Footer dialog box, from which you can add and style headings and footings.

Command	Description
Column Totals	Adds a grand total row to the bottom of the report to sum numeric data in each column. This option is disabled for charts in Document mode.
Row Totals	Adds a grand total column to the right side of the report to sum numeric data in each row. This option is disabled for charts in Document mode.

Insert Tab

Command	Description
Pages Group	
Page	Adds a new page to a document.
Reports Group	
Report	Inserts a report placeholder on the canvas.
Chart	Inserts a chart placeholder on the canvas.
Existing Report	Opens the Open dialog box, where you can browse to the report that you want to insert in the upper-left corner of the canvas.
Objects Group	
Text Box	Inserts an inline text object in the upper-left corner of the canvas.
Image	Opens the Open dialog box, where you can browse to the image that you want to insert in the upper-left corner of the canvas.
active dashboard Prompts group	
Drop Down	Inserts a drop-down control placeholder in the upper-left corner of the canvas.

Command	Description
List	Inserts a list control placeholder in the upper-left corner of the canvas.
Checkbox	Inserts a check box control placeholder in the upper-left corner of the canvas.
Radio Button	Inserts a radio button control placeholder in the upper-left corner of the canvas.
Text	Inserts a text area control placeholder in the upper-left corner of the canvas.

Format Tab

Command	Description
Destination Group	
InfoMini	Enables the creation of an InfoMini application. For more information on using InfoMini, see Building InfoMini Applications on page 475.
Report	Makes report-specific functionality available. In Document mode, if you select a report object, the Report option is enabled on the Home tab and the ribbon options change. For more information, see Ribbon Commands for Reports on page 497.
Chart	Makes chart-specific functionality available. In Document mode, if you select a chart object, the Chart option is enabled on the Home tab and the ribbon options change. For more information, see Ribbon Commands for Charts on page 511.
File	Creates a data file from a report component of a document.
Navigation Group	

Command	Description
Table	Generates standard browser output. This is the default. This option is only available for reports in Document mode.
Table of Contents	<p>Generates output by displaying a table of contents icon in the upper-left corner where report output typically appears. Clicking <i>Table of Contents</i> opens a menu that enables you to select (view) individual values of the first Sort By (By) field, one value at a time.</p> <p>You can also select options to view the entire report or remove the table of contents. This option is unavailable for charts in Document mode, and is disabled for reports in Document mode.</p>
Freeze	Generates output with column titles that freeze (remain in view) when you scroll through pages of the report output. This option is unavailable for charts in Document mode, and is disabled for reports in Document mode.
Pages On Demand	Provides access to two distinct features depending upon the output type that you have selected. This option is available for reports in Document mode.
Auto Drill	This option is disabled in Document mode.
Features Group	
Title Popup	Displays pop-up titles when the mouse pointer hovers over a column title in the report output.
Accordion	Creates expandable views of data for each vertical sort field. This option displays data values only for the first vertical sort field when you first view the output. You can manually expand your view to expose the data values of lower-level sort fields.
Repeat Sort Value	Displays all repeated sort values instead of blanks in the output after the first instance of a new sort value, which is the default behavior.

Command	Description
Stack Measures	Displays all numeric measure field names in a column of the report output with the corresponding numeric data values.
active report Options	Opens the active report options dialog box where you can configure your active report options such as menu items, graph engine, and colors. For more information, see Using Active Technologies on page 416.
Accessibility	Allows a title to be added to a report, chart, or document that is Section 508-compliant.

Data Tab

Command	Description
Calculation Group	
Detail (Define)	Opens the Detail Field (DEFINE) dialog box, where you can create a defined field, type a name for the field, and enter a format. A Define field is an optional attribute used to create a virtual field for reporting. You can derive the virtual field value from information already in the data source (that is, from permanent fields).
Summary (Compute)	Opens the Summary Field (COMPUTE) dialog box, where you can create a computed field, type a name for the field, and enter a format.
Join Group	
Join	Opens the Join dialog box, where you can create a new join, edit or delete existing joins, and add data sources to a join.
Filter Group	

Command	Description
Filter	Opens the Filter dialog box, enabling you to set filtering options. Filter options include Where, Where Total, the And conjunction, and the Or conjunctions in a single expression.
Display Group	
Missing Data	Includes options for how to display missing values in charts.
Data Source Group	
Add	Opens the Open dialog box, where you can add additional data sources to a document, enabling you to insert reports from different data sources into the same document.
Switch	Opens a drop-down list of all the data sources that have been added. You can choose which data source is currently active and being used to create new reports.

Slicers Tab

Command	Description
Options Group	
New Group	Creates a new group of similar slicers.
Clear Slicers	Resets all slicers so that no filtering is done.
Update Preview	Applies slicers to preview.
Record Limit Group	
Preview	Sets the number of records retrieved from the data source for preview.
Run Time	Sets the number of records retrieved at run time.

Command	Description
Group Number Group	
Group <i>n</i>	Contains a group for each Slicer group that is added. Group 1 is the default slicer group to which you can drag fields to create slicers.

Layout Tab

Command	Description
Page Setup Group	
Margins	This option is disabled in Document mode.
Orientation	Enables you to set the orientation of your report to portrait or landscape.
Size	Enables you to select the size of the paper for printing output. You can choose A3, A4, A5, Letter, Tabloid, Legal, PowerPoint, or Large Size (34 x 44 Inches).
Units	Enables you to select the unit of measurement used for customizing the dimension fields of your report or chart. You can choose Inches, Centimeters, or Points.
Page Numbers	Enables you to select page numbering options. You can choose one of the following: <input type="checkbox"/> No Lead (no space for headers) <input type="checkbox"/> On (page numbers only in headers) <input type="checkbox"/> Off (space for headers, but no page numbering) The Page Numbers value is overridden by header and footer text options.
Size & Arrange Group	
Height	Sets the height of the selected document component.

Command	Description
Width	Sets the width of the selected document component.
Auto Overflow	Automatically expands the query area to show all data.
Aspect Ratio	Lock the height and width aspect ratio.
AutoFit	In Document mode, this option is disabled.
Align	Opens a drop-down menu of available alignment options, when two or more document components are selected.
Relative Position	Positions the top-left corner of the lower component, to the bottom-left corner of the higher component, when two or more document components are selected.
Size and Arrange	Opens the Size and Position dialog box where you can set size and position options for the object in your document.
Report Group	
Cell Padding	Opens the Cell Padding dialog box, where you can set specific values to control the amount of space inserted between rows and columns in a report. For more information, see How to Use Cell Padding in a Report on page 565.
Autofit Column	When working with a report component, this option automatically compresses the columns in the report to the width of the widest data instance. Autofit Column is selected, by default.

View Tab

Command	Description
Design Group	
Query (Design view)	This option is unavailable in Document mode.

Command	Description
Live Preview (Design view)	This option is unavailable in Document mode.
Document	Enables Document mode by default.
Show/Hide Group	
Resources	Minimizes the Resources panel and expands the size of the canvas to also occupy the area where the Resources panel typically appears. The canvas can display a preview of a report, output of a report, or the Query Design pane.
Ruler	Displays a ruler above the canvas and to the left of the canvas for a document.
Grid	Displays a grid as a visual aid for aligning objects in a document.
Relationships	Shows the relative positioning relationship among objects.
Data Panel Group	
Logical	Displays the data source fields by type. This is the default view. The Logical view options include Title, Description, Field, and Alias.
List	Displays the data source fields in a tabular list format. This list contains a header row. You can sort fields differently by clicking a column header. The List view options include Title, Description, Field, Alias, Format, Segment, Filename, and Reference.
Structured	Displays the hierarchical structure of the data source files. The Structured view options include Title, Description, Field, and Alias.
Query Panel Group	
Areas 2x2	Displays data in a two column by two-row grid. When working with the new field container syntax, this option is unavailable.

Command	Description
Areas 1x4	Displays data in a one column by four-row grid. When working with the new field container syntax, this option is unavailable.
Tree	Displays data in a tree. This is the default.
Output Window Group	
Arrange	Opens a drop-down menu where you can choose how to display multiple output windows. The options are Cascade, Tile Horizontally, and Tile Vertically.
Output Location	Opens a drop-down menu where you can choose how to direct new output. The options are Single tab (default), New Tab, Single Window, and New Window.
Switch Output	Opens a drop-down menu for choosing to view any active output window.
Report Group	
Switch Report	Lists any active report or chart to which you can switch.

Field Tab

Command	Description
Filter Group	
Filter	Opens the Filter dialog box for creating filters. Filters enable you to select only the data that you want and to exclude unwanted data.
Exclude	Removes, but does not delete, the filter from the report or chart.
Include	Restores a filter that was previously excluded from a report or chart.

Command	Description
Prompt	<p>Opens the Create a filtering condition dialog box for creating an auto prompting parameter that you can select when you run a report. The Create a filtering condition dialog box is used to create both filters and auto prompting parameters. The following prompt options are available when Parameter is selected from the Type drop-down menu:</p> <ul style="list-style-type: none"><input type="checkbox"/> Simple. This is used for prompts using Text Input. This is the default value.<input type="checkbox"/> Static. This is used for prompts using Selection. This option allows you to select multiple values at run time.<input type="checkbox"/> Dynamic. This is used for prompts using Data Values. This option allows you to select multiple values at run time.<input type="checkbox"/> Optional. This is used for prompts using Single or Multiselect parameters.
Sort Group	
Up	Sorts the selected field in ascending order.
Down	Sorts the selected field in descending order.
Rank	Inserts a rank column immediately to the left of the report if a Sort By field is selected. It also adds a rank column to the left of the Sort By field if a Measure field is selected. Ranking a Measure field results in two copies of the field, the original Measure field, and the Sort By field that is created during ranking.
Group	Opens the Create a Group dialog box where you can create a group to combine values together.

Command	Description
Limit	Opens a drop-down menu that allows you to specify the number of unique values to display for a sort group that has been added.
Break Group	
Page Break	Starts a new page when the primary sort field changes. Clicking the drop-down icon enables you to select <i>Reset Page Numbers</i> , which allows you to reset page numbers on a page break to start at 1.
Line Break	Inserts a line in the report output when the primary sort field changes.
Subtotal	Inserts a line, total text (TOTAL FIELD Value), and subtotals for all numeric fields when the primary sort field changes.
Sub Header	Opens a dialog box where you can type text to add a subheading just below the column titles in the report output when the primary sort field changes.
Sub Footer	Opens a dialog box where you can type text to add a subfooting at the end of the data on each page of the report output when the primary sort field changes.
Style Group	
Note: The options in this group are only available for reports in Document mode.	
Font	Opens the Font list, which you can use to change the font.
Font Size	Opens the Font Size list, which you can use to change the numeric value for the font size.
Font Color	Opens the Color dialog box, where you can select the font color.
Style Reset	Resets all settings to the default settings from the template.

Command	Description
Bold	Applies bold font formatting to the selected text.
Italic	Applies italic font formatting to the selected text.
Underline	Underlines the selected text.
Justify Left	Aligns the text to the left of the visual.
Justify Center	Aligns the text to the center of the visual.
Justify Right	Aligns the text to the right of the visual.
Background Color	Opens the Color dialog box, where you can select the background color for the visual.
Data Style	Styles only the data for the selected data source field.
Title Style	Styles only the column title for the selected data source field.
Data + Title	Styles both the data and the column title for the selected data source field.

Format Group

Note: The options in this group are only available for reports in Document mode.

Change currency options	Changes the currency options for the selected field.
Percent	Specifies the value of the field as a percentage.
Comma	Specifies the use of commas for the selected field.
Increase Decimal Places	Increases the number of decimal places that display for the selected field.
Decrease Decimal Places	Decreases the number of decimal places that display for the selected field.

Display Group

Hide Field	Allows you to hide a selected field.
------------	--------------------------------------

Command	Description
Hide Missing	Allows you to hide fields that have no value.
Aggregation	Opens a drop-down menu of the following options: None (default), Sum, Average, Count, Count Distinct, Percent of Count, First Value, Last Value, Maximum, Minimum, Total, Percent, Row Percent, Median, Average Square.
Traffic Lights	Opens the Traffic Light Condition dialog box. From this dialog box, you can add new conditional styling by applying traffic light (and other) colors to a selected field in the output when the field meets specified criteria, modify existing conditional styling, and enable conditional drill-down.
Within	Allows you to use specific aggregation tasks at different report levels. You can use the Within phrase to manipulate display field values as they are aggregated within a sort group rather than a report column.
Data Bars	Adds a data visualization column to the right of a selected numeric field. The column displays values in each row using horizontal bars that extend from left to right in varying lengths, depending on the corresponding data values.
Column(s)	Allows you to indicate the number of columns in which you wish to display multiple graphs. The value can be between 1 and 512. The default is 1. This option is also available from the Query Design pane shortcut menu for a Multi-graph component.
Links Group	

Command	Description
Drill Down	Opens the Drill Down dialog box, where you can configure a hyperlink or a drill-down procedure for the selected field. Clicking that field in the report output, at run time, redirects you to the URL you specified or executes the indicated procedure. This option is disabled in Document mode.

Series Tab

In Document mode, the Series tab is enabled for chart components.

Command	Description
Select Group	
Series drop-down list	Lists the available series in the current chart.
Style Group	
Style	Opens the Format Series dialog box, where you can edit the styling options for the selected series. You can also open this dialog box by right-clicking a series, and then clicking <i>More Style Options</i> .
Properties Group	
Data Labels	<p>Adds data labels to the chart. The drop-down menu contains the following data position options for selecting where to display data values as labels on a chart: Above (default), On top edge, Below top edge, Center, and Base. If you are working with a Pie chart, the options are: On Slice, Outside Slice, and Outside with feeler lines.</p> <p>Clicking <i>More Data Label Options</i> opens the Format Labels dialog box, where you can further edit your data labels.</p>
Trendline	Opens a drop-down menu that provides options for adding a trendline to a chart.

Command	Description
Line Group	
Smooth Line	Draws the chart using smooth lines.
Connect Lines	Controls the display of connecting lines between markers on a line or scatter chart. By default, lines are connected on a line chart and disconnected on a scatter chart.
Marker	Opens a drop-down menu from which you can select options to change the display of the default data and legend markers on line and scatter chart types. For more information, see How to Change the Appearance of a Marker on page 581.
Pie group	
Note: The following options are only enabled when you are working with a pie chart.	
Expand	Expands pie slices.
Hide	Hides pie slices.

Ribbon Commands for Visualizations

When creating and customizing visualizations in Visualizations mode, you can use the following ribbons and commands to customize visualization functionality.

Home Tab

Command	Description
Clipboard Group	
Paste	Enables you to paste a text, report, or chart object that you have copied to or placed on the clipboard.
Cut	Enables you to cut a text, report, or chart object from your document, placing it on the clipboard.

Command	Description
Copy	Enables you to copy a text, report, or chart object to the clipboard.
Duplicate	Enables you to duplicate a text, report, or chart object in your document, placing it on the clipboard.
Data Group	
Calculation	Opens a menu from which you can choose to create Define or Compute fields.
Join	Opens the Join dialog box, where you can create a new join, edit or delete existing joins, and add data sources to a join. You can also create blends, which allow you to combine data from local or system resources into your current data source.
Visual Group	
Insert	Inserts a new visual. The left side of the Insert button inserts the default visual, which is a stacked bar chart. Click the down arrow next to the Insert button to specify a visual type, such as a chart, grid, or text.
Change	Opens the Select a Visual menu, from which you can select the type of chart, map, or grid that you want to add to your visualization.

Command	Description
Swap	<p>Changes the vertical or horizontal orientation of data in a visual. When you add one or more fields to the canvas, Swap is enabled. When you have data fields in the vertical and horizontal field containers, clicking Swap switches the axis of these data fields to display on the opposite axis. When working with matrix charts, the rows and columns are similarly switched when you click Swap. Swap is available for bar, line, area, scatter, bubble, and matrix marker charts. It is also available for grids.</p> <p>Note:</p> <ul style="list-style-type: none"> <input type="checkbox"/> Swap is disabled when the canvas is empty. <input type="checkbox"/> Swap is unavailable for maps.
Clear	<p>Clears the current visual. You can use the split button to select the option to clear a component, or the entire visualization, which clears all visuals on the canvas. If you have created a filter, you can also clear it. The Clear button is disabled until you begin developing a visual on the canvas.</p>
Storyboard Group	
Add	<p>Takes a snapshot of the current visual or visualization, adding it to the storyboard.</p>
Show	<p>Opens your storyboard as a PowerPoint presentation, where you can choose to view or save your storyboard. All storyboards display in Microsoft PowerPoint format.</p>

Format Tab

Command	Description
Report Group	

Command	Description
Theme	<p>Opens the Templates dialog box, where you can select a theme to style your grid. You can use the default style sheet by clicking the <i>Use Default Stylesheet</i> button.</p> <p>You can also select a styling theme for your grid or an application theme to style all visualizations created.</p>
Header & Footer	Opens the Header & Footer dialog box, from which you can add and style headings and footings.
Column Totals	Adds a grand total row to the bottom of the grid object to sum numeric data in each column.

Features Group

Note: These options do not display for grids or maps.

Reference	Opens the reference menu, where you can access the Add Reference Line to Y-Axis and Add Reference Line to X-Axis options. Selecting one of these options opens the Reference Line dialog box, where you can set the following: the X-axis or Y-axis value, the text that you want to appear, and the position of the reference line on a chart.
Grid	Opens the grid menu, where you can access the Horizontal or Vertical Gridlines options. Both selections allow you to enable or disable Major and Minor Gridlines. Clicking <i>More Grid Lines Options</i> opens the Format Grid Lines dialog box.
Frame & Background	Opens the Frame & Background dialog box, where you can edit the background style and frames for charts. This dialog box contains different options depending on the chart type that you have selected.

Labels Group

Note: These options do not display for grids or maps.

Command	Description
Axes	Opens the Axes menu, where you can enable and rotate horizontal and vertical axis labels, as well as stagger horizontal axis labels. You can also edit the axis labels by clicking <i>More Vertical Axis Options</i> or <i>More Horizontal Axis Options</i> .
Legend	Opens the Legend menu, where you can select the Show Legend option to display the legend on the chart. You can also clear your selection to hide the legend. In addition, you can change the default legend position and orientation.

Interactive Group

Note: These options do not display for grids or maps.

Interactive Options	Opens the Interactive Options dialog box, which enables you to specify animation and mouse over effects in your chart.
---------------------	--

View Tab

Command	Description
Show/Hide Group	
Resources	Minimizes the Resources panel, expanding the size of the canvas to occupy the area where the Resources panel typically appears. When you click the Resources button again, the Resources panel displays and the chart, map, or grid adjusts accordingly.
Data Panel Group	
Logical	Displays the data source fields by type. This is the default view. The Logical view options include Title, Description, Field, and Alias.

Command	Description
List	Displays the data source fields in a tabular list format. This list contains a header row. You can sort fields differently by clicking a column header. The List view options include Title, Description, Field, Alias, Format, Segment, Filename, and Reference.
Structured	Displays the hierarchical structure of the data source files. The Structured view options include Title, Description, Field, and Alias.
Report Group	
Switch Report	Lists any active report or chart to which you can switch.

Field Tab

Command	Description
Filter Group	
Note: Options on the Field tab are contingent on the visual type that you select.	
Filter	Opens the Filter dialog box for creating filters. Filters enable you to select only the data that you want and to exclude unwanted data.
Exclude	Select an existing filter in the Filter pane and choose <i>Exclude</i> to remove, but not delete, the filter from the visual.
Include	Select an existing filter in the Filter pane and choose <i>Include</i> to restore the filter, which was previously excluded from the visual.
Sort Group	
Up	Displays the data source fields by type. This is the default view. The Logical view options include Title, Description, Field, and Alias.

Command	Description
Down	Displays the data source fields in a tabular list format. This list contains a header row. You can sort fields differently by clicking a column header. The List view options include Title, Description, Field, Alias, Format, Segment, Filename, and Reference.
Group	Opens the Create a Group dialog box where you can create a group to combine values together. For grids, this option is activated when you select a dimension.
Display Group	
Hide Field	Hides a selected field.
Aggregation	Specifies an aggregation for a specific field. Commonly used options include: Sum, Average, Count, Minimum, and Maximum.
Traffic Lights	Specifies conditional styling to the selected field.

Series Tab

Command	Description
Select Group	
Note: This group is not available for grids. For maps, this group is disabled.	
Series drop-down list	Lists the available series in the current visualization.
Style Group	
Style	Opens the Format Series dialog box, where you can edit the styling options for the selected series. You can also open this dialog box by right-clicking a series, and then clicking <i>More Style Options</i> .
Properties Group	

Command	Description
Data Labels	Adds data labels to the chart. The drop-down menu contains the following data position options for selecting where to display data values as labels on a chart: Above (default), On top edge, Below top edge, Center, and Base. If you are working with a Pie chart, the options are: On Slice, Outside Slice, and Outside with feeler lines.
Trendline	Opens a drop-down menu that provides options for adding a trendline to a chart.
Line Group	
Smooth Line	Draws the chart using smooth lines.
Connect Lines	Controls the display of connecting lines between markers on a line or scatter chart. By default, lines are connected on a line chart and disconnected on a scatter chart.
Marker	Opens a drop-down menu from which you can select options to change the display of the default data and legend markers on line and scatter chart types. For more information, see How to Change the Appearance of a Marker on page 581.
Pie Group	
Note: The following options are only enabled when you are working with a pie chart.	
Expand	Expands pie slices.
Hide	Hides pie slices.

Customizing Content

After you create your content, you can use the WebFOCUS Business User Edition's vast array of features and styling options to customize your content.

In this chapter:

- ☐ [Customizing Reports](#)
 - ☐ [Customizing Charts and Visualizations](#)
 - ☐ [Using Storyboards](#)
 - ☐ [Animating Visualizations](#)
 - ☐ [Using Filters to Customize the Display of Data](#)
 - ☐ [Using Auto Drill](#)
 - ☐ [Using the Auto Linking Feature to Link Content](#)
 - ☐ [Using Multi Drill](#)
 - ☐ [Adding Parameters for Data Selection at Run Time](#)
 - ☐ [Working With Slicers](#)
 - ☐ [Changing a Field Format](#)
 - ☐ [Dynamic Grouping](#)
-

Customizing Reports

You can customize reports using the various options on the Layout tab and Field tab.

- ☐ **Cell Padding.** Opens the Cell Padding dialog box, where you can set specific values to control the amount of space inserted between rows and columns in a report. For more information, see [How to Use Cell Padding in a Report](#) on page 565.

- ☐ **AutoFit.** Limits the width of columns in a report to be no wider than the largest value in each column. When additional fields are added, the report automatically grows at design time. AutoFit Column is selected, by default.
- ☐ **Page Break.** Starts a new page when the primary sort field changes. Clicking the drop-down icon enables you to select *Reset Page Numbers*, which allows you to reset page numbers on a page break to start at 1.
- ☐ **Line Break.** Inserts a line in the report output when the primary sort field changes.
- ☐ **Sub Header.** Opens a dialog box where you can type text to add a subheading just below the column titles in the report output when the primary sort field changes.
- ☐ **Sub Footer.** Opens a dialog box where you can type text to add a subfooting at the end of the data on each page of the report output when the primary sort field changes.
- ☐ **Subtotal.** Inserts a line, total text (TOTAL FIELD Value), and subtotals for all numeric fields when the primary sort field changes.

The drop-down menu on the *Field* tab, in the *Format* group provides three field-type options for the selected column, which are Alphanumeric, Integer, and Decimal. Selecting the fourth option, More options, opens the Field Format Options dialog box, which provides further formatting options for the selected field. For more information, see [Changing a Field Format](#) on page 673.

You can also apply floating and non-floating currency, percent signs, and commas. You can increase and decrease decimal places by clicking the appropriate buttons under the drop-down menu.

The Currency button is a split button that turns the currency symbol on or off. Click the *Currency* button to turn on the default floating currency option. Click the *Currency* button a second time to turn off the floating currency option.

To turn on the non-floating currency option, open the drop-down menu and select *Non floating currency*. To turn off the non-floating currency option, click the *Currency* button.

The following options on the Field tab, in the Display group:

- ☐ **Hide Field.** Allows you to hide a selected field.
- ☐ **Hide Missing.** Allows you to hide fields that have no value.
- ☐ **Aggregation.** Opens a drop-down menu of the following options:
 - ☐ None (default)

- ☐ Sum
- ☐ Average
- ☐ Count
- ☐ Count Distinct
- ☐ Percent of Count
- ☐ Distinct Values
- ☐ First Value
- ☐ Last Value
- ☐ Maximum
- ☐ Minimum
- ☐ Total
- ☐ Percent
- ☐ Row Percent
- ☐ Median
- ☐ Average Square

Sum is the default aggregation type value for all numeric fields added to the Measure Query field container in the Query pane. Changing the Measure Query field container from Sum to Print, Count, or List overrides all assigned aggregation type values. For more information related to charts, see [How to Display Aggregations on Measure Data](#) on page 583.

- ☐ **Traffic Lights.** Opens the Traffic Light Condition dialog box. From this dialog box you can do the following:
 - ☐ add new conditional styling by applying traffic light (and other) colors to a selected field in the output when the field meets specified criteria
 - ☐ modify existing conditional styling
 - ☐ enable conditional drill-down

For more information related to reports, see [Styling Reports](#) on page 556. For more information related to charts, see [Traffic Light Condition Dialog Box](#) on page 575.

- ☐ **Data Bars.** (Reports only.) Adds a data visualization column to the right of a selected numeric field. The column displays values in each row using horizontal bars that extend from left to right in varying lengths, depending on the corresponding data values.
- ☐ **Within.** Allows you to use specific aggregation tasks at different report levels. You can use the Within phrase to manipulate display field values as they are aggregated within a sort group rather than a report column.
- ☐ **Column(s).** Allows you to indicate the number of columns in which you wish to display multiple graphs. The value can be between 1 and 512. The default is 1. This option is also available from the Query pane shortcut menu for a Multi-graph component.

Styling Reports

You can apply custom styling to specific areas of a report. When creating a report, you can perform the following styling customization.

- ☐ Global styling for the entire report.
- ☐ Style data and column titles.
- ☐ Style headings and footings. For more information, see [How to Style Headings and Footings in a Report](#) on page 561.
- ☐ Style the rows of data with alternating colors. For more information, see [How to Style Rows of Data With Alternating Colors in a Report](#) on page 562.
- ☐ Apply traffic light conditional styling to data. For more information, see [How to Apply Traffic Light Conditional Styling to a Report \(By Constant\)](#) on page 562 and [How to Apply Traffic Light Conditional Styling to a Report \(By Field\)](#) on page 564.
- ☐ Increase or decrease the amount of space inserted between rows and columns. For more information, see [How to Use Cell Padding in a Report](#) on page 565.

Procedure: How to Apply Styling to an Entire Report

1. Create a report, or open an existing report.
2. On the *Home* tab, in the *Report* group, click *Style*.
The Report Style dialog box opens.
3. Select any of the following styling options that are available in the Style group.
 - ☐ **Font.** Opens a drop-down list, which you can use to change the font.

- ☐ **Font size.** Opens a drop-down list, which you can use to change the font size value.
 - ☐ **Bold.** Applies bold font formatting to the selected text.
 - ☐ **Italic.** Applies italic font formatting to the selected text
 - ☐ **Underline.** Underlines the selected text.
 - ☐ **Justify Left.** Aligns the text to the left of the canvas.
 - ☐ **Justify Center.** Aligns the text to the center of the canvas.
 - ☐ **Justify Right.** Aligns the text to the right of the canvas.
 - ☐ **Font color.** Opens the Color dialog box, where you can select the font color.
 - ☐ **Background color.** Opens the Color dialog box, where you can select the background color for the report.
- Note:** If you have chosen to specify a color, you must click OK to return to the Report Style dialog box.
- ☐ **Currency Symbol.** Opens a drop-down menu, from which you can choose a currency symbol. Options are US dollar, British Pound, Japanese Yen, Euro, New Israeli Shekel.
 - ☐ **Reset to Quick Styles from Template.** Resets all settings to the default settings from the template.

Note: Reset only works while the Report Style dialog box is open. Once you click *OK*, all changes are committed. To undo global styling after it has been committed, you must use the Undo command on the Quick Access Toolbar.

Preview. Displays the text as you have formatted it.

4. Click *OK*. The report is styled accordingly.

***Procedure:* How to Perform Field-Level Styling in a Report**

You can style data, column titles, or both, in the report output for the selected data source field.

1. Create a report.
2. In the Query pane, select a data source field.

The Field tab appears on the ribbon.

3. In the Style group, click one of the following:

- ☐ **Data Style.** Styles only the data for the selected data source field.
- ☐ **Title Style.** Styles only the column title for the selected data source field.
- ☐ **Data + Title.** Styles both the data and the column title for the selected data source field.

4. Select any of the following styling options that are available in the Style group.

- ☐ **Font.** Opens a drop-down list, which you can use to change the font.
- ☐ **Font size.** Opens a drop-down list, which you can use to change the font size value.
- ☐ **Font color.** Opens the Color dialog box, where you can select the font color.
- ☐ **Reset to Quick Styles from Template.** Resets all settings to the default settings from the template.

Note: Reset only works while the Report Style dialog box is open. Once you click *OK*, all changes are committed. To undo global styling after it has been committed, you must use the Undo command on the Quick Access Toolbar.

- ☐ **Bold.** Applies bold font formatting to the selected text.
- ☐ **Italic.** Applies italic font formatting to the selected text
- ☐ **Underline.** Underlines the selected text.
- ☐ **Justify Left.** Aligns the text to the left of the canvas.
- ☐ **Justify Center.** Aligns the text to the center of the canvas.
- ☐ **Justify Right.** Aligns the text to the right of the canvas.
- ☐ **Background color.** Click the button to open the Color dialog box, where you can select the background color for the report.

Note: When working with font or background colors, you must click *OK* on the Color dialog box to return to the Report Style dialog box.

5. Click *OK*. The report is styled accordingly.

Procedure: How to Add Headings and Footings to a Report

You can make a report more meaningful by adding headings and footings. Headings and footings supply context and key information about a report, such as its purpose and audience. Headings and footings also provide structure, helping you navigate to the detail sought. They enhance visual appeal.

In this procedure, you will add and style a report heading and page heading. The procedure uses sample values, but you can supply values that apply to your own, reports.

Headings and Footings are available in Query Design view, Live Preview, and Document view.

1. Create a report.
2. On the *Home* tab, in the *Report* group, click the *Header & Footer* button.

The Header & Footer dialog box opens.

3. Click the tab for the heading or footing element that you want to add.

For a report, you can add a report heading, page heading, page footing, or report footing. By default, the Report Header tab is selected. In this procedure, accept the default.

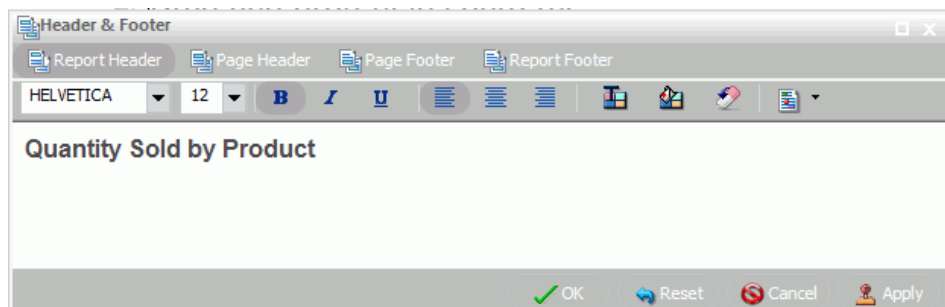
4. Click inside the design area of the dialog box, and type the text for the heading.

For example, the text for a sample report heading might be Quantity Sold by Product.

5. Using the styling ribbon, apply styling to the report heading text.


For example, click *HELVETICA* from the Font drop-down list. From the Font size drop-down list, click 12.

The sample report heading with the selected styling values is shown in the following image.

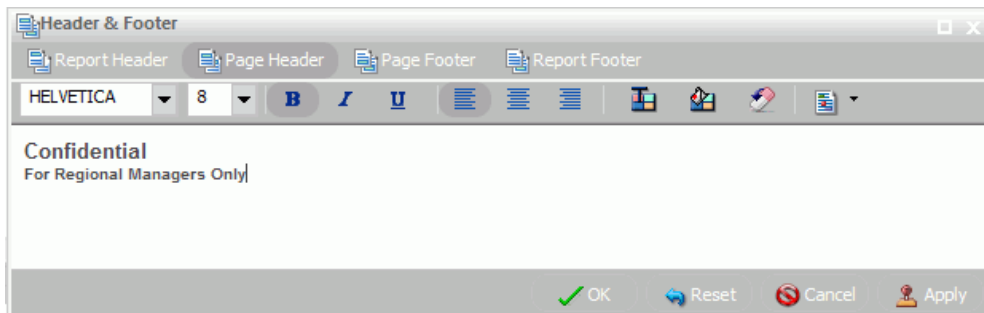


6. Click *Apply* to save the changes you have made so far, without closing the dialog box.
7. To add a page heading, click the *Page Header* tab.

In this procedure, you are going to add one of the supplied quick text options.

8. Click the last icon  (Insert preformatted text content for headers/footers) on the right of the styling ribbon, and click *Confidential* in the list.
9. Change the font and font size. For example, change the font to *HELVETICA* and the size to 10.

You can add your own text before or after the supplied text, for example, For Regional Managers Only.



10. Click OK to save the report heading and page heading and close the Header & Footer dialog box.

The report heading and page heading that you added and styled are shown in Live Preview in the following image.

Quantity Sold by Product	
Confidential For Regional Managers Only	
Product Category	Quantity Sold
Accessories	511,667
Camcorder	455,244
Computers	351,777
Media Player	771,934
Stereo Systems	1,114,332
Televisions	105,188
Video Production	199,749

11. To make changes, to either the report, or page heading, right-click the heading and click *Edit*. You can also double-click on the header to open the Header & Footer dialog box.

Procedure: How to Style Headings and Footings in a Report

You can style headings and footings in the report output for the selected heading or footing field.

1. Create a report.
2. With the report opened in Live Preview, select the heading or footing text that you want to style.
3. On the *Home* tab, in the *Report* group, click *Header & Footer*.

The Header & Footer dialog box opens.

4. Select any of the following styling options that are available in the Style dialog box.

- ☐ **Font.** Opens a drop-down list, which you can use to change the font.
- ☐ **Font size.** Opens a drop-down list, which you can use to change the font size value.
- ☐ **Bold.** Applies bold font formatting to the selected text.
- ☐ **Italic.** Applies italic font formatting to the selected text
- ☐ **Underline.** Underlines the selected text.
- ☐ **Justify Left.** Aligns the text to the left of the canvas.
- ☐ **Justify Center.** Aligns the text to the center of the canvas.
- ☐ **Justify Right.** Aligns the text to the right of the canvas.
- ☐ **Font color.** Opens the Color dialog box, where you can select the font color.
- ☐ **Background color.** Opens the Color dialog box, where you can select the background color for the report.
- ☐ **Reset to Quick Styles from Template.** Resets all settings to the default settings from the template.

Note: Reset only works while the Report Style dialog box is open. Once you click *OK*, all changes are committed. To undo global styling after it has been committed, you must use the Undo command on the Quick Access Toolbar.

5. Click *OK*. The report is styled accordingly.

Procedure: How to Style Rows of Data With Alternating Colors in a Report

You can style rows of data in a report with alternating colors.

1. Create a report.
2. Open the report in Live Preview.
3. On the *Home* tab, in the *Report* group, click *Banded*.

The Color dialog box opens.

4. Select a color.
5. Click *OK*.

The selected color provides an alternating color scheme for the report. The report output displays alternating rows of data, using a white background for one row and a background of the selected color for the next row. This pattern continues throughout the report, as shown in the following image of Banded report output.

Product Category	Quantity Sold
Accessories	511,667
Camcorder	455,244
Computers	351,777
Media Player	771,934
Stereo Systems	1,114,332
Televisions	105,188
Video Production	199,749

Procedure: How to Apply Traffic Light Conditional Styling to a Report (By Constant)

You can apply traffic light conditional styling to data for a selected measure field. By default the report displays the values that satisfy the first condition in green, and the values that satisfy the second condition in red.

1. Open a report in Live Preview.
2. Open the Traffic Light Condition dialog box in one of the following ways:
 - ☐ **Ribbon:** Select a field on the report, and then on the *Field* tab, in the *Display* group, click *Traffic Lights*.
 - ☐ **Shortcut Menu:** Right-click a field on the report, point to *More*, and then click *Traffic Light Conditions*.

The Traffic Light Condition dialog box opens.

3. From the Relational Operators drop-down menu below the field name, click a relational operator. For example, Equal to.
4. In the field to the right of the Relational Operators drop-down menu, click the down arrow for the Type drop-down menu.

The Type menu opens.

5. In the Type drop-down list, click *Constant*.
6. Enter a value in the Value field.

or

- a. From the Get Values drop-down menu, select one of the following values: All, First, Last, Minimum, Maximum, or From File. The value that you select appears in the Get Values field.
 - b. Select the value in the Get Values field. The value that you selected appears in the Value field.
7. Click *OK*.

The value that you selected appears in the field to the right of the Relational Operators drop-down menu.

8. Click the Style button.

The Style menu opens.

9. From the Style menu, click the *Font Color* or *Background Color* button.

The Color dialog box opens.

10. Select a color.
11. Click *OK*.

The color appears in the Preview box.

12. Click Apply to apply the colors to the report.
13. Click the *Drill Down* button.

The Drill Down dialog box opens.

14. In the Drill Down dialog box, use the radio buttons to specify the action you wish to perform. For example, drill down to a report or webpage, create an auto link target, or refresh a BI portal.
15. Click *OK* to close the dialog box.

16. In the Traffic Light Condition window, click the *New* button to set traffic light conditions for additional fields.

Procedure: How to Apply Traffic Light Conditional Styling to a Report (By Field)

You can apply traffic light conditional styling to data for a selected measure field. By default the report displays the values that satisfy the first condition in green, and the values that satisfy the second condition in red.

1. Open a report in Live Preview.
2. Open the Traffic Light Condition dialog box in one of the following ways:
 - ☐ **Ribbon:** Select a field on the report, and then on the *Field* tab, in the *Display* group, click *Traffic Lights*.
 - ☐ **Shortcut Menu:** Right-click a field on the report, point to *More*, and then click *Traffic Light Conditions*.

The Traffic Light Condition dialog box opens.

3. From the drop-down menu below the field name, select a relational operator. For example, *Equal to*.
4. In the field to the right of the Relational Operators drop-down menu, click the arrow for the *Type* drop-down menu.

The *Type* dialog box opens.

5. In the *Type* drop-down list, select *Field*.

The *Type* dialog box displays a list of the data fields that you can choose from.

6. Select a data field from the list.
7. Click *OK*.

The field that you selected appears in the field to the right of the Relational Operators drop-down menu.

8. Click the *Style* button.

The *Style* menu opens.

9. From the *Style* menu, click the *Color* button.

The *Color* dialog box opens.

10. Select a color.

The color appears in the *Preview* box.

11. Click *OK*.
12. Click the *Drill Down* button.

The Drill Down dialog opens.
13. In the Drill Down dialog, specify each of the following:
 - ☐ Drill down to a report or a webpage
 - ☐ URL of the webpage
 - ☐ An alternate comment
 - ☐ Target (New Window, Same Window)
 - ☐ Parameters that you want to use (Name, Value)
14. Click *OK*.
15. Click the *New* button to set traffic light conditions for additional fields.

Procedure: **How to Use Cell Padding in a Report**

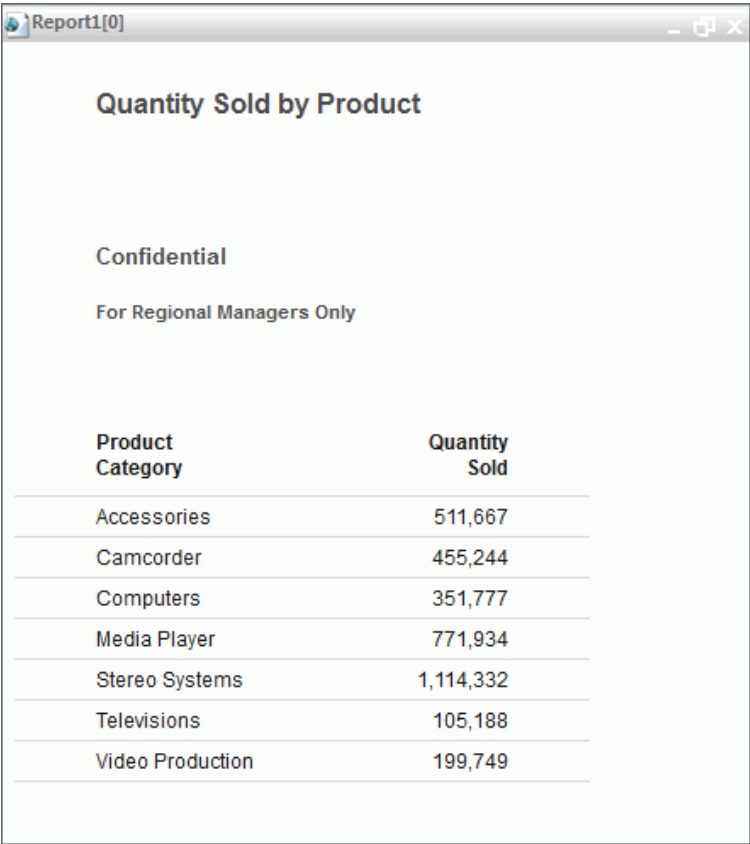
You can customize the amount of space inserted between rows and columns in a report.

1. Open a report in Live Preview.
2. On the *Layout* tab, in the *Report* group, click *Cell Padding*, and then click *Custom*.

The Cell Padding dialog box opens.
3. Type the cell padding values that you want in the Top, Bottom, Left, and Right fields.
4. Click *OK*.

The report reflects the cell padding that you set.

The following image shows a report with custom cell padding.



The screenshot shows a report window titled "Report1[0]". The report content includes a title "Quantity Sold by Product", a confidentiality notice "Confidential For Regional Managers Only", and a table with two columns: "Product Category" and "Quantity Sold". The table lists seven product categories and their corresponding quantities sold.

Product Category	Quantity Sold
Accessories	511,667
Camcorder	455,244
Computers	351,777
Media Player	771,934
Stereo Systems	1,114,332
Televisions	105,188
Video Production	199,749

Report Style Dialog Box

The Report Style dialog box provides options to style your report, as described below.

- ☐ **Font.** Use the drop-down menu to change the font.
- ☐ **Font size.** Use the drop-down menu to change the numeric value for the font size.
- ☐ **Font style.** Click the appropriate button (bold, italic, underline) to style the selected text.
- ☐ **Text alignment.** Click the appropriate button (left, center, right) to align the selected text.
- ☐ **Font color.** Click the button to open the Color dialog box, where you can select the font color. For more information, see [Color Dialog Box](#) on page 567.

☐ **Background.** Click the button to open the Color dialog box, where you can select the background color for the report.

☐ **Currency Symbol.** Click the button to access the following currency symbols US Dollar, British Pound, Japanese Yen, Euro, New Israeli Shekel.

Note: The New Israeli Shekel currency symbol can be displayed with the following settings only:

☐ Server codepage = 1255, 65001, 424, or 65002.

☐ Client codepage = 1255 or 65001.

☐ Application server encoding = Cp1255 or UTF8.

☐ Font name = Lucida Sans Unicode or Arial Unicode MS.

☐ **Reset to Quick Styles from Template.** Click the button to reset all settings to the default settings from the template.

Note: Reset only works while the Report Style dialog box is open. Once you click OK, all changes are committed. To undo global styling after it has been committed, you must use the Undo command on the Quick Access Toolbar.

Color Dialog Box

The Color dialog box provides options for working with color in your report.

The Color dialog box options are:

☐ **Standard Color Swatches.** Provides a set of 48 predefined colors from which to choose on the left side of the dialog box. Your selection appears in the Selected Color field.

☐ **Custom Color Palette.** Provides a palette on the right side of the dialog box to define a custom color. Your selection appears in the Selected Color field.

☐ **Luminosity bar.** Drag the slider to change the luminosity (relative lightness or darkness) of a color. The corresponding numerical value appears in Lum.

☐ **Hue.** Indicates the hue value of your selected color. You can enter a value, or increase or decrease the hue value by using the up and down arrows, respectively.

☐ **Sat.** Indicates the saturation value of your selected color. You can enter a value, or increase or decrease the saturation value by using the up and down arrows, respectively.

- ☐ **Lum.** Indicates the luminosity (lightness or darkness) of your selected color. You can enter a value, or increase or decrease the luminosity value by using the up and down arrows, respectively.
- ☐ **Red.** Represents the numeric value of red (0 to 255).
- ☐ **Green.** Represents the numeric value of green (0 to 255).
- ☐ **Blue.** Represents the numeric value of blue (0 to 255).
- ☐ **Selected Color.** Displays the color that you selected.
- ☐ **Transparent.** Makes the color transparent. This is only available for background colors.

Report Group

The Report group on the Home tab contains commands to enhance a report.

- ☐ **Theme.** Opens a dialog box where you can select a theme to style your report. You can use the default style sheet by clicking the *Use Default Stylesheet* button.

You can also select a document styling theme or an application theme to style all reports created. Use the Environment and Styling section of the Options window, which is accessible by clicking *Options* in the Application main menu.

- ☐ **Style.** Opens a Report Style dialog box for applying global styling to the entire report. For more information about the Report Style dialog box, see [Report Style Dialog Box](#) on page 566. For more information about styling reports, see [Styling Reports](#) on page 556.
- ☐ **Banded.** Opens a Color dialog box for choosing a color that provides an alternating color scheme for the report. The report output displays alternating rows of data, using a white background for one row and a background of the selected color for the next row. This pattern continues throughout the report. For more information about the Color dialog box, see [Color Dialog Box](#) on page 567. For more information about banded styling, see [How to Style Rows of Data With Alternating Colors in a Report](#) on page 562.
- ☐ **Header & Footer.** Opens the Header & Footer dialog box, from which you can add and style headings and footings.
 - ☐ For a report, you can add and style report headings, page headings, page footings, and report footings. In the dialog box, the tab for Report Header is active by default.
 - ☐ For a chart, you can add and style page headings and page footings. Page Header is active by default.


- ☐ You can drag fields from the Data pane into the Header & Footer dialog box.

Another way to access the Header & Footer dialog box is to click the down arrow next to the Header & Footer button. It opens a drop-down menu from which you can select the heading or footing that you want to work with (Report Header, Page Header, Page Footer, Report Footer). After you make your selection, the Header & Footer dialog box opens, and the heading or footing that you selected is active.

From the Header & Footer dialog box, you can add and style the active heading or footing, or choose a different heading and footing to work with by selecting the applicable tab. You can switch among tabs, but InfoAssist+ does not save changes made on the tabs until you click Apply or OK. If you click Apply, the Header & Footer dialog box remains open. If you click OK, the dialog box closes.

You can style a selected heading or footing using the options on the styling toolbar. From left to right, you can customize the font type, font size, and font style (bold, italic, or underline). You can justify text (left, center, or right), select the font color and background color, and restore styling settings to their default value from the template.

You can also add page footings to your reports or charts. From the Header & Footer dialog box, select Page Footer. When working with reports, you may use the Additional alignment


options icon , to specify whether you want to align the footer relative to the data (default) or relative to the page. The Align Relative to Data option places the footer directly below the data, and the Align Relative to Page option positions the footer at the bottom of the page. When working with charts, only the default page footer alignment option is supported.

Note: The Align Relative to Page option works with reports that use positioned formats (for example, PDF, PS, DHTML, PPT, and PPTX).

You can also insert *quick text* into a heading or footing. Quick text is supplied for you. It includes information that is typically useful in identifying a report or chart. From the preformatted text drop-down menu, you can select:

- ☐ Draft
- ☐ Page X of Y
- ☐ Confidential
- ☐ Date (multiple formats)
- ☐ Time (multiple formats)

- ☐ Created by (followed by a name)

For charts, an icon  on the far right of the styling ribbon is enabled. This icon provides two options for controlling the way in which the page heading and page footing are rendered. The default option, Create Header and Footer as text, renders the heading and footing as text elements that are separate from the chart image. The option, Embed Header and Footer in the chart, renders the heading and footing text as part of the chart image.

Once you have added a heading or footing to a report or chart, you can double-click it on the canvas in design mode to reopen the Header & Footer dialog box. You can also right-click an existing heading or footing in design mode and click *Edit* from the menu to open the dialog box.

For more information on adding and styling headings and footings, see [How to Add Headings and Footings to a Report](#) on page 559.

- ☐ **Column Totals.** (Reports only) Adds a grand total row to the bottom of the report to sum numeric data in each column.
- ☐ **Row Totals.** (Reports only) Adds a grand total column to the right side of the report to sum numeric data in each row.

Report Features

The options on the Format tab, in the Features group, while in Report mode, allow you to add pop-up titles to columns, stack measures, and more.

- ☐ **Title Popup.** Displays pop-up titles when the mouse pointer hovers over a column title in the report output.
- ☐ **Accordion.** Creates expandable views of data for each vertical sort field. This option displays data values only for the first vertical sort field when you first view the output. You can manually expand your view to expose the data values of lower-level sort fields.
Note: You cannot use the Table of Contents with the Accordion feature.
- ☐ **Repeat Sort Value.** Displays all repeated sort values instead of blanks in the output after the first instance of a new sort value, which is the default behavior.
- ☐ **Stack Measures.** Displays all numeric measure field names in a column of the report output with the corresponding numeric data values.
- ☐ **active report options.** Opens the active report options dialog box where you can configure your active report options such as menu items, graph engine, and colors.

The following table lists the output for which each feature is available. Yes means the output is available for the feature. No means that the output is not available for the feature.

	HTML	HTML5	active report	PDF	Excel	PowerPoint
Title Popup	yes	no	no	no	no	no
Accordion	yes	no	yes	no	no	no
Repeat Sort Values	yes	yes	yes	yes	yes	yes
Stack Measures	yes	no	no	no	no	no
active report options	no	no	yes	no	no	no

Customizing Charts and Visualizations

Once you have created a visualization or chart, you can customize and style your content. For example, you can add axis titles or a header. The customization options vary depending on the type of chart or visual you are developing.

Chart Features

The options on the Format tab, in the Features group allow you to add effects and different functionality to your charts.

- ☐ **3D Effect.** Sets the three-dimensional view to on or off. The 3D Effect feature is disabled for 3D, stock, gauge, gauge thermometer, Pareto, spectral map, and funnel chart types. By default, 3D effect is off for all chart types.
- ☐ **Rotate.** Toggles between a vertical display or horizontal display of a chart. For more information, see [How to Rotate a Chart](#) on page 639. The Rotate feature is disabled for pie, scatter, 3D, stock, gauge, gauge thermometer, Pareto, spectral map, and funnel chart types.
- ☐ **Reference.** Opens a drop-down menu that provides the Add Reference Line to Y-Axis and Add Reference Line to X-Axis options. Selecting one of these options opens the appropriate Reference Line dialog box, where you can set the specific X-axis or Y-axis value, type the text that you want, and position the reference line on a chart. For more information, see [How to Display a Static Reference Line](#) on page 639. The Reference feature is disabled for pie, 3D, stock, gauge, gauge thermometer, Pareto, spectral map, and funnel chart types.

- ❑ **Annotate.** Opens a drop-down menu that provides the Add an annotation option. Selecting this option opens the Annotation dialog box, where you can type the text that you want and position the annotation on a chart.
- ❑ **Grid.** Opens a drop-down menu allowing you to expand options for Horizontal or Vertical Gridlines. Both selections allow you to enable or disable Major and Minor Gridlines. Clicking *More Options* opens the Format Grid Lines dialog box. For more information, see [Formatting Gridlines](#) on page 598.
- ❑ **Frame & Background.** Opens the Frame & Background dialog box where you can edit the background style and frames for charts. The dialog contains different options depending on the chart type selected. For more information, see [Formatting a Frame and a Background](#) on page 617.
- ❑ **Gauges.** Opens the Gauge dialog box where you can edit your gauge chart. This button is only available when a gauge chart type is selected. For more information, see [How to Style a Gauge Needle](#) on page 634.
- ❑ **active report Options.** Opens the active report options dialog box where you can configure your active report options, such as menu items, graph engine, and colors. This button is available when the output type is set to active report.

Note: Annotate is not available in HTML5.

Labeling Charts

You can add labels to charts and visualizations using the Labels group on the Format tab.

- ❑ **Axes.** Opens a drop-down menu, where you can enable, stagger, and rotate horizontal and vertical axis labels. You can also edit the axis labels by clicking *More Horizontal Axis Options* or *More Vertical Axis Options*. For more information, see [Formatting Axis Labels](#) on page 608.
- ❑ **Legend.** Opens a drop-down menu, where you can select the *Show Legend* option to display the legend on the chart, or clear your selection to hide the legend, change the default legend position, and change the default legend orientation. For more information, see [Format Legend Dialog Box](#) on page 594.

Accessing Chart Formatting Tools

Your presentation of data on a chart or visualization is successful when it communicates to your audience the message that you intend. InfoAssist+ helps you meet the needs of your audience and convey your message by providing numerous chart features. For example, you can adjust the appearance of a chart, add layers of information, or customize the labels that identify the data on the chart.

Using Live Preview

In Live Preview, the canvas on the right of the window provides a preview of the content that you can interact with. The preview is context sensitive, meaning that depending on what portion you select different options become available.

In Live Preview, when you hover the mouse over a graph element (for example, legend, axis label, title), the bounding area is highlighted with a dotted line.

In Live Preview, when you select a graph element (for example, legend, axis label, title), the bounding area is highlighted with a solid line.

Once you select a chart element, you can access all available design options on the ribbon, or you can right-click an element to open a shortcut menu of frequently-used design options. Once you have selected your design option from the ribbon or the menu, InfoAssist+ instantly applies it to the chart element, so that you see the result immediately.

Shortcut menus are enabled for charts that are generated with either sample data, or live data from your data source.

The following sections describe the chart elements and the ribbon options that you can work with to design your charts in Live Preview.

Related Information:

❏ [Using Field Containers](#) on page 91

Formatting a Series

A series is a measure field that is included in a chart or visualization. You can format a series in a variety of ways. For example, you can change the color of a series, add a trendline to a series, or change the appearance of markers on a series.

You can access the full set of formatting options on the Series tab and Field tab. For more information, see [Series Tab](#) on page 81 and [Field Tab](#) on page 80.

You can also access a subset of frequently-used options by right-clicking a series element on a chart to open a menu of those options.

Tip: The options that you see on the menu depend on the type of chart that you are creating. For example, the Series Type option would not appear on the menu for a pie chart, but it would appear for a bar, line, and area chart.

Associated Dialog Boxes

Whether you access series options from the ribbon or the shortcut menu, you are presented with a dialog box of options. The following dialog boxes are commonly used for formatting a series:

- ☐ Format Series
- ☐ Edit Title
- ☐ Traffic Light Condition

For Instructions on how to open these dialog boxes, see the procedures in [Using Series Properties](#) on page 578.

Format Series Dialog Box

The Format Series dialog box contains options to format the fill and border of each series on a chart. To access this dialog box, on the *Series* tab, in the *Style* group, click *Style*.

The Format Series dialog box contains the following tabs:

- ☐ Fill
- ☐ Border

Use the Fill tab to modify the color of a chart series.

The Fill tab contains the following options:

- ☐ **No fill.** Select this option to remove the color from the series.
- ☐ **Solid fill.** Select this option to display the Color and Transparency options.
 - ☐ **Color.** Click this icon to open the Color dialog box, where you can select a color for the series.
 - ☐ **Transparency.** Move the slider to make the bands opaque (0%) or transparent (100%). The default is 0%.

- ☐ **Gradient fill.** Select this option to display the direction of the gradient, the color pattern of the gradient, and the degrees of transparency for the two colors that make up the gradient. A gradient is a smooth color transition or blending of one color to another. The number of colors to use in a gradient is defined by the *stop* or *pin* elements.
- ☐ **Direction.** Select from this drop-down menu to set the direction of the gradient fill. For example, Gradient right or Gradient left.

Use the Border tab to specify a border for a chart series.

The Border tab contains the following options:

- ☐ **Show Border Color.** Select this option to show a border color around each series.
- ☐ **Border Color.** Click this icon to open the Color dialog box, where you can select a color for the border.

Edit Title Dialog Box

To edit the title of a series, right-click a series on the canvas, and click *Change Title*. The Enter Title dialog box contains a text field in which you can type the title for a series on a chart. Click *OK* and the title appears on the chart.

Traffic Light Condition Dialog Box

The Traffic Light Condition dialog box contains fields for adding new conditional styling or modifying existing conditional styling by applying a traffic light color to the selected field.

The Traffic Light Condition dialog box contains the following fields.

- ☐ **Relational Operators.** Select from this drop-down menu to set the relational operator. For example, Equal to.
- ☐ **Type/Value.** Click this unlabeled field to open a dialog box that contains the following fields:
 - ☐ **Type.** Opens a drop-down menu of the following values Constant and Field. Select *Constant* to enter a constant value. Select *Field* to open a visual display of the fields in your data source.
 - ☐ **Value:** Enables you to specify a value based on the Type that you select.

Note: If you are creating a Traffic Light condition on a full date field, the Value field will have a calendar icon adjacent to it. You can use this icon to select a date using a calendar control.

☐ **Get Values.** Select a value option from this drop-down list. For example, All or First.

The Traffic Light Condition dialog box contains the following buttons:

- ☐ **Selected Condition.** Click this icon to select a condition to work on.
- ☐ **New.** Creates a new rule.
- ☐ **Delete.** Deletes a rule.
- ☐ **Color.** Opens the Color dialog box.
- ☐ **Drill Down.** Opens the Drill Down dialog box, where you can drill down to a webpage or a URL. Specify the following:
 - ☐ URL of the webpage or location of the report
 - ☐ A description
 - ☐ Target (New Window, Same Window, a value that you enter)
 - ☐ Parameters that you want to use (Name, Value)

Series Elements Shortcut Menu

When you right-click a series, a menu of options opens. The menu contains options that are available on the Field and Series tabs.

The menu options are described in the following table. The table provides links to the sections of this document in which those options are also discussed.

Option	Description
Filter Values	Enables you to create or modify a WHERE statement, using the Filter dialog box. With a WHERE statement, you select only the data that you want to display, and exclude unwanted data. For information on filtering your data, see Data Tab on page 78 and Field Tab on page 80.
Sort	Enables you to sort the series in either ascending or descending order.

Option	Description
Visibility	<p>Controls the display of the selected series (field) on a chart. The value Hide suppresses the display of the series, and the default value Show displays the series.</p> <p>For instructions, see How to Hide a Field in a Series on page 583.</p>
Change Title	<p>Enables you to edit the title of the selected series. In the Edit Title dialog box, type the new title in the Enter Title field, and click <i>OK</i>.</p>
Series Type	<p>Changes the chart type of the selected series to bar, line, or area. The option None (default) returns the series to the chart type that was in effect before you changed it. This option applies to bar, line, and area chart types only.</p>
Series Color	<p>Enables you to specify the color of the selected series, using the Color dialog box. For more information, see Color Dialog Box on page 567.</p>
More Style Options	<p>Opens the Format Series dialog box. For more information, see Format Series Dialog Box on page 574.</p>
Data Labels	<p>Controls the display of data labels (values) on the selected series. The default value Hide suppresses the display of labels, and the value Show displays labels.</p> <p>This option does not apply to the gauge chart type.</p> <p>For instructions, see How to Show and Hide Data Labels on page 592.</p>
Color Mode	<p>Controls how color is applied to a series (measure field) on a chart. The possible settings are By Series (default) and By Group. For example, assume that there is only one series on a sample bar chart. The By Series setting applies the same color to all the bars in the series. The By Group setting applies a different color to each bar.</p> <p>For instructions, see How to Control the Color Mode on page 587.</p>

Option	Description
Add Trendline	<p>Draws a line on a chart to indicate a statistical trend.</p> <p>This option does not apply to the pie, funnel, 3D, gauge, or stock chart type.</p> <p>For an example of a chart with a trendline, see How to Add a Trendline on page 580.</p>
Drill Down	<p>Opens the Drill Down dialog box, where you can configure a hyperlink or a drill-down procedure for the selected field.</p> <p>Clicking that field in the report output, at run-time, redirects you to the URL you specified or executes the indicated procedure.</p>
More	<p>Contains the Aggregation Functions, Traffic Light Conditions, and Missing options.</p> <p>Aggregation Functions assign an aggregation value to a numeric measure field in a report. For instructions, see How to Display Aggregations on Measure Data on page 583.</p> <p>Traffic Light Conditions enables you to specify the color of numeric measure fields in the output, depending on conditions that you set. You can use the Traffic Light Condition dialog box to specify the conditions and colors.</p> <p>For instructions, see How to Apply Traffic Light Conditions With Drill-Down to a Numeric Measure Field (By Constant) on page 584 and How to Apply Traffic Light Conditions With Drill-Down to a Numeric Measure Field (By Field) on page 585.</p> <p>The Missing option allows you to show or hide fields with no value.</p>
Delete	<p>Removes the selected series from the report and updates the Live Preview accordingly.</p>

Using Series Properties

The following sections contain procedures for customizing a series.

Procedure: How to Select a Series

1. Create a chart or visualization.
2. On the *Series* tab, in the *Select* group drop-down menu, select the Series that you want to customize.

The Series appears in the drop-down menu field.

Procedure: How to Format the Fill and Border of a Series

1. Create a chart or visualization.
2. Open the Format Series dialog box in one of the following ways:

☐ **Ribbon:** On the *Series* tab, in the *Style* group, click *Style*.

☐ **Shortcut Menu:** Right-click a series on the chart, and click *More Style Options*.

The Format Series dialog box opens.

3. Use the fill and border options to format the series.

For more information, see [Format Series Dialog Box](#) on page 574.

4. Click *OK* to close the dialog box.

The Format Series dialog box closes. The series fill and border are formatted accordingly.

5. Click *Run* to generate the report.

Enhancing Series Using the Series Tab

The *Properties* group contains commands for enhancing charts, such as changing the type or adding a trendline, for the selected series.

Procedure: How to Change the Type of a Series

1. Create a bar, line, or area chart.
2. Access the list of series types in one of the following ways:

☐ **Ribbon:** On the *Series* tab, in the *Properties* group, open the *Type* drop-down menu.

☐ **Shortcut Menu:** Right-click a series on the chart, and point to *Series Type*.

3. Select the type that you want the series to become.

The chart contains the new series type.

Procedure: How to Add a Trendline

A trendline is a line that is drawn over the plot area of a chart or visual to show the pattern of data points. The pattern reveals a statistical trend. In particular, the slope of the trendline, which is calculated by subtracting and dividing two different x, y coordinate values, is a value that indicates the rate at which the y value of a line rises or falls as the x value increases. Once the slope of your chart or visual is determined, you can further extrapolate your results and gain further insight into your data.

Note: The mathematical equation for the selected trendline option is only available in chart mode. It is not available in visualization mode.

1. Create a chart or visualization.
2. Access the menu of trendline types in one of the following ways:
 - ☐ **Ribbon:** On the *Series* tab, in the *Properties* group, open the *Trendline* drop-down menu.
 - ☐ **Shortcut Menu:** Right-click a series on the chart or visual, and point to *Add Trendline*.
3. Select the type of trendline that you want to display.

The trendline appears on the canvas.
4. Optionally, to display the mathematical equation for the selected trendline option, on the *Series* tab, in the *Properties* group, click *Equation*.

Formatting Charts Using the Series Tab

The *Series* tab contains options for formatting charts.

Procedure: How to Apply Smooth Line Effect to a Line Chart

1. Create a line chart.
2. Select a series on the line chart.
3. On the *Series* tab, in the *Line* group, click *Smooth Line*.

The Smooth Line effect is applied to the series.

Procedure: How to Hide a Series Line Between Markers

Lines appear between markers by default.

To hide a series line between a marker:

1. Create a line chart.
2. Select a series on the line chart.

3. On the *Series* tab, in the *Series* group drop-down menu, select the series that you want to hide.
4. In the *Line* Group, click *Connect Lines*.

The series line between the markers disappears.

To make the series line reappear, click *Connect Lines* again.

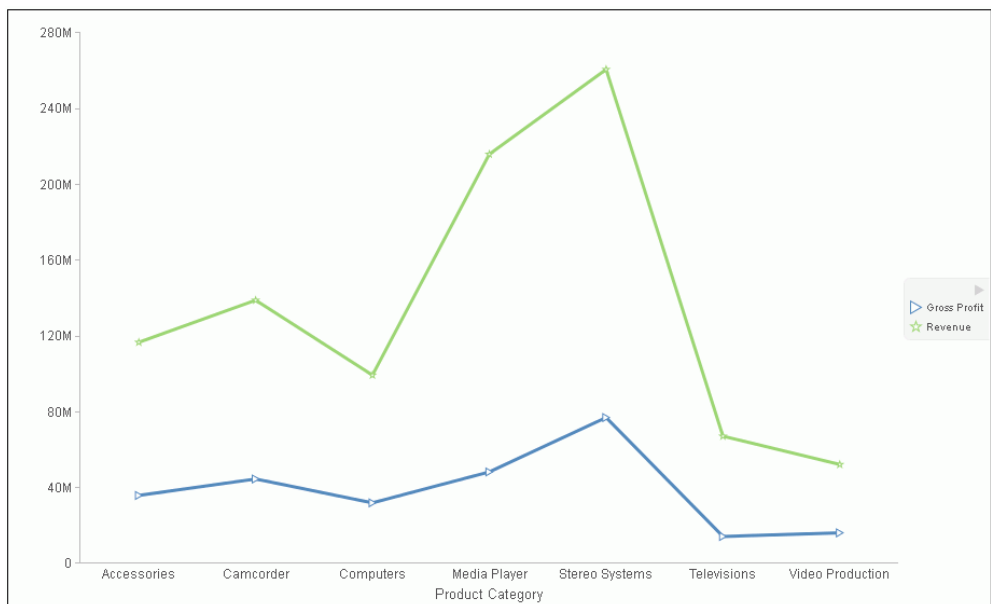
Procedure: How to Change the Appearance of a Marker

Markers are used to display points of data on a line chart. They are also used in the legend to identify the data that is on the chart. The different marker shapes distinguish one series from another.

1. Create a line chart.
2. Select a series on the line chart.
3. On the *Series* tab, in the *Line* group, click *Marker* to open a drop-down menu of options.
4. From the Marker drop-down menu, select the marker shape. For example, Diamond or Hourglass.

The markers are formatted.

The following image shows a triangle marker for gross profit data and a star marker for revenue data.



Procedure: How to Expand Pie Slices

1. Create a pie chart.
2. On the *Series* tab, in the *Select* group, from the drop-down menu, select one of the following:
 - ☐ **All Series** expands all slices out from the center of the pie.
 - ☐ **A specific series** expands that particular slice out from the center of the pie.
3. In the *Pie* group, click *Expand*.
The pie expands accordingly.

Procedure: How to Hide a Pie Slice

1. Create a pie chart.
2. On the *Series* tab, in the *Select* group, from the drop-down menu, select the series that you want to hide. Then, in the *Pie* group, click *Hide*.
The slice is hidden.

Procedure: How to Filter Values in a Series

1. Create a chart.
2. Select a series on the chart.
3. Open the Filter dialog box in one of the following ways:
 - ☐ **Ribbon:** On the *Field* tab, in the *Filter* group, click *Filter*.
 - ☐ **Shortcut Menu:** Right-click a series, and click *Filter Values*.The Filter dialog box opens.
4. Select values for values and prompts.
5. Click *OK* to close the dialog box.
The series values are filtered.

Procedure: How to Sort the Fields in a Series

1. Create a chart.
2. Select a series.

3. Sort the series in one of the following ways:

- ☐ **Ribbon:** On the *Field* tab, in the *Sort* group, click *Up* to sort the series values from smallest to largest, or click *Down* to sort the series values from largest to smallest.
- ☐ **Shortcut Menu:** Right click a series on the chart, and point to *Sort*, and then *Sort* again. Click *Ascending* to sort the series values from smallest to largest, or click *Descending* to sort the series values from largest to smallest. Select *Limit* to open a list of values to display for a sort group.

The chart appears with the series sorted accordingly.

Procedure: How to Hide a Field in a Series

1. Create a chart or visualization.
2. Hide a field in a series in one of the following ways:
 - ☐ Select the field in the Query pane.
 - ☐ Right-click the field in the chart.
 - ☐ **Ribbon:** Select the field in the Query pane or by right-clicking it in the chart. On the *Field* tab, in the *Display* group, click *Hide Field*. Click *Hide Field* again to make the series reappear.
 - ☐ **Shortcut Menu:** Right-click a series in the Query pane, or in the chart, point to *Visibility*, and then click *Hide*. Right-click the same series, point to *Visibility*, and then click *Show* to make the series reappear.

The field is hidden.

Procedure: How to Display Aggregations on Measure Data

You can display numeric measure data using a variety of aggregation values.

1. Create a chart or visualization.
2. Open the list of Aggregation options in one of the following ways:
 - ☐ **Ribbon:** On the *Field* tab, in the *Display* group, click *Aggregation*.
 - ☐ **Shortcut Menu:** Right-click a series, point to *More*, and then *Aggregation Functions*.
3. Select an aggregation function.

The aggregation function is applied to the series.

Note: If you change the Measure (Sum) Query field container in the Query pane from Sum to Print, Count, or List, the change overrides all assigned aggregation values.

Procedure: How to Apply Traffic Light Conditions With Drill-Down to a Numeric Measure Field (By Constant)

1. Create a chart or visualization.
2. Open the Traffic Light Condition dialog box in one of the following ways:
 - ☐ **Ribbon:** In the Query pane, select a field, and then on the *Field* tab, in the *Display* group, click *Traffic Lights*.
 - ☐ **Shortcut Menu:** Right-click a series on the chart, point to *More*, and then click *Traffic Light Conditions*.

The Traffic Light Condition dialog box opens. For more information, see [Traffic Light Condition Dialog Box](#) on page 575.

3. From the Relational Operators drop-down menu below the field name, select a relational operator. For example, Equal to.
4. In the field to the right of the Relational Operators drop-down menu, click the down arrow for the Type drop-down menu.

The Type dialog box opens.

5. In the Type dialog box, select *Constant*.
6. Enter a value in the Value field, or
 - a. From the Get Values drop-down menu, select one of the following values All, First, Last, Minimum, Maximum, From File. The value that you select appears in the Get Values field.
 - b. Select the value in the Get Values field. The value that you selected appears in the Value field.
7. Click *OK*.

The value that you selected appears in the field to the right of the operator drop-down menu.

8. Click the *Color* button.

The Color dialog box opens.

9. Select a color.
10. Click *OK*.

The color appears in the Preview box.

11. Click *OK*.
12. Click the *Drill Down* button.
The Drill Down dialog box opens.
13. In the Drill Down dialog box, specify each of the following:
 - ☐ Drill down to a report or a webpage
 - ☐ URL of the webpage
 - ☐ An alternate comment
 - ☐ Target (New Window, Same Window)
 - ☐ Parameters that you want to use (Name, Value)
14. Click *OK* to close the dialog box.
15. Click the *New* button to set traffic light conditions for additional fields.

Procedure: How to Apply Traffic Light Conditions With Drill-Down to a Numeric Measure Field (By Field)

1. Create a chart or visualization.
2. Open the Traffic Light Condition dialog box in one of the following ways:
 - ☐ **Ribbon:** In the Query pane, select a field, and then on the *Field* tab, in the *Display* group, click *Traffic Lights*.
 - ☐ **Shortcut Menu:** Right-click a series on your chart, point to *More*, and then click *Traffic Light Conditions*.

The Traffic Light Condition dialog box opens. For more information, see [Traffic Light Condition Dialog Box](#) on page 575.

3. From the Relational Operators drop-down menu below the field name, select a relational operator. For example, Greater than.
4. In the field to the right of the operator drop-down menu, click the arrow for the Type drop-down menu.
The Type dialog box opens.
5. In the Type dialog box, select *Field*.

The Type dialog box displays the Dimensions, and Measures and Properties of your data. You can display the data in the following ways:

- ☐ View fields in business order. Select from the following options: Title, Description, Name, or Alias.
- ☐ View fields in a sortable grid. Select from the following options: Name, Title, Alias, Format, Segment, Filename, Description, or Reference.
- ☐ View the hierarchical structure of the data. Select from the following options: Title, Description, Name, or Alias.

6. Select a field.

7. Click *OK*.

The field that you selected appears in the field to the right of the operator drop-down menu.

8. Click the *Color* button.

The Color dialog box opens.

9. Select a color.

The color appears in the Preview box.

10. Click *OK*.

11. Click the Drill Down button.

The Drill Down dialog box opens.

12. In the Drill Down dialog box, specify each of the following:

- ☐ Drill down to a report or a webpage
- ☐ URL of the webpage
- ☐ An alternate comment
- ☐ Target (New Window, Same Window)
- ☐ Parameters that you want to use (Name, Value)

13. Click *OK* to close the dialog box.

14. Click the *New* button to set traffic light conditions for additional fields.

Procedure: How to Change the Title of a Series

1. Create a chart or visualization.

2. Open the Edit Title dialog box in one of the following ways:

☐ **Shortcut Menu:** Right-click a series on the chart, and click *Change Title*.

☐ **Query Pane:** Right-click a series, and click *Change Title*.

The Edit Title dialog box opens.

3. In the Enter Title field, type the new name for the series.
4. Click *OK* to close the dialog box.

The series has a new title.

Procedure: How to Control the Color Mode

When you create a single-series chart, all series groups appear in the same color. To use a different color for each group, set the color mode to *By Group*.

1. Create a chart or visualization.
2. Right-click a series on the chart, point to *Color Mode*, and then click *By Group*.

A different color is applied to each group in the series. To return to the default display of the series in one color, right-click the series, point to *Color Mode*, and then click *By Series*.

3. Click *Run* to generate the report.

Procedure: How to Delete a Series

1. Create a chart or visualization.
2. Right-click a series on the chart, and click *Delete*.

The series is deleted.

Formatting Data Labels

Data labels highlight important data points on a chart. They identify exact numbers. You can customize data labels in a variety of ways to make them stand out more clearly on the chart. For example, you can change the position, angle, color, or size of data labels.

Associated Dialog Boxes

Whether you access data label options from the ribbon or the shortcut menu, you are presented with a dialog box of options. The following dialog boxes are commonly used for formatting data labels:

- ☐ **Format Labels**

- ☐ Style
- ☐ Line Style

For instructions on how to open these dialog boxes, see the procedures in [Using Data Labels Properties](#) on page 592.

Format Labels Dialog Box

The Format Labels dialog box contains options for editing data labels. The Format Labels dialog box offers different options depending on the chart type that you are using. Bar, line, and area charts share the same tabs.

The Format Labels dialog box contains the following tabs:

- ☐ General Options
- ☐ Advanced
- ☐ Pie Title (for pie charts only)
- ☐ Pie Labels (for pie charts only)
- ☐ Funnel Labels (for funnel and pyramid charts)

General Options Tab

Use the General Options tab to add data labels to a chart and set their position, angle, and radius.

The General Options tab contains the following options:

- ☐ **Show Data Labels.** Select this option to show data labels on a chart. Clear this option to suppress data labels.
- ☐ **Position.** Select an option from this drop-down menu to determine where the data label will be positioned. The options are:
 - ☐ Above
 - ☐ Below top edge
 - ☐ Center
 - ☐ Base
 - ☐ Center back
- ☐ **Format Labels.** Select from this drop-down menu of preset formats that can be applied to the labels. Some of the options include Use Pattern, Currency General, and Date Full.

- ☐ **Custom Format.** Enter a standard number format pattern for the data label. This option is only available when you select the *Use Pattern* option from the Format Labels drop-down menu.

The following table describes the characters that you can use in a custom format.

Character	Description
#	Is a digit.
0 (zero)	Shows as absent.
. (period)	Is a placeholder for decimal separator.
, (comma)	Is a placeholder for grouping separator.
; (semicolon)	Separates formats.
- (dash)	Is the default negative prefix.
% (percent)	Divides by 100 and shows as a percentage.
x	Determines that any other characters can be used in the prefix or suffix.
' (apostrophe)	Is used to quote special characters in a prefix or suffix.

- ☐ **Style Labels.** Click this icon to open the Style dialog box, where you can style text. For more information, see [Style Dialog Box](#) on page 591.
- ☐ **Show Cumulative Sums.** Select this option to have the data text labels show cumulative sums. Clear this option to have data text labels show individual sums. This option is available for stacked charts.
- ☐ **Show Stacked Total.** Select this option to display stacked totals. Data position should be set to *Center* to display a stacked total. This option is available for stacked charts.

Advanced Tab

Use the Advanced tab to modify additional data labels properties.

The Advanced tab contains the following options:

- ☐ **Show Zero Labels.** Select this option to display zero values in a chart. Clear this option to display all data values except zero.

- ☐ **Apply color to negative data labels.** Select this option to style negative data labels separately from positive data labels.
- ☐ **Color.** Click this icon to open the Color dialog box, where you can select a color for the negative number.

Pie Title Tab

Use the Pie Title tab to create and style a pie title.

The Pie Title tab contains the following options:

- ☐ **Show Pie Title.** Clear this option to suppress a pie title. Select this option to display a pie title. This is the default option.
- ☐ **Style Title.** Click this icon to open the Style dialog box, where you can style pie title text.

Pie Labels Tab

Use the Pie Labels tab to customize your pie data labels.

The Pie Labels tab contains the following options:

- ☐ **Label Position.** Select from this drop-down menu an option to control the display of feeler lines and labels on a pie chart.
- ☐ **Label Display.** Select from this drop-down menu an option to control the format of labels displayed next to feelers on a pie chart.
- ☐ **Format Labels.** Select from this drop-down menu of preset formats that can be applied to labels.
- ☐ **Custom Format.** Select this option to use a custom format from a list of preset formats. See the following table for a list and description of the characters that you can use in a custom format.
- ☐ **Style Labels.** Click this button to open the Style dialog box, where you can style text.

Ring Label

- ☐ **Show Ring Label.** Select this option to control the display of the total label on a pie ring chart.
- ☐ **Format Labels.** Select from this drop-down menu of preset formats that can be applied to labels.
- ☐ **Custom Format.** Enter a standard number format pattern for the data label. This option is only available when you select the *Use Pattern* option from the Format Labels drop-down menu.

- ☐ **Style Labels.** Click this button to open the Style dialog box, where you can style text.

Feeler Line

- ☐ **Line Style.** Click this button to open the Line Style dialog box, where you can edit the color, weight, and style of the feeler line.

Funnel Labels Tab

Use the Funnel Labels tab to customize the labels on a funnel or a pyramid chart.

The Funnel Labels tab contains the following options:

- ☐ **Label Position.** Select an option from this drop-down menu to control the display of feeler lines and labels on a funnel chart.
- ☐ **Format Labels.** Select from this drop-down menu of preset formats that can be applied to labels.
- ☐ **Custom Format.** Select this option to use a custom format. See the table in the previous section for a list and description of the characters that you can use in a custom format.
- ☐ **Style Labels.** Click this button to open the Style dialog box, where you can style text.
- ☐ **Style Value.** Opens the Style dialog box, where you can style the value.

Feeler Line

- ☐ **Show Feeler Lines.** (Default) Clear this option to suppress feeler lines. Select this option to display feeler lines.
- ☐ **Line Style.** Click this button to open the Line Style dialog box, where you can edit the color, weight, and style of the feeler line.

Note: While some style options, such as Show Pie Title and Show Feeler Lines, are enabled by default, a stylesheet applied to the chart may contain different settings that will override these default settings.

Style Dialog Box

The Style dialog box contains options to style the data labels.

The Style dialog box contains the following options:

- ☐ **Font.** Use the drop-down menu to change the font.
- ☐ **Font size.** Use the drop-down menu to change the numeric value for the font size.

- ☐ **Font style.** Click the appropriate button (bold, italic, underline) to style the selected text.
- ☐ **Text alignment.** Click the appropriate button (left, center, right) to align the selected text.
- ☐ **Font color.** Click the button to open the Color dialog box, where you can select the font color.
- ☐ **Reset to Quick Styles from Template.** Click the button to reset all settings to the default settings from the template.

Note: Reset only works while the Style dialog box is open. Once you click *OK*, all changes are committed. To undo global styling after it has been committed, you must use the *Undo* command on the Quick Access Toolbar.

Line Style Dialog Box

The Line Style dialog box contains options to style lines on a chart.

The Line Style dialog box contains the following options:

- ☐ **Color.** Click this button to open the Color dialog box, where you can select the color for the line.
- ☐ **Weight.** Click this button to open a drop-down menu of line weight options.
- ☐ **Style.** Click this button to open a drop-down menu of line style options.
- ☐ **Reset.** Click this button to reset the line to the default options.

Data Labels Elements Shortcut Menu

When you right-click a data label on a bar, line, or area chart, a menu of the following options opens:

- ☐ **Data Labels.** Point to this option to toggle between Show and Hide.
- ☐ **More Label Options.** Click this option to open the Format Labels dialog box.

The shortcut menu contains options that are available on the Series tab.

Using Data Labels Properties

The following sections contain procedures for customizing data labels.

***Procedure:* How to Show and Hide Data Labels**

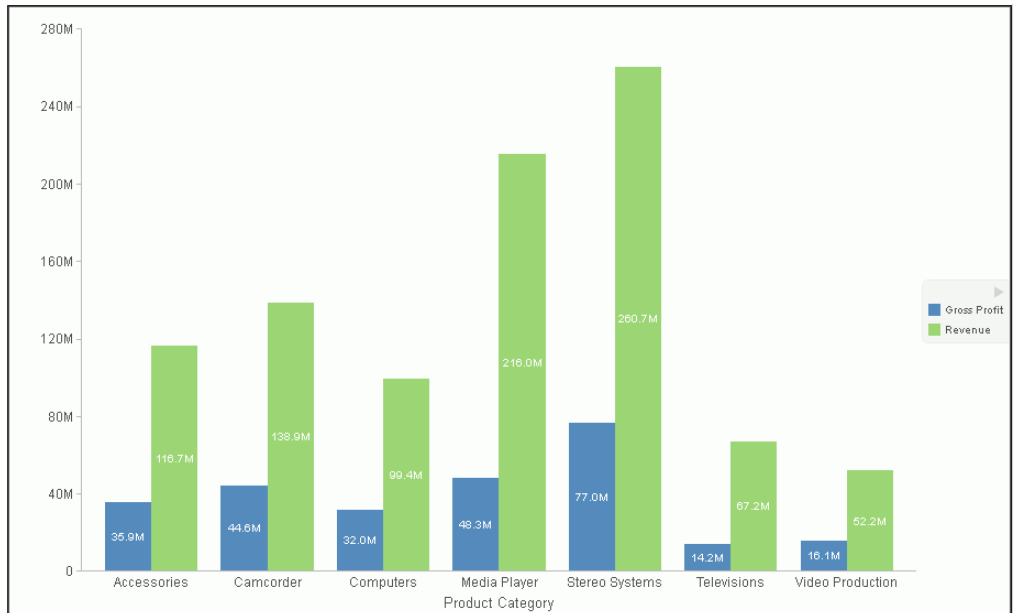
1. Create a chart or visualization.

2. You can access the option to show data labels in one of the following ways:

- ❑ **Ribbon:** On the *Series* tab, in the *Properties* group, click *Data Labels*, and from the drop-down menu, click *More Data Label Options*. The *Format Labels* dialog box opens. On the *General Options* tab, select the *Show Data Labels* check box, and click *OK* to close the dialog box. You can use this dialog box to format and style the data labels. For more information, see [Format Labels Dialog Box](#) on page 588. To hide data labels, clear this option.
- ❑ **Shortcut Menu:** Right-click a series on the chart, point to *Data Labels*, and then click *Show*. To hide data labels, right-click a series on the chart, point to *Data Labels*, and then click *Hide*.

The data labels appear, and are formatted and styled accordingly.

The following image shows a chart with data labels.



Procedure: How to Change the Position of Data Labels

1. On the *Series* tab, in the *Properties* group, click *Data Labels*.
2. On the menu, select the position for the data labels.

The data labels are positioned accordingly.

Formatting a Legend

A legend contains information that is necessary to accurately interpret the data on a chart. By default, a chart displays either a vertical axis title if there is a single measure field, or a legend if there are multiple measure fields.

Format Legend Dialog Box

Whether you access legend options from the ribbon or the shortcut menu, you are presented with the Format Legend dialog box of options. For instructions on how to open this dialog box, see [Using Legend Properties](#) on page 596.

The Format Legend dialog box contains options for formatting a legend on a chart or visualization. It contains the following tabs:

- ☐ Legend Options
- ☐ Markers & Labels
- ☐ Fill
- ☐ Border Styles

For instructions on how to open this dialog box, see the procedures in [Using Legend Properties](#) on page 596.

Use the Legend Options tab to customize the appearance of a legend on a chart or visualization.

The Legend Options tab contains the following options.

- ☐ **Show Legend.** When selected, a legend displays on a chart or visualization. Clear this option to suppress a legend on a chart or visualization.
- ☐ **Legend Position.** Opens a drop-down menu of options to position the legend. For example, top or left.
- ☐ **Reverse Legend Order.** Specifies that the legend be drawn in reverse order. Clear this option to specify that the legend be drawn in normal order.

Use the Markers & Labels tab to customize the appearance of markers and labels on legends.

The Markers & Labels tab contains the following options:

- ☐ **Style Labels.** Opens the Style dialog box, where you can style text.
- ☐ **Marker Position.** Opens a drop-down menu of options to set the position of text relative to the legend marker. For example, Left of Text or Above Text.

Use the Fill tab to modify the color of the legend area. For more information, see [Format Series Dialog Box](#) on page 574.

Use the Border Styles tab to place a border around a legend. For more information, see [Format Series Dialog Box](#) on page 574.

The Border Styles tab contains the following options:

- ☐ **Show Border.** Select this option to place a border around a legend.
- ☐ **Color.** With the Show Border option selected, you can click this button to open the Color dialog box, where you can select a color for the border.

Legend Elements Shortcut Menu

When you right-click a legend on a chart, a menu of options opens. The menu contains options that are available on the Format tab.

The shortcut menu options are described in the following table. The table provides links to the sections of this document in which those options are also discussed.

Option	Description
Show legend	<p>Controls the display of the legend. InfoAssist+ displays the legend by default. When you clear this option, InfoAssist+ suppresses the legend. For instructions, see Using Legend Properties on page 596.</p> <p>The background shortcut menu has an option to restore the legend after it has been suppressed.</p>
Legend Position	<p>Controls the placement of the legend on the chart. For instructions see, Using Legend Properties on page 596.</p>
Legend Area Color	<p>Enables you to specify the color of the legend background area using the Color dialog box.</p> <p>This option is available only when you right-click the area around the legend. For instructions see, Using Legend Properties on page 596.</p>

Option	Description
Legend Border Color	Enables you to specify the color of the border around the legend background area using the Color dialog box. This option is available only when you right-click the area around the legend. For instructions see, Using Legend Properties on page 596.
More Legend Options	Opens the Format Legend dialog box.

Using Legend Properties

The following sections contain procedures for customizing legend. The procedures are organized by the tab and group in which their associated options appear on the ribbon.

Procedure: How to Hide a Legend

1. Create a chart or visualization with multiple measure fields.
2. Clear the *Show legend* option in one of the following ways:
 - ☐ **Ribbon:** On the *Format* tab, in the *Labels* group, click *Legend*, then click *Show legend* to clear the option.
 - ☐ **Shortcut Menu:** Right-click the legend, and clear the *Show legend* option.The legend is hidden.

Procedure: How to Position a Legend

1. Create a chart or visualization with multiple measure fields.
2. Open the menu of label position options in one of the following ways:
 - ☐ **Ribbon:** On the *Format* tab, in the *Labels* group, click *Legend*, then point to *Legend Position*.
 - ☐ **Shortcut Menu:** Right-click the legend, and point to *Legend Position*.
3. Select a position for the legend. For example, *Auto* or *Right*.

Procedure: How to Specify the Color of a Legend Border

1. Create a chart or visualization with multiple measure fields.

2. Open the Format Legends dialog box in one of the following ways:

☐ **Ribbon:** On the *Format* tab, in the *Labels* group, click *Legend*, then click *More Legend Options*.

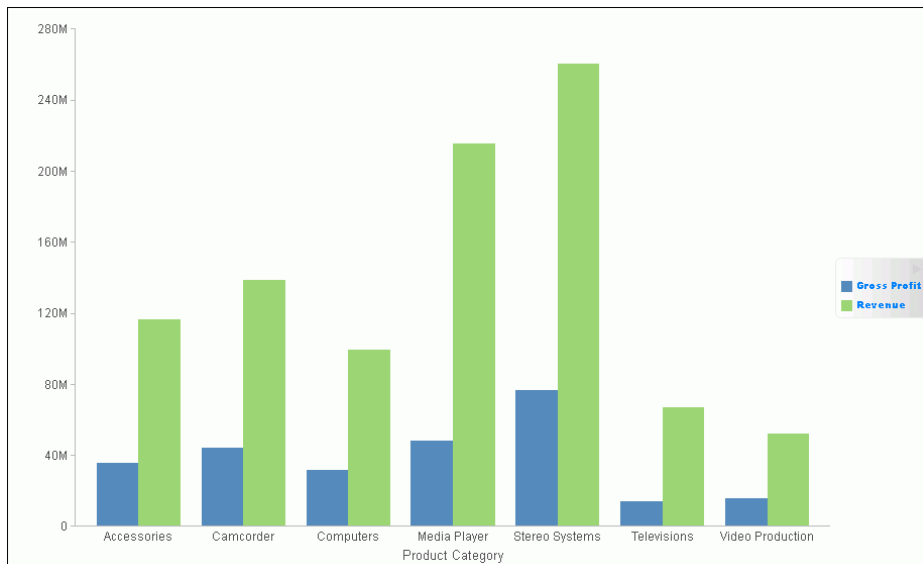
☐ **Shortcut Menu:** Right-click the legend, and click *More Legend Options*.

The Format Legends dialog box opens. For more information, see [Format Legend Dialog Box](#) on page 594.

3. On the Border Styles tab, select the option to *Show Border*.
4. Click the *Color* icon to open the Color dialog box, where you can specify the color of the legend border.
5. Click *OK* to close the Color dialog box.
6. Click *OK* to close the Format Legend dialog box.

The legend border is formatted accordingly.

The following image is an example of a bar chart with a styled legend.



Formatting Gridlines

Gridlines are used on a chart as a reference to help you understand the quantities and values of your data and decode information on the axis. There are four types of gridlines that you can display and edit on your chart. They are:

- ☐ Horizontal major gridlines
- ☐ Horizontal minor gridlines
- ☐ Vertical major gridlines
- ☐ Vertical minor gridlines

Major gridlines enhance the display of values, while minor gridlines supplement major gridlines. If a plot point falls in between major gridlines, you can use minor gridlines for more precise interpretation of the data.

- ☐ Vertical gridlines in a vertical chart run on the X axis.
- ☐ Horizontal gridlines in a vertical chart run on the Y axis.
- ☐ Horizontal major gridlines enhance the display of values, compared to the Y-axis scale alone. They are enabled by default on many charts. They do not apply to the pie, 3D, gauge, spectral map, or funnel chart type.
- ☐ Horizontal minor gridlines are disabled by default. They do not apply to the pie, 3D, gauge, spectral map, or funnel chart type.
- ☐ Vertical major gridlines enhance the display of values, compared to the X-axis scale alone. They are enabled by default. They do not apply to the pie, 3D, gauge, spectral map, or funnel chart type.
- ☐ Vertical minor gridlines are disabled by default. They do not apply to the pie, 3D, gauge, spectral map, or funnel chart type.

Note: The orientation of a chart determines the available gridline options.

Format Grid Lines Dialog Box

You can format horizontal and vertical gridlines, color bands, and frames on a chart using the options in the Format Grid Lines dialog box. For instructions on how to open this dialog box, see the procedures in [Using Gridline Properties](#) on page 601.

Color bands come in a pair, with each band uniquely colored. They appear in a continually repeating pattern behind a series on a chart. The contrast of colors is designed to make the chart easier to read.

Alternate formatting can be used to apply different colors to sections, called regions, of an axis.

The Format Grid Lines dialog box contains the following tabs:

- ☐ Major Grid Lines
- ☐ Minor Grid Lines
- ☐ Color Bands
- ☐ Frames

Use the Major Grid Lines tab to format the major gridlines on the chart.

The Major Grid Lines tab contains the following options:

- ☐ **Show Grid Lines.** Select this option to display major gridlines on a chart if minor gridlines are the default for the chart.
- ☐ **Line Style.** Click this icon to open the Line Style dialog box, where you can edit the color, weight, and style of the gridline.
- ☐ **Show Ticks.** Select this check box to enable or disable the tick mark functionality.
- ☐ **Tick Style.** Select from this drop-down menu of tick styles (Inside, Outside, Spanning).
- ☐ **Line Style.** Click this icon to open the Line Style dialog box, where you can edit the color, weight, and style of the gridline.

Use the Minor Grid Lines tab to format the minor gridlines on your chart.

The Minor Grid Lines tab contains the following options:

- ☐ **Show Grid Lines.** Select this option to display minor gridlines on a chart. By default, this option is enabled.
- ☐ **Line Style.** Click this icon to open the Line Style dialog box, where you can edit the color, weight, and style of the gridline.
- ☐ **Grid count.** Set the number of minor gridlines that will appear between major gridlines.
- ☐ **Show Ticks.** Select this check box to enable or disable the tick mark functionality.
- ☐ **Tick Style.** Select from this drop-down menu of tick styles (Inside, Outside, Spanning).
- ☐ **Line Style.** Click this icon to open the Line Style dialog box, where you can edit the color, weight, and style of the gridline.

Use the Color Bands tab to format the color bands on your chart.

The Color Bands tab contains the following options:

- ☐ **Band 1.** Select this option to add Band 1 to a chart.
 - ☐ **Color.** Click this icon to open the Color dialog box, where you can edit the color of Band 1.
 - ☐ **Transparency.** Move the slider to make Band 1 opaque (0%) or transparent (100%). The default is 0%.
 - ☐ **%.** Enter or select the percentage of the transparency of Band 1.
- ☐ **Band 2.** Select this option to add Band 2 to a chart.
 - ☐ **Color.** Click this icon to open the Color dialog box, where you can edit the color of Band 2.
 - ☐ **Transparency.** Move the slider to make the Band 2 opaque (0%) or transparent (100%). The default is 0%.
 - ☐ **%.** Enter or select the percentage of the transparency of Band 2.

Use the Quadrant Lines tab to control the placement and style of the quadrant lines in your scatter and bubble chart.

The Quadrant Lines tab contains the following options:

- ☐ **Show Quadrant Line.** Select this option to show the quadrant line. Clear this option to suppress the quadrant line.
- ☐ **Value.** Enter a value to control where the quadrant line will be placed on the X axis.
- ☐ **Line Style.** Click this icon to open the Line Style dialog box, where you can edit the color, weight, and style of the quadrant line.

Use the Frames tab to enable or disable frame regions, and to set the location and style of the frame text.

The Frames tab contains the following options:

- ☐ **Show Frame Regions.** Select this option to show a frame region. Clear this option to suppress a frame region.
- ☐ **Region.** Select from this drop-down list, the region that you want to format.
- ☐ **Add.** Click this button to add a region.
- ☐ **Remove.** Click this button to remove a region.

- ☐ **Location.** Enter the location of the region.
- ☐ **Color.** Click this icon to open the Color dialog box, where you can edit the color of the frame.
- ☐ **Border Color.** Click this icon to open the Color dialog box, where you can edit the color of the frame border.
- ☐ **Text.** Enter the text that you want to appear on the frame.
- ☐ **Style Text.** Click this icon to style the frame text.

For instructions on how to open this dialog box, see the procedures in [Using Gridline Properties](#) on page 601.

Gridline Elements Shortcut Menu

When you right-click a gridline on a chart, a menu of options opens. The options for the gridline elements are described in the following table.

Element	Option	Description
Horizontal Major Gridlines Horizontal Minor Gridlines	Delete	Removes the gridline from the chart and updates the Live Preview accordingly.
Vertical Major Gridlines Vertical Minor Gridlines	Set Line Color	Enables you to specify the color of the gridline, using the Color dialog box. For more information, see Color Dialog Box on page 567.
	More Grid Lines Options	Opens the Format Gridlines dialog box.

Using Gridline Properties

The following sections contain procedures for customizing gridlines.

Procedure: How to Display Horizontal Major Gridlines

If your chart does not display gridlines by default, use this procedure to generate gridlines.

1. Create a chart or visualization.

2. On the *Format* tab, in the *Features* group, open the *Grid* drop-down menu, point to *Horizontal Gridlines*, and then click *Major Gridlines*.

Horizontal major gridlines are added to the chart.

Procedure: How to Display Horizontal Minor Gridlines

1. Create a chart or visualization.
2. Access the option to show gridlines in one of the following ways:
 - ❑ **Ribbon:** On the *Format* tab, in the *Features* group, open the *Grid* drop-down menu. On the *Grid* drop-down menu, point to *Horizontal Gridlines*, and then click *Minor Gridlines*.
 - ❑ **Shortcut Menu:** Right-click the gridlines on the chart, and click *More Grid Lines Options*. The *Format Horizontal Grid Lines* dialog box opens. On the *Minor Grid Lines* tab, select *Show Grid Lines*.

Horizontal minor gridlines are added to the chart.

Procedure: How to Display Vertical Major Gridlines

1. Create a chart or visualization.
2. Access the option to show gridlines in one of the following ways:
 - ❑ **Ribbon:** On the *Format* tab, in the *Features* group, open the *Grid* drop-down menu. On the *Grid* drop-down menu, point to *Vertical Gridlines*, and then click *Major Gridlines*.
 - ❑ **Shortcut Menu:** Right-click the gridlines on the chart, and click *More Grid Lines Options*. The *Format Vertical Grid Lines* dialog box opens. On the *Major Grid Lines* tab, select *Show Major Grid Lines*.

Vertical major gridlines are added to the chart.

Procedure: How to Display Vertical Minor Gridlines

1. Create a chart or visualization.
2. Access the option to show gridlines in one of the following ways:
 - ❑ **Ribbon:** On the *Format* tab, in the *Features* group, open the *Grid* drop-down menu. On the *Grid* drop-down menu, point to *Vertical Gridlines*, and then click *Minor Gridlines*.
 - ❑ **Shortcut Menu:** Right-click the gridlines on the chart, and click *More Grid Lines Options*. The *Format Vertical Grid Lines* dialog box opens. On the *Minor Grid Lines* tab, select *Show Minor Grid Lines*.

Vertical minor gridlines are added to the chart.

Procedure: How to Set the Color, Weight, and Style of a Gridline

1. Open the Format Grid Lines dialog box in one of the following ways:

☐ **Ribbon:** On the *Format* tab, in the *Features* group, open the Grid drop-down menu, point to the gridline type that you want to format, and click *More Grid Lines Options*.

☐ **Shortcut Menu:** Right-click a gridline, and click *More Grid Lines Options*.

The Format Grid Lines dialog box opens.

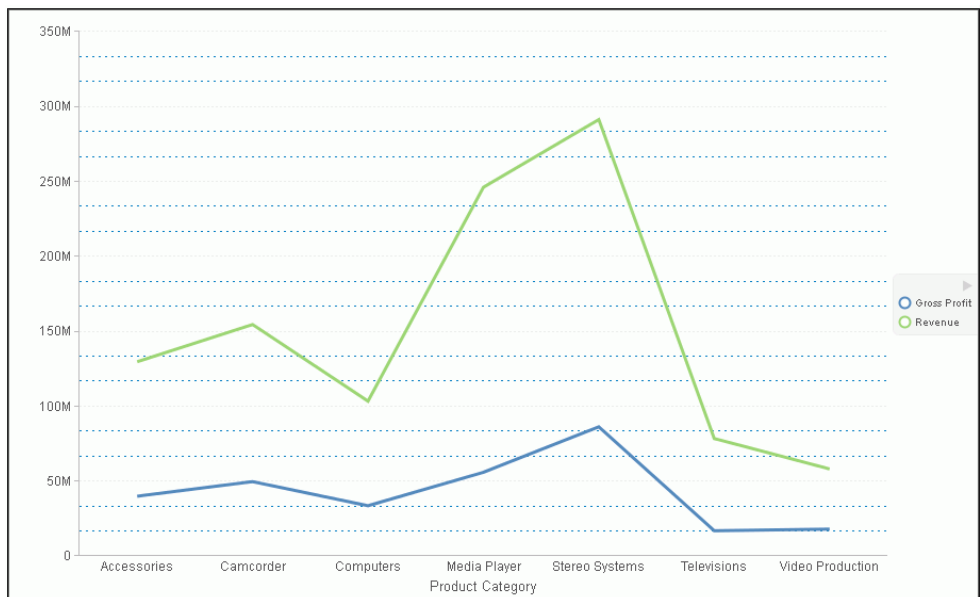
2. Click the Line Style icon.

The Line Style dialog box opens.

3. Set the color, weight, and style of the gridline.
4. Click *OK* to close the Line Style dialog box.
5. Click *OK* again to close the Format Grid Lines dialog box.

The gridline is formatted accordingly.

The following image shows a line chart with styling applied to the gridlines.



Procedure: How to Set Ticks

Ticks are short lines which are perpendicular to a gridline. They are used to tick off specific increments along the gridline.

1. Create a chart or visualization with gridlines.
2. Open the Format Grid Lines dialog box in one of the following ways:
 - ☐ **Ribbon:** On the *Format* tab, in the *Features* group, open the Grid drop-down menu, point to the gridline type that you want to format, and click *More Grid Lines Options*.

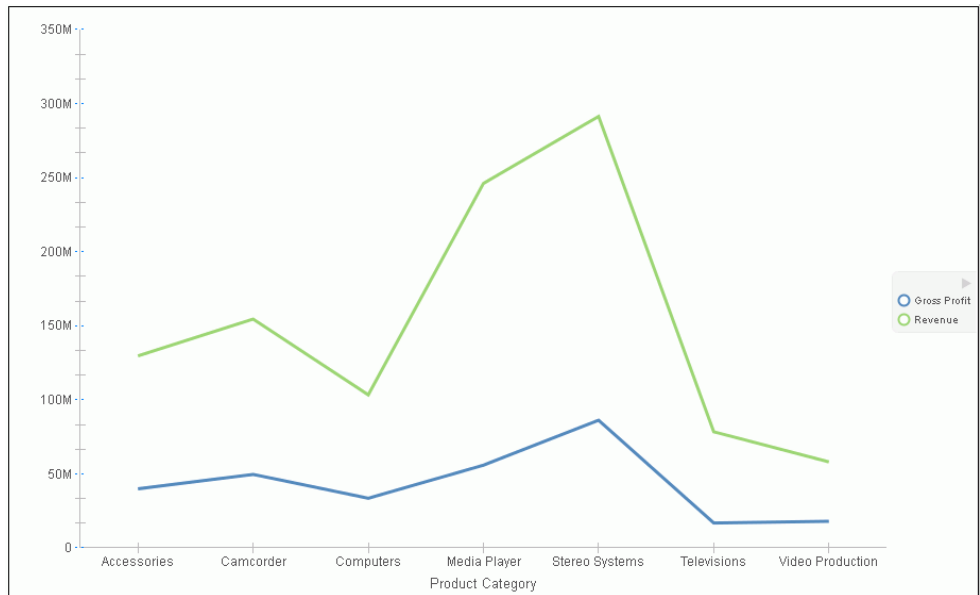
- ☐ **Shortcut Menu:** Right-click a gridline, and click *More Grid Lines Options*.

The Format Grid Lines dialog box opens.

3. From the Tick Style drop-down menu select a tick style option. The options are:
 - ☐ Inside
 - ☐ Outside
 - ☐ Spanning
4. Click the *Line Style* icon to open the Line Style dialog box, where you can edit the color, weight, and style of the gridline.
5. Click *OK* to close the Line Style dialog box.
6. Click *OK* again to close the Format Grid Lines dialog box.

The tick marks are formatted accordingly.

The following image shows a chart with tick marks spanning both the vertical axis and horizontal axis.



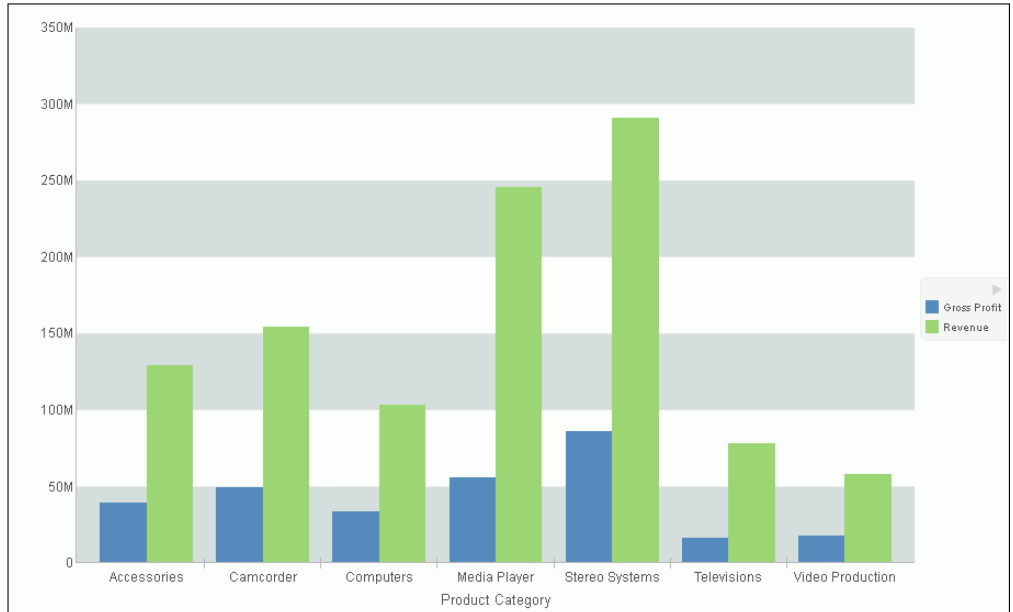
Procedure: How to Set Color Bands

1. Create a chart or visualization with gridlines.
2. Open the Format Grid Lines dialog box in one of the following ways:
 - ☐ **Ribbon:** On the *Format* tab, in the *Features* group, open the Grid drop-down menu, point to the gridlines that you want to format, and click *More Grid Lines Options*.
 - ☐ **Shortcut Menu:** Right-click a gridline, and click *More Grid Lines Options*.

The Format Grid Lines dialog box opens.

3. On the Color Bands tab, click the *Color* icon to open the Color dialog box, where you can set the color of the color bands.
4. Use the transparency slider, or the percentage box, to type or select the percentage of the transparency that you want to apply to the color.
5. Click *OK* to close the Color dialog box.
6. Click *OK* again to close the Format Grid Lines dialog box.

The following image shows a chart with color bands along the horizontal gridline.



Procedure: How to Delete a Gridline

1. Create a chart or visualization with gridlines.
2. Select a gridline.
3. Delete the gridline in one of the following ways:
 - ☐ **Ribbon:** On the *Format* tab, in the *Features* group, open the *Grid* drop-down menu. From the menu, point to the gridline that you want to format, and click *More Grid Lines* to open the Format Grid Lines dialog box. On the tab for the gridline that you want to delete, clear the *Show Grid Lines* option, and click *OK*.
 - ☐ **Shortcut Menu:** Right-click the gridline, and from the menu, select *Delete*.

The gridline is deleted from the chart.

Procedure: How to Display Quadrant Lines

Use the options on the Quadrant Lines tab to control the placement and style of the quadrant lines in your scatter and bubble chart.

1. Create a scatter chart.

2. On the *Format* tab, in the *Features* group, click *Grid*.
3. On the drop-down menu, point to either Horizontal Gridlines or Vertical Gridlines, and then click *More Grid Lines Options*.

The Format Grid Lines dialog box opens.

4. On the Quadrant Lines tab, select *Show Quadrant Line*.

The quadrant line displays on the chart.

5. Click *OK* to close the Format Grid Lines dialog box.

***Procedure:* How to Add Frames to Regions on a Chart**

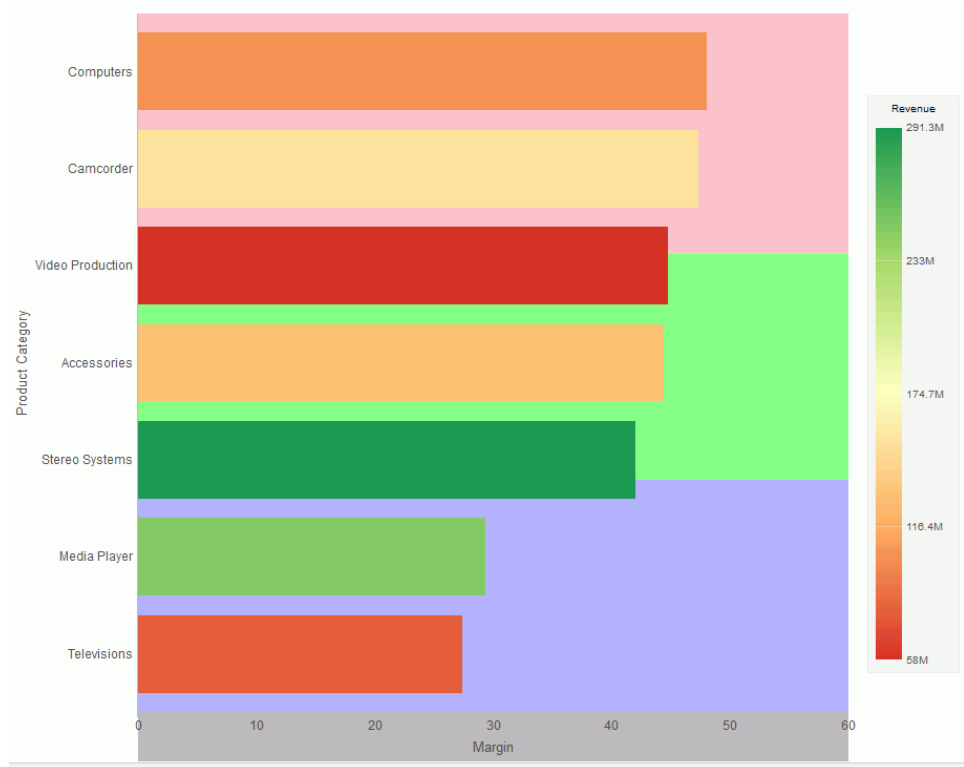
1. Create a chart or visualization.
2. On the *Format* tab, in the *Features* group, click *Grid*.
3. On the drop-down menu, point to either Horizontal Gridlines or Vertical Gridlines, and then click *More Grid Lines Options*.

The Format Grid Lines dialog box opens.

4. On the Frames tab, select the *Show Frame Regions* check box.
5. Click *Apply*.

Colored frames are added to the chart, using the default styling options. Optionally, you may use the color and border color options to change these.

The following image shows a bar chart with colored frames around each region.



Formatting Axis Labels

Vertical and horizontal axes are based on the orientation of the graph. For example, in a vertical graph, the horizontal axis refers to the X axis and the vertical axis refers to the Y axis. In a horizontal graph, the horizontal axis refers to the Y axis and the vertical axis refers to the X axis. This is important to consider, since options could change depending on the orientation of the graph.

A chart can contain the following types of axis labels:

- ☐ Horizontal axis labels represent the X axis. They do not apply to pie, funnel, or gauge charts.
- ☐ Vertical axis labels represent the Y1 axis in a single axis chart. They represent a numeric scale, usually located on the left side of a vertical chart.

- ☐ Secondary horizontal and vertical labels can only be used when dual axes charts are selected.

Format Axis Dialog Box

Whether you access axis options from the ribbon, or the shortcut menu, you are presented with the Format Axis dialog box of options for formatting for both vertical and horizontal axes. The Format Axis dialog box contains the following tabs:

- ☐ General
- ☐ Scale
- ☐ Title
- ☐ Labels
- ☐ Advanced

Reference: Format Vertical Axis Dialog Box

You use the Format Vertical Axis dialog box to specify formatting options for the vertical axis in your chart.

Use the Scale tab to modify scale properties.

The Scale tab contains the following options:

- ☐ **Automatic Minimum.** Supplies the minimum value on the Y-axis scale automatically. To use manual scaling, clear this option. You can then set the minimum value by entering a number into the Value text box.
 - ☐ **Value.** Type the minimum value in this text box if you have not selected Automatic Minimum.
- ☐ **Automatic Maximum.** Supplies the maximum value on the Y-axis scale automatically. To use manual scaling, clear this option. You can then set the maximum value by entering a number into the Value text box.
 - ☐ **Value.** Type the maximum value in this text box if you have not selected Automatic Maximum.

- ☐ **Automatic Grid Step.** Calculates the number of major grid steps automatically. To use manual scaling, clear this option. You can then set the value by typing the number into the Value text box.
- ☐ **Value.** Type the value in this text box if you have not selected Automatic Grid Step.
- ☐ **Logarithmic Scale.** Controls whether or not the Y-axis scale progresses logarithmically instead of linearly. This option is disabled by default. When selected, the logarithmic base is set to 10.0, but can be changed by entering another value.
- ☐ **Log scale base.** Type the base value to be shown on the logarithmic scale.
- ☐ **Include zero on scale.** Controls whether or not a zero (0) value appears on the scale. This option is enabled by default.

Use the Title tab to create and style the title for the axis.

The Title tab contains the following options:

- ☐ **Text.** Type a title for the axis in the Text field.
- ☐ **Style text.** Opens the Style dialog box, where you can style the text.

Use the Labels tab to format the layout of the axis labels.

The Labels tab contains the following options:

- ☐ **Show Labels.** Displays labels next to the axis. This is enabled by default. Clear this option to suppress labels.
- ☐ **Axis side.** Contains a list of position options for the labels on the axis. The options are Left (default), Right, or Both.
- ☐ **Style labels.** Opens the Style dialog box, where you can style text.
- ☐ **Format Labels.** Contains a list of preset formats that can be applied to the labels.
- ☐ **Custom Format.** Allows you to use a custom format. This option is activated when *Use Pattern /100* and *Use Pattern* are selected as the format labels.

The Format Labels drop-down menu provides a list of preset formats that you can apply to labels. When you select a custom format, it must be defined using a custom format pattern. See the following table for a list and description of the characters that you can use in a custom format.

Character	Description
#	Is a digit.
0 (zero)	Shows as absent.
. (period)	Is a placeholder for a decimal separator.
, (comma)	Is a placeholder for a grouping separator.
; (semicolon)	Separates formats.
- (dash)	Is the default negative prefix.
% (percent)	Divides by 100 and shows as a percentage.
x	Determines that any other characters can be used in the prefix or suffix.
' (apostrophe)	Is used to quote special characters in a prefix or suffix.

Use the Advanced tab to modify additional axis properties.

The Advanced tab contains the following options:

- ☐ **Exclude Minimum Label.** Excludes the label with the lowest axis value from the chart.
- ☐ **Exclude Maximum Label.** Excludes the label with the highest axis value from the chart.
- ☐ **Descending Axis.** Draws the axis in descending order.
- ☐ **Show axis line.** Controls the display of the axis baseline.
 - ☐ **Line Style.** Opens the Line Style dialog box, where you can edit the color, weight, and style of the axis line.
- ☐ **Show zero line.** Controls the display of the zero line.
 - ☐ **Line Style.** Opens the Line Style dialog box, where you can edit the color, weight, and style of the zero line.
- ☐ **Custom Baseline.** Controls the display of the custom baseline.
 - ☐ **Value.** Type a value for the custom baseline.
 - ☐ **Line Style.** Opens the Line Style dialog box, where you can edit the color, weight, and style of the custom baseline.

For instructions on how to open this dialog box, see the procedures in [Using Axis Properties](#) on page 614.

Reference: Format Horizontal Axis Dialog Box

You use the Format Horizontal Axis dialog box to specify formatting options for the horizontal axis in your chart.

The General tab contains the Show axis line check box, which controls the display of the axis line. When this check box is selected, you can style the axis line.

The General tab contains the following options:

- ☐ **Show axis line.** Enables the display of the axis line.
 - ☐ **Line Style.** Opens the Line Style dialog box, where you can edit the color, weight, and style of the axis line.

Use the Title tab to create and style the title for the axis.

The Title tab contains the following options:

- ☐ **Text.** Type a title for the axis in the Text field.
- ☐ **Style.** Opens the Style dialog box, where you can style the text.

Use the Labels tab to format the layout of the axis labels.

The Labels tab contains the following options:

- ☐ **Show Labels.** Displays labels next to the axis. This is enabled by default. Clear this option to suppress labels.
 - ☐ **Axis side.** Contains a list of position options for the labels on the axis. The options are Left (default), Right, or Both.
 - ☐ **Style labels.** Opens the Style dialog box, where you can style text.
- ☐ **Stagger Labels.** Sets the labels to appear staggered.

Use the Advanced tab to modify additional axis properties.

The Advanced tab contains the following options:

- ☐ **Exclude Minimum Label.** Excludes the label with the lowest axis value from the chart.
- ☐ **Exclude Maximum Label.** Excludes the label with the highest axis value from the chart.
- ☐ **Reverse Groups.** Reverses the order of the display of the groups on the horizontal axis.

For instructions on how to open this dialog box, see the procedures in [Using Axis Properties](#) on page 614.

Secondary Axes Options

Formatting options are available for secondary axes in dual-axis charts. For example, in a vertical dual-axis chart, the secondary vertical axis refers to the Y2 axis.

The Format Secondary Axis dialog box contains the following tabs for both vertical and horizontal axes:

- ☐ **General.** For more information, see the Scale tab options in [Format Axis Dialog Box](#) on page 609.
- ☐ **Title.** For more information, see the equivalent tab in [Format Axis Dialog Box](#) on page 609.
- ☐ **Labels.** For more information, see the equivalent tab in [Format Axis Dialog Box](#) on page 609.
- ☐ **Advanced.** For more information, see the equivalent tab in [Format Axis Dialog Box](#) on page 609.

Axis Elements Shortcut Menu

When you right-click an axis label or title in a chart in Live Preview, a menu of options opens. The options for the right-click axis label elements are described in the following table.

Element	Option	Description
Axis Title	Delete	Deletes the axis title from the chart and updates the Live Preview accordingly.
	Change Title	Enables you to change the axis title.
	Style Title	Enables you to apply styling to the axis title, using the Style dialog box. For more information, see Style Dialog Box on page 591.

Element	Option	Description
Horizontal Labels	Delete	Deletes labels from the chart and updates the Live Preview accordingly.
	Stagger	Controls the positioning of the labels. The On value positions the labels in a zigzag pattern. The Off default value positions the labels in a straight row.
	Rotate	Rotates the labels a specified number of degrees.
	Style Labels	Enables you to apply styling to the labels, using the Style dialog box. For more information, see Style Dialog Box on page 591.
	More Axis Options	Opens the Format Axis dialog box. For more information, see Formatting Axis Labels on page 608.
Vertical Labels	Delete	Deletes labels from the chart and updates the Live Preview accordingly.
	Rotate	Rotates the labels a specified number of degrees.
	Format Labels	Formats the labels according to the value that you specify.
	Style Labels	Enables you to apply styling to the labels, using the Style dialog box. For more information, see Style Dialog Box on page 591.
	More Axis Options	Opens the Format Axis dialog box. For more information, see Formatting Axis Labels on page 608.

Using Axis Properties

The following sections contain procedures for customizing an axis. The procedures are organized by the tab and group in which their associated options appear on the ribbon.

Axis labels appear by default.

Procedure: How to Delete Axis Labels

1. Create a chart or visualization.

2. You can delete axis labels in one of the following ways:

- ☐ **Ribbon:** On the *Format* tab, in the *Labels* group, open the *Axes* drop-down menu, point to the axis that you are working with, and clear the *Show Labels* option.
- ☐ **Shortcut Menu:** Right-click an axis label on the chart, and click *Delete*.

The axis labels are deleted from the chart.

Procedure: How to Stagger Axis Labels

Note: You can only apply a staggered effect to horizontal axis labels.

1. Create a chart or visualization with at least one axis label on display.
2. Access the *Stagger* option in one of the following ways:

- ☐ **Ribbon:** On the *Format* tab, in the *Labels* group, open the *Axes* drop-down menu, point to the axis that you are working with, and click *Stagger Labels*.
- ☐ **Shortcut Menu:** Right-click an axis label on the chart, point to *Stagger* and select *On*.

The axis labels are staggered.

Procedure: How to Rotate Axis Labels

1. Create a chart or visualization with axis labels.
2. Access the *Rotate* option in one of the following ways:

- ☐ **Ribbon:** On the *Format* tab, in the *Labels* group, open the *Axes* drop-down menu, point to the axis that you are working with, point to *Rotate Labels*, and then click the degree to which you want to rotate the axis labels.
- ☐ **Shortcut Menu:** Right-click an axis label on the chart, point to *Rotate*, and then click the degree to which you want to rotate the axis labels.

The axis labels are rotated.

Procedure: How to Format Axis Labels

1. Create a chart or visualization with an axis label.
2. Access the list of axis label options in one of the following ways:

- ☐ **Ribbon:** On the *Format* tab, in the *Labels* group, open the *Axes* drop-down menu, point to the axis that you are working with, and click *More Axis Options* to open the *Format Axis* dialog box. Open the *Labels* tab, and from the *Labels* option drop-down menu, select the formatting option that you want.

- ❑ **Shortcut Menu:** Right-click an axis label, point to *Format Labels*, and click the formatting option that you want.

The axis labels are formatted accordingly.

Procedure: How to Manually Set the Scale of an Axis

1. Create a chart or visualization.
2. Open the *Format Axis* dialog box in one of the following ways:
 - ❑ **Ribbon:** On the *Format* tab, in the *Labels* group, open the *Axes* drop-down menu, point to the axis that you are working with, and click *More Axis Options*.
 - ❑ **Shortcut Menu:** Right-click an axis value on the chart, and click *More Axis Options*.

The *Format Axis* dialog box opens.

3. On the *scale* tab, clear the *Automatic Minimum* option and enter your own minimum value in the *Value* text box.
4. Clear the *Automatic Maximum* option, and enter your own maximum value in the *Value* text box.
5. Clear the *Automatic Grid Step* option, and enter your own grid step value in the *Value* text box.
6. Optionally, you can select the *Logarithmic Scale* option and enter the log scale base in the text field of that name. You can also clear the *Include zero on scale* option if you do not want zero to appear on the axis.
7. Click *OK* to close the dialog box.

The axis scale is set accordingly.

Procedure: How to Add an Axis Title

1. Create a chart or visualization with an axis label.
2. Open the *Format Axis* dialog box in one of the following ways:
 - ❑ **Ribbon:** On the *Format* tab, in the *Labels* group, open the *Axes* drop-down menu, point to the axis that you are working with, and click *More Axis Options*.
 - ❑ **Shortcut Menu:** Right-click an axis label, and click *More Axis Options*.

The *Format Axis* dialog box opens.

3. On the *Title* tab, type the axis title in the *Text* field.
4. Click the *Style text* icon to open the *Style* dialog box, where you can style the text.

The axis title is styled accordingly.

Note: Right-click the axis title to delete, change, or style the axis title.

Procedure: How to Set Advanced Axis Properties

1. Create a chart or visualization with an axis label.
2. Open the Format Axis dialog box in one of the following ways:
 - ☐ **Ribbon:** On the *Format* tab, in the *Labels* group, open the *Axes* drop-down menu, point to the axis that you are working with, and click *More Axis Options*.
 - ☐ **Shortcut Menu:** Right-click an axis label, and click *More Axis Options*.

The Format Axis dialog box opens.

3. On the *Advanced* tab, set the following options:
 - ☐ Exclude Minimum Label
 - ☐ Exclude Maximum Label
 - ☐ Descending Axis
 - ☐ Show axis line
 - ☐ Show zero line
 - ☐ Custom Baseline (Value)

You can edit the color, weight, and style of all the lines that you set in the *Line Style* dialog box.

4. Click *OK*.

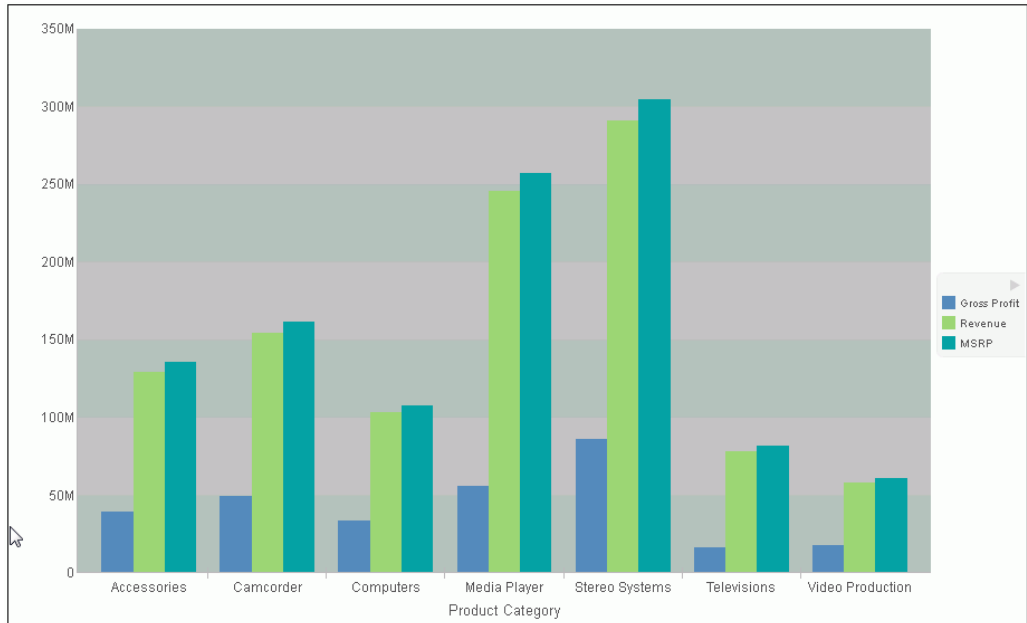
The axis advanced options are set accordingly.

Formatting a Frame and a Background

The frame of a chart is the area that contains the plot points. It is also the area in which horizontal and vertical gridlines are drawn.

The background of a chart is the area around the frame.

You can adjust the appearance of the frame and background to achieve different visual effects for your charts. For example, you can change the appearance of the chart frame line, or you can change the color of the background of your chart, as shown in the following image.



Frame & Background Dialog Box

Whether you access frame and background options from the ribbon, or the shortcut menu, you are presented with the Frame & Background dialog box of options for formatting the frame and background of a chart. The Frame & Background dialog contains the following tabs:

- ☐ Frame
- ☐ Frame Edge
- ☐ Background

The Frame & Background dialog box and pertinent tabs are discussed in this section for the following types of charts:

- ☐ 2D Charts and 2D Charts with 3D Effects (Frame and Frame Edge tabs)
- ☐ Pie Charts (Frame and Frame Edge tabs)
- ☐ 3D Charts (Left Wall, Right Wall, Floor, and Advanced tabs)

Note: The options in the Frame tab vary based on the chart type selected.

Use the Frame tab for 2D charts and 2D charts with 3D effects to set a frame depth angle and depth radius, select a fill for a frame, and set a shadow for a frame for these types of charts.

The Frame tab for 2D charts and 2D charts with 3D effects contains the following options:

- ☐ **Depth Angle.** Enter the angle from the front of the chart to the back where the chart risers and frames are drawn. You can set the depth angle from zero to 180 degrees, but it must be used along with Depth Radius.
- ☐ **Depth Radius.** Enter how far out the extruded frame will be extended. Small values, such as zero, produce very narrow charts. Large values, the maximum being 100, produce thicker charts.

Fill

- ☐ **No fill.** (Default). Select this option to keep the legend colorless.
- ☐ **Solid fill.** Select this option to display the Color and Transparency options.
 - ☐ **Color.** Click this icon to open the Color dialog box, where you can select a color for a frame.
 - ☐ **Transparency.** Move the slider to make the frame opaque (0%) or transparent (100%). The default is 0%.
- ☐ **Gradient fill.** Select this option to display the direction of the gradient, the color pattern of the gradient, and the degrees of transparency for the two colors that make up the gradient. A gradient is a smooth color transition or blending of one color to another. The number of colors to use in a gradient is defined by the *stop* or *pin* elements.
 - ☐ **Direction.** Select from this drop-down menu to set the direction of the gradient fill. For example, Gradient right or Gradient left.

Gradient style

- ☐ **Color Pattern.** Select the color pattern for the data series. The color pattern option on the left is an AB wash that uses two colors in the pattern color1 - color2, for example, red-green. The color pattern option on the right is an ABA wash that uses two colors in the pattern color1 - color2 - color1, for example, red-green-red.
- ☐ **First Color.** Click this icon to open the Color dialog box, where you can select a color for the first color.
- ☐ **Second Color.** Click this icon to open the Color dialog box, where you can select a color for the second color.

Shadow

- ☐ **Show Shadow.** Select this option to set a shadow.

Use the Frame Edge tab for 2D charts and 2D charts with 3D effects to set a frame depth angle and depth radius, select a fill for a frame, and set a shadow for a frame for these types of charts.

The Frame tab for 2D charts and 2D charts with 3D effects contains the following options:

- ☐ **Automatically Shade Frame Edge.** Select this option to automatically shade the frame edge. Clear this option to enable the Side Frame and Bottom Frame options.

Side Frame

- ☐ **Show Color.** Select this option to show the color of the side frame.
- ☐ **Color.** Click this icon to open the Color dialog box, where you can select a color for the side frame.
- ☐ **Show Border Color.** Select this option to show the color of the border of the side frame.
- ☐ **Border Color.** Click this icon to open the Color dialog box, where you can select a color for the border of the side frame.

Bottom Frame

- ☐ **Show Color.** Select this option to show the color of the bottom frame.
- ☐ **Color.** Click this icon to open the Color dialog box, where you can select a color for the bottom frame.
- ☐ **Show Border Color.** Select this option to show the color of the bottom frame.
- ☐ **Border Color.** Click this icon to open the Color dialog box, where you can select a color for the border of the bottom frame.

Use the Frame tab for pie charts to set pie depth and tilt, select a fill and color for a pie frame, and set a shadow for the frame.

The Frame tab for pie charts contains the following options:

- ☐ **Pie Depth.** Set the depth of the edge of a pie chart. You can select a value from zero to 100 to set the thickness of an edge.
- ☐ **Pie Tilt.** Set the tilt of the pie chart. The smaller the value you add, the flatter the pie chart appears. The larger the value you add, more of the pie edge appears.

Fill

- ☐ **No fill.** (Default). Results in no color added to the edge of the pie.
- ☐ **Solid fill.** Select this option to display the Color and Transparency options.
 - ☐ **Color.** Click this icon to open the Color dialog box, where you can select a color for the frame.
 - ☐ **Transparency.** Move the slider to make the fill opaque (0%) or transparent (100%). The default is 0%.
- ☐ **Gradient fill.** Select this option to display the direction of the gradient, the color pattern of the gradient, and the degrees of transparency for the two colors that make up the gradient. A gradient is a smooth color transition or blending of one color to another. The number of colors to use in a gradient is defined by the *stop* or *pin* elements.
- ☐ **Direction.** Select from this drop-down menu to set the direction of the gradient fill. For example, Gradient right or Gradient left.
- ☐ **Gradient Style**
 - ☐ **Color Pattern.** Select the color pattern for the data series. The color pattern option on the left is an AB wash that uses two colors in the pattern color1 - color2, for example, red-green. The color pattern option on the right is an ABA wash that uses two colors in the pattern color1 - color2 - color1, for example, red-green-red.
 - ☐ **First Color.** Click this icon to open the Color dialog box, where you can select a color for the first color.
 - ☐ **Second Color.** Click this icon to open the Color dialog box, where you can select a color for the second color.

Shadow

- ☐ **Show Shadow.** Select this option to set a shadow around the frame.
- Use the Frame Edge tab for pie charts to set the edge of a pie frame.

The Frame Edge tab for pie charts contains the following options:

- ☐ **Automatically Shade Frame Edge.** Select this option to automatically shade the frame edge. Clear this option to enable the Side Frame and Bottom Frame options.
- ☐ **Show Color.** Select this option to show the color of the side frame.
- ☐ **Color.** Click this icon to open the Color dialog box, where you can select a color for side frame.

- ☐ **Show Border Color.** Select this option to show the color of the border of the side frame.
- ☐ **Border Color.** Click this icon to open the Color dialog box, where you can select a color for the border of the side frame.

Bottom Frame

- ☐ **Show Color.** Select this option to show the color of the bottom frame.
- ☐ **Color.** Click this icon to open the Color dialog box, where you can select a color for the bottom frame.
- ☐ **Show Border Color.** Select this option to show the color of the bottom frame.
- ☐ **Border Color.** Click this icon to open the Color dialog box, where you can select a color for the border of the bottom frame.

Use Wall and Floor tabs to set the walls and floor of a 3D chart.

These Wall and Floor tabs contain the following options:

- ☐ **Show Wall** or **Show Floor.** Select this option to show the wall or floor (depending on the tab) of the 3D chart.
- ☐ **Wall Thickness.** Specify the thickness of the wall or floor.
- ☐ **Cube Size.** Specify the cube size of the wall or floor.
- ☐ **Color.** Click this icon to open the Color dialog box, where you can select a color for the wall or floor.
- ☐ **Border Color.** Click this icon to open the Color dialog box, where you can select a color for the border of the wall or floor.
- ☐ **Show Wall (Left Wall).** Select this option to show the left wall.
- ☐ **Show Wall (Right Wall).** Select this option to show the right wall.
- ☐ **Show Floor.** Select this option to show the floor.

Use the Advanced tab for 3D charts to modify additional properties for 3D frames.

The Advanced tab contains the following options:

- ☐ **Viewing Angles.** Select from a drop-down menu of viewing angles for three-dimensional charts. For example, Standard or Group View.
- ☐ **Isometric Projection.** Select this option to ignore perspective distortion in a project graph cube from an isometric view.

- ☐ **Proportional Cube.** Select this option to define the axis size proportional to the number of series or groups.
- ☐ **Automatic shading of walls.** Select this option to shade chart walls.
- ☐ **Automatically Shade Frame Edge.** Select this option to automatically shade the frame edge.
- ☐ **3D Zoom Factor.** Enter a value in the text box to set the global scaling factor for zooming in and out in a 3D chart. Smaller values zoom out and produce a smaller display of the chart within a frame. Larger values zoom in and produce a larger display of the chart within a frame.
- ☐ **Pan Horizontally.** Enter a value in the text box to pan a 3D chart in the horizontal direction. Smaller values move the frame of the chart to the left. Larger values move the frame of the chart to the right.
- ☐ **Pan Vertically.** Enter a value in the text box to pan a 3D chart in the vertical direction. Smaller values move the frame of the chart upward. Larger values move the frame of the chart downward.

Use the Background tab to set and customize a border for a chart.

The Background tab contains the following options:

- ☐ **Show Border Color.** Select this option to show the color of the border.
- ☐ **Border Color.** Click this icon to open the Color dialog box, where you can select a color for the border.
- ☐ **Solid fill.** Select this option to display the Color and Transparency options.
 - ☐ **Color.** Click this icon to open the Color dialog box, where you can select a color for the frame.
 - ☐ **Transparency.** Move the slider to make the background opaque (0%) or transparent (100%). The default is 0%.
- ☐ **Gradient fill.** Select this option to display the direction of the gradient, the color pattern of the gradient, and the degrees of transparency for the two colors that make up the gradient. A gradient is a smooth color transition or blending of one color to another. The number of colors to use in a gradient is defined by the *stop* or *pin* elements.
- ☐ **Direction.** Select from this drop-down menu to set the direction of the gradient fill. For example, Gradient right or Radial.

Gradient Style

- ☐ **Color Pattern.** Select the color pattern for the data series. The color pattern option on the left is an AB wash that uses two colors in the pattern color1 - color2, for example, red-green. The color pattern option on the right is an ABA wash that uses two colors in the pattern color1 - color2 - color1, for example, red-green-red.
- ☐ **First Color.** Click this icon to open the Color dialog box, where you can select a color for the first color.
- ☐ **Second Color.** Click this icon to open the Color dialog box, where you can select a color for the second color.

For instructions on how to open this dialog box, see the procedures in [Using Frame and Background Properties](#) on page 625.

Frame and Background Shortcut Menu

When you right-click a chart background, a menu of options opens. The options for the background and frame elements are described in the following table.

Element	Option	Description
Background	Background Color	Enables you to specify the color of the background, using the Color dialog box. For instructions, How to Change the Color of the Background on page 626.
	Show legend	Controls the display of the legend on the background. When selected, it displays the legend. When cleared, it suppresses the display of the legend.
	More Frame and Background Options	Opens the Frame & Background dialog box. For more information, see Frame & Background Dialog Box on page 618.

Element	Option	Description
Frame	Frame Color	Enables you to specify the color of the frame, using the Color dialog box. For more information, see Color Dialog Box on page 567.
	Show 3D	Controls the depth of the frame. The value On renders the frame in 3D depth. The value Off renders the frame in one dimension.
	More Frame and Background Options	Opens the Frame & Background dialog box. For more information, see Frame & Background Dialog Box on page 618.

Using Frame and Background Properties

The following sections contain procedures for customizing frame and background properties. The options for the following procedures are found in the Features group of the Format tab.

Procedure: How to Change the Color of the Frame

The default color of the frame in a chart is determined by the Document Theme selected on the Options window. For more information, see [Changing InfoAssist+ User Preferences](#) on page 238.

This procedure describes how to change the color of the frame.

1. Create a chart or visualization.
2. Access the Frame & Background dialog box in one of the following ways:
 - ☐ **Ribbon:** On the *Format* tab, in the *Features* group, click *Frame & Background*.
 - ☐ **Shortcut Menu:** Right-click the chart, and click *More Frame & Background Options*.

The Frame & Background dialog box opens.

3. On the Frame tab, in the Fill area, click the *Color* icon.

Note: Solid fill must be selected for the Color icon to appear.

The Color dialog box opens. Select a new color for the frame. For more information on the Color dialog box, see [Color Dialog Box](#) on page 567. You can also set the depth angle and depth radius for the frame, as well as set a shadow for the frame.

4. Click *OK*.

The chart displays the new frame color.

***Procedure:* How to Change the Color of the Background**

The default color of the background of a chart is determined by the Document Theme selected on the Options window. For more information, see [Changing InfoAssist+ User Preferences](#) on page 238.

1. Create a chart or visualization.
2. Open the Frame & Background dialog box in one of the following ways:

☐ **Ribbon:** On the *Format* tab, in the *Features* group, click *Frame & Background*.

☐ **Shortcut Menu:** Right-click the background of the chart, and click *Background Color*.

The Frame & Background dialog box opens.

3. On the Background tab, in the Fill area, click the *Color* icon.

The Color dialog box opens. Select a new color for the background. For more information on the Color dialog box, see [Color Dialog Box](#) on page 567.

4. Click *OK*.

The chart displays the new background color.

Formatting a Gauge Chart

A gauge chart is a circular chart that indicates the current position of a single data value within a given spectrum.

You can change the appearance of a gauge chart by using the gauge chart options found on the Format Gauge dialog box. To access the Format Gauge dialog box, on the *Format* tab, in the *Features* group, click *Gauges*.

Format Gauge Dialog Box

Whether you access gauge options from the ribbon, or the shortcut menu, you are presented with the Format Gauge dialog box of options for formatting a gauge chart. Such options include setting and styling a title for the gauge chart, setting tick marks, enabling and styling color bands, as well as setting advanced options, such as the gauge start and stop angle.

The Format Gauge dialog box contains the following tabs:

- ☐ General Options

- ☐ Axis Scale & Labels
- ☐ Tick Marks
- ☐ Bands
- ☐ Advanced

Use the General Options tab to set general gauge options.

The General Options tab contains the following options:

- ☐ **Gauges per row.** Enter or select how many gauges appear on each row. The default setting displays two gauge charts. A maximum of 32 gauge charts can appear on each row.

Note: The number of charts displayed per row can differ, based on the output format selected. For example, HTML format will display two charts per row, and HTML5 will display three.

Gauge Needle

- ☐ **Style.** Select a style from a drop-down menu of options to be applied to the gauge needle. For example, Pencil.
- ☐ **Color.** Click this icon to open the Color dialog box, where you can select a color for the gauge needle.

Note: If you are working with a gauge chart with more than one series, then you use the Format Series dialog box to format the color of the gauge needle and its border.

- ☐ **Border Color.** Click this icon to open the Color dialog box, where you can select a color for the gauge needle border.

Note: If you are working with a gauge chart with more than one series, then you use the Format Series dialog box to format the color of the gauge needle and its border.

Gauge Title

- ☐ **Show title.** (Default) Displays a gauge title appear for each gauge. Clear this option to suppress the title.
- ☐ **Position.** Select a position for the gauge title. Above is the default value.
- ☐ **Style.** Click this icon to open the Style dialog box, where you can style text.

Gauge value

- ☐ **Show value.** (Default) Clear this option to suppress the gauge value.

- ☐ **Style.** Click this icon to open the Style dialog box, where you can style text.
- ☐ **Format Value.** Contains a list of preset formats that can be applied.
- ☐ **Custom Format.** Allows you to use a custom format. This option is activated when *Use Pattern /100* and *Use Pattern* are selected as the format labels.

Use the Axis Scale & Labels tab to set the axis scale and label properties for the gauge needle.

The Axis Scale & Labels tab contains the following options:

Scale

- ☐ **Automatic Minimum.** (Default) Sets the engine to automatically supply the minimum value on the scale. Clear this option to manually set the minimum value by entering a number into the Value text box.
- ☐ **Value.** Enter the minimum value in this text box if you have not selected Automatic Minimum.
- ☐ **Automatic Maximum.** (Default) Sets the engine to automatically supply the maximum value on the scale. Clear this option to manually set the maximum value by entering a number into the Value text box.
- ☐ **Value.** Enter the maximum value in this text box if you have not selected Automatic Maximum.

Labels

- ☐ **Show Labels.** (Default) Displays labels next to the axis. Clear this option to suppress labels.
- ☐ **Style Labels.** Click this icon to opens the Style dialog box, where you can style text.
- ☐ **Format Labels.** Select from a drop-down menu of present formats that can be applied to the labels. When a custom format is selected, the format must be defined using a custom format pattern. For a list and description of the characters that you can use in a custom format, see the table in [Formatting Data Labels](#) on page 587.
- ☐ **Custom Format.** Text field to enter the custom format that you want to use.

The Tick Marks tab contains options to format tick marks on a gauge chart.

The Tick Marks tab contains the following options:

- ☐ **Automatic Grid Step.** (Default) Automatically calculates the number of major grid steps in a gauge chart. Clear this option to manually set the value by entering a number in the Value text box.
- ☐ **Value.** Enter the value in this text box if you have not selected Automatic Grid Step.
- ☐ **Tick Color.** Click this icon to open the Color dialog box, where you can select a color for the tick marks.
- ☐ **Tick Mark Length.** Enter a value for the relative length of major tick marks in a gauge chart. The valid range is from 0.0 to 0.5. If you set the smallest value, major tick marks do not appear.

The Bands tab contains options to format the color of the scale background on a gauge chart.

The Bands tab contains the following options:

- ☐ **Band 1.** Opens a drop-down menu of available bands.
 - ☐ **Add.** Adds the band selected from the drop-down menu of bands to the gauge chart. You can create up to five bands for a gauge chart.
 - ☐ **Remove.** Removes the band selected from the drop-down menu of bands to the gauge chart.
- ☐ **Minimum Value.** Enter a minimum value to a quality band in the gauge chart.
- ☐ **Maximum Value.** Enter a maximum value to a quality band in the gauge chart.
- ☐ **Border Color.** Opens the Color dialog box, where you can edit the color of the gauge band border.

Fill

- ☐ **No fill.** Removes fill from the quality band.
- ☐ **Solid fill.** (Default) Applies a solid color to the quality band.
- ☐ **Color.** Click this option to open the Color dialog box, where you can select the color for the quality band.
- ☐ **Transparency.** Move the slider to make the band opaque (0%) or transparent (100%). The default is 0%.

The Advanced tab contains options to set additional properties for the gauge needle.

The Advanced tab contains the following options:

- ☐ **Descending Axis.** Select this option to draw the gauge scale in descending order. When this option is cleared (default), the gauge scale is drawn in ascending order.
- ☐ **Show Zero Label.** Displays the zero label on the axis scale. Clear this option to start the gauge at another value. This option is selected by default.
- ☐ **Gauge Center by Quality.** Select this option to have the center of the gauge needle appear in the same the color as the band to which it is pointing.
- ☐ **Gauge Start Angle.** Enter a value to rotate the gauge start angle to a specified number of degrees. Values can range from 0 to 359 degrees. The default is 220 degrees.
- ☐ **Gauge Stop Angle.** Enter a value to rotate the gauge stop angle to a specified number of degrees. Values can range from 0 to 359 degrees. The default is 320 degrees. Setting a start angle to 0, and a stop angle to 180, creates a semi-circle.
- ☐ **Relative Inner Radius.** Enter a value to define the inner radius of the gauge bands and labels relative to the outer background of the gauge. Smaller values, such as 0.0, place the inner radius closer to the center of the gauge. A maximum value of 1.0, places the inner radius close to the gauge outline.
- ☐ **Relative Thickness.** Enter a value to define the relative thickness of the gauge bands. Values can range from 0.0 to 1.0.
- ☐ **Same Size Gauges.** This option applies to multi-category gauge charts. If you are working with three or more gauges per row, you can select this option to have the gauges display in the same size.

Gauge Elements Shortcut Menu

When you right-click a gauge chart, a menu containing the More Gauge Options becomes available. Select this option to open the Format Gauge dialog box. This is only available when using the HTML output format.

Using Gauge Properties

The following sections contain procedures for gauge properties. The Gauges option can be found on the *Format* tab, in the *Features* group.

Note: This option is only available when you are working with a gauge chart.

Procedure: How to Set the Number of Gauges Per Row

If you have multiple gauges on a chart, you might want to specify how many gauges appear on each row.

1. Create a gauge chart that has multiple values.
2. Open the Format Gauge dialog box in one of the following ways:

☐ **Ribbon:** On the *Format* tab, in the *Features* group, click *Gauges*.

☐ **Shortcut Menu:** Right-click the gauge chart and select *More Gauge Options*.

The Format Gauge dialog box opens.

3. On the General options tab, in the Gauges per row field, enter or select the number of gauges that you want to appear on one row.
4. Click *OK*.

The gauges now appear on one row.

After changing the setting from the default of 2 gauges per row to 3 gauges per row, all the charts now appear on one row.

Procedure: How to Set the Minimum and Maximum Axis Scale Values

1. Create a gauge chart.
2. Open the Format Gauge dialog box in one of the following ways:

☐ **Ribbon:** On the *Format* tab, in the *Features* group, click *Gauges*.

☐ **Shortcut Menu:** Right-click the gauge chart, and click *More Gauge Options*.

The Format Gauge dialog box opens.

3. On the Axis Scale & Labels tab, in the Scale area, either accept the minimum and maximum values for the scale automatically supplied by the chart engine, or enter your own in the *Value* fields.
4. Click *Apply*.

The gauge scale reflects the selections that you have made.

Procedure: How to Style Axis Labels

1. Create a gauge chart.

2. Open the Style dialog box in one of the following ways:

☐ **Ribbon:** On the *Format* tab, in the *Features* group, click *Gauges* to open the Format Gauge dialog box. In the Format Gauge dialog box, open the Axis Scale & Labels tab. In the Labels area, click the *Style Labels* icon.

☐ **Shortcut Menu:** Right-click an axis label, and click *Style*.

The Style dialog box opens.

3. Select your styling options.
4. Click *OK*.

The scale labels reflect the style selections that you have made.

Procedure: How to Format Axis Labels

1. Create a gauge chart.
2. Access the menu of available format options for axis labels in one of the following ways:

☐ **Ribbon:** On the *Format* tab, in the *Features* group, click *Gauges* to open the Format Gauge dialog box. In the Format Gauge dialog box, open the Axis Scale & Labels tab. In the Labels area, open the *Format Labels* drop-down menu.

☐ **Shortcut Menu:** Right-click an axis label, and point to *Format*.

The menu of available format options opens.

3. Select your format option.

Note: If you select a custom format (for example, User Pattern, or Use Pattern /100), from the Format Labels menu on the Format Gauge dialog box, the Custom Format menu becomes available. When you select a custom format, it must be defined using a custom format pattern. For a list and description of the characters that you can use in a custom format, see the table in [Formatting Data Labels](#) on page 587.

4. Click *OK*.

The scale labels reflect the format selection that you have made.

Procedure: How to Rotate Axis Labels

1. Create a gauge chart.
2. Right-click an axis label.
3. On the shortcut menu, point to *Rotate*, then select the degree to which you want the labels rotated.

Procedure: How to Set Gauge Tick Marks

This procedure describes how to set the grid step, tick color, and tick length for the major and minor tick marks.

1. Create a gauge chart.
2. Open the Format Gauge dialog box in one of the following ways:
 - ☐ **Ribbon:** On the *Format* tab, in the *Features* group, click *Gauges*.
 - ☐ **Shortcut Menu:** Right-click the gauge chart and select *More Gauge Options*.

The Format Gauge dialog box opens.

3. On the *Tick Marks* tab, for the *Automatic Grid Step*, either accept the value automatically supplied by the chart engine, or enter your own, in the *Value* fields.
4. Click the *Tick Color* icon, to open the Color dialog box, where you can select a color for the tick mark.
5. For the *Tick Mark Length*, either accept the value automatically supplied by the chart engine, or enter your own, in the *Value* fields.

Procedure: How to Set Gauge Color Bands

1. Create a gauge chart.
2. Open the Format Gauge dialog box in one of the following ways:
 - ☐ **Ribbon:** On the *Format* tab, in the *Features* group, click *Gauges*.
 - ☐ **Shortcut Menu:** Right-click the gauge chart, and click *More Gauge Options*.

The Format Gauge dialog box opens.

3. Set the minimum and maximum value for each band that you want to appear on the gauge chart.
4. Set the color of the fill and border for each color band.

Note: The gradient fill option is not available for color bands.
5. Click *OK* to close the Format Gauge dialog box.

The bands are formatted accordingly.

Procedure: How to Match Needle Center Color to Band Color (Gauge Center by Quality)

This option matches the needle center color to the color of the band that the needle is pointing to.

1. Create a gauge chart.
2. On the *Format* tab, in the *Features* group, click *Gauges*.

The Format Gauge dialog box opens.

3. On the Bands tab, set the minimum and maximum value for each band that you want to appear on the gauge chart
4. Set the color of the fill and border for each color band.

Note: The gradient fill and transparency options are not available for gauge needles.

5. On the Advanced tab, select *Gauge Center by Quality*.

Note: This option displays in HTML, but not in HTML5.

6. Click *OK* to close the Format Gauge dialog box.

The color of the center of the gauge needle now matches the color of the band.

Procedure: How to Style a Gauge Needle

This procedure explains how to change the gauge needle from the Normal style to the Pencil style.

1. Create a gauge chart.
2. Open the Format Gauge dialog box in one of the following ways:

☐ **Ribbon:** On the *Format* tab, in the *Features* group, click *Gauges*.

☐ **Shortcut Menu:** Right-click the gauge chart, and click *More Gauge Options*.

3. The Format Gauges dialog box opens.
4. In the General Options tab, in the Gauge Needle section, from the Style drop-down menu, select *Pencil*.
5. Click *OK*.

The gauge needle changes to a pencil.

Procedure: How to Set Gauge Needle Colors in a Multi Series Gauge Chart

You cannot change the fill color and border color of the needles in a multi-series gauge chart through the General Options tab of the Format Gauge dialog box. You can do so through the Format Series dialog box. However, you can still use the General Options tab to change the style of the needle.

1. Create a gauge chart with more than one measure field.
2. On the *Series* tab, from the Series drop-down menu, select the series that you want to style.
3. In the Properties group, click *Style*.
The Format Series dialog box opens.
4. On the Fill tab, select the fill and color options for the gauge needle for the selected series.
5. On the Border tab, select the *Show Border Color* option, if you want the gauge needle to have a border. You can select the color for the border, as well.
6. Click *Apply*.
7. Repeat steps 2 through 5 for each series gauge needle that you want to style.
8. When you are finished, click *OK* to close the Format Series dialog box.
The color of the gauge needles and their borders changes accordingly.

Formatting Page Headings and Page Footings

Page headings and page footings supply context and key information about a chart, such as its purpose, audience, and author. Page headings and page footings also enhance visual appeal.

Procedure: How to Add a Page Heading and Page Footing to a Chart

The procedure uses sample values, but you can supply values that apply to your own charts.

This feature is available in Query Design view, Live Preview, and Document view.

1. Create a chart or visualization.
2. On the *Home* tab, in the *Report* group, click *Header & Footer*.

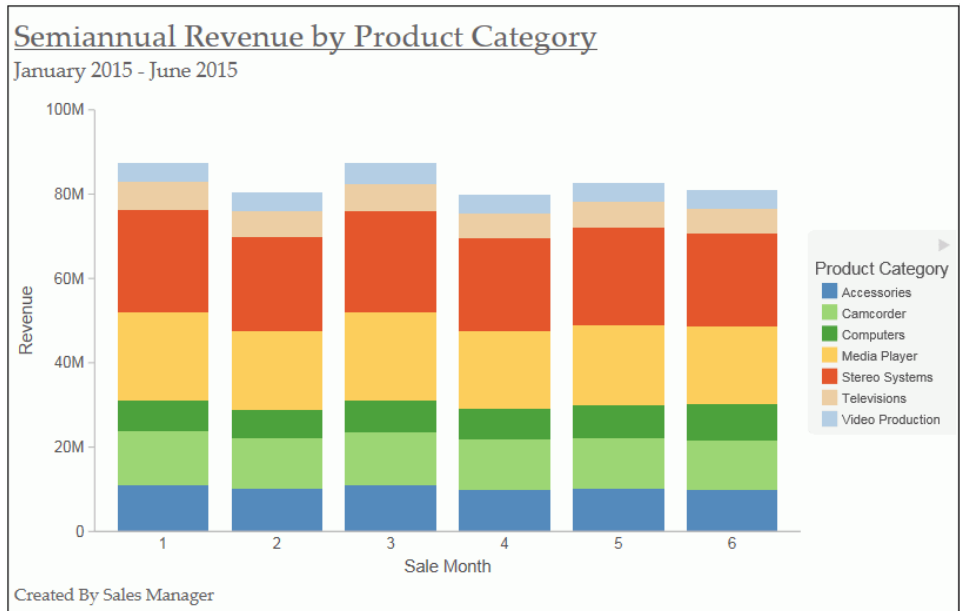
The Header & Footer dialog box opens.

Tip: Another way to access the Heading & Footing dialog box is to click the arrow next to the Header & Footer button. Doing so opens a drop-down menu from which you can select the heading or footing that you want to work with. After you make your selection, the Header & Footer dialog box opens, and the heading or footing that you selected is active.

3. Click the tab for the page heading or page footing, depending on which you want to add.
By default, the Page Heading tab is selected. In this procedure, accept the default to add a page heading first.
4. Click inside the design area of the dialog box, and type the text for the page heading.
For example, the text for a sample page heading might be Customers By Occupation.
5. Using the styling options, apply styling to the page heading text.
For example, 14 pt. bold Helvetica with center alignment.
6. Click *Apply* to save the changes that you have made, without closing the dialog box.
7. To add a page footing, click the *Page Footer* tab.
For this procedure, you are going to add one of the supplied quick text options, followed by some text of your own.
8. Click the down arrow next to the preformatted text button, and click *Created by* from the drop-down list.
9. Complete the text by typing the applicable value within the supplied text, for example, Sales Manager.
10. Change the styling as you want.
11. Click *OK* to save the page heading and page footing and close the Header & Footer dialog box.

The chart contains the page heading and page footing that you added and styled.


The following image shows a chart with a styled header and footer.



12. To make changes to either the page heading or page footing, return to Design view. Right-click the header or footer, and click *Edit* from the drop-down menu.

Procedure: How to Control the Rendering of a Page Heading and Page Footing

You can control the way in which a page heading and page footing are rendered on a chart at run time.

1. Create a chart or visualization.
2. Open the Header & Footer dialog box, as described in [How to Add a Page Heading and Page Footing to a Chart](#) on page 635.
3. Click the *Text options for chart* button .

A drop-down menu, with two options for rendering the page heading and page footing, opens.

Tip: Your selection for a page heading also applies to a page footing, and vice versa.

- ☐ Create Header and Footer as text renders the heading and footing as text elements that are separate from the chart image.
- ☐ Embed Header and Footer in the chart renders the heading and footing text as part of the chart image. This is the default value.

4. Click **OK** to save your selection and close the Heading & Footing dialog box.

Using Additional Formatting Features

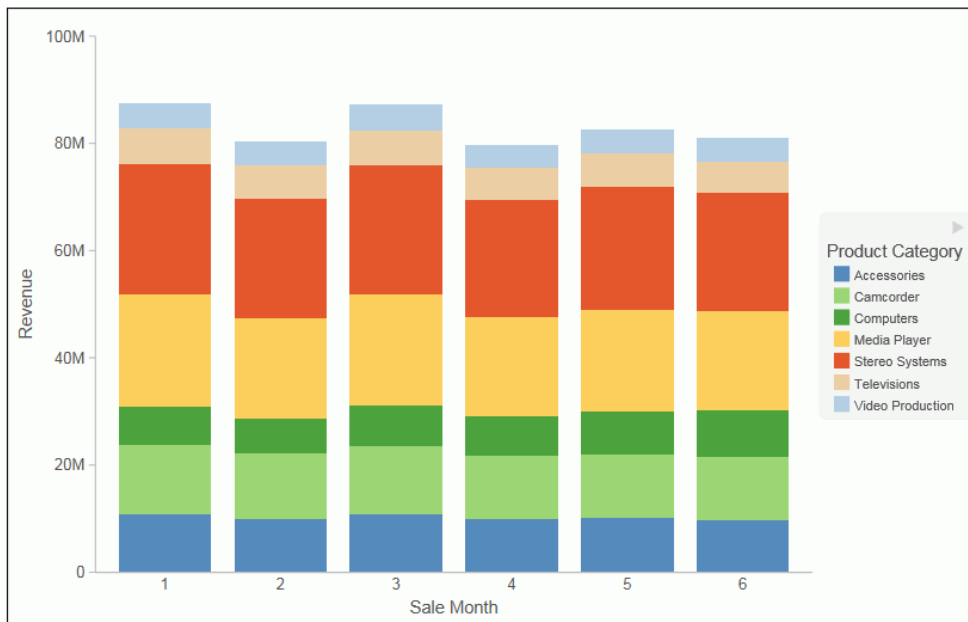
This section describes additional formatting features that are available for charts.

Procedure: How to Set 3D Depth on a Bar Chart

You can apply a 3D-effect to a 2D-chart.

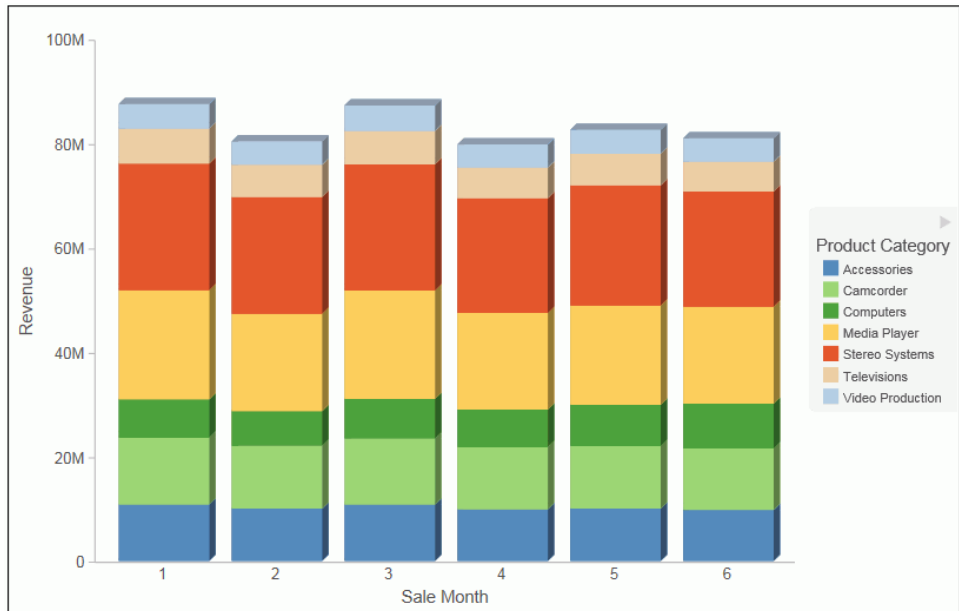
Note: If you find that the depth of the 3D effect makes it difficult to distinguish the values on the Y-axis scale, then you can turn this option off. This procedure explains how to set the 3D effect for a default vertical bar chart.

The following image shows a bar chart before the 3D effect is applied.



1. Create a 2D chart.
2. In Live Preview, apply the 3D effect in one of the following ways:
 - ☐ **Ribbon:** On the *Format* tab, in the *Features* group, click **3D Effect**.
 - ☐ **Shortcut Menu:** Right-click the frame of the chart, point to **Show 3D**, and click **On**.

The 3D effect is applied to the chart, as shown in the following image.



Procedure: How to Rotate a Chart

You can rotate bar, line, and area charts to change the orientation of the data.

1. Create a chart or visualization.
2. On the *Format* tab, in the *Features* group, click *Rotate*.

The chart is rotated 90 degrees clockwise.

Procedure: How to Display a Static Reference Line

Reference lines draw attention to specific data locations on a chart. You can add up to three horizontal (X axis) and three vertical (Y axis) reference lines to a chart.

Reference lines can be created using either a constant or a computed field.

Field values allow a property to be dynamic, so that the reference line can change position dynamically on the chart depending on the value of the field that is assigned to it.

Note: A reference line value should be a single value, such as 10K, for example. In the Reference line dialog box, if you select a field with multiple values, (Quantity Sold, for example), then InfoAssist+ plots the last value on the chart.

1. Create a chart or visualization.
2. On the *Format* tab, in the *Features* group, click *Reference*.
3. In the drop-down menu that opens, select one of the following:

☐ Add Reference Line to Y-Axis

☐ Add Reference Line to X-Axis

The Reference Line dialog box opens.

4. Create the reference line with a static value that you set.
 - a. Select the *Value* option.
 - b. In the *Value* text field, type the value that you want to distinguish with the reference line.
5. In the *Text* field, type the text for the reference line.
6. In the *Position* drop-down menu, select the position for the reference line on the chart. For example, *Above Center*.
7. Click the *Style* button to set the style for the reference line. For example, *Solid*.
8. Click the *Color* button to open the *Color* dialog box, where you can select a color for the reference line and the text.
9. Click the *Weight* button to select the weight of the reference line. For example, *1px - Light*.
10. Click *OK* to save the options that you have selected, and close the *Reference Line* dialog box.

The reference line is added to the chart.

Procedure: How to Display a Dynamic Reference Line

Reference lines can be created using either a constant or a computed field.

Field values allows a property to be dynamic, so that the reference line can change position dynamically on the chart depending on the value of the field that is assigned to it.

Note: A reference line value should be a single value, such as 10K, for example. In the Reference line dialog box, if you select a field with multiple values, (Quantity Sold, for example), then InfoAssist+ plots the last value on the chart.

The following procedure provides an example of how to create a dynamic reference line, where the value changes based on the date. In this example, you can track the number of estimated delivery days per shipping company, where the number of days estimated for delivery are based on whether or not the sale date is a holiday.

1. Create a COMPUTE field to calculate the Reference Line.
 - a. On the *Data* tab, in the *Calculation* group, click *Summary (Compute)*.
The Summary Field (COMPUTE) dialog box opens.
 - b. In the Field text box, enter *Reference*.
 - c. Set the format to *D12.2*.
 - d. Enter *IF WF_RETAIL_LITE.WF_RETAIL_TIME_SALES.TIME_HOL EQ 'Y' THEN 5 ELSE 2* into the expression.
 - e. Click *OK* to close the dialog box.
2. Hide the *Reference* field from the chart.
3. Add a dynamic reference line.
 - a. On the *Format* tab, in the *Features* group, click *Reference*.
 - b. From the drop-down menu, select *Add Reference Line to Y-Axis*.
The Reference Line dialog box opens.
 - c. Select the *Field* option.
 - d. From the Field drop-down menu, select *Reference*.
 - e. In the Text field, enter *Acceptable*.
 - f. From the Weight drop-down menu, select *2px - Medium*.
 - g. From the Position drop-down menu, select *Above Left*.
 - h. Optionally, you can change the color and style of the Reference line.
 - i. Click *OK* to close the Reference Line dialog box.
4. Run the chart.
Auto Prompt appears.
 - a. Select *20100101* (January 1, 2010) and click *Run* to see that five days are acceptable since the date is a holiday.
 - b. Select *20100202* (January 2, 2010) and click *Run* again to see that two days are acceptable since the date is not a holiday.
5. Click *Save*.

Procedure: How to Display Annotations

Annotations are explanatory notes or comments. You can add up to eight annotations on a chart.

1. Create a chart or visualization.
2. On the *Format* tab, in the *Features* group, click *Annotate*.
3. In the drop-down menu that opens, click *Add an annotation*.
The Annotation dialog box opens.
4. In the text input field, type the text for the annotation.
5. In the Position drop-down menu, select the position for the annotation on the chart. For example, Top Left.
6. Click *OK* to save the options that you selected, and close the Annotation dialog box.
The annotation is added to the chart.
7. Click *Run* to generate the report.

Collation in Charts

Charts that have the new attribute syntax respect collation, which provides alphabetical sorting. Collation is also used in other areas, such as filtering.

Collation is set to *CODEPAGE* by the WebFOCUS Business User Edition installation, by default. This can be changed in the Custom Settings section of the Administration Console. For more information, see [How to Configure Collation Sequence Settings](#) on page 114.

Using Storyboards

The Storyboard feature allows you to capture and preserve snapshots of your visualizations as you create them.

Each time you add something to the storyboard, it is placed onto a unique PowerPoint slide. When you click *Show*, you can open and review your slides in PowerPoint.

Since storyboards give you the ability to turn your visualizations into a PowerPoint presentation, you can modify your storyboard to create a customized demo. For example, you can rearrange or delete slides, or add titles and text.

You can also make your visualization portable by saving the PowerPoint files that you create and sharing the slides through email or other electronic distribution.

Procedure: How to Create a Storyboard

1. Create a visual, for example, a bar chart or a grid.
2. On the *Home* tab, in the *Storyboard* group, click *Add*.

Note: When you receive the *Added to Storyboard* message, click *OK*.

3. Insert another visual. On the *Home* tab, in the *Visual* group, click *Insert*.

Note: You can also click the down-arrow on the *Insert* button to insert a chart, grid, or text.

4. On the *Home* tab, in the *Storyboard* group, click *Add*.

Repeat steps 2 through 4 until you complete your data analysis.

5. On the *Home* tab, in the *Storyboard* group, click *Show*.

A new browser window opens, and you are prompted to open or save the PowerPoint file that contains your storyboard.

- ☐ If you click *Open*, the storyboard opens in PowerPoint. You can save the file as a PowerPoint presentation on your machine.
- ☐ If you click *Save*, the storyboard will be saved to a PowerPoint (.pptx) file and placed in the Downloads folder on your machine. Click the down arrow on the right side of the *Save* button to perform a *Save as*. This allows you to specify the drive and directory to which the file will be saved.

In PowerPoint, each storyboard frame displays as a PowerPoint slide, as shown in the following image.



Animating Visualizations

You can animate visuals in a visualization. Each individual visual can have different animation settings. The animations for each visual execute when you run the visualization.

Procedure: How to Add Animation to a Visual

1. Create a visualization with one or more visuals.
2. Select the visual that you want to animate.
3. On the *Format* tab, in the *Interactive* group, click *Interactive Options*.
4. On the Interactive Options dialog box, select the *Show Animation* check box.

This animates the visual.

5. Set the duration for the animation, which determines the speed at which the animation occurs. The default value is 1400ms. Increase the value to make the animation slower, and decrease the value to make it faster.
6. Optionally, clear the *Enable Hover Over Effect* check box.

This option lightens or adds a border to the segment of the visual that you move your mouse over.

7. Click *OK*.

8. Click *Run*.

Your visual animates automatically based on the options that you specified.

Using Filters to Customize the Display of Data

Filters enable you to customize the display of data in your report, chart, document, or visualization. This gives you the advantage of viewing only the data that you want to see and use.

When creating a report, you refer to fields in several parts of the request. For example, in display commands (PRINT, SUM), in sort phrases (BY, ACROSS), and in selection criteria (WHERE, WHERE TOTAL, IF).

The WHERE phrase selects records from the data source to be included in a report. The data is evaluated according to the selection criteria before it is retrieved from the data source. You can use as many WHERE phrases as necessary to define your selection criteria.

In WHERE TOTAL tests, however, data is selected after all the data has been retrieved and processed.

You can group conditions and expressions within filter criteria. In addition, you can apply functions and calculations within criteria. For more information on filtering options, see [Field Tab](#) on page 80.

You can create Where and Where Total filters in the Filter dialog box by clicking *WHERE*.

Double-clicking the *Double-click or press F2 to edit!* text opens drop-down menus for Fields and Subqueries, Operators, and Values.

You can retrieve fields and values from the Master File and data source and Subqueries from a HOLD file.

The Field drop-down menu provides a field list from the Master File. The Field list can be viewed as follows:

- ☐ In business order (default)
- ☐ In a sortable grid
- ☐ In a hierarchical structure of the data
- ☐ If you select Subquery, then the Operator drop-down menu is replaced by a menu with the options of *In list* and *Not in list*.
- ☐ **In list.** Activates a drop-down menu in the right-most part of the expression. The drop-down menu contains a list of all subqueries that you have created from the Master file in use, and an EXISTING option.

Click *EXISTING* to open the Open dialog box, where you can select a subquery from a different Master file.

- ☐ **Not in list.** Opens the Open dialog box, where you can select a subquery from a different Master file.

The Operator (default) drop-down menu provides various operators for your filter. For example, Equal to.

The Value drop-down menu opens a dialog box with multiple options.

Note: If you are creating a filter on a full date field, the Value field will have a calendar icon adjacent to it. You can use this icon to select a date using a calendar control.

The Type drop-down menu contains the following options:

- ☐ **Constant.** Enables the entry of a literal constant value.
- ☐ **Parameter.** Enables you to specify a parameter by entering a name and description in the provided text input areas, as well as selecting the type of parameter (Simple, Static, Dynamic, or Optional).
- ☐ **Field.** Enables the specification of a field name to compare against.

The value area generally contains a text input box that you can use to manually insert values. However, if you are working with a date field, the Value field is set to Today. You can optionally select Beginning of Month, End of Month, Beginning of Quarter, End of Quarter, Beginning of Year, End of Year, or you can specify a Custom date using the calendar that displays when you make that selection.

Note: The default date of Today applies to Report, Chart, and Document modes only.

The value area also contains a Get Values drop-down menu, which supplies the following options:

- ☐ **All.** Retrieves all the values from the selected field.
- ☐ **First.** Retrieves the first value from the selected field.
- ☐ **Last.** Retrieves the last value from the selected field.
- ☐ **Minimum.** Retrieves the minimum value from the selected field.
- ☐ **Maximum.** Retrieves the maximum value from the selected field.
- ☐ **From File.** Retrieves a value that you specify. Selecting this option opens the Select From File dialog box, where you can browse to a local file and specify a file format, such as a flat file (CSV) or an Excel spreadsheet (XLS or XLSX).

Note: The Get Values drop-down menu is only accessible if you have already selected a field.

After selecting the values that you want, you can move them into and out of the Multiple Values area with the left and right arrows. You can also change the value order and delete values with the up and down arrows and the Delete icon.

After creating a condition, you can insert additional conditions before and after the selected condition by using the *Insert Before* and *Insert After* buttons at the top of the Filter dialog box. You can use either the *And* or the *Or* conjunction to link conditions and the *Group* and *Ungroup* buttons to nest and organize conditions.

You can create additional filters by clicking the *New Filter* button at the top of the Filter dialog box.

After creating the filters that you want, click *OK* to save and apply the filters. You can access them from the Filter pane of the Resources panel.

Including or Excluding a Filter

Once you have created a filter, you can make decisions about when and where you want to include it in your report. For example, you might want to include one filter and exclude another. Using the Include and Exclude options in the Filter group on the Home or Field tabs, you can set filters accordingly.

You can use the following information to make decisions about your filters:

- ☐ **Exclude.** Removes, but does not delete, the filter from the report.
- ☐ **Include.** Restores a filter that was previously excluded from a report.

Using the Prompt Functionality to Select Field Information at Run Time

Using the prompt functionality, you can create an auto prompting parameter for which you can define a value at run time. This feature is located in the Filter Group on the Field tab.

To create an auto prompting parameter:

1. Select a field for which you want to create a parameter in your report.
2. On the Field tab, in the Filter group, click *Prompt*.

The Create a filtering condition dialog box displays.

3. Double-click the *Value* field.

Note: Parameter is selected, by default, when you are defining a prompt.

The following options display, from which you can make a selection:

- ☐ **Simple.** This is used for prompts using Text Input. This is the default value.
 - ☐ **Static.** This is used for prompts using Selection. This option allows you to select multiple values at run time.
 - ☐ **Dynamic.** This is used for prompts using Data Values. This option allows you to select multiple values at run time.
 - ☐ **Optional.** This is used for prompts using Single or Multiselect parameters.
4. Click *OK*, and click *OK* again to close the Create a filtering dialog box.

When you run your report, you will be prompted for information based on the parameters you created.

Using Auto Drill

Auto Drill enables you to navigate through different levels within the dimension hierarchy of your data source. This allows you to review underlying data for a particular area, and move through the structure of your data source based on your informational needs.

In order to use Auto Drill, you must enable it by clicking the Auto Drill button on the Format tab, in the Navigation group.

Once you have enabled Auto Drill, you can develop your report or chart and run it to activate the hyperlinks that Auto Drill creates. This enables you to navigate up or down through the related hierarchy of your data source by clicking the links that display.

When you click a hyperlink, the option to drill up or drill down displays, depending on where you are in the hierarchy of your data. You can then select one of these options to begin navigating your data. For example, if you have Product Category in your hierarchy, you would be able to drill down to Product SubCategory. When you drill down, you can subsequently drill back up to the originating dimension sort field. If you have selected a data source that has multiple levels and your report or chart uses a component in the middle of the hierarchy, both the Drill up and Drill down options will display. Once you have started navigating your data using the Drill up and Drill down options, the Restore Original option displays, enabling you to start your data analysis over by re-executing the original Auto Drill request.

At run-time, the Auto Drill functionality displays a breadcrumb header. This defines your current location in the hierarchy of your data source and enables you to navigate back and forth between different levels in your data.

Important:

- ☐ The Auto Drill functionality is only available for data sources that have a dimension hierarchy.
- ☐ If you have Auto Drill enabled and you attempt to run your report or chart in deferred mode, the hyperlinks will not display. This is also true for Auto Drill enabled reports that are distributed through ReportCaster.
- ☐ With the exception of Visualization mode, Auto Drill is available at run-time, but not in Live Preview.
- ☐ Auto Drill is supported for HTML and active report output format reports.
- ☐ Auto Drill is automatically enabled in Visualization mode for data sources that have a dimension hierarchy.
- ☐ Auto Drill functionality is not available in reports distributed by ReportCaster, because Auto Drill uses live data, in an interactive session, for data drilling. Data values and totals may not be the same if the data has changed since the last distribution. Mixing past data with current data could impact data analysis.
- ☐ Auto Drill is not supported for HTML5 output format charts that do not support the new chart attributes syntax.
- ☐ Auto Drill is not supported on Across fields in active reports.
- ☐ You must use dimension fields as a sorting field, either BY or ACROSS.

Procedure: How to Use Auto Drill to Navigate the Hierarchy of Your Data Source

1. Select a data source that has a dimension hierarchy. For example, wf_retail_lite.mas.
2. Create a report or chart with one or more hierarchical fields (for example, Product,Category).
3. On the *Format* tab, in the *Navigation* group, click *Auto Drill*.
4. Click *Run*.
The report or chart displays.
5. For reports, click a hyperlinked field within your report. For charts, hover over a chart aspect (for example, a bar).
A menu appears.
6. Select a hierarchical value to which to drill down to.
Once you have drilled down on a field, you can subsequently drill up.

7. To return to the default state of the report or chart, click a hyperlink or hover over a chart aspect, and then click *Restore Original*.

Using the Auto Linking Feature to Link Content

Auto Linking makes it easy to connect reports and charts in your development environment, expanding the reporting capabilities of your organization. Using Auto Linking, you can dynamically link HTML reports, active reports, and HTML5 charts with a single report or chart of any format, based on their common sort (BY) fields and parameters referenced in any filters. It is this commonality that dynamically links content in your WebFOCUS repository, allowing you to discover new possibilities in your data, and explore new relationships within your reporting enterprise. In addition, Auto Linking saves development time and effort, because drilldowns do not need to be manually created and maintained.

To use Auto Linking, you must use the Enable Auto Linking option in InfoAssist+ to set the reports or charts that can be auto-linked. In addition, you must also set reports or charts to be Auto Link targets. The options to enable and disable Auto Linking and Auto Link Target functionality are available on the Format tab, in the Auto Linking group, in InfoAssist+.

Note:

- ☐ The Auto Link Enabled option is activated only when the HTML, HTML5, and active report output format is selected.
- ☐ You can open an existing report or chart and enable Auto Linking or set the item as an Auto Link target.

When Auto Link functionality is activated, your reports or charts display a link for each sort (BY) field with qualifying target reports or charts at run time. In Auto Link enabled reports, you must add sort (BY) fields in the BY field container. In Auto Link enabled charts, you can add sort (BY) fields to the Horizontal Axis, Vertical Axis, or Color field containers.

To qualify a report or chart as an Auto Link target, you must include filters that contain the parameters that you selected as the sort field or fields in the enabled report or chart. The parameter names defined in these filters must be the same as the sort (BY) field names in the Auto Link enabled report. When you select a report or chart as the Auto Link Target, it specifies that the parameter information should be catalogued, and will be evaluated when an Auto Link enabled report or chart is run.

For an Auto Link enabled report at run time, the qualifying target reports and charts are those that have filters with parameters for the selected sort field and the sort fields that precede it, which appear to the left. For an Auto Link enabled chart at run time, the target reports and charts are those that have filters with parameters for all sort fields. For run time for both reports and charts, the linked sort field values in the Auto Link enabled report or chart are passed to the Auto Link target report so that it can be filtered by the sort (BY) field values.

For example, you may have an Auto Link enabled report that contains sort (BY) fields Product Category and Model, with a measure (Sum) field, Revenue. To qualify as an Auto Link target, other reports or charts in your repository can contain a single filter with a parameter for Product Category, or two filters with parameters for both Product Category and Model. When you run the Auto Link enabled report, the Product Category field will link to target reports or charts that have a filter with a parameter for Product Category and the Model field will link to target reports or charts that have filters for both Product Category and Model.

If you use the same example for an Auto Link enabled chart instead of a report, you can access the target reports or charts from a tooltip option that displays when you point your mouse over an area of the chart, such as a bar that represents Revenue by Product Category and by Model.

Note:

- ☐ The linked reports displayed are limited to those that you are authorized to run or run deferred.
- ☐ When running an HTML report with On-demand Paging enabled, the qualifying Auto Link targets appear on the menu in the order that they are specified in the request. This is also true for a chart request that is a legacy graph format (PFJ-based formats, such as PNG and non-bucket HTML5). In reports, horizontal lines appear in the menu to separate user specified drilldowns and Auto Link navigation options.
- ☐ Consideration should be given to how many reports or charts are indicated as Auto Link targets, as the run time Multi-drill menu for the qualifying target reports may become long in length. In these cases, some browsers may display a script processing warning message.
- ☐ The Auto Link Enabled and an Auto Link Target options can be set individually or both can be set on the same report or chart if that item meets the Auto Linking requirements.

Procedure: How to Set an Existing Report or Chart as Auto Link Enabled

1. In the Resources tree, right-click a report or chart and click *Edit*.
InfoAssist+ opens in the relevant mode.

2. Verify that there is a sort (BY) field in the report or chart.
 - ☐ For reports, sort fields are added to the BY field container.
 - ☐ For charts, sort fields are added to the Horizontal, Vertical, or Color field containers.
3. On the *Format* tab, in the Auto Linking group, click *Enable Auto Linking*.
4. Save the report or chart.

Your existing report or chart is now Auto Link enabled.

Procedure: How to Set an Existing Report or Chart as an Auto Link Target

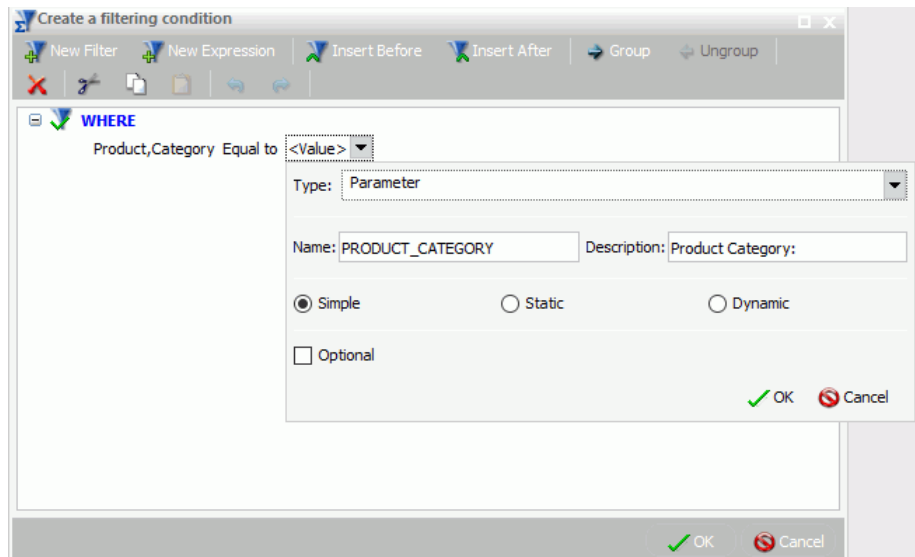
1. In the Resources tree, right-click a report or chart and click *Edit*.

InfoAssist+ opens in the relevant mode.

2. On the *Format* tab, in the Auto Linking group, click *Auto Link Target*.

Note: Selecting Auto Link Target specifies that the parameter information for this report or chart will be stored, adding it to the repository of reports and charts that will be evaluated when an Auto Link enabled report is run.

3. Add a filter with a simple or optional parameter to the target report or chart. Qualifying target reports are those that have filters with parameters for the sort (BY) fields in Auto Link enabled reports or charts.
 - a. From the Data pane, drag a sort (BY) field into the Filter pane.
 - b. Using the Create a filtering condition dialog box, add the selected parameter as simple, as shown in the following image.

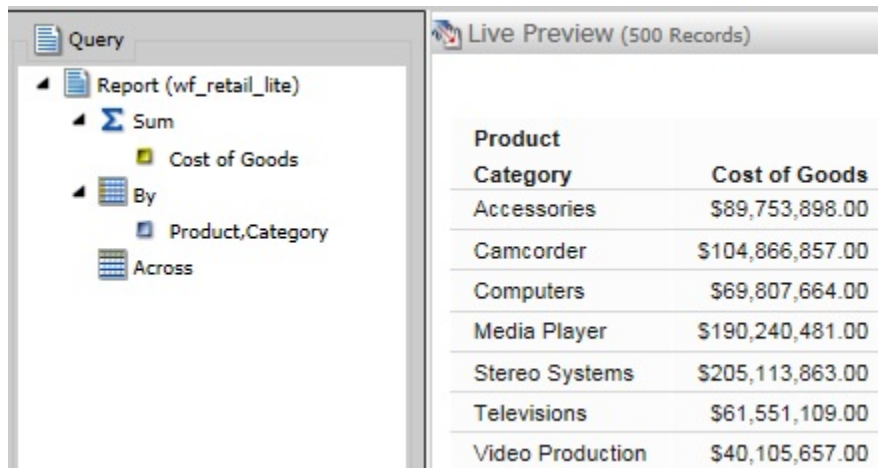


- c. Click **OK**, and then click **OK** again to exit the Create a filtering dialog box.
4. Save the report or chart.

Your existing report or chart is now set as an Auto Link target.

Procedure: How to Create a New Auto Link Enabled Report or Chart

1. Open InfoAssist+ in report or chart mode.
2. From the Open dialog box, select a data source.
3. On the *Format* tab, in the Auto Linking group, click *Enable Auto Linking*.
4. Add fields to the report or chart, ensuring that one is a sort (BY) field.
 - ☐ For reports, sort fields are added to the BY field container.
 - ☐ For charts, sort fields are added to the Horizontal, Vertical, or Color field containers.



The screenshot shows the InfoAssist+ interface. On the left, the 'Query' pane displays a tree structure for a report named 'Report (wf_retail_lite)'. The structure includes a 'Sum' container with 'Cost of Goods' and a 'By' container with 'Product,Category'. The 'Across' container is also visible. On the right, the 'Live Preview (500 Records)' pane shows a table with two columns: 'Product Category' and 'Cost of Goods'.

Product Category	Cost of Goods
Accessories	\$89,753,898.00
Camcorder	\$104,866,857.00
Computers	\$69,807,664.00
Media Player	\$190,240,481.00
Stereo Systems	\$205,113,863.00
Televisions	\$61,551,109.00
Video Production	\$40,105,657.00

5. Save the report or chart.
- Your report or chart is now set as Auto Link Enabled.

Procedure: How to Create a New Auto Link Target Report or Chart

1. Open InfoAssist+ in report or chart mode.
2. From the Open dialog box, select a data source.
3. On the *Format* tab, in the Auto Linking group, click *Auto Link Target*.

Note: Selecting Auto Link Target specifies that the parameter information for this report or chart will be stored, adding it to the repository of reports that will be evaluated when an Auto Link enabled report is run.

4. Add fields to the report or chart.
5. Add a filter with a simple or optional parameter to the target report or chart.
 - a. From the Data pane, drag a sort (BY) field into the Filter pane.
 - b. Using the Create a filtering condition dialog box, add the selected parameter as simple or optional.

Note: When creating a parameter for a field, the parameter name defaults to the fieldname that you select. In the previous image, the creation of a filter with parameter Product Category is shown. To ensure that the name of the parameter is the same as the sort (BY) field in the Auto Link enabled report or chart, do not change the name of the parameter that is generated by your field selection.

- c. Click *OK*, and then click *OK* again to exit the Create a filtering dialog box.
6. Save the report or chart.

Your report or chart is stored as an Auto Link target.

***Procedure:* How to Run an Auto Link Enabled Report or Chart**

You can run an Auto Link enabled report or chart from within InfoAssist or from the Portal.

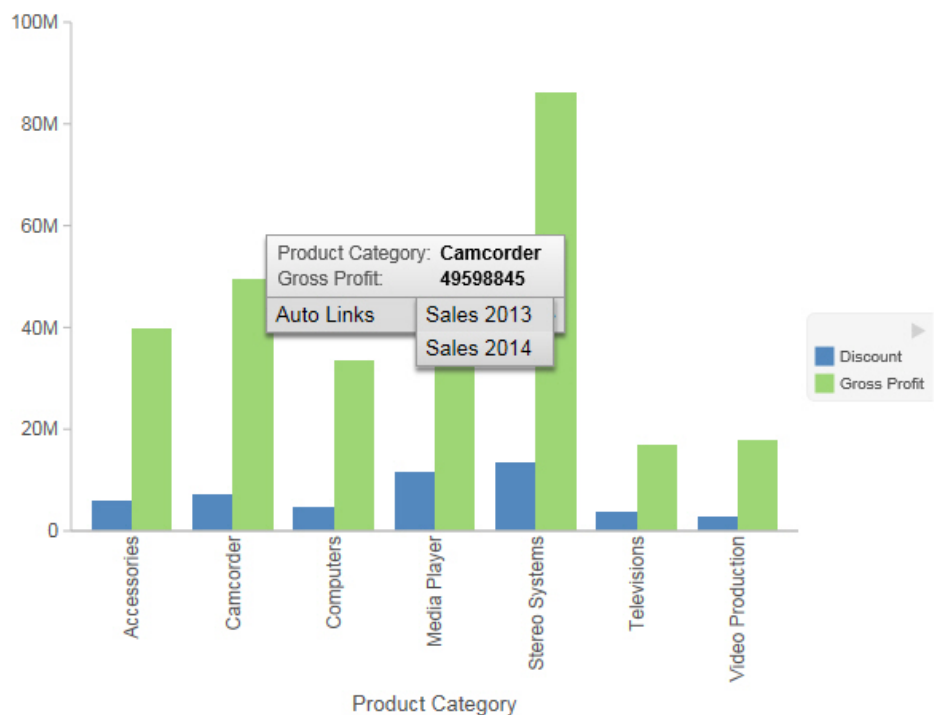
1. Open and run an Auto Link enabled report or chart in one of the following ways:
 - ☐ In InfoAssist+, open or create an Auto Link enabled report and on the Quick Access toolbar, click *Run*.
 - ☐ From the Portal, right-click a report or chart that is Auto Link enabled and click *Run*.

2. Use the hyperlinks to link to other reports and charts, based on the following information:

- ❑ **Reports.** Displays with hyperlinks on the sort (BY) fields that have qualifying target reports. Click a hyperlink to display the menu with the Auto Links option, which cascades to list the qualified Auto Link target reports and charts, as shown in the following image.

Product Category	Revenue	Quantity Sold	Cost of Goods	Gross Profit
Accessories	\$103,316,482.12	511,667	\$89,753,898.00	\$39,854,440.53
Camcorder	\$103,316,482.12	455,244	\$104,866,857.00	\$49,598,845.24
Computers	\$103,316,482.12	351,777	\$69,807,664.00	\$33,508,818.12
Media Player	\$246,073,059.36	771,934	\$190,240,481.00	\$55,832,578.36
Stereo Systems	\$291,294,933.52	1,114,332	\$205,113,863.00	\$86,181,070.52
Televisions	\$78,381,132.81	105,188	\$61,551,109.00	\$16,830,023.81
Video Production	\$58,053,276.62	199,749	\$40,105,657.00	\$17,947,619.62

- Charts.** Displays with aspects over which you can hover (for example, a bar) in the chart. A tooltip displays with the Auto Link target reports and charts, as shown in the following image.



Using Multi Drill

The Multi Drill functionality enables you to create multiple drill down links on a data field in a report or chart. This is useful when you want to define custom links to other reports or websites.

Using the Multi Drill functionality, you can define conditions for drilling down or through the data element that you select. Once drill down information is saved and you run the report or chart, the drill down options display when you click or hover over the defined area of a report or chart, respectively.

You can create drill down links by selecting a field in the Query pane. On the *Field* tab, in the *Links* group, click *Drill Down* to open the Drill Down dialog box. You can also right-click a field in the Query pane, and click *Drill Down*. The Drill Down dialog box is shown in the following image.

Drill Down - Revenue

Drill Down 1

☒ Report
 ☐ Refresh BI Portal
 ☐ Auto Link Target
 ☐ Web Page

Report*

Browse

Description

Drill Down 1

Target

New Window

Parameters

Name	Value
------	-------

* Required Field

OK Delete all and exit Cancel

When you access the Drill Down dialog box, the name of the field that you selected displays in the title bar. This enables you to identify the current field and is useful when working with Drill Down parameters.

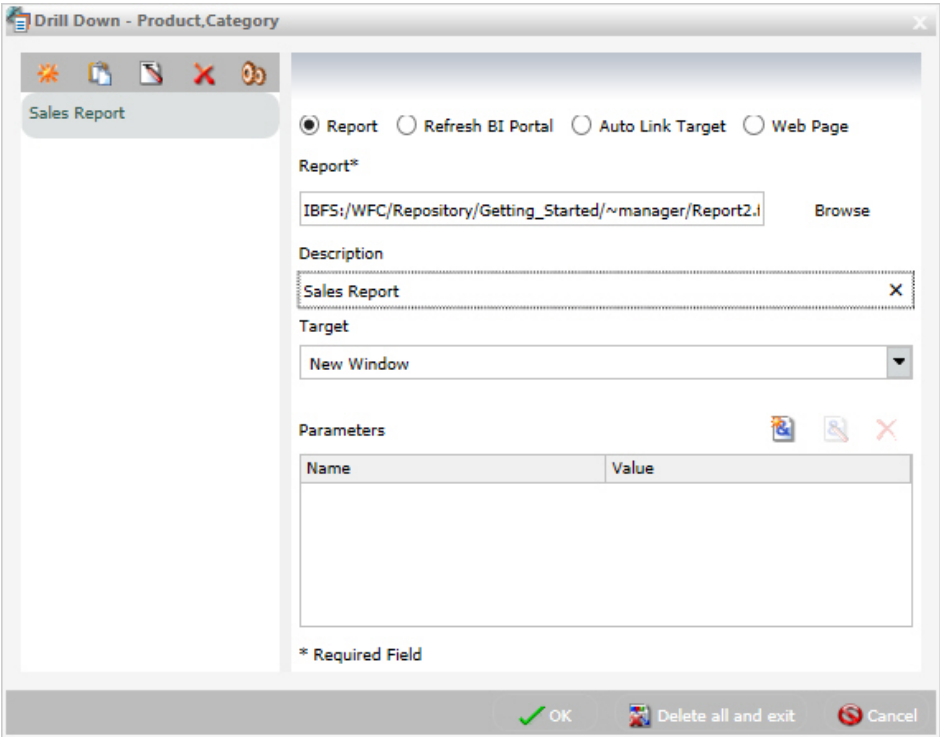
In the Drill Down dialog box, you can configure a hyperlink or drill-down procedure. When you run the report, you can click the link that is created, which would take you to the specified URL or execute the specified procedure.

You can also create multiple drilldowns for a selected field. The drill down links that you define display as separate entries in the drill down list, also known as the drill menu, at run-time. The display of these entries is based on the naming conventions that you applied when creating the multi-drill scenario.

Once your drilldown options are defined, you must run your report in order to access the drill down links that you have created. For reports, the links display when you click on a hyperlink for the data field that you selected. For charts, they display when you hover over an area of the chart for which the drill down has been defined (that is, the field that you selected when you defined the drill down).

These links enable you to drill directly through to an external procedure or URL.

For example, you can create one drill down link for an existing procedure, and another for a webpage. Both drill downs are stored for the underlying field that you selected. Once you define the first drill down, additional options display on the toolbar, as shown in the following image.



Note: You can rename a drill down using the Description field in the right pane of the Drill Down dialog box.

The following icons are available on the Drill Down dialog box toolbar:

- ☐ **Create a new drill down.** This option is enabled when all required information is provided for the first drill down. When you click *Create a new drill down*, a new drill down item is created. This switches the drill down functionality from a regular drill down to multi-drill. By default, the name of the drilldown is Drill Down n, where n is a consecutive integer, such as Drill Down 1, Drill Down 2, etc. This option is initially disabled, but is enabled after you have created at least one drill down link.
- ☐ **Duplicate the selected drill down.** When an existing drill down item is selected, clicking this option duplicates the item. A (Copy n) is appended to the name of the drill down, where n is the consecutive number assigned to the duplicated drill down. Once assigned, the name can subsequently be changed in the Description field. Consecutive copies can also be rearranged by dragging the drill down items into the desired order.

Note: The Duplicate button is only active when sufficient information is available for the selected drill down item.

- ☐ **Rename the selected drill down.** This option enables you to rename an existing drilldown and is active by default. When this option is selected, the Description field is selected, allowing you to provide a new name for the drill down link. You can enter a description up to 100 characters long. If you remove the description, the drilldown is renamed to the default name.
- ☐ **Remove the selected drill down.** If you have a drill down item selected, the Remove the selected drill down option is active. When you select this option, the current drilldown is deleted. If a drill down item is removed, the next drill down item is automatically selected. If you remove the last drill down item, the Drill Down dialog box resets to its initial state, restoring Drill Down 1 as the default value.
- ☐ **Options.** The Options functionality enables you to change the order of your drill down items. This allows you to control the order in which the items are presented in the drill menu when you run your report and click on a hyperlink. You can change the order by selecting a defined drill down in the list that is presented under Edit Order and then clicking the Move Up or Move Down button. You can also remove one or more drill down items from the list by selecting the desired items and then clicking the *Remove from List* button. If you multi-select your drill down items using the CTRL key, the Move Up and Move Down buttons are disabled.

Note: Move Up, Move Down, and Delete are also available as context menu options when you right-click a drill down item in the Edit Order list.

The Drill Down dialog box contains the following options for specifying drill down information for the selected field:

Note: When specifying parameters for your drill down, click *Add Parameter* to add them to your parameters list.

- ☐ **Report.** Select this option to choose a report for the current drill down. This option is selected by default.
 - ☐ **Report.** Enables you to browse to a stored report.
 - ☐ **Description.** Provides a field in which to describe the report. If you selected a report by browsing, this field is automatically populated with the title of the procedure or the name where procedure titles are not accessible. This field is also used to rename a specified drill down.
 - ☐ **Target.** Allows you to specify a target for the drill down. Options include: New Window, Same Window, or a value that you specify. The default value for report is New Window.
 - ☐ **Parameters.** Displays the parameters that you have indicated for the current drill down.
- ☐ **Refresh BI Portal.** Select this option to refresh the components of the BI Portal and pass parameters to them.
 - ☐ **Refresh.** Provides options by which to refresh the page. Options include Current Page and All Pages.
 - ☐ **Description.** Provides a field in which to describe the refresh process. This field is also used to rename a specified drill down.
 - ☐ **Target.** Reflects the target for the drill down. The only applicable value for Refresh BI Portal is Same Window.
 - ☐ **Parameters.** Displays the parameters that you have indicated for the current drill down.
- ☐ **Auto Link Target.** Select this option to create an explicit list of Auto Link targets. Upon selection, these will be available in the drill down menu at run time.

Note: When working with reports, the Auto Link Target drill down option is only available for a selected dimension (BY) field. For more information, see *Using the Auto Linking Feature to Link Content* .

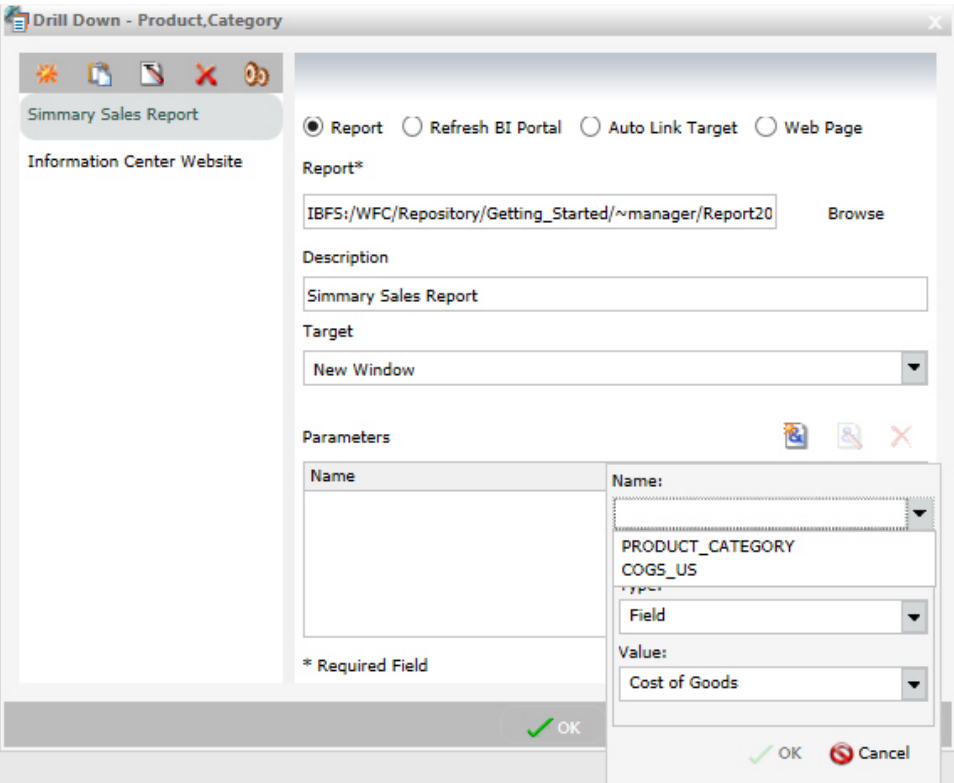
- ☐ **Auto Link Target.** Enables you to browse to a stored target report.
- ☐ **Description.** Provides a field in which to describe the target report. If you originally selected a report by browsing, this field is automatically populated with the report title or the name where titles are not accessible. This field is also used to rename a specified drill down.

- ☐ **Target.** Enables you to specify a target for the drill down. Options include: New Window, Same Window, or a value that you specify. The default value for Auto Link Target is New Window.
- ☐ **Parameters.** Displays the parameters that meet the column-to-parameter requirement for the selected target report.

For more information, see *Using the Auto Linking Feature to Link Content* .

- ☐ **Web Page.** Select this option to indicate that you want to drill through to a webpage. Enter a value in the URL field.
- ☐ **URL.** Enables you to specify the URL for the webpage to which to drill down. This option is available only when Web Page is selected.
- ☐ **Description.** Enables you to enter additional comments related to the drill down. This field is also used to rename a specified drill down.
- ☐ **Target.** Enables you to specify a target for the drill down. Options include: New Window, Same Window, or a value that you specify. The default value is New Window.
- ☐ **Parameters.** Displays the parameters that you have indicated for the current drill down.

If you are working with a report that has parameters, the parameter names are automatically retrieved and made available from the Name control in the Parameters section of the Drill Down dialog box, as shown in the following image.



By default, the field is labeled with a value of Name. If there are multiple parameters to choose from, you can use the drop down list to make a selection. You can also manually enter a name to create a new parameter. The parameters that display are populated from the underlying report procedure (.fex) that you selected for your drill down.

The Type field enables you to specify a field or constant value. The options in the Value list are based on the fields that you selected for your report. The item that you select determines the value for the parameter (for example, Discount or Revenue).

Note: If you switch the Type field from Field to Constant or from Constant to Field, the Value field is reset, enabling you to specify a unique value.

Procedure: How to Create a Single Drill Down Link

You can create a single drill down link on a measure field.

1. Create a report or chart with at least one measure field.
2. In the Query pane, do one of the following:
 - ☐ Right-click the measure field and click *Drill Down*.
 - ☐ Click the measure field and on the *Field* tab, in the *Links* group, click *Drill Down*.

The Drill Down Dialog box displays, as shown in the following image.

Drill Down - Revenue

Drill Down 1

☒ Report
 ☐ Refresh BI Portal
 ☐ Auto Link Target
 ☐ Web Page

Report*

Browse

Description

Drill Down 1

Target

New Window

Parameters

Name	Value

* Required Field

OK Delete all and exit Cancel

3. Select one of the following Drill Down options:

- ☐ Report
- ☐ Refresh BI Portal
- ☐ Auto Link Target
- ☐ Web Page

4. Populate the fields based on the drill down option you selected.

Note: Descriptions for these fields are provided earlier in this topic.

5. Optionally, rename the Drill Down using the Description field. This dictates how the Drill Down will appear in the left pane of the Drill Down dialog box.
6. Click *OK*.

The drill down link displays when you hover over the defined area of a report or chart, at run time.

Procedure: How to Create Multi Drill Links

You can create a multiple drill down links on a single dimension field.

1. Create a report or chart with at least one dimension field.
2. In the Query pane, do one of the following:
 - ☐ Right-click the dimension field and click *Drill Down*.
 - ☐ Click the dimension field and on the *Field* tab, in the *Links* group, click *Drill Down*.

The Drill Down Dialog box displays, as shown in the following image.

Drill Down - Product,Category

Drill Down 1

☒ Report
 ☐ Refresh BI Portal
 ☐ Auto Link Target
 ☐ Web Page

Report*

Browse

Description

Drill Down 1

Target

New Window

Parameters

Name	Value
------	-------

* Required Field

OK Delete all and exit Cancel

3. Select one of the following Drill Down options:

- ☐ Report
- ☐ Refresh BI Portal
- ☐ Auto Link Target
- ☐ Web Page

4. Populate the fields based on the drill down option you selected.

Note: Descriptions for these fields are provided earlier in this topic.

5. Optionally, rename the Drill Down using the Description field. This dictates how the Drill Down will appear in the left pane of the Drill Down dialog box.
6. Repeat steps 3-5 until you have added all of the drill down links required.

Note: The following image defines three unique drill down links for the selected dimension field.

Drill Down - Product,Category

Usage Analysis Summary

Sales Report

Company Website

☒ Report ☐ Refresh BI Portal ☐ Auto Link Target ☐ Web Page

Report*

IBFS:/WFC/Repository/Getting_Started/~manager/Report2.i

Browse

Description

Sales Report

Target

New Window

Parameters

Name	Value
------	-------

* Required Field

OK

Delete all and exit

Cancel

7. Click OK.

The links display when you hover over the defined area of a report or chart, at run time, as shown in the following image.

Product Category	Cost of Goods	Discount
Accessories	\$89,753,898.00	\$6,014,845.52
Camcorder		\$7,108,408.27
Computers		\$4,808,910.10
Media Player		\$1,519,142.41
Stereo Systems	\$205,113,863.00	\$13,530,873.57
Televisions	\$61,551,109.00	\$3,635,693.38
Video Production	\$40,105,657.00	\$2,695,890.76

Adding Parameters for Data Selection at Run Time

You can add new parameters to your report or chart to create parameter options for selection at run time. This enables you to dynamically select and review different scenarios with your data, offering you on-demand, dynamic report or chart creation at run time.

When you add a parameter, it becomes a field container into which you can add one or more measures and dimensions. You create a new parameter by right-clicking on a field container in the Query pane and clicking *New Parameter*. This creates a numbered parameter (for example, Parameter 1). In cases where you have multiple parameters defined, the numbering is applied sequentially.

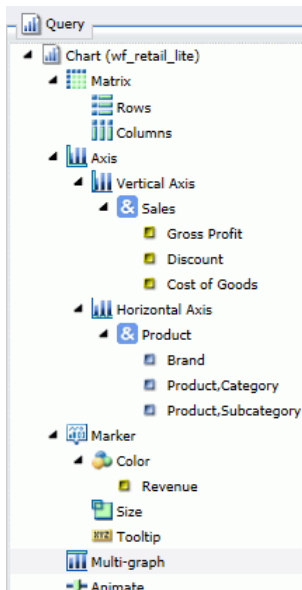
Once you add a parameter, you can rename the variable by right-clicking the *Parameter-n* field container and clicking *Rename Variable*, where *n* is the number of the parameter that was added. Once you rename a parameter field container, the new title displays in the Query pane and at run-time. This allows you to easily identify the parameters while categorizing the data fields that you want each to contain. For example, if you add a number of Sales-related fields to a parameter, you can rename the parameter with a title of Sales to encapsulate the data fields that you selected. Parameters should be renamed before you run the report or chart so that they are easy to identify.

In order to populate parameters in your report or chart, data fields must be placed in the parameter field container. You can do this by selecting fields from the Data pane and dragging them into the relevant parameter field containers that you create.

Note: Double-clicking the data fields in the Data pane places the data field in the default field containers in the Query pane, not the parameter field container. For example, in a report, if you double-click *Discount*, the data field is placed in the Sum container (outside of the parameter field container that you created). You can always drag a data field into a parameter container from another field container in the Query pane if you wish to consider it a parameter.

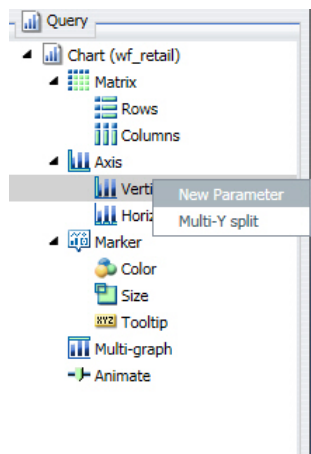
You can also reorder the data fields that you have placed in a given parameter field container by dragging them into the order that you wish to display them at run time. Parameters can be specified in any field container, with the exception of Multi-graph. Once you have defined your parameters and selected the relevant data fields, run the report or chart. Using the drop-down lists for each parameter, you can select different options from the parameter lists to view the data in your report or chart dynamically. To load different scenarios, select the values and then click *Run*.

The following image shows the Query pane with a number of parameters and data fields specified.



Procedure: How to Add a Parameter

1. Create a new report or chart or open an existing one.
2. Add a new parameter by right-clicking a field container and clicking *New Parameter* , as shown in the following image.



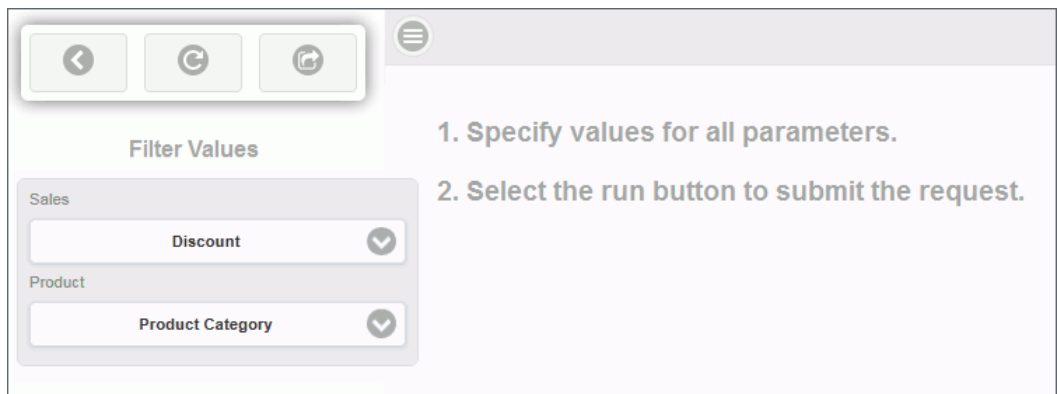
A new parameter field container is added.

3. Add the data fields that you would like to display in a parameter list at run time to the new parameter field container.
4. Optionally, rename the parameter by right-clicking the parameter and clicking *Rename Variable*.

This is not required, but it does help in differentiating the various numbered parameters that display, depending on how many you define.



5. Optionally, add one or more additional parameters, and relevant data fields, into the field containers for which you would like to select data values at run time.
6. On the Quick Access toolbar, click *Run*.



You can use the Parameter drop-down lists that display at run-time to create different views of your data, as shown in the following image.



Note: You can optionally run your report or chart using the default parameters without making a selection. Click the down arrow on the Run button in the Quick Access toolbar and select Run with Default Parameter Values.

The following navigational icons display at run time:

Icon	Description
	Close Filter Panel. Click this option to collapse the filter panel, displaying more of the report or chart in the right pane.
	Reset filter values. Click this option to reset the filter values to their default values, as indicated when the parameter was initially set up.

Icon	Description
	Run with filter values. Click this option to run the report or chart based on your selections.
	Show filter panel. Click this option to show the filter panel when viewing a report or chart. When viewing the filter panel, you can close it by clicking <i>Close Filter panel</i> .

Working With Slicers

Slicers provide a quick and easy way to add dynamic user controls to a chart or report, giving you the ability to filter the data in use as you create or view content. Slicers are dynamic filters that can be changed by making a selection from a menu of fields. You can use them to filter your content by modifying a slicer in the Slicers tab and running the chart or report, or you can use them at run time when running a report in InfoMini with the Slicers tab selected.

Creating Slicers

You can create slicers by dragging fields to the Slicers tab or through the context menu of a field. When you create a slicer group, it appears on the Slicers tab.

You can use slicers to dynamically filter reports at design time.

You can dynamically filter reports at run time when they are included in an InfoMini application. For more information on InfoMini, see [Building InfoMini Applications](#) on page 475.

Procedure: How to Create a Slicer

1. Click the *Slicers* tab.
2. Create a new slicer by using one of the following methods:

- ☐ Click the *New Group* button to create a new slicer group.

Select a field from the Data pane and drag it onto the *Drag Fields Here to Create Slicers* text on the Slicers tab.

Note: You cannot drag parent-child hierarchies onto the Slicers tab.

The field is added to the new group.

or

- ☐ Right-click a field in the Data pane, point to *Slicers*, and then click *New Group*.

The field is added to the new group.

If you are creating a slicer on a full date field, a calendar icon displays adjacent to the field. You can use this icon to select a date using a calendar control.

Procedure: How to Add a Field to an Existing Slicer Group

1. Click the *Slicers* tab.
2. Add a field to an existing slicer group using one of the following methods:
 - ☐ Select a field from the Data pane and drag it onto an existing slicer group.
or
 - ☐ Right-click a field in the Data pane, point to *Slicers*, and then click *Existing Group*.
Select an existing group from the menu, and then click *OK*.
The field is added to the existing group.

Procedure: How to Add a Hierarchy as a Slicer Group

Note: You cannot drag parent-child hierarchies onto the *Slicers* tab.

1. Click the *Slicers* tab.
2. Add a hierarchy as a slicer group using one of the following methods:
 - ☐ Select a hierarchy from the Data pane and drag it onto an existing slicer group.
The hierarchy is added as a new group, not as a slicer in the existing group. The new group is given the same name as the hierarchy.
or
 - ☐ Click the *New Group* button to create a new slicer group.
Select a hierarchy from the Data pane and drag it onto the *Drag Fields Here to Create Slicers* text of the *Slicers* tab.
The hierarchy is added to the new group. The name of the new group changes to the name of the hierarchy automatically.

Note: Hierarchies cannot be added to existing slicer groups. If a hierarchy is dragged onto an existing slicer group, a new group is created automatically.

There is no right-click context menu for hierarchies. To add a hierarchy as a slicer group, you must drag it into a new slicer group.

Filtering With Slicers

Once you add slicers to an InfoAssist+ report, you can use them to filter the report. You can select values from the slicers menus, change the number of records that appear, create new slicer groups, clear existing slicer groups, and update the report preview.

Slicers that are not selected have their values filtered by the selected slicers. Only values that meet the criteria for previously selected slicers will appear in the menu for the next slicer. Slicers are not filtered in the order they appear in the slicer group, but by the order they are selected. Cascading of slicers is only valid for hierarchies.

Procedure: How to Change the Relationship Operator in a Slicer

1. With at least one slicer added to the report, click the *Slicers* tab.
2. Click the operators button on the slicer for the operation you want to change.

A list of operators (numeric) appears in the menu.

Note: Alpha fields just flip between equal and not equal.

3. Select the operator that you want from the menu. You can rest the mouse on an operator to display a tooltip describing the operation.

How Slicers Cascade Together

Slicers cascade within a hierarchy (cube or dimension builder-based), rather than in the order of user interaction, eliminating potential performance issues.

Procedure: How to Cascade Slicers

Controls with fields from the same hierarchy are cascaded together.

1. Create a report.
2. Create multiple slicer groups, as described in [How to Create a Slicer](#) on page 670.

Note: You can select multiple values by holding down the CTRL key while selecting items in the drop-down list box. In cases where multiple items are selected, the label for the drop-down list is set to Multiple. You can click the drop-down list to review the selections.

Once you click *OK*, the new control appears in the drop-down menu.

The 1 prior to the decimal point indicates that this is the first cascade you have interacted with. The 1 after the decimal point indicates that it is the first control in that cascade.

3. Filter the report using the controls in the slicer groups that you created.

Note: The order of a cascade is dynamic. For example, if you selected Camcorder from Product Subcategory first, then the Product Category control would have a value of Video Production.

Changing a Field Format

You can change the format of any field except a Sort field. You can change the format of measure fields, as well as any calculated fields.

The following procedures show how to change a field format from the Field Format Options dialog box. For more information on how to use the Format group options, see [Field Tab](#) on page 80.

Procedure: How to Assign an Alphanumeric Format

1. With a report open in Live Preview, right-click a virtual field or column measure field, and select *Edit Format*.

The Field Format Options dialog box opens.

2. From the Field type list, select *Alphanumeric*.
3. To assign a different length, specify a number between 1 and 4095 in the *Total length* field. The default value is 20.
4. Click *OK* to close the Field Format Options dialog box and return to the canvas.

The new format appears in the previously selected column.

Procedure: How to Assign a Numeric Format

1. With a report open in Live Preview, right-click a virtual or column measure field, and click *Edit Format*.

The Field Format Options dialog box opens.

2. From the Field type list, click one of the following options:

☐ *Floating Point* (default length 7.2)

☐ *Integer* (default length 5)

☐ *Decimal* (default length 12.2)

☐ *Packed* (default length 12.2)

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

- ☐ To assign a different length, specify it in the *Field Length* field for format types as follows: 1-9 for Floating Point, 1-11 for Integer, 1-20 for Decimal, and 1-33 for Packed.
 - ☐ To assign a different number of decimal places for Floating Point, Decimal, or Packed, specify the number in the *Decimals* field.
3. Click *OK* to close the Field Format Options dialog box and return to the canvas.
The new format appears in the previously selected column.

Procedure: How to Assign a Date Format

1. With a report open in Live Preview, right-click a virtual or column measure field, and select *Edit Format*.
The Field Format Options dialog box opens.
2. From the Field type list, click *Date*.
3. To assign a different date display format, select a date format from the *Date Format* list.
The default date display format is MDY.
4. Click *OK* to close the Field Format Options dialog box and return to the canvas.
The new format appears in the previously selected column.

Procedure: How to Assign a Date-Time Format

1. With a report open in Live Preview, right-click a virtual or column measure field, and select *Edit Format*.
The Field Format Options dialog box opens.
2. From the Field type list, click *Date-Time*.
3. To assign a different date-time format, select the date format you want to use from the *Date Format* list. Expand Year First, Month First, or Day First to see the available options.
4. Select a time format (if applicable) from the Time Format list.

If you select the *Time Only* check box then you can only set a time format. Expand Hour First, Minute First, or Seconds First to see the available options.

Note: These expansion options are only visible when you select Time Only.

5. Click *OK* to close the Field Format Options dialog box and return to the canvas.

The new format appears in the previously selected column.

Procedure: How to Add a Percent Sign to a Numeric Field

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

1. With a report open in Live Preview, right-click a virtual or column measure field, and select *Edit Format*.

The Field Format Options dialog box opens.

2. From the Field type list, click a numeric value format (Floating point, Integer, Decimal, or Packed), and then select the *Percent (%)* check box.
3. Click *OK* to close the Field Format Options dialog box and return to the canvas.

The new format appears in the previously selected column.

Dynamic Grouping

Dynamic grouping allows you to create groups of elements based on the field data type that you select. For example, in the wf_retail_lite database, there are a number of brands of televisions. Using the dynamic grouping functionality, you can create groups based on the popularity of a particular brand. The first group might include top sellers such as LG and Sony. The second group might contain the remaining brands (Panasonic, GPX, Supersonic, Tivax, and Audiovox). This would allow you to group top sellers into one group, and the remaining brands into another group.

Note: The Create Group option is only available for dimension fields of non-numeric format or attribute.

You can also specify multiple, unique groupings in the same session. For example, you might want to group the data to indicate groups or products, or specific regions.

Note: If you want to exclude a specific data element from your analysis, you can use the filter functionality.

The grouping that you specify is applied and this new group then replaces the original field that you selected in the Query pane. The name that you specify when creating the group is reflected in the Query pane.

Note: You can edit the group once it has been created by right-clicking on the higher-level field and clicking *Edit Group*.

Procedure: How to Create a Dynamic Group

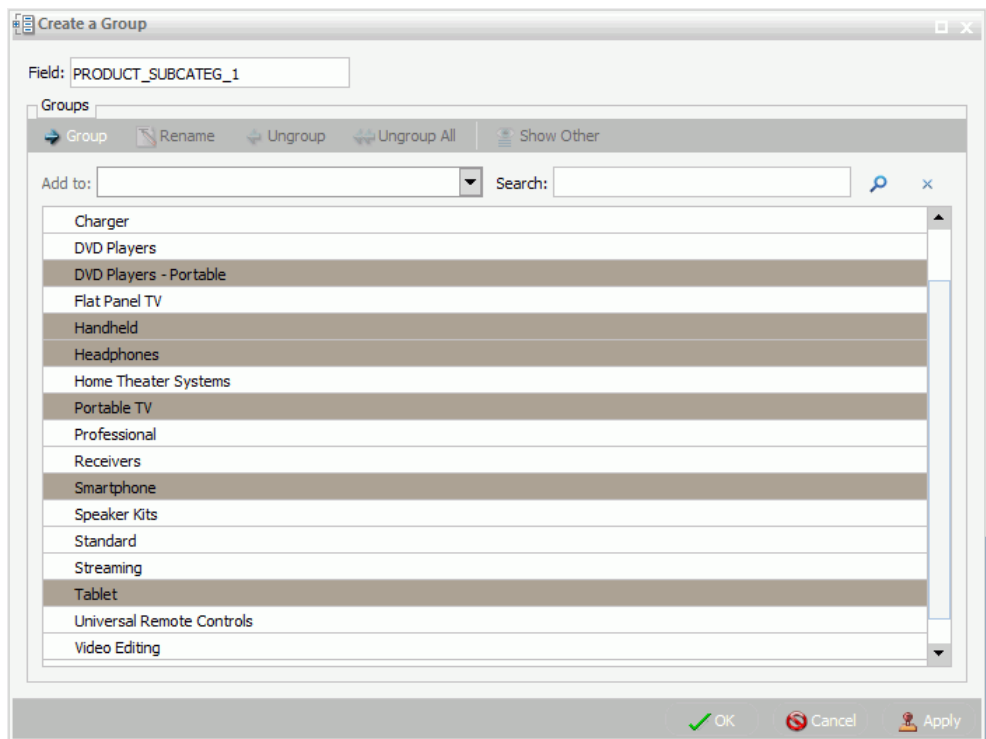
1. From the Data pane, add one or more data fields to your report or chart.
2. In the Query pane, right-click the data field for which you want to apply dynamic grouping.

Note: The Create Group option is also available in the Data pane shortcut menu.

3. Click *Create Group*.

The Create a Group dialog box displays.

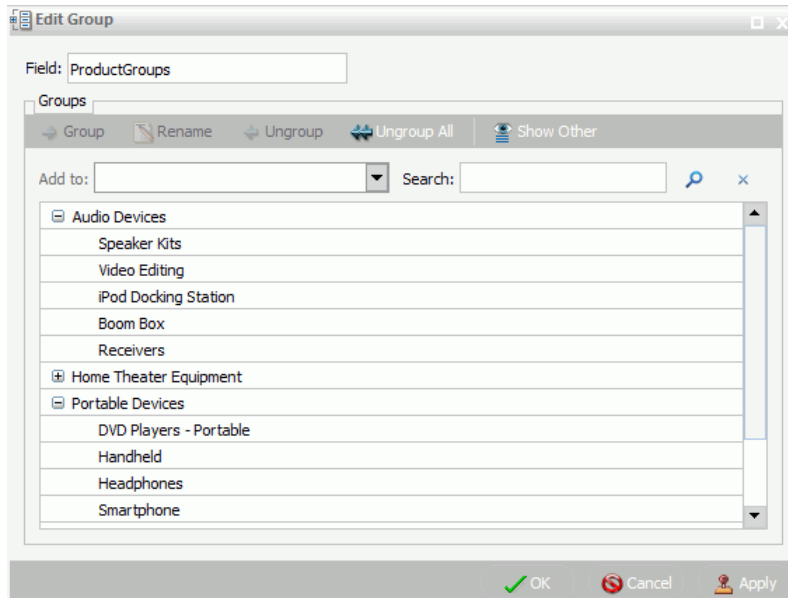
4. In the Field text box, optionally type a name for the new group.
5. Select the data values that you want to group. Use CTRL + click to select more than one value, as shown in the following image.



6. Click *Group*.

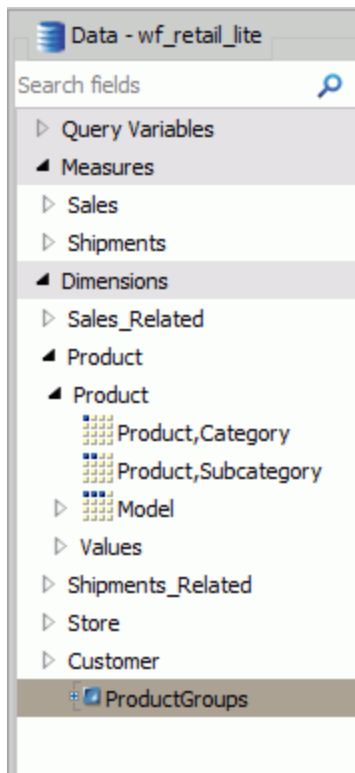
Note: To ungroup values, click a group and then click *Ungroup*.

7. Optionally, create additional groups, as shown in the following image.



8. Click OK.

Your grouped data will display in the Data pane when your report, chart, or visualization refreshes, as shown in the following image.



Using Content

After you have created and customized your content, learn how to distribute the content and create schedules.

In this chapter:

- ☐ [Creating Distribution Lists](#)
 - ☐ [Maintaining Distribution Lists](#)
 - ☐ [Creating Schedules](#)
 - ☐ [Maintaining Schedules](#)
 - ☐ [Tracking Schedules](#)
 - ☐ [Using Favorites](#)
 - ☐ [Publishing Content](#)
 - ☐ [Using Visualizations at Run-Time](#)
 - ☐ [Using the Deferred Report Status Interface](#)
-

Creating Distribution Lists

A Distribution List is an easy way to distribute content to multiple recipients by specifying the name of a list stored in the Repository that contains the individual recipients rather than entering each recipient separately into a schedule. A Distribution List can be made available to other users by sharing it or changing its ownership to be managed or published.

Creating a Distribution List

You create distribution list to distribute content to multiple recipients at one time. If you have an LDAP data source, you can access Email information that is stored in it by configuring the LDAP Email Setting in the ReportCaster Console Configuration tool. Once configured, you can select email addresses from within the email address book of your company, eliminating the need to type them in manually. For more information on the configuration of this functionality, see [LDAP Setting](#) on page 170.

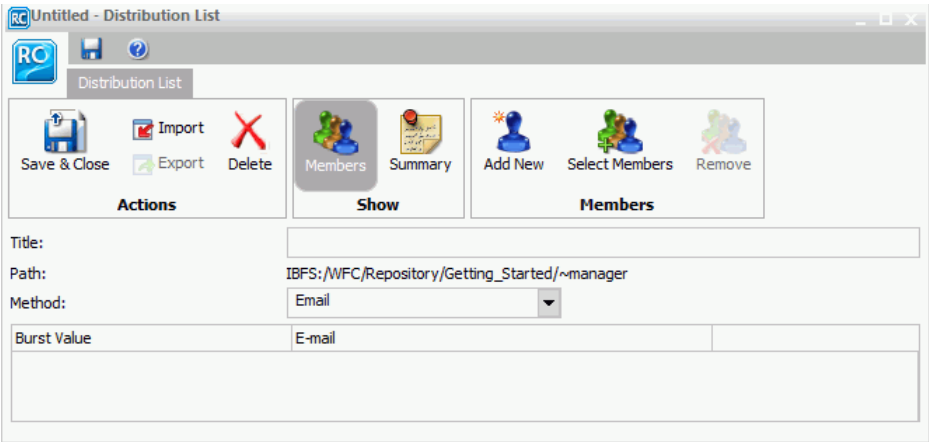
For more information on using this feature, see [How to Retrieve Email Addresses from an LDAP Data Source](#) on page 681.

Before creating a Distribution List, understand what groups or individual users require access to it to determine the folder in which to create it.

Procedure: How to Create a Distribution List

1. Right-click a Repository folder and select *New, Distribution List*.

The Distribution List window opens, as shown in the following image.



2. In the *Title* box, type a descriptive name for the Distribution List.
3. From the Method drop-down list, select the distribution method for the Distribution List. Email is the default distribution method.

☐ **If you select Email**, you must provide a list of email addresses and can optionally burst values associated with an address. For details on entering burst values, see [Bursting a Report](#) on page 686.

In the *Address* box, specify the email addresses of the recipients (for example, chuck_hill@ibi.com for an individual user or #sales@ibi.com for a email server list that contains multiple email addresses). Be careful typing this information because there is no edit checking. The maximum number of email addresses you can specify in a Distribution List is 9999. You can specify a maximum of 800 characters within a single Address line.

You can specify multiple email addresses within a single Address field. For more information, see [Specifying Multiple Email Addresses](#) on page 690.

If the Email Delivery, Restrict Email Domains option is set to yes in the Server Configuration tool, then only those email domains (the portion of the email address following the at (@) symbol) listed in Allowed Email Domains (also in the Server Configuration tool) are valid email recipients.

- ❑ **If you select Print**, you must specify the printers that will receive the distribution and, optionally, burst values associated with the printer. For details on entering burst values, see [Bursting a Report](#) on page 686.

In the Printer input field, specify the printer using the following format:

queue@printserver

where:

queue

Is the name of the printer queue.

printserver

Is the host name or IP address of the printer.

Although ReportCaster supports specifying only the print server (host name or IP address), we recommend that you specify both the print queue and print server. (ReportCaster differentiates between the printer queue and the printer server by detecting the presence of the '@' separator.)

4. If you are finished creating a Distribution List, click *Save & Close*.

Procedure: How to Retrieve Email Addresses from an LDAP Data Source

Note: In order to use this procedure, LDAP must be configured. For more information, see [Using the ReportCaster Console](#) on page 137.

1. From the Distribution List toolbar, click *Add New*.

The Add New Member dialog box displays, as shown in the following image.

RC Add New Member

Burst value:

Pattern:

(* = any thing, ? = any character, = escape for literals: *?)

E-mail address:

2. Click the *Search* button adjacent to the email address field.

The Find dialog box displays, as shown in the following image.

Find

Search Criteria:

Last Name Equals

(Maximum Retrieval: 300) Search

Name	Value

To ->

OK Cancel

3. Select the criteria (Last Name, First Name, or Email) and the operator (Equals, Starts with, Ends with, or Contains). Enter the search string (for example, M* to locate all records beginning with that letter).
4. Click *Search*.

ReportCaster retrieves and displays the specified names from the LDAP data source.

Note: You can select one record at a time or use the CTRL key to select multiple records simultaneously.

5. Click the *To* button to add the selected email address or addresses to your selection.
6. Click *OK* to save them in the Distribution List.
7. On the Add New Member dialog box, click *OK*.

The email recipients are then listed in the Email column on the Distribution List dialog box.

Maintaining Distribution Lists

A Distribution List is an easy way to distribute content to multiple recipients by specifying the name of a list stored in the Repository that contains the individual recipients rather than entering each recipient separately into a schedule. A Distribution List can be made available to other users by sharing it or changing its ownership to be managed or published.

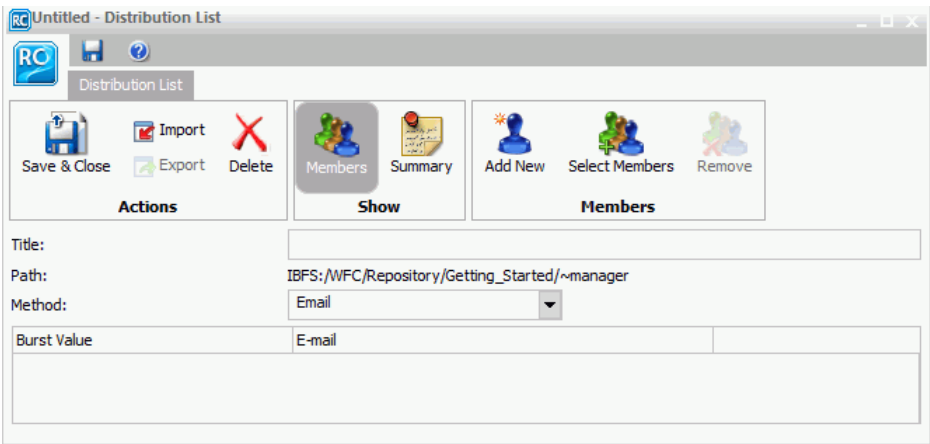
Editing and Deleting a Distribution List

If you are authorized to access the Distribution List tool, you can view and edit the Distribution Lists of which you are the owner. If the Distribution List is owned by a group or is published, you have to be authorized to edit it.

Procedure: How to Edit a Distribution List

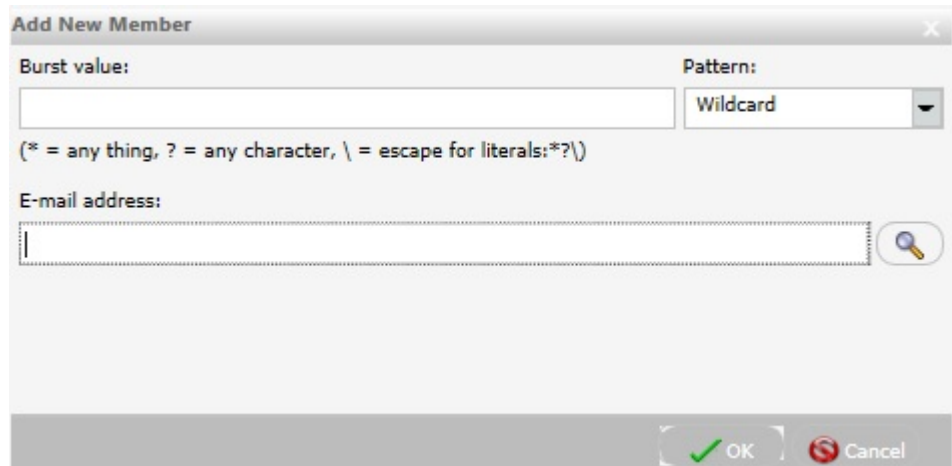
1. From the Resources tree, select the Distribution List you want to edit, right-click and click *Edit*. Optionally, you can double-click the distribution list to open it.

A window opens displaying the properties of the selected Distribution List, as shown in the following image.



2. From this window, you can perform the following:
 - ☐ Change the name of the Distribution List by typing a new name in the *Title* field.
 - ☐ Change the Distribution Method.

- ❑ Click *Add New* or double-click within the area below the Burst Value column to add a New Member to the Distribution List below Burst Value and Email. The Add New Member dialog box appears, as shown in the following image.



The image shows a dialog box titled "Add New Member". It has two input fields at the top: "Burst value:" and "Pattern:". The "Pattern:" field has a dropdown menu currently showing "Wildcard". Below these fields is a text label: "(* = any thing, ? = any character, \ = escape for literals:*?\\)". Below that is a large text input field labeled "E-mail address:". To the right of the email field is a magnifying glass icon. At the bottom of the dialog box are two buttons: "OK" with a green checkmark icon and "Cancel" with a red X icon.

- ❑ Delete a Distribution List entry by selecting the item to be deleted, then click *Delete*.
3. When you have completed your changes, click *Save & Close*.
To exit the editing window without making changes, click *Close*.

Accessing Distribution Lists

Sharing your private Distribution Lists allows you to permit groups and users to see its content and use it in schedules while you retain ownership. Authorized users can share their private Distribution Lists with groups and users with whom they are permitted to share. The Distribution List can be accessed by users authorized to access the folder in which the shared Distribution Lists are located.

Authorized users can publish and unpublish Distribution Lists to make them available to users authorized to access the folder in which they are located. Publishing requires the folder the item is located in to be published in order to publish an item within the folder. If the folder is subsequently unpublished, all items in that folder are unpublished.

Procedure: How to Share a Distribution List

From the Resources tree, if you are authorized to share your private content, you can share a Distribution List with users authorized to access the folder in which the Distribution List is located by right-clicking the folder or Distribution List in the Resources tree and selecting *Share*.

Note: In order to share a Distribution List in the BUE, it must reside in the My Content folder. If you have created the Distribution List in a different folder, simply drag the Distribution List into the My Content folder to enable the sharing options that are available to you.

If you are authorized for advanced sharing, you can share your private content with specific groups and users with whom you are authorized to share by performing the following steps.

1. In the Resources tree, select the Distribution List you want to share.
2. Right-click the Distribution List and select *Share with*.

The Set Sharing dialog box displays, as shown in the following image.

3. Use the left and right arrow buttons to select which Groups will have access to the Distribution List.
4. Use the left and right arrow buttons to select which Users will have access to the Distribution List.
5. Click *OK*.

Procedure: How to Publish Distribution Lists

The following steps allow you to publish a Distribution List or folder.

Note: A folder must be published in order to publish items within it. Published items or folders display in color. Those that are unpublished display in black and white.

1. From the Resources tree or ReportCaster Explorer, select the Distribution List or folder that you want to publish.
2. Right-click the folder or Distribution List and select *Publish*.

Procedure: How to Unpublish Distribution Lists

From the Resources tree or ReportCaster Explorer, you can unpublish a Distribution List or folder.

1. From the Resources tree or ReportCaster Explorer, select the Distribution List or folder that you want to unpublish.
2. Right-click the published folder or item that you want to unpublish and select *Unpublish*.

Bursting a Report

Instead of distributing an entire report from a scheduled report procedure (FEX), you can use the ReportCaster burst feature to break the report into sections to be distributed separately to the same or different destinations. Bursting enables you to target relevant sections of a report to individual users. Each report section is saved to a separate file.

If you are distributing a burst tabular report, the burst value is determined by the first BY field. If you are distributing a burst graph report, the burst value is determined by the second BY field. The burst value is automatically determined by the internal matrix, which is a memory area that stores each database field value and calculates values referenced by the TABLE or GRAPH request.

You can send several report sections to one recipient by specifying the destination of that recipient (email addresses and files or printers) for each section you want to send. You can also send several report sections to one destination. The burst values you specify in the Distribution List must exist in the data source you are reporting against.

Note: If you want to burst a report, you must enable the bursting option within the Task for a schedule. The burst values specified in the Burst Value column in the Distribution List are ignored unless the Task specifies to burst the report.

Example: Specifying Burst Values in a Distribution List

You can specify sort field burst values and destinations (email addresses or printers) when creating or editing a Distribution List. The following image shows burst values and the destination email addresses specified in the Distribution List window.

Burst Value	E-mail
South Sales	Tom_Gregory@ibi.com
Midwest Sales	Tom_Gregory@ibi.com
Northeast Sales	Chuck_Hill@ibi.com

Using the primary sort field values (Northeast Sales, South Sales, and Midwest Sales), the email address of each representative is associated with the relevant sales report data. Since Chuck Hill needs only the data for the Northeast branch, the sort value Northeast is listed in the Burst Value column and is associated with his email address in the E-mail column.

However, Tom Gregory works in both the Midwest and South regions. Since he requires data for both regions, his email address is listed in the E-mail column twice, next to a Burst Value column entry for each region.

Note: You can click on a column heading to sort the data in that column.

Tip: You can specify multiple email addresses on a single Address line. For details, see [Specifying Multiple Email Addresses](#) on page 690.

Bursting Guidelines and Limitations

This section provides detailed information to assist you in defining burst values.

When a schedule task specifies to burst a report procedure (FEX), all data values generated for each burst section are returned to the Distribution Server.

- ❑ For the Repository each burst section is distributed to the repository. The owner of the schedule must have write access to the repository folder specified in the schedule when the scheduled job runs for the report output to be successfully distributed. Access to the report sections is controlled by the repository security rules that are created when the report is distributed.
- ❑ For the email and printer distribution methods, specific burst sections are distributed based on the burst values specified when creating the Distribution List or Single Address used by the schedule.

The following are guidelines and limitations that apply to the burst feature:

- ❑ **Case.** Burst values are case-sensitive.
- ❑ **Keywords.** Burst values can contain the following keywords:
 - ❑ **Wildcard Characters.** Use an asterisk (*) and a question mark (?) as wildcards to represent characters at the beginning, end, or middle of the burst values. The asterisk represents one or more characters, while the question mark represents any single character. Precede each instance of a burst value using a wildcard with the wildcard keyword enclosed in brackets followed by a colon, [wildcard];, as shown in the following examples.

[wildcard]:abc* = all values that start with 'abc'.

[wildcard]:a?c = all three-character values that start with 'a' and end with 'c'.

[wildcard]:a?c* = all values that start with 'a' and have a 'c' as the third character.
 - ❑ **Java Regular Expressions.** Use to identify strings of text. Precede each instance of a burst value using a Java regular expression with the regular expression keyword enclosed in brackets followed by a colon, [regexp];, as shown in the following examples.

[regex]:[bcr]at = values that are bat, cat, or rat.

[regex]:[^bcr]at = any value that is not bat, cat, or rat.

- ☐ **Default Distribution.** You can provide a default destination for burst values that are not specified in the Distribution List. To do this, enter the following in the burst value column of the Distribution List.

[elsesend] = reports for burst values not contained in the Distribution List will be sent to the named recipient.

- ☐ **'%BURST' Syntax.** You can include a burst value in the name of a distributed file by using the '%BURST' syntax in the name. The use of '%BURST' is not supported in a zip file name when the Packet email setting is Yes.

The following are example entries in an email Distribution List that illustrate the use of the wildcard and default distribution keywords in burst values.

Burst Value	Address
[wildcard]:*an*	sml@company.com
England	ray@company.com
[elsesend]:	jt@company.com

Using a scenario where the report output from a scheduled report procedure (FEX) is burst on the Country field that contains values of Germany, USA, France, Canada, Italy, Chile, England, and Japan, then:

- ☐ Report information for Germany, France, Canada, England, and Japan will be delivered to sml@company.com.
- ☐ Report information for England will be delivered to ray@company.com.
- ☐ Report information for USA, Italy, and Chile will be delivered to jt@company.com.
- ☐ **Formats.** All formats support bursting except XML and EXCEL. Each burst section of the report output from a scheduled report procedure (FEX) will be named *burstvalue_filename.format* (for example, Northeast_Sales.pdf).
- ☐ **ACROSS command.** This command is not evaluated as a primary sort field. To burst report output from a scheduled report procedure (FEX), you must also include a BY field. Bursting occurs on the BY field.

- ❑ **TABLEF.** No internal sort processing is performed. The specification of a BY field requires that the data already be sorted in the data source.
- ❑ **ON TABLE SUBHEAD/ON TABLE SUBFOOT.** Creates a SUBHEAD for only the first page of the report output from a scheduled report procedure (FEX), and a SUBFOOT for only the last page of the report output from a scheduled report procedure (FEX). When bursting report output from a scheduled report procedure (FEX), the SUBHEAD and SUBFOOT should occur for each sort break. Therefore, specify the primary sort field in place of TABLE in the ON command. For example:

```
ON primarysortfield SUBHEAD
```

- ❑ **A *n* V field types.** Bursting is not supported on a field with the AnV (where *n* is an integer value) field type.

Specifying Multiple Email Addresses

When creating a schedule or Distribution List, you can specify multiple email addresses within a single field, row, or record.

When creating a schedule or Distribution List, you can separate each email address with a comma (,) or a semicolon (;).

The multiple email addresses will appear in the To line of a single email when the scheduled output is distributed.

Note:

- ❑ To distribute separate emails for each address, specify the email addresses on separate lines within the Distribution List.
- ❑ If the Email Delivery, Restrict Email Domains option is set to yes in the Server Configuration tool, then only those email domains (the portion of the email address following the at (@) symbol) listed in Allowed Email Domains (also in the Server Configuration tool) are valid email recipients.

Example: Specifying Multiple Burst Email Addresses

If you are using the default configuration (Packet Email = YES), one email is distributed for multiple burst values specified for the same email address. The email address values specified on each row are treated as a string that is a key. If there are multiple rows with the same address value (key), one email is distributed with all the burst values. For example, consider the following Distribution List:

Burst Value Address

A	user1@abcd.com;user2@abcd.com
B	user1@abcd.com
C	user1@abcd.com

In this example, user1@abcd.com receives two emails when the scheduled output is distributed. In the first email, user1@abcd.com; user2@abcd.com appears in the email To line and one attachment is distributed for burst value A. In the second email, user1@abcd.com appears in the To line and two attachments are distributed, one for burst value B and one for burst value C.

If you are using the configuration that specifies to distribute a single email for each row (Packet Email = NO), then the following behavior occurs for our example. Three separate emails are distributed. In the first email, user1@abcd.com; user2@abcd.com appears on the To line and one attachment is distributed for burst value A. The second email is sent to user1@abcd.com with one attachment for burst value B. The third email is sent to user1@abcd.com with one attachment for burst value C.

If a schedule has Packet Email = BURST, then for each burst value the output for each task is combined and distributed. In our example, three separate emails are distributed. In the first email, user1@abcd.com; user2@abcd.com appears on the To line and all output from the tasks for burst value A are distributed. The second email is sent to user1@abcd.com with all output from all tasks for burst value B. The third email is sent to user1@abcd.com with all output from all tasks for burst value C.

Another consideration is when using the default configuration (Packet Email = YES) and the same burst value is specified multiple times for the same Address (key) value. For example, consider the following Distribution List:

Burst Value Address

A	user1@abcd.com;user2@abcd.com
B	user1@abcd.com
B	user1@abcd.com

In this Distribution List, only two emails are distributed for user1@abcd.com. In the first email, user1@abcd.com; user2@abcd.com appears in the To line and the attachment is for burst value A. In the second email, user1@abcd.com appears in the To line and the attachment is for burst value B. The third row in the Distribution List is ignored since it contains the same key and the same burst value, B, as the second row.

As a best practice, be sure to review your distribution information to make sure you have not duplicated the same burst and address value pairs.

Example: Specifying Multiple Non-Burst Email Addresses

Consider the following sample Distribution List, which does not contain burst values:

Address

`user1@abcd.com;user2@abcd.com`

`user1@abcd.com`

`user2@abcd.com`

`user3@abcd.com`

`user1@abcd.com;user2@abcd.com;user3@abcd.com`

In this Distribution List, an email is distributed for each address line regardless of whether Packet Email is set to YES or NO. This is because each address value is unique. For the first email, `user1@abcd.com; user2@abcd.com` appears in the To line and the attachment is for the full report output from a scheduled report procedure (FEX). The second email is distributed to `user1@abcd.com`, and so on.

If one of the address lines is repeated in the Distribution List (for example, if `user3@abcd.com` is added as the sixth line in the example Distribution List), the behavior would work as follows. If Packet Email = YES, only one email is distributed for `user3@abcd.com`. However, if Packet Email = NO, two separate emails are distributed to `user3@abcd.com`.

Creating Schedules

A schedule allows you to specify when to run a report, the format in which to create the output, and how it will be distributed.

About the Basic Scheduling Tool

The Basic Scheduling tool provides the ability to create a schedule for a procedure (FEX) from the Resources tree when a report procedure (FEX) is selected from the BUE portal. The WebFOCUS Client security authorization model controls whether users are authorized to access the application content stored in the Repository and the Scheduling tools.

To create a new schedule for a report procedure (FEX), navigate to the Resources tree and expand a folder to display your report procedures (FEX). Right-click the report procedure (FEX) you want to schedule, select *Schedule*, and then select the method to distribute the report procedure (FEX). You can distribute the report procedure (FEX) by Email, Printer, or Repository, as shown in the following image.

The following image shows the Basic Scheduling tool that consists of a Quick Access Toolbar and the ribbon from which you access the schedule options.

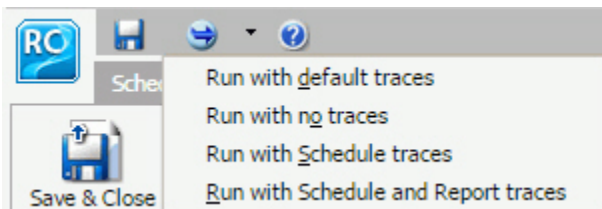
Basic Scheduling Tool Quick Access Toolbar

The Quick Access Toolbar, located at the top of the Basic Scheduling tool, is always visible no matter which options are selected. It provides access to the most commonly used functions. From the Quick Access Toolbar, you can select the ReportCaster button to access the New Distribution List, Save, Save As, Delete, and Close options. You can also access the Save, Run, and Help options from the toolbar.

Run options are available for selection from the Run drop-down list.

Note: The schedule must be saved for Run options to be available or enabled.

The Run options you can select include: Run with default traces, Run with no traces, Run with Schedule traces, and Run with Schedule and Report traces, as shown in the following image.



Note:

☐ If you have the Session Traces privilege, you will have the Run with Traces options in the schedule tools. If you are not authorized to run with traces, these options will not display.

☐ Online help is available by clicking the online help icon



Basic Scheduling Tool Ribbon

The Basic Scheduling tool ribbon partitions the scheduling options into the following categories:

☐ **Actions**

The Actions portion of the ribbon is shown in the following image.

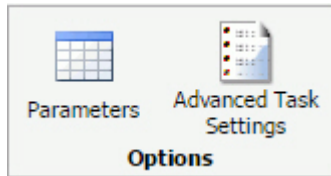


- ☐ **Save & Close.** Saves and closes the schedule.
- ☐ **Delete.** Deletes the schedule and closes the Scheduling tool.
- ☐ **Show**

The Show portion of the ribbon is shown in the following image.

- ☐ **Properties.** Provides a Title, Path where the schedule will be created or was opened from, Summary, Job Priority Level, No Report to Distribute, and other settings for the schedule. The *Delete this schedule if it is not scheduled to run again* check box specifies to delete the schedule if, after it is run, it is not scheduled to run again. The *Enabled (Scheduled job runs at specified time)* check box specifies to run the schedule as specified by the Recurrence settings within the schedule. The *No Report to Distribute* drop-down list box specifies whether to process a No Report to Distribute result as an Error or a Warning.
- ☐ **Recurrence.** Provides run-time intervals for distribution and repeat options.
- ☐ **Distribution.** Provides options to specify the recipients or location to which the report will be distributed.
- ☐ **Task.** Provides information on the report procedure (FEX) that is being scheduled.
- ☐ **Notification.** Provides the options to set up notification of the schedule status.
- ☐ **Log Reports.** Shows the Number of Jobs and the Log Report for individual jobs.
- ☐ **Options**

The Options portion of the ribbon is shown in the following image.



- ☐ **Parameters.** Specifies values for parameters that are required at run time by the report procedure (FEX) being scheduled.
- ☐ **Advanced Task Settings.** Allows you to input the report language and additional FOC Errors to be processed as warnings.

Creating a Schedule in the Basic Scheduling Tool

This section provides the overall procedure to create a new schedule for a report procedure (FEX). Some steps in the procedure contain details on the associated options, while other steps direct you to a separate section that contains detailed descriptions of the options and additional information, such as tips in making a selection.

To save a schedule, the required information in the Properties, Recurrence, Distribution and Notification tabs must be provided. If required schedule information is missing when you save the schedule, a message will display informing you of the schedule information that needs to be entered.

Procedure: How to Create a Schedule

1. Open the Basic Scheduling tool, as described earlier in this section. For more information, see [About the Basic Scheduling Tool](#) on page 692.
2. In the Properties tab, you can edit the name for the schedule in the *Title* box.
This is a required field and a default name is provided.
3. Type a descriptive summary in the *Summary* box.
Note: This is an optional field.
4. Select a Job Priority Level.
Normal - 3 Job Priority Level is the default.
5. Check the *Delete this schedule if it is not scheduled to run again* check box if you do not want this schedule to be stored in the Repository if it will not run again as specified in the Schedule recurrence settings.
6. Leave the *Enabled (Scheduled job runs at specified time)* check box checked if you want scheduled jobs to run as specified in the recurrence settings.

7. Select the Recurrence tab and make the following selections.
 - a. From the Settings radio button list, select a time interval that the schedule will use to run the report procedure (FEX).

You can set the interval to Run Once, Minutes, Hourly, Daily, Weekly, Monthly, Yearly, or at Custom intervals.
 - b. From the Start Schedule options, select the date (from the drop-down calendar) and time you want the schedule to begin running.

Note: To change the time setting, select either the hour or minutes and use the arrows to increase or decrease the value.
 - c. If applicable to the Run Interval selection, from the End Schedule options, select the date and time you want the schedule to stop running.
 - d. If applicable to the Run Interval selection, from the Advanced settings, click the *Repeat schedule every:* check box to enable custom intervals.

Note: This option is disabled for the Run Once, Minutes, and Hourly settings options.
8. Select the *Task* tab. The Path, Procedure, Server Name, and Save Report As fields are populated according to the report procedure (FEX) you selected to schedule. For a description of the Task tab, see [About Tasks in the Basic Scheduling Tool](#) on page 697.
9. Select the *Notification* tab and specify whether or not you want to send a notification when the schedule runs and under what conditions to send it. The notification options are:
 - ☐ **Never.** ReportCaster will not send a notification of the schedule status under any circumstances. This is the default value.
 - ☐ **Always.** Send a notification each time the schedule runs.
 - ☐ **On Error.** Only send a notification when there is an error running the schedule.

For more information, see [Notification Options in the Basic Scheduling Tool](#) on page 718.
10. In the Log Reports tab, you can view log and manage log reports for the schedule.
11. To save the schedule, click *Save & Close* on the ribbon.

Note: You can optionally click *Save* on the toolbar to remain in the scheduling tool. The Save dialog box appears.
12. Select the folder that will contain the schedule.

Note:

 - ☐ You must be authorized to create content in this folder.

- ❑ The default location for saving a schedule is dependent upon your privileges. If you can create content in the folder where the scheduling tool is launched, the Save dialog will be positioned in that folder. If you are not allowed to create content in that folder, the Save dialog will be positioned in the My Content folder located under that folder. If a My Content folder is not available, the Save dialog will be positioned in the first writable folder found.

13. Enter a name for the schedule and click Save.

About Tasks in the Basic Scheduling Tool

When you access the Basic Scheduling tool, the information for the Task tab options is predefined with the selected report procedure (FEX) information. You can complete the available task options, such as specifying parameter values, as well as select whether or not to burst the report.

The following image shows an example of a predefined Task tab in the Basic Scheduling tool.

Note: You can use parameters (ampers variables) when specifying a value in the Save Report As field of a schedule. These parameters can be system variables, such as &YYMD or any ampers variable for which a value is returned by the Reporting Server when the scheduled procedure is executed. To use the dot file extension separator after a variable, terminate the variable with the '|' character, for example, &YYMD|.htm. Similarly, to use the ampersand character itself, follow the ampersand with the '|' character (for example, Smith&|Jones). If a value for a parameter specified in a schedule is not returned by the Reporting Server when the procedure executes, the schedule will fail with a "No report to distribute" error. If the scheduled task is burst, you can also use the '%BURST' symbol to include the burst value in the Save Report As field.

Task Options in the Basic Scheduling Tool

The options available when you select the Task tab are:

- ❑ **Path.** Shows the report path in the Repository or on the Reporting Server.
- ❑ **Procedure.** The name of the procedure you are scheduling.
- ❑ **Server Name.** The Reporting Server to which the report procedure (FEX) will be submitted.
- ❑ **Execution ID.** The user ID running the report procedure (FEX) on the Reporting Server. Click on the *Password* button to the right of this field to set the password.
- ❑ **Password.** Enter a password for the Execution ID.
- ❑ **Alert.** Specifies how to reactivate the alert or to deactivate the alert condition when it is triggered. Click *Alert* to specify the alert options.

Select one of the following options from the Alert Options dialog box.

- ☐ **Automatically Reset.** After the alert is triggered, reactivate the alert when the condition is no longer true. The system will keep checking the condition after the alert has been triggered. As soon as the condition is no longer true, it will reactivate the alert. This is the default value.
- ☐ **Continue After Alert.** After the alert is triggered, reactivate the alert immediately.
- ☐ **Deactivate Schedule After Alert.** Deactivate the schedule after the alert is triggered.
- ☐ **Delay.** Restart the alert after a specified period. You can specify to restart the alert after a maximum of 99 hour(s), day(s), week(s), month(s), or year(s).

Caution: Be sure that you set the schedule interval to a time period greater than the time it takes to run the scheduled procedure. When the schedule interval is less than the time it takes to run the procedure and the Delay option is selected, an alert schedule will distribute (based on the Distribution that you selected) more frequently than expected. For email distribution, this can result in unnecessary distribution of emails which can impact the business or operational goals of the Alert report distribution.

- ☐ **Enabled.** Selected by default, this option allows you to disable a task.

Report Properties

- ☐ **Burst Report.** If you want to burst the report, select the *Burst Report* check box. The burst feature enables you to instruct the Reporting Server to create the report in sections so that they can be distributed separately.
- ☐ **Override the Format Specified in the Procedure.** Select this check box to indicate a format other than the one specified in the procedure.
- ☐ **Save Report As.** Allows you to specify a different name for the report, which defaults to the name of the report that you selected to schedule.

Specifying Parameter Values

You can customize a report and control its execution with parameters. When scheduling a report procedure (FEX), the schedule task Parameters section allows you to supply a value for parameters referenced in the report procedure or add a parameter to the schedule by creating a new parameter and specifying the parameter name and value.

Procedure: How to Specify Parameter Values

If the scheduled procedure contains parameters for which values must be supplied at run time, these parameters are displayed in the parameters section of the scheduling tool.

In the Basic Scheduling tool, click on the *Parameters* tab in the ribbon to open the Task Parameters dialog box. The Task Parameters dialog box is shown in the following image.

Name	Description	Display Value	Value	Use Default

Up Down New Delete Refresh

Parameter Properties

Type:

Name:

Description:

Value:

Display Value:

Default Value:

☒ Always Use Default Value Specified in the Procedure

Data Type: Minimum: Maximum:

OK Cancel

Parameters may have default values and may have values that can be selected from a static or dynamic list. For more information about selecting parameters in the Scheduling Task Parameters dialog box, see the examples provided in this chapter.

Reference: Considerations When Specifying Parameter Values

The following are considerations when specifying parameter values for a procedure:

- ☐ The maximum number of characters for each individual parameter value is 3200. You can store multiple values for a single parameter. Multiple values for a parameter are stored as one entry, which must not exceed the 3200 maximum character limit.
- ☐ ReportCaster displays the description for the parameter when it is specified in the procedure. Otherwise, ReportCaster displays the parameter name.
- ☐ ReportCaster displays default variable values, as well as static or dynamic single-select and multiselect lists.

Note:

- ☐ The No Selection option is displayed for dynamic multi-select lists. When selected, this option does not perform any data selection test on that field.
- ☐ ReportCaster does not support using the -HTMLFORM command to create a dynamic selectable list of parameter values.
- ☐ ReportCaster displays global variables that are used in FILTERS defined in Master Files and referenced by a procedure. For example, if a Master File contains

```
FILENAME=CAR,SUFFIX=FOC
VARIABLE NAME=&&COUNTRY1, USAGE=A10, DEFAULT=ENGLAND,$
FILTER FILTER1=COUNTRY EQ '&&COUNTRY1'; $
```

and the procedure being scheduled contains

```
WHERE FILTER1
```

then ReportCaster displays COUNTRY1 in the Parameters window.

- ☐ ReportCaster will not prompt for variables with defaults set by the -DEFAULTH command. The purpose of the -DEFAULTH command is to assign a default value to amper variables and not be dynamically prompted for that variable.
- ☐ ReportCaster does not prompt for the value of its internal variables that are set by the Distribution Server at schedule execution time. For example, &DSTOWNER is the user ID of the schedule owner. If this parameter is referenced in a scheduled procedure, the value will be available because the Distribution Server sets the value to the owner of the schedule at schedule execution time. However, the scheduling tool will not prompt for this value in the Task Parameters dialog box.

- ❑ When specifying parameters with special characters (for example, %, &, |):
 - ❑ If you are specifying the entire WHERE condition as the parameter value, you must enclose the value within two single quotation marks rather than a double quotation mark. For example, "WHERE CAR NOT LIKE MOTO%".
 - ❑ If you are only specifying a value as the parameter value, you do not need to enclose the parameter value within quotation marks. For example, O&DINFO.
- ❑ Once a parameter is stored for a schedule, ReportCaster will continue to display that parameter in the Parameter list and submit the parameter to the Reporting Server when the schedule is run even if the parameter is removed from the underlying report. To remove the parameter from the schedule information, edit the schedule and delete the parameter. For information on how to delete a parameter from a schedule, see [Deleting a Parameter](#) on page 709.

Example: Specifying a Default Parameter Value in the Report Procedure (FEX)

Specifying default parameter values can be done with the -DEFAULT or -DEFAULTH command or within the WHERE statement. Default values specified with the -DEFAULTH command are not prompted for.

The following procedure sets a default value of NY for the STATE (2-3 letters for US State) parameter.


```

                                -DEFAULT &STATE=NYTABLE FILE GGSales
SUM DOLLARS UNITS
BY ST
BY CATEGORY
BY PRODUCT
ON TABLE SUBHEAD
"Product Sales Report"
WHERE ST EQ '&STATE.2-3 letters for US State.'
END
  
```

The parameters that have default values defined in the report procedure (FEX) are listed within the Parameters tab with the default value in the Value column. When parameters are stored in a schedule, ReportCaster adds a -SET for the parameter to the schedule procedure sent to the Reporting Server to be run. A -SET command overrides a value specified in a -DEFAULT command.

The following image shows the Task Parameters dialog box with parameter STATE that has a default value specified. The STATE parameter has the value Yes displayed in the Task Parameters table *Use Default* column. The STATE parameter is indicated in the Parameter Properties section, displaying a value of NY in the *Display Value* field.

Task Parameters

Name	Description	Display Value	Value	Use Default	
 State	2-3 letters for US State	NY		Yes	

Up

Down

New

Delete

Refresh

Parameter Properties

Type:

Report Defined

Name:

State

Description:

2-3 letters for US State

Value:

NY

Display Value:

Default Value:

_FOC_NULL

☒ Always Use Default Value Specified in the Procedure

Data Type:

Minimum:

Maximum:

OK

Cancel

To use a value other than the default value at schedule execution time, so that it will be used even if the procedure is changed, click the parameter and change the parameter value within the Parameter Properties section. Values for simple parameters are specified by entering a value for the parameter in the *Value* field. There are also static and dynamic parameter types that allow selection of a single or multiple values, which are explained in the following examples.

When parameters are stored in a schedule, ReportCaster adds a -SET for the parameter to the schedule procedure sent to the Reporting Server to be run. A -SET command overrides a default value specified in a -DEFAULT command.

Example: Adding a Static Single-Select List of Parameter Values

The following procedure provides a list of static values that are valid for the CATEGORY (Category) parameter.

```
-DEFAULT &STATE=NY
TABLE FILE GGSales
SUM DOLLARS UNITS
BY ST
BY CATEGORY
BY PRODUCT
ON TABLE SUBHEAD
"Product Sales Report"
WHERE ST EQ '&STATE.2-3 letters for US State.'
WHERE CATEGORY EQ '&CATEGORY. (Coffee,Food,Gifts) .Category. '
END
```

The following image shows the CATEGORY (Category) parameter selected in the Task Parameters table. The CATEGORY parameter has a default value of Coffee. The Task Parameter *Use Default* column is blank for CATEGORY and the Parameter Properties section *Value* field is blank for STATE.

Task Parameters

Name	Description	Display Value	Value	Use Default	
STATE	2-3 letters for US State			Yes	
CATEGORY	Category	Coffee	Coffee		

Up

Down

New

Delete

Refresh

Parameter Properties

Type:

Report Defined

Name:

CATEGORY

Description:

Category

Value:

Coffee

Display Value:

Default Value:

_FOC_NULL

☐ Always Use Default Value Specified in the Procedure

Data Type:

Minimum:

Maximum:

OK

Cancel

To supply a value for a single select parameter, select the parameter in the Task Parameters table and enter a value in the *Value* field in the Parameter Properties section. From the list of the values, select a value to assign to the parameter. You can select only one value for a static single-select parameter.

Example: Adding a Dynamic Single-Select List of Parameter Values

The following procedure provides a single select list of values that are valid for the PRODUCT (Product Name) field. This list is dynamically populated with values from the GGSALES data source.

```

-DEFAULT &STATE=NY;
TABLE FILE GGSALES
SUM DOLLARS UNITS
BY ST
BY CATEGORY
BY PRODUCT
ON TABLE SUBHEAD
"Product Sales Report"
WHERE ST EQ '&STATE.2-3 letters for US State.'
WHERE PRODUCT EQ '&PRODUCT.(FIND PRODUCT IN GGSALES).Product Name.'END

```

The following image shows the PRODUCT (Product Name) parameter selected in the Task Parameters table. The PRODUCT parameter does not have a default value specified so the Task Parameter *Use Default* column and the Parameter Properties section *Value* field are blank.

Name	Description	Display Value	Value	Use Default
STATE	2-3 letters for US State			Yes
CATEGORY	Category			Yes
PRODUCT				

Up Down New Delete Refresh

To supply a value, click the PRODUCT (Product Name) parameter in the table and change the parameter value within the Parameter Properties section, as shown in the following image.

Task Parameters

Name	Description	Display Value	Value	Use Default	
STATE	2-3 letters for US State			Yes	
CATEGORY	Category			Yes	
PRODUCT	Product Name	Latte	Latte		

Up

Down

New

Delete

Refresh

Parameter Properties

Type:

Report Defined

Name:

PRODUCT

Description:

Product Name

Value:

Latte

Display Value:

Default Value:

☐ Always Use Default Value Specified in the Procedure

Data Type:

Minimum:

Maximum:

OK

Cancel

Example: Adding a Static Multi-Select List of Parameter Values

The following procedure provides a static multi-select list of values that are valid for the CATEGORY (Category) field.

```
-DEFAULT &STATE=NY
TABLE FILE GGSales
SUM DOLLARS UNITS
BY ST
BY CATEGORY
BY PRODUCT
ON TABLE SUBHEAD
"Product Sales Report"
WHERE ST EQ '&STATE.2-3 letters for US State.'
WHERE CATEGORY EQ '&CATEGORY. (OR(Coffee,Food,Gifts)).Category.'END
```

The following image shows the CATEGORY parameter selected in the Task Parameters table. When the Value field is selected in the Parameter Properties section, a drop-down list of available values is displayed.

The screenshot shows the 'Task Parameters' dialog box. At the top is a table with columns: Name, Description, Display Value, Value, and Use Default. The 'CATEGORY' row is selected. Below the table are buttons for 'Up', 'Down', 'New', 'Delete', and 'Refresh'. Below these is the 'Parameter Properties' section. On the left, there are labels for 'Type:', 'Name:', 'Description:', 'Value:', 'Display Value:', 'Default Value:', and 'Data Type:'. On the right, there is a 'Report Defined' section with a text box containing 'CATEGORY'. Below this, there is a 'Value:' field with a drop-down list showing 'Coffee', 'Food', and 'Gifts'. The 'Coffee' option is currently selected. At the bottom right of the dialog are 'OK' and 'Cancel' buttons.

Name	Description	Display Value	Value	Use Default
STATE	2-3 letters for US State			Yes
CATEGORY	Category			Yes
PRODUCT	Product Name	Latte	Latte	

Parameter Properties

Type: Report Defined

Name: CATEGORY

Description: Category

Value: Coffee

Display Value: Coffee

Default Value: Food

Data Type: Gifts

Example: Adding a Dynamic Multi-Select List of Parameter Values

The following procedure provides a dynamic multi-select list of values that are valid for the PRODUCT (Product Name) field. This list is dynamically populated with values from the GGSales data source.

```
-DEFAULT &STATE=NY;
TABLE FILE GGSales
SUM DOLLARS UNITS
BY ST
BY CATEGORY
BY PRODUCT
ON TABLE SUBHEAD
"Product Sales Report"
WHERE ST EQ '&STATE.2-3 letters for US State.'
WHERE PRODUCT EQ '&PRODUCT.(OR(FIND PRODUCT IN GGSales)).Product
Name.'END
```

The following image shows the PRODUCT parameter selected in the Task Parameters table. When the Value field is selected in the Parameter Properties section, a drop-down list of available values is displayed.

The screenshot shows the 'Task Parameters' dialog box. At the top is a table with columns: Name, Description, Display Value, Value, Use Default, and an empty column. The table contains three rows: STATE (2-3 letters for US State), CATEGORY (Category), and PRODUCT (Product Name). The PRODUCT row is highlighted. Below the table are buttons: Up, Down, New, Delete, and Refresh. Below these is the 'Parameter Properties' section. It has fields for Type (Report Defined), Name (PRODUCT), Description (Product Name), Value (Latte), Display Value, Default Value, and Data Type. A drop-down list is open next to the Value field, showing a list of coffee-related items: Biscotti, Capuccino, Coffee Grinder, Coffee Pot, Croissant, Espresso, Latte (highlighted), Mug, Scone, and Thermos. At the bottom right are OK and Cancel buttons.

Name	Description	Display Value	Value	Use Default	
STATE	2-3 letters for US State			Yes	
CATEGORY	Category			Yes	
PRODUCT	Product Name	Latte	Latte		

Up Down New Delete Refresh

Parameter Properties

Type: Report Defined

Name: PRODUCT

Description: Product Name

Value: Latte

Display Value:

Default Value:

Data Type:

Biscotti
Capuccino
Coffee Grinder
Coffee Pot
Croissant
Espresso
Latte
Mug
Scone
Thermos

OK Cancel

Deleting a Parameter

It is important to verify that the parameters you delete when scheduling a report procedure (FEX) will be handled, as follows, so that the scheduled job will run successfully:

- ☐ A default value is specified in the report procedure (FEX) being scheduled.
- ☐ A value will be dynamically assigned to the parameter by the report procedure (FEX) processing when the scheduled job runs on the Reporting Server.
- ☐ The parameter will not be referenced when the report procedure (FEX) is processed by the Reporting Server.

To delete a parameter so that it is not stored with the schedule information, highlight the parameter in the parameter table you want to delete, and click the *Delete* button.

Creating a New Parameter

If you need to have the schedule job send a parameter and value that is not defined in the report procedure being scheduled, you can create a new parameter in the Task Parameters dialog box. Parameters referenced during processing by the Reporting Server must be assigned a value for the scheduled job to run successfully. The schedule job log report will contain information when required parameter values were not provided.

Procedure: How to Create a New Parameter

You can create parameters for a task as follows:

1. On the ribbon, click *Parameters* to open the Task Parameters dialog box. The Task Parameters dialog box is shown in the following image.

Name	Description	Display Value	Value	Use Default	
------	-------------	---------------	-------	-------------	--

Up Down

New Delete Refresh

Parameter Properties

Type:

Name:

Description:

Value:

Display Value:

Default Value:

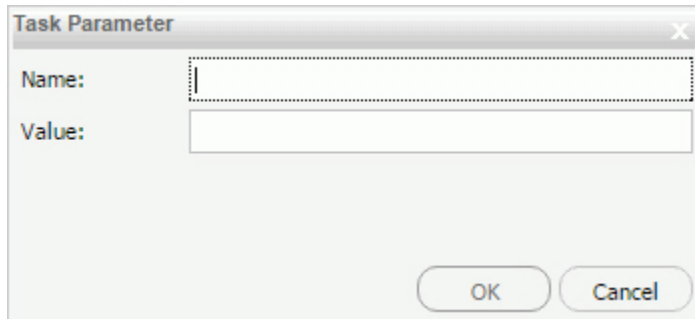
☒ Always Use Default Value Specified in the Procedure

Data Type: Minimum: Maximum:

OK Cancel

2. Click the *New* button located above the Parameter Properties section.

The Task Parameter dialog box displays, as shown in the following image.



The image shows a dialog box titled "Task Parameter" with a close button (X) in the top right corner. Inside the dialog, there are two text input fields. The first field is labeled "Name:" and the second field is labeled "Value:". Both fields are currently empty. At the bottom right of the dialog, there are two buttons: "OK" and "Cancel".

3. Enter a value in the *Name* and *Value* boxes.
4. Click *OK*.

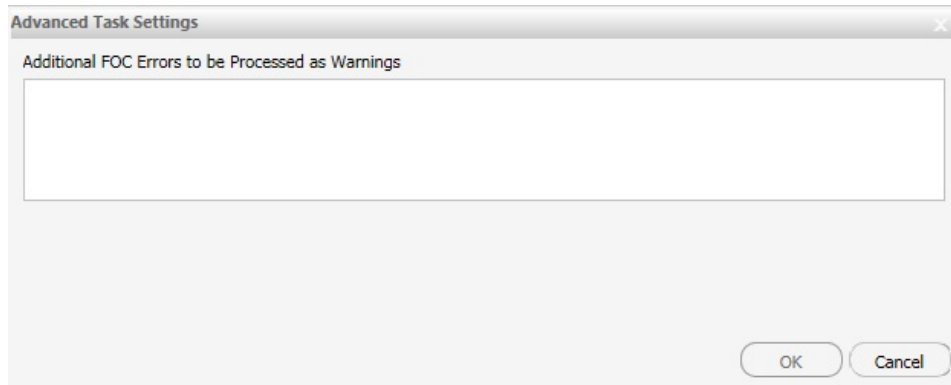
The *Name* and *Value* boxes are populated in the Task Parameters dialog box Parameters table.

5. If you would like to change the parameter value, select the parameter in the Parameters table and specify the values in the *Description* and *Value* fields located in the Parameter Properties section.
6. Click *OK* when you have completed entering parameter settings for the task.

Advanced Task Settings

The Advanced Task Settings option in the Options group provides access to the Additional FOCUS Errors to be Processed as Warnings dialog box, where you can specify one or more FOCUS error message numbers, separated by a comma. If one of these FOCUS error numbers is encountered during schedule execution, ReportCaster will process it as a warning rather than an error. For example, if a FOC1517 error is generated by the procedure, add 1517 to the text box to convert this error to a warning. This means that if a report is produced, it will be distributed in spite of the FOCUS error number. It also means that this FOCUS error number will not trigger error notification if the schedule is configured to notify on error.

The Advanced Task Settings dialog box is shown in the following image.



Distribution Options in the Basic Scheduling Tool

The Distribution tab in the Basic Scheduling tool provides the options available for distributing the report output for the scheduled report procedure (FEX). You can distribute a report output using one of the following methods.

- ☐ Email
- ☐ Printer
- ☐ Repository

Note:

- ☐ Distribution methods can be limited globally (for all users) in ReportCaster configuration and for groups or individual users by security operations.
- ☐ Maps can be distributed to the WebFOCUS Repository only. Maps distributed by Email do not open correctly.

Using the Email Distribution Option in the Basic Scheduling Tool

When you distribute a report through email, you can include the report in the body of the email (known as an inline email message) or send it as an attachment. Only the HTML, DHTML, WP, and DOC formats can be distributed as an inline email message.

Note: You must be authorized to distribute by Email in order to create a schedule that uses Email distribution.

Distributing a report as an inline email message is particularly useful when the report is distributed to mobile devices or through email systems that do not support attachments.

You can also distribute a report to a fax machine, as explained later in this section. The following image shows the email distribution options in the Distribution tab of the Basic Scheduling tool when the email method is selected.

Note:

- ☐ The availability of the inline message option when you create a schedule depends on the *Inline Report Distribution* setting in the ReportCaster Configuration tool accessible from the ReportCaster Console.
- ☐ The display of a report that is distributed as an inline email message can be affected by settings and restrictions of your email server or email client.
- ☐ When distributing HTML reports by email, the scheduled report (.fex) must set a fully qualified FOCEXURL and FOCHTMLURL for the report styling options defined below. These reference the JavaScript components located on the web or application server where the Client is configured. The Distribution Server will set a fully qualified FOCEXURL and FOCHTMLURL for email distribution using the host name and port provided at installation time. This can be overridden by setting these values in the scheduled procedure. For example:

```
SET FOCEXURL='hostname:port/ibi_apps/'
```

```
SET FOCHTMLURL='hostname:port/ibi_apps/ibi_html'
```

Styling options include:

- ☐ Accordion reports
- ☐ Table of Contents (TOC) reports
- ☐ Peer Graphics/Data Visualization graphical reporting
- ☐ Multi-drill reports
- ☐ HFREEZE options

Procedure: How to Use the Email Distribution Option

Note: You can use parameters (ampers variables) when specifying values in the Email settings of a schedule. These parameters can be system variables, such as &YYMD, or any ampers variable for which a value is returned by the Reporting Server when the scheduled procedure is executed. To use the dot file extension separator after a variable, terminate the variable with the '|' character (for example, &YYMD|.htm). Similarly, to use the ampersand character itself, follow the ampersand with the '|' character (for example, Smith&|Jones). If a value for a parameter specified in a schedule is not returned by the Reporting Server when the procedure executes, the schedule will fail with a "No report to distribute" error. If the scheduled task is burst, you can also use the '%BURST' symbol to include the burst value. If the Packet Email setting is Yes, the burst value will not be substituted in the Zip filename.

1. Right-click a report procedure (FEX), select *Schedule*, and then select *Email*.

The Basic Scheduling tool appears.

2. Select the *Distribution* tab.
3. From the Type drop-down list, select the method you will use to provide the email addresses that will receive the distribution. The options are Distribution List, Distribution File, Dynamic Distribution List, and Email Address(es). These options all show where the report procedure (FEX) distributes To, From, Reply Address, and Subject. Additionally, in the Email Information section, select whether you would like to send all reports as attachments or send a report as an inline message.

☐ **Distribution List.** The report will be sent to all email addresses in the selected email Distribution List.

☐ **Email Address(s).** This is the default method for supplying email addresses in the scheduling tools. The default value of this field is the email address of the user that is creating the schedule. You can specify multiple email addresses in the Email Address(s) field. Separate each email address with a comma (,) or a semicolon (;). The email addresses will appear in the To line of a single email when the scheduled output is distributed. Each individual email address can be a maximum of 130 characters, according to the SMTP specification. The total maximum length of this field is 800 characters.

Additionally, you can use group mail lists (defined on your mail server) with the Email Address(es) option. You can use group mail lists to distribute a report or notification to multiple recipients without having to maintain multiple email addresses in the Repository. The format of the group mail list depends on the mail server being used. For example, if you are using a Microsoft Exchange Server and your group mail list is defined as #group1, you would enter group1@listdomain in the Email Address(es) field. If the group mail list contains a space within its name, enclose the space with double quotation marks (" "). For more information, see your mail server administrator.

4. In the To box, type the email address of the recipient.
5. In the From box, type any value (for example, the name of the person creating the schedule). ReportCaster does not require a value for this field, but your email system may require one.
6. In the Reply Address box, type a valid email address. If recipients reply to the email, their messages will be sent to this address. If your email system is unable to deliver the content, the undeliverable output message is also returned to this address. ReportCaster requires this field.
7. In the Subject box, type the text that you want to appear in the email subject line. ReportCaster may not require this information, but it may be needed by your email system. The value you entered in the schedule Title field is the default Subject value.
8. Specify whether you want to send the report as an email attachment, or within the body of the email inline by selecting or clearing the *Send all reports as attachments* option. For more information, see [Using the Email Distribution Option in the Basic Scheduling Tool](#) on page 712.
9. Optionally, you can type a message to appear in the email body to replace the default message, *Please see attachment(s)*.

Note: You can also provide a custom default distribution message using the Email Distribution and Notification settings in the ReportCaster Configuration tool in the ReportCaster Console. This allows you to override the default message, *Please see attachment(s)*, by enabling you to create a custom message specific to your organization. The message displays when creating a schedule using Email distribution. For more information, see [ReportCaster Configuration](#) on page 146.

10. Optionally, select a file from the WebFOCUS Repository that contains the message you want to distribute with your Email distribution.

Note: When you use the file option, the information can be maintained in one common place. It can also be revised among multiple schedules. There is no size limit to the contents of the file, other than what the email client and server support.

11. Optionally, specify a value for the Packet Email or accept the default value. The Packet Email option is set to the default value configured by the Manager. For a schedule that has a task that is burst, you can decide how many emails to send to each recipient. Options include:

- ☐ **No.** Sends each attachment in a separate email.
- ☐ **Yes.** Sends one email that contains all attachments.
- ☐ **By Burst Values.** Sends one email with multiple attachments for each burst value.

12. Specify whether to send the report as a compressed (.zip) file by selecting or clearing the *Add Report to Zip File* check box.

You have the option of converting your distributed report into a password-protected compressed file. By default, the Add Report to Zip File check box is not selected and the report will not be compressed. You can optionally password protect a zipped file by including a password in a Distribution File or Dynamic Distribution List.

13. If the *Add Report to Zip File* check box is selected, enter the name of the Zip file in the *Zip File Name* box.

Note: Support of Unicode characters in a Zip file name and content within a Zip file on Windows 7 and Windows 2008 Server R2 requires the following hotfix from Microsoft: <http://support.microsoft.com/kb/2704299/en-us>. Without this hotfix, file names are corrupted after you decompress a .zip file in Windows 7 or in Windows Server 2008 R2.

14. Additionally, you can specify the minimum size (in KBs) a file must exceed before it is automatically added to a Zip file. The Zip minimum size is set to the default value configured by the administrator. To automatically zip an attachment that exceeds a certain size, set this value to the desired size.
15. Select the *Notification* tab to specify whether to send an email notification of the schedule job status. If you select Always or On Error, then you have to specify the Reply Address, Subject, Brief Message To, and Full Message To.
16. Next, select the *Properties* tab to specify the Title, Priority, whether to delete the schedule if it will not run again, and whether the schedule is enabled to run. For more information, see [About Properties in the Basic Scheduling Tool](#) on page 720.
17. Select the *Recurrence* tab to specify how often to run the schedule. If you want the schedule to run on the current day, set the Start Date and Start Time to values later than the current time. For more information, see [About Recurrence in the Basic Scheduling Tool](#) on page 721.
18. Select *Save & Close* to save the schedule.

Using the Printer Distribution Option in the Basic Scheduling Tool

The report formats that support printing are DOC, PDF (when you configure ReportCaster to enable PDF to print and the printer has the appropriate driver), PS, and WP.

Note:

- ☐ You must be authorized to distribute to a Printer in order to create a schedule that uses Printer distribution.
- ☐ Problems may occur in printed output if the distributed reports contain UTF-8 characters.
- ☐ Since a Printer schedule will not work unless the report output format is a valid print format, ReportCaster always sets the format of a schedule with distribution to a printer to a valid format. If the default configuration of PDF as a valid print format is in place, the override format is set to PDF. Otherwise, the override format is set to DOC. You can change this format on the Task tab.

Procedure: How to Use the Printer Distribution Option

Note: You can use parameters (amper variables) when specifying a value for the Printer Name field of a schedule. These parameters can be system variables, such as &YYMD, or any amper variable for which a value is returned by the Reporting Server when the scheduled procedure is executed. To use the dot file extension separator after a variable, terminate the variable with the '|' character (for example, &YYMD|.htm). Similarly, to use the ampersand character itself, follow the ampersand with the '|' character (for example, Smith&|Jones). If a value for a parameter specified in a schedule is not returned by the Reporting Server when the procedure executes, the schedule will fail with a "No report to distribute" error. If the scheduled task is burst, you can also use the '%BURST' symbol to include the burst value.

1. Right-click a report procedure (FEX), select *Schedule*, and then select *Printer*.
The Basic Scheduling tool appears.
2. Click the *Distribution* tab.
3. From the Type drop-down menu, select the method in which you will provide the file names to distribute to the printer. The options are:
 - ☐ **Distribution List.** The report will be sent to all printers in the selected Distribution List. To select a Distribution List, click the icon next to the Distribution List field.
 - ☐ **Printer Name.** Specify the printer using the following format.

queue@printserver

where:

queue

Is the name of the printer queue.

printserver

Is the host name or IP address of the printer.

ReportCaster can differentiate between the printer queue and the printer host name or IP address due to the presence of the '@' separator. Although ReportCaster supports specifying only the host name or IP address of the printer, we recommend that you specify both the printer queue and host name or IP address when distributing ReportCaster output to a printer. The maximum length of this field is 800 characters.

4. If you select *Distribution List*, click the *Name* button, which will display the Open dialog box to allow you to select a Distribution List. If you select *Distribution File*, specify in the Name field the fully qualified path and file name where the external Distribution File is located. If you select *Printer*, specify the printer name in the Name field.
5. Select the *Notification* tab to specify whether to send an email notification of the schedule job status. If you select *Always* or *On Error*, then you must specify the Reply Address, Subject, Brief Message To, and Full Message To.
6. Select the *Properties* tab to specify the Title, Priority, whether to delete the schedule if it will not run again, and whether the schedule is enabled to run. For more information, see [About Properties in the Basic Scheduling Tool](#) on page 720.
7. Select the *Recurrence* tab to specify how often to run the schedule. If you want the schedule to run on the current day, set the Start Date and Start Time to values later than the current time. For more information, see [About Recurrence in the Basic Scheduling Tool](#) on page 721.
8. Select *Save & Close* to save your changes.

Notification Options in the Basic Scheduling Tool

The Notification tab in the Basic Scheduling tool, shown in the following image, provides the options to send a notification of the schedule status to specific email recipients.

Note:

- ☐ Notification will not work unless a mail server is configured. If a mail server is not configured, notification will fail and an error message will be recorded in the schedule job log. For more information about configuring a mail server, refer to the *Installation* topic in the Information Center.

- ☐ You can use parameters (amper variables) when specifying values for the Notification fields of a schedule. These parameters can be system variables, such as &YYMD, or any amper variable for which a value is returned by the Reporting Server when the scheduled procedure is executed. To use the dot file extension separator after a variable, terminate the variable with the '|' character (for example, &YYMD|.htm). Similarly, to use the ampersand character itself, follow the ampersand with the '|' character (for example, Smith&|Jones). If a value for a parameter specified in a schedule is not returned by the Reporting Server when the procedure executes, the schedule will fail with a "No report to distribute" error.

The notification options are:

- ☐ **Never.** ReportCaster will not send a notification of the schedule status under any circumstances. This is the default value.
- ☐ **Always.** The specified users are always notified when the schedule runs.
- ☐ **On Error.** The specified users are notified when errors are encountered while running the schedule. Information Builders recommends the use of the On Error notification option.

Setting On Error and Always Notification in the Basic Scheduling Tool

When you select the On Error or Always notification option, additional options become available, as shown in the following image.

The On Error and Always notification options are:

- ☐ **Reply Address.** Type the email address of the sender. If report recipients reply to the report sender, then their messages are sent to this address. If your email system is unable to deliver a report, then the undeliverable report message is also returned to this address.

Note:

- ☐ If you are authenticating to the mail server with your user ID and password, then the reply address will be the email address associated with that user ID.
- ☐ If the mail server is configured with authentication and the Reply Address is configured in the ReportCaster Configuration tool, then the Reply Address field in the scheduling tools will be disabled. If a Reply Address is not configured, then the field is enabled to allow a Reply Address to be sent to the email server, however the actual Reply Address of the delivered email will be that of the authenticating account.
- ☐ **Subject.** Type the text you want to display in the subject line of the email notification. There is a limit of 255 alphanumeric characters. By default, this field contains the report name and date and time stamp.

- ❑ **Brief Message To.** Type the email address where you want a full notification sent. There is no syntax error checking for this field.

Tip: Use the Brief Message To option when you are sending notification to devices that have limited memory, such as pagers and cell phones. If you want to notify multiple recipients, you can use group mail lists defined on your mail server provided that you append an at sign (@) followed by a valid domain.

- ❑ **Full Message To.** Type the email address to which you want a full notification sent. There is no syntax error checking for this field.

About Properties in the Basic Scheduling Tool

When you access the Basic Scheduling tool, the Title and Path Properties options are predefined for the selected report procedure (FEX). The following image displays the Properties tab in the Basic Scheduling tool.

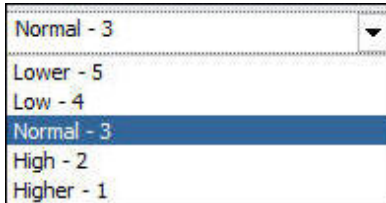
The options within the Properties tab are:

- ❑ **Title.** This allows you to provide a brief description of the purpose of the schedule. It is pre-populated with the Title of the report procedure being scheduled when creating schedules with the Basic Scheduling tool. You can edit the Title while creating the schedule or after saving the schedule from within the Schedule tools. You can also edit the title from the Properties option from the Resources tree after saving the schedule.

The Title of the schedule is the default name assigned when saving the schedule. If the name value already exists in the selected tree folder, a message is displayed informing you that the name already exists. You can change the Title field in the Save dialog box, which when saved, will also update the Title field within the schedule information.

- ❑ **Path.** This is the Repository path of the report procedure (FEX) you selected to schedule.
- ❑ **Summary.** This allows you to insert a detailed description for the schedule. This is an optional field.

- ☐ **Job Priority Level.** This specifies the priority the scheduled job will be given when processed by the Distribution Server. The default Job Priority Level is set to Normal - 3. However, you can use the drop-down list to set the priority level, as shown in the following image.



- ☐ **No Report to Distribute.** This option is set to the default value configured by the Manager. To trigger error notification if no report is generated, set this value to Error. If you do not want to trigger notification when no report is generated, set this value to Warning.
- ☐ **Delete this schedule if it is not scheduled to run again.** This check box allows you to specify that the schedule should be deleted after the scheduled job processing is completed if the schedule is not scheduled to run again. Selecting this option for schedules you will not utilize again is recommended as it will improve overall performance within the Resources tree listing folder contents and within the ReportCaster Explorer tool when listing schedules.
- ☐ **Enabled (Scheduled job runs at specified time).** This check box is selected by default to specify that the schedule should be evaluated by the Distribution Server when polling for scheduled jobs to run. If you do not want to distribute the schedule based on its NEXTRUNTIME value, clear this check box.

About Recurrence in the Basic Scheduling Tool

When you access the Basic Scheduling tool, the Recurrence tab options allow you to define how often to run the schedule. The following image displays the Recurrence tab in the Basic Scheduling tool.

Options that users must determine include frequency of distribution, start and end times, and Advanced interval settings. Select one of the following frequency of distribution settings:

- ☐ Run Once
- ☐ Minutes
- ☐ Hourly

- ☐ Daily
- ☐ Weekly
- ☐ Monthly
- ☐ Yearly
- ☐ Custom

You can assign start and end times by using the drop-down lists. When you click on the down arrow, a calendar will display that enables an authorized user to set the date for schedule distribution. Use the up and down arrows to set a specific time for schedule distribution. Alternatively, you can enter the time manually.

If the user has the privilege to set Advanced settings, check the *Repeat schedule every* check box to enable Advanced interval setting options. Set how often you want to repeat schedule distribution, when you want to stop distributing the schedule (Until Time), and the duration to distribute the schedule (Last For). Enter this information manually or use the up and down arrows to set parameters.

The Run Once Interval

The Run Once option sets the job to execute immediately. This is the default value. You can modify the date or time if you do not want the schedule to run immediately. You can specify the date and time you want the schedule to run using the Start Schedule options, as shown in the following image.

The screenshot shows a software interface with two panels. The top panel, titled "Settings", has a left sidebar with radio buttons for "Run Once", "Minutes", "Hourly", "Daily", "Weekly", "Monthly", "Yearly", and "Custom". The "Run Once" option is selected. The main area of the "Settings" panel displays the text "The time zone for the Start Time, End Time and Until Time is Eastern Standard Time (GMT-04...)". Below this text are two rows of controls: "Start:" with a date dropdown set to "Monday, April 4, 2016" and a time dropdown set to "10: 47 AM"; and "End:" with a date dropdown set to "Thursday, December 31, 2099" and a time dropdown set to "11: 59 PM". The bottom panel, titled "Advanced settings", contains a checkbox labeled "Repeat schedule every:" which is unchecked. To its right is a numeric input set to "10" and a dropdown set to "Minute(s)". Below this are two radio buttons: "Until Time:" (unchecked) and "Last For:" (checked). The "Last For:" section has a numeric input set to "1", a dropdown set to "hour(s)", and another numeric input set to "0" with a dropdown set to "minute(s)".

To select a date, choose a date from the drop-down date calendar. To select a time, select either the hour or minutes, and use the up and down arrows to increase or decrease the value. Alternatively, you can enter the time manually.

The Minutes Interval

The Minute(s) option sets the schedule to run every n minutes.

In the Every minute(s) field, type or select the minutes interval (1 to 59), check the days of the week on which you want to run the schedule, and select the *Start* and *End* date and time to define the time period in which the schedule will run. For example, the following image shows a schedule that will run every 30 minutes on Mondays beginning at noon May 16, 2012 and ending 6:00 PM October 30, 2012.

The screenshot shows a configuration window with two main sections: **Settings** and **Advanced settings**.

Settings:

- On the left, a list of options: Run Once, Minutes (selected), Hourly, Daily, Weekly, Monthly, Yearly, and Custom.
- Text: "The time zone for the Start Time, End Time and Until Time is Eastern Standard Time (GMT-0400)."
 - Start:** Wednesday, May 16, 2012, 12: 00 AM
 - End:** Tuesday, October 30, 2012, 6: 00 AM
- Every 30 minute(s) on:**
 - ☒ Mon
 - ☐ Tue
 - ☐ Wed
 - ☐ Thu
 - ☐ Fri
 - ☐ Sat
 - ☐ Sun

Advanced settings:

- ☐ Repeat schedule every: 10 Minute(s)
- ☐ Until Time: 7: 27 PM
- ☒ Last For: 1 hour(s) 0 minute(s)

Tip: Selecting this option may affect system performance if you choose to run the schedule every 5 minutes or less. We recommend specifying a minimum of 30 minutes. The minute interval option is primarily for alert schedules.

The Hourly Interval

The Hourly option sets the schedule to run every n hours.

In the Every hour(s) field, type or select the hours interval (1 to 24), check the days of the week on which you want to run the schedule, and select the *Start* and *End* date and time to define the time period in which the schedule will run. For example, the schedule shown in the following image will run every three hours on Mondays and Fridays beginning at noon May 16, 2012 and ending 6:00 PM October 30, 2012.

Chart1- Schedule (Basic)

Actions

Save & Close Delete Properties Recurrence Task Distribution Notification Log Reports Parameters Advanced Task Settings

Show

Options

Settings

Run Once
Minutes
Hourly
Daily
Weekly
Monthly
Yearly
Custom

The time zone for the Start Time, End Time and Until Time is Eastern Standard Time (GMT-0400) .

Start: Wednesday, May 16, 2012 12: 00 AM

End: Tuesday, October 30, 2012 6: 00 AM

Every 3 hour(s) on: ☒ Mon ☐ Sat
☐ Tue ☐ Sun
☐ Wed
☐ Thu
☒ Fri

Advanced settings

☐ Repeat schedule every: 10 Minute(s)

☐ Until Time: 7: 27 PM

☒ Last For: 1 hour(s) 0 minute(s)

The Daily Interval

The Daily option in the Run Interval drop-down list sets the schedule to run every n days. In the Every day(s) field, type or select the days interval to run the schedule and select *Start* and *End* date and time to define the period in which the schedule will run. For example, the schedule shown in the following image will run every five days beginning at noon May 16, 2012 and ending 6:00 PM October 30, 2012.

Chart1- Schedule (Basic)

Actions

Save & Close Delete Properties Recurrence Task Distribution Notification Log Reports Parameters Advanced Task Settings

Settings

Run Once Minutes Hourly **Daily** Weekly Monthly Yearly Custom

The time zone for the Start Time, End Time and Until Time is Eastern Standard Time (GMT-0400) .

Start: Wednesday, May 16, 2012 12:00 AM

End: Tuesday, October 30, 2012 6:00 AM

Every 1 day(s)

Advanced settings

☒ Repeat schedule every: 10 Minute(s)

☐ Until Time: 7:27 PM

☒ Last For: 1 hour(s) 0 minute(s)

You can also set a secondary run interval. For information about this setting, see [Advanced Settings](#) on page 731.

The Weekly Interval

The Weekly option in the Run Interval drop-down list sets the schedule to run every n weeks.

In the Every week(s) field, type or select the weekly interval to run the schedule, check the days of the week on which you want to run the schedule, and select the *Start* and *End* date and time to define the time period in which the schedule will run. The following image shows a schedule that will run every two weeks on both Monday and Friday beginning at noon May 16, 2012 and ending at 6:00pm on October 30, 2012.

Chart1- Schedule (Basic)

Actions: Save & Close, Delete

Show: Properties, **Recurrence**, Task, Distribution, Notification, Log Reports

Options: Parameters, Advanced Task Settings

Settings:

- ☐ Run Once
- ☐ Minutes
- ☐ Hourly
- ☐ Daily
- ☒ **Weekly**
- ☐ Monthly
- ☐ Yearly
- ☐ Custom

The time zone for the Start Time, End Time and Until Time is Eastern Standard Time (GMT-0400) .

Start: Wednesday, May 16, 2012 12:00 AM

End: Tuesday, October 30, 2012 6:00 AM

Every 2 week(s) on:

- ☒ Mon
- ☐ Tue
- ☐ Wed
- ☐ Thu
- ☒ Fri
- ☐ Sat
- ☐ Sun

Advanced settings:

☒ Repeat schedule every: 10 Minute(s)

☐ Until Time: 7:27 PM

☒ Last For: 1 hour(s) 0 minute(s)

Note: When selecting the *Weekly* interval, set the *Start* to the date of the first day (current or future) of the week you want the schedule to run. If you select the current date, then you must make sure that the *Start* time is later than the current time when you save the schedule. If the *Start* time is less than or equal to the current time, the calculation for the next run time results in the schedule not running on the current date.

You can also set a secondary run interval. For more information about this settings, see [Advanced Settings](#) on page 731.

The Monthly Interval

The Monthly option sets the schedule to run every n months. You can then refine the monthly interval with one of the following options. Note that these options are mutually exclusive.

- ☐ Every first, second, third, fourth, or last n day of the week (where n is Monday, Tuesday, Wednesday, Thursday, Friday, Saturday, or Sunday) every n months.
- ☐ Specific days every n months.

Additionally, select the *Start* and *End* date and time to define the time period in which the schedule will run. The following image shows a schedule set to run on the first Monday of every month beginning at noon May 16, 2012 and ending 6:00 PM October 30, 2012.

The screenshot shows the 'Chart1 - Schedule (Basic)' dialog box. The 'Schedule' tab is active, and the 'Recurrence' button is highlighted. The 'Settings' section is expanded, showing the 'Monthly' interval selected. The 'Start' date is 'Wednesday, May 16, 2012' at '12: 00 AM', and the 'End' date is 'Tuesday, October 30, 2012' at '6: 00 AM'. The time zone is set to 'Eastern Standard Time (GMT-0400)'. The 'Every' field is set to '1' month(s). The 'Day(s)' section is expanded, showing a calendar grid where the first Monday of the month is selected. The 'Advanced settings' section is also expanded, showing 'Repeat schedule every: 10 Minute(s)', 'Until Time: 7: 27 PM', and 'Last For: 1 hour(s) 0 minute(s)'.

The following image shows a schedule set to run on the 2nd, 9th, 16th, 23rd, and 30th of every month, regardless of the day of the week on which those dates fall.

The screenshot displays a 'Settings' window for a scheduling application. The 'Run Once' option is selected. The time zone is set to Eastern Standard Time (GMT-0400). The start date is Wednesday, May 16, 2012, at 12:00 AM, and the end date is Tuesday, October 30, 2012, at 6:00 AM. The interval is set to 'Every 1 month(s) on:'. The 'Day(s)' option is selected, and a calendar grid shows the 2nd, 9th, 16th, 23rd, and 30th of the month selected. The 'Advanced settings' section shows 'Repeat schedule every: 10 Minute(s)', 'Until Time: 7:27 PM', and 'Last For: 1 hour(s) 0 minute(s)'.

You can also select the *Last Day of the Month* option at the end of the calendar to run the schedule on the last day of the month.

Note: When selecting the *Month(s)* interval, set the *Start* date to the date of the first day (current or future) of the month you want the schedule to run. If you select the current date, then you must make sure that the *Start* time is later than the current time when you save the schedule. If the *Start* time is less than or equal to the current time, the calculation for the next run time results in the schedule not running on the current date.

You can also set a secondary run interval. For information about this setting, see [Advanced Settings](#) on page 731.

The Yearly Interval

The Yearly option sets the schedule to run every n years during a specific time period. The following image shows a schedule set to run every two years beginning at noon May 16, 2012 and ending 6:00 PM October 30, 2022.

The image shows a software interface with two panels: 'Settings' and 'Advanced settings'.

Settings Panel:

- On the left, a list of radio buttons for scheduling intervals: Run Once, Minutes, Hourly, Daily, Weekly, Monthly, **Yearly** (selected), and Custom.
- On the right, a text label states: "The time zone for the Start Time, End Time and Until Time is Eastern Standard Time (GMT-0400) .".
- Below the label, there are two rows of date and time pickers:
 - Start:** Wednesday, May 16, 2012 | 12: 00 AM
 - End:** Tuesday, October 30, 2012 | 6: 00 AM
- Below the end time picker, there is a field labeled "Every" with a value of "2" and a unit of "year(s)".

Advanced settings Panel:

- At the top, a checkbox labeled "Repeat schedule every:" is checked.
- Below the checkbox, there are two radio buttons:
 - Until Time:** (unselected) with a value of "7: 27 PM".
 - Last For:** (selected) with a value of "1" hour(s) and "0" minute(s).

You can also set a secondary run interval. For information about this setting, see [Advanced Settings](#) on page 731.

The Custom Interval

The Custom option allows you to select a set of dates that do not follow a specific pattern. For example, if you want to run a quarterly report on a different day of each quarter, then you can use the Custom run interval to set the schedule to run on dates, such as March 3 (Saturday), June 4 (Monday), September 7 (Friday), and December 2 (Sunday).

The following image shows the Custom Run Interval options, which includes Start (initially set to the current date and time) and the End date and time options, the Custom calendar, and the Custom Date List node that appears in the right panel.

Settings

☐ Run Once
☐ Minutes
☐ Hourly
☐ Daily
☐ Weekly
☐ Monthly
☐ Yearly
☒ Custom

The time zone for the Start Time, End Time and Until Time is Eastern Standard Time (GMT-0400) .

Start:

Wednesday, May 16, 2012

12: 00 AM

End:

Tuesday, October 30, 2012

6: 00 AM

<<

December 2012

>>

<

<<

Sun	Mon	Tue	Wed	Thu	Fri	Sat
25	26	27	28	29	30	1
2	3	4	5	6	7	8
9	10	11	12	13	14	15
16	17	18	19	20	21	22
23	24	25	26	27	28	29
30	31	1	2	3	4	5

Advanced settings

☒ Repeat schedule every:

10

Minute(s)

☐ Until Time:

7: 27 PM

☒ Last For:

1

hour(s)

0

minute(s)

Define the time period in which the schedule will run by selecting the *Start* date and time and the *End* date and time. Select the specific days on which to run the schedule by clicking the day in the calendar. (Use the forward and back buttons at the top of the calendar to move through the months and years.) As you select a date, it appears in the Custom Date List. If it does not already exist, a folder for the year and month of the date you select is automatically created under the Custom Date List node. The following image shows an example of a selection of Custom dates in the calendar and the Custom Date List.

The screenshot shows two panels: 'Settings' and 'Advanced settings'.

Settings Panel:

- Run Time:** Run Once, Minutes, Hourly, Daily, Weekly, Monthly, Yearly, Custom (selected).
- Time zone:** The time zone for the Start Time, End Time and Until Time is Eastern Standard Time (GMT-0400).
- Start:** Wednesday, May 16, 2012, 12: 00 AM
- End:** Tuesday, October 30, 2012, 6: 00 AM
- Calendar:** A calendar for December 2012. Dates 9, 16, 23, and 30 are highlighted in red. Navigation buttons (left arrow, right arrow, double left arrow, double right arrow) are present.
- Custom Date List:** A list of dates: Saturday, March 3, 2012; Monday, June 4, 2012; Friday, September 7, 2012; Sunday, December 2, 2012.

Advanced settings Panel:

- Repeat schedule every:** 10 Minute(s)
- Until Time:** 7: 27 PM
- Last For:** 1 hour(s) 0 minute(s)

To remove a date from the list, click the date in the calendar. The date is no longer highlighted in the calendar and will not appear in the Custom Date List. You can optionally use the left single arrow to remove a date from the list. The double arrow button removes all defined lists.

You can also set secondary run interval. For information about this setting, see [Advanced Settings](#) on page 731.

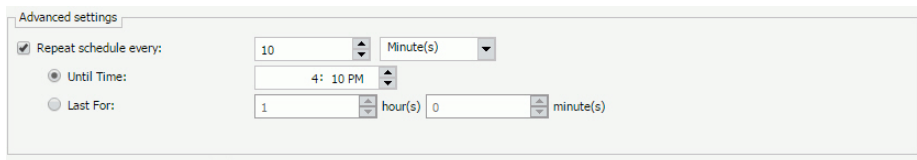
Advanced Settings

The Advanced settings option enables you to create a secondary run interval within the day the schedule runs. You can apply the secondary run interval every *n* minutes or hours for a specified number of hours and minutes or until a specified time. This option is available for schedules that run every day(s), week(s), month(s), or year(s).

Note: When a schedule is saved, the hours setting is converted to minutes. When a schedule is edited, this value appears as minutes.

The secondary run interval will not be validated when the schedule is created. Instead, validation is performed every time the next run time of the schedule is calculated when running within the secondary run interval. The secondary run interval cannot exceed the next run time for the primary run interval. For example, a daily schedule cannot have a secondary run interval greater than Every 1 day(s). If you schedule a secondary run interval to run after the next primary run interval of the schedule, the secondary run interval is stopped and an error message appears. This error message is also written to the log file.

The Advanced settings section allows you to specify repeat options, as well as time intervals. The following image shows an example of set secondary run interval options.



The screenshot shows a dialog box titled "Advanced settings". It contains three radio button options: "Repeat schedule every:" (which is selected), "Until Time:", and "Last For:". The "Repeat schedule every:" option has a value of "10" and a unit dropdown set to "Minute(s)". The "Until Time:" option has a value of "4:10 PM". The "Last For:" option has a value of "1" for "hour(s)" and "0" for "minute(s)".

The Apply secondary run interval options are:

- ☐ **Repeat schedule every.** Applies the secondary run interval every n minutes or hours (in this example, every 10 minutes) within the day the schedule runs.
- ☐ **Until Time.** The time up until which the secondary run interval will be applied. In this example, the schedule will rerun every 10 minutes until 4:10 PM when the Until Time option is selected.
- ☐ **Last For.** The duration, specified in hours and minutes, during which the secondary run interval will be applied. This option and the Until Time option are mutually exclusive.

Note: When a schedule is updated, the next run time is recalculated based only on the primary run interval. This means that if a schedule that includes a secondary run interval is updated before the secondary schedule is able to run, then the secondary run interval is ignored and the NEXTRUNTIME is calculated based on the primary interval.

For example, a schedule exists that is set to run daily at 2:00 PM with a secondary run interval of every 10 minutes from 2:00 PM to 3:00 PM. When the schedule runs at 2:00 PM, the NEXTRUNTIME resets to run at 2:10, which honors the secondary run interval. If this schedule is updated at 2:03 PM, the NEXTRUNTIME is recalculated to be 2:00 PM the next day, rather than 2:10 PM on the current day.

Maintaining Schedules

Maintaining a schedule allows you to edit schedule properties or delete the schedule when it is no longer needed. If a schedule contains properties that you want to use in a new schedule, the duplicate or copy option creates a template with those properties for the new schedule. You can also check the status of a schedule and run a log report to obtain detailed information about the schedule.

About Maintaining a Schedule in the Basic Scheduling Tool

As a user with the ability to access the Basic Scheduling tool, you can perform various maintenance functions on the schedules you are authorized to access.

If you right-click a schedule in the Resources tree, the options shown in the following image are available.

Note: The options shown are dependent on your security effective policy.

Edit

Allows you to open and edit an existing schedule.

Run

Runs the schedule.

View Log

Allows you to view a log report for one or more selected schedules.

Duplicate

Creates a new schedule with the same properties in the same folder. The new schedule is disabled automatically, because it is a duplicate of an existing schedule.

Cut

Allows you to move the schedule from the original folder to a target folder using the Paste operation.

Copy

Allows you to create a new schedule by copying an existing schedule.

Create Shortcut

Allows you to create a shortcut to the schedule.

Delete

Deletes the existing schedule.

Change Title

Allows you to rename the schedule.

Publish and Unpublish

An owner of a schedule can make a schedule available to other members of the top-level folder in which the schedule resides. The schedule owner remains the execution ID. The following table describes the shortcut menu options permitted to the different groups associated with a top-level folder.

	Basic	Advanced	Developer	Group Administrator	Manager
Run	✗	✓	✓	✗	✓
View Log	✗	✓	✓	✗	✓
Edit	✗	✓	✓	✗	✓
Publish/Unpublish	✗	✗	✓	✓	✓
Security	✗	✗	✓	✗	✓
Hide/Show	✗	✗	✓	✗	✓
Copy/Duplicate	✗	✓	✓	✗	✓
Properties	✓	✓	✓	✓	✓
Change Title	✗	✗	✓	✗	✓
Delete	✗	✗	✓	✗	✓

Note: The default setting is Unpublish.

Hide and Show

Once an owner publishes a schedule, they can elect to Hide the schedule from groups not authorized to create content in the folder. To hide or show a schedule, right-click on a schedule and select the *Hide* or *Show* option.

Note: The default setting is Show.

Share

Allows you to share the selected schedule. You can subsequently Unshare the schedule via the same menu.

Share with

Allows you to specify the groups and users with whom you want to share the selected schedule.

Unshare

Allows you to remove sharing on a shared schedule. This option only displays if you have previously shared the schedule.

Security

Allows you to view and modify the ownership of the schedule. Options include: Rules, Rules on this Resource, and Effective Policy.

Properties

Allows you to view properties of the schedule, as well as modify title and summary information. You can also change the name of a schedule using the Properties dialog box. In addition, you can access Security features, including Owner and Sharing.

Editing a Schedule in the Basic Scheduling Tool

From the Resources tree, you can edit the schedules you are authorized to access.

***Procedure:* How to Edit a Schedule in the Basic Scheduling Tool**

1. Right-click the schedule you want to edit and select *Edit*.
2. Make the required changes to the schedule. For details on the Basic Scheduling tool options, see [Creating a Schedule in the Basic Scheduling Tool](#) on page 695.
3. Click *Save & Close*.

***Reference:* Considerations When Editing a Schedule in the Basic Scheduling Tool**

- ☐ When you open a schedule that uses unavailable options, such as a distribution method, information is displayed that describes the change or changes that you must make for the schedule to use available options. Changes to the schedule cannot be saved until the schedule uses available options.
- ☐ If you selected *Run once* for the run interval, the schedule runs immediately unless you change the Start Time to a time later than the current time. All other run intervals run at the next primary run-time interval of the schedule.

- ☐ You must save your changes in order to be able to run from within the scheduling tool.
- ☐ If you want your selected schedule or schedules to run immediately, click *Run*.

Copying a Schedule in the Basic Scheduling Tool

For your convenience, you can copy an existing schedule for use as a template to create a new schedule. The copied schedule is created disabled because it is a duplicate schedule.

Procedure: How to Copy a Schedule in the Basic Scheduling Tool

Note: The cut, copy, and paste options display when you are authorized to use these options.

1. From the Resources tree, right-click the schedule you want to copy and select *Copy*.
2. Right-click the folder you wish to copy the schedule into and select *Paste*.

Deleting a Schedule in the Basic Scheduling Tool

From the Resources tree, you can delete schedules using the following procedure.

Procedure: How to Delete a Schedule in the Basic Scheduling Tool

Note that the Delete option displays when you are authorized to delete the selected schedule.

1. Right-click the schedule you want to delete and select *Delete*.

Note: To select multiple schedules, use the Shift key and Control (Ctrl) key as in a standard Windows interface. A message appears asking to confirm if you want to delete the selected schedules.

2. Click *OK* to delete the schedule or schedules.


Sharing Schedules

The owner of a schedule that is located in a My Content folder can share that schedule with other users.

Procedure: How to Share Schedules

From the Resources tree, you can share one or more schedules by performing the following steps.

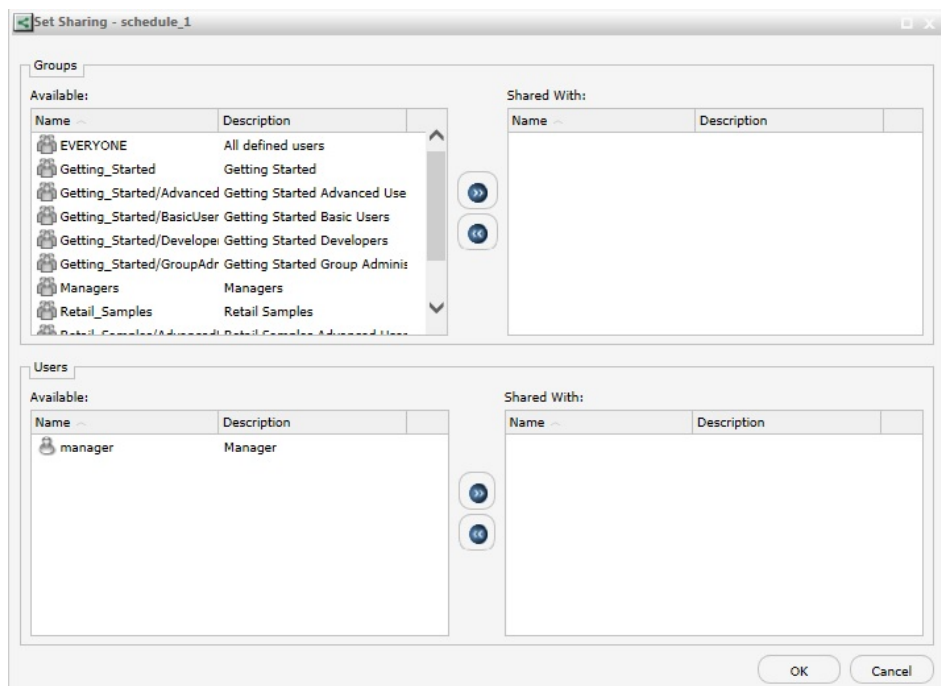
1. In the Resources tree, select one or more Schedules that you want to share.
2. Right-click the selected schedule or schedules and select *Share*.

If you are a group administrator or user with the share schedule privilege, the schedule or schedules are shared. The Schedule icon  changes to denote that it is shared.

Procedure: How to Share Schedules With Specific Groups or Users

From the Resources tree, you can specify the groups or users with whom you want to share the selected schedule or schedules.

1. In the Resources tree, select one or more schedules that you want to share.
2. Right-click the selected schedule or schedules and select *Share with*.



3. Use the left and right arrow buttons to select which Groups will have access to Schedules.
4. Use the left and right arrow buttons to select which Users will have access to Schedules.
5. Click *OK*.

Publishing Schedules

A published schedule is visible to all users with access to the folder in which it resides. The shortcut menu options that appear on the schedule depend upon the privileges of the user that is signed in. For example, a user with the Run privilege is able to run the published schedule. When a published schedule runs, it runs as the creator of the schedule and not as the signed in user that initiated the run.

Note: A published schedule runs as the creator of the schedule. A user that is allowed to edit a published schedule may make a change to the schedule that the creator of the schedule is not permitted to make. In this case, the schedule will fail at run time. For example, a user may change the Distribution List used in the schedule to a private Distribution List that is not available to the creator of the schedule. When the changed schedule runs, it will fail due to inability of the creator to retrieve the Distribution List from the WebFOCUS Repository.

Procedure: How to Publish a Schedule

From the Resources tree, you can publish a schedule by performing the following steps.

1. In the Resources tree, select a schedule that you want to publish.
2. Right-click the selected schedule and select *Publish*.

The schedule is published.

Tracking Schedules

Information about a schedule, such as date, time, execution status, and recipients of a distributed job, can be accessed by running a log report and checking the job status in the ReportCaster Console Job Status tool.

Log Reports

Log reports are stylized HTML format and appear in a separate browser window. You can search, print, or save the log report. The log report displays information according to your specifications in a separate browser window. One log record is produced for each scheduled job run in the specified time frame.

Tracking Schedules in the Console

Information about a schedule, such as date, time, execution status, and recipients of a distributed job, can be accessed by running a log report and checking the job status. For more information, see [Using the ReportCaster Console](#) on page 137.

Using Schedule Logs

Log reports enable you to view information about a distributed job, such as whether or not the job executed successfully, when the scheduled output was distributed, in what format the distributed output was sent, and the method of distribution. Log reports are stylized HTML format and display in a separate browser window. You can search, print, or save the log report. Log reports can be accessed within the scheduling tool when editing a schedule or by right-clicking on a schedule from the Resources tree.

The log file accumulates information. You should periodically purge log records to manage the number of log reports stored in the Repository, as well as the performance of log report information that is displayed.

The list in the right panel provides basic information about the job execution, including the job ID, the time the job started running, the amount of time it took to complete the execution of the job, and the general status of the job. To view a full log report for a job, double-click the job in the job list.

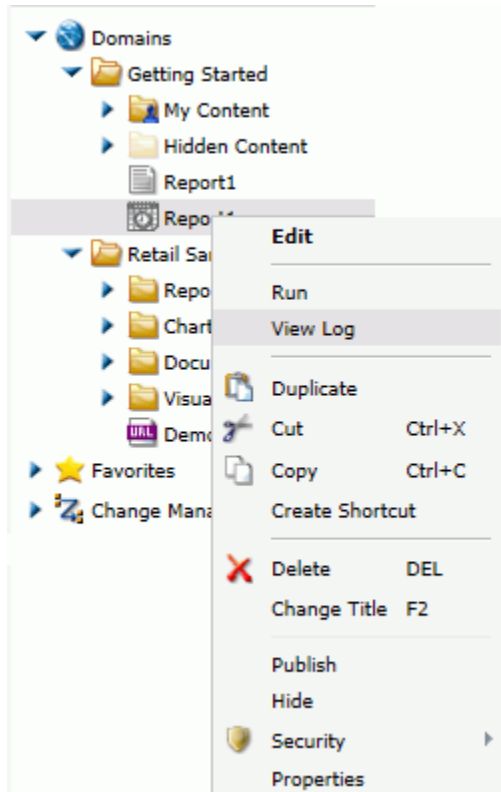
Checking the Job Status

Another resource for tracking schedules is the schedule job status. The schedule status provides a list of scheduled jobs that are in the Distribution Server queue. Status information includes the schedule ID, the time it started running, and the status of the job.

To access the schedule job status information, see the Job Status tab in the ReportCaster Console. For more information, see [Using the ReportCaster Console](#) on page 137.

Procedure: How to View a Log Report From the Resources Tree

1. Right-click the schedule from the Resources tree and select *View Log*, as shown in the following image.



The Schedule Log Options dialog box appears, as shown in the following image.

Schedule Log Options

☒ Last Executed
☐ All
☐ Date

Start Date/Time

Start date: April 7, 2016
Start time: 4: 18 PM

End Date/Time

End date: December 31, 2099
End time: 4: 18 PM

The time zone for the start and end times of the schedule log is Eastern Standard Time (GMT-0400)

OK Cancel

2. Specify which log report you would like to view by selecting *Last Executed*, *All*, or *Date*.
If you select *Date*, you will have the option to specify your search using start date and time and end date and time parameters.
3. Click *OK*.

The log reports that match your search criteria appear, as shown in the following image.

Job Process Log Report

Job Description: Chart2 (IBFS:/WFC/Repository/Getting_Started/Chart2.sch)
User: manager
Procedure: Jca03305cj1869j48dj855dj327d6661ee87
Schedule ID: Sdf5cb5a8si13als4a2asa18cs7bfda42959f2
Start Time: 2016-04-06 05:28:07 PM EDT
End Time: 2016-04-06 05:28:14 PM EDT

Schedule Executed Due To NEXTRUNTIME at YA117742:26030 (YA117742/172.30.234.99)
Job placed in the waiting queue at 2016-04-06 17:28:07.683-0400 (1,459,978,087,683)
Job started running at 2016-04-06 17:28:08.119-0400 (1,459,978,088,119)
Job remained in waiting queue for 0.436 seconds
Starting task: Task 1
Task type: WebFOCUS Report
WebFOCUS Report: /WFC/Repository/Getting_Started/Chart1.fex
Task domain: Getting_Started
Connecting to server EDASERVE with execution id manager at 2016-04-06 17:28:10.503-0400 (1,459,978,090,503)
Executing focexec.
(FOC007) THE REQUEST STATEMENT DOES NOT CONTAIN A VERB
(FOC802) MULTIPLE PARENTAGE IS NOT SUPPORTED FOR THIS OPERATION
Task error: IBFSException 6000: Error executing procedure: Chart1.fex
No report to distribute.
Job finished at 2016-04-06 17:28:13.798-0400 (1,459,978,093,798)
Job time on distribution server after the report completed was 0.013 seconds
Total running time was 5.679 seconds
Total elapsed time (including the queue time) was 6.115 seconds

The log report first lists the job description for the record, which is the unique description identifier that you specified when you created the schedule. Underneath the Job Description, the left column of the log report includes the following information:

- ☐ **User.** ReportCaster user ID, indicating the owner of the schedule.
- ☐ **Procedure.** Unique key generated by ReportCaster that identifies a specific execution of a scheduled job.
- ☐ **Schedule ID.** Unique key generated by ReportCaster that was assigned to the job when it was scheduled.
- ☐ **Start Time.** Date and time the job started running.
- ☐ **End Time.** Date and time the job finished running.

In the second column, the log report specifies messages consisting of the following:

- ☐ General information, such as the method of distribution for a particular job (for example, email distribution).
- ☐ Processing information, indicating that the request started, distribution was successful, and the request was completed. Processing information also includes reasons why a request failed, such as the unavailability of a data source.

Procedure: How to View a Log Report in the Scheduling Tool

1. From the Resources tree, right-click the schedule for which you want to see the logs, and click *Edit*.

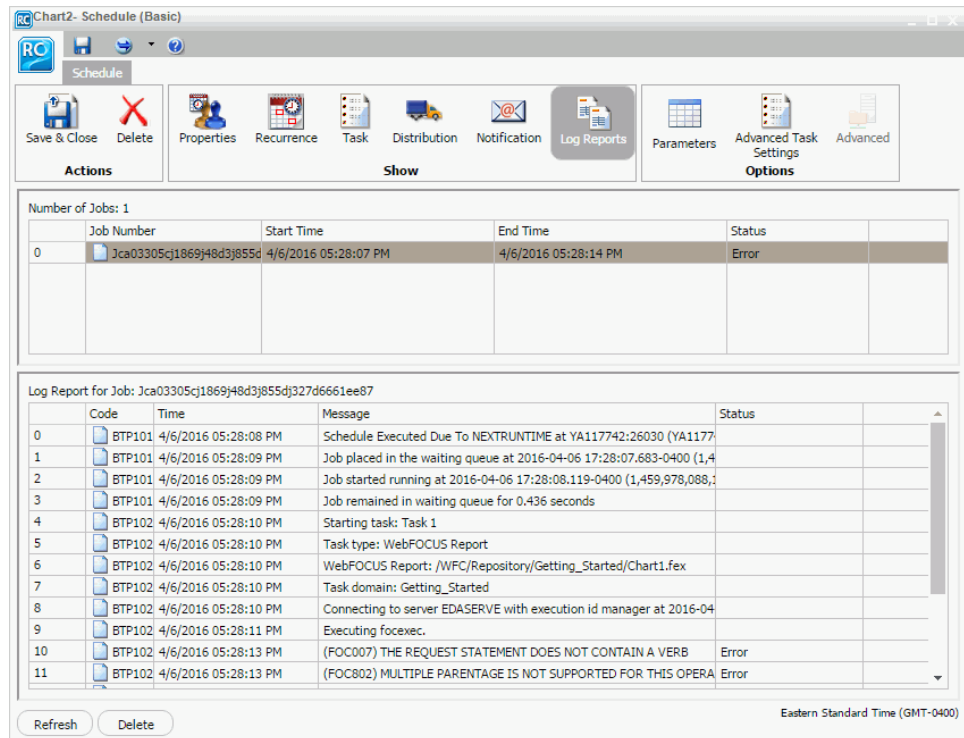
The Basic Scheduling tool opens.

742

Information Builders

- From the Basic Scheduling tool, click the *Log Reports* tab.

The Log Reports panel appears, as shown in the following image.



- Observe the Number of Jobs that have run.
- Click the Job Number to view the log report information for that job in the panel below the job listing.

Reference: Considerations When Viewing a Log Report

When viewing a log report, be aware of the following considerations.

Task and Report Names

The ReportCaster Log references WebFOCUS folders and procedures (FEXs) by their path and file names and not their descriptions.

Email Addresses

ReportCaster cannot validate email addresses since email validation is performed by the mail server. The log report will include any email addresses validated by the mail server and returned to ReportCaster.

Burst Reports

- ❑ If a valid burst value is omitted in a Distribution List, Distribution File, or Dynamic Distribution List, ReportCaster treats the blank value as if it is a valid burst value and no entries indicating a blank burst value appear in the log file. This will significantly reduce the size of the log file, particularly when the database contains many values for the primary field and only a small subset of those values are burst.

- ❑ If a burst value is specified in a Distribution List, Distribution File, or Dynamic Distribution List and it is not found in the database, the following message appears in the log file:

Burst Value: value is not in the database.

- ❑ When a report procedure (FEX) is successfully burst, the log file will include the following message for each burst value:

FILE filename SUCCESSFULLY DISTRIBUTED TO destination FOR burst value.

Unavailable Options

- ❑ When schedules with unavailable task types or distribution methods are not permitted to run, an error notification is triggered. The error is shown in red text within the job process log report. The log report, as well as the full and brief notifications, contains information on the unavailable options that the owner of the schedule must change.
- ❑ When schedules with unavailable task types or distribution methods are permitted to run, normal job execution occurs and a message appears in the log report indicating that existing schedules using the unavailable task types or distribution methods are able to run. For more information on configuring available task types and distribution methods, see [Configuration Tab Folders](#) on page 147.

Using Favorites

You can further customize your Personal Portal, by using the Favorites feature. When you save content as a Favorite, it automatically appears in the Favorites node in the Resources tree. Here, you can interact with your content and view its properties.

Procedure: How to Add Content to Favorites

1. In the Resources tree, right-click a report, chart, document, visualization, shortcut, or URL, and then click *Add to Favorites* .

The item is added to the Favorites node, as shown in the following image.



2. Right-click the item inside the Favorites node to access the shortcut menu for this item.

***Procedure:* How to Remove a Favorite**

1. In the Resources tree, under the Favorites node, right-click an item.
2. Click *Remove Favorite*.

The item is removed from the Favorites node.

Publishing Content

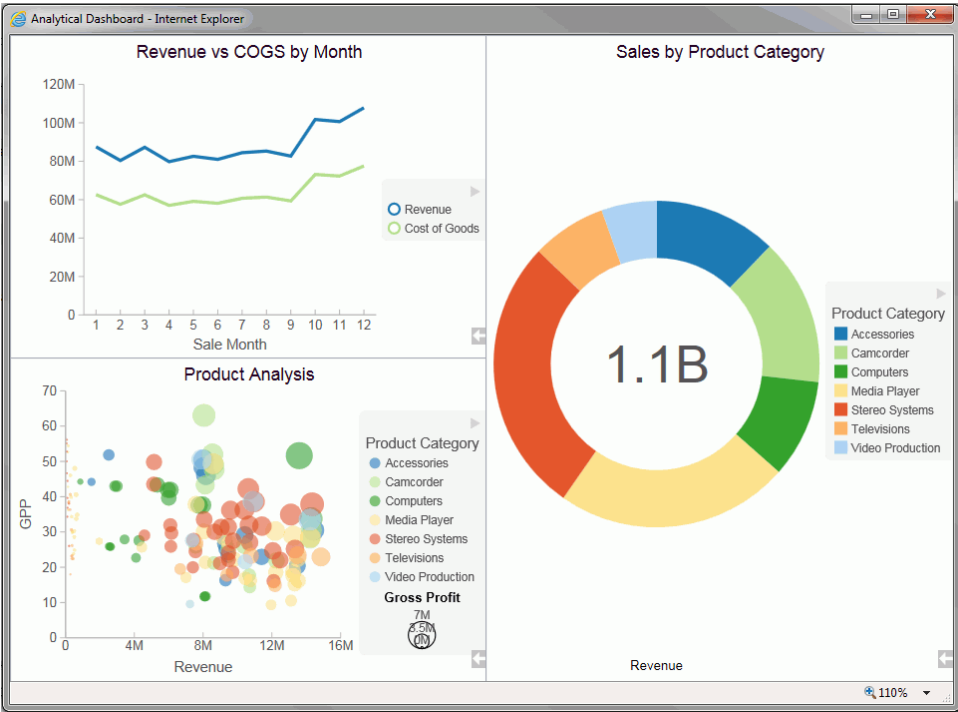
When you create a new item (for example, folder, report, or chart), it is private. Only you can see it, run it, edit it, and so on. A private item is illustrated with a gray icon.

When you are ready to allow others to access the item you can publish it. This turns on any rules set on the item itself or inherited from a higher level. The icon of the item appears in full color to indicate the item is live.

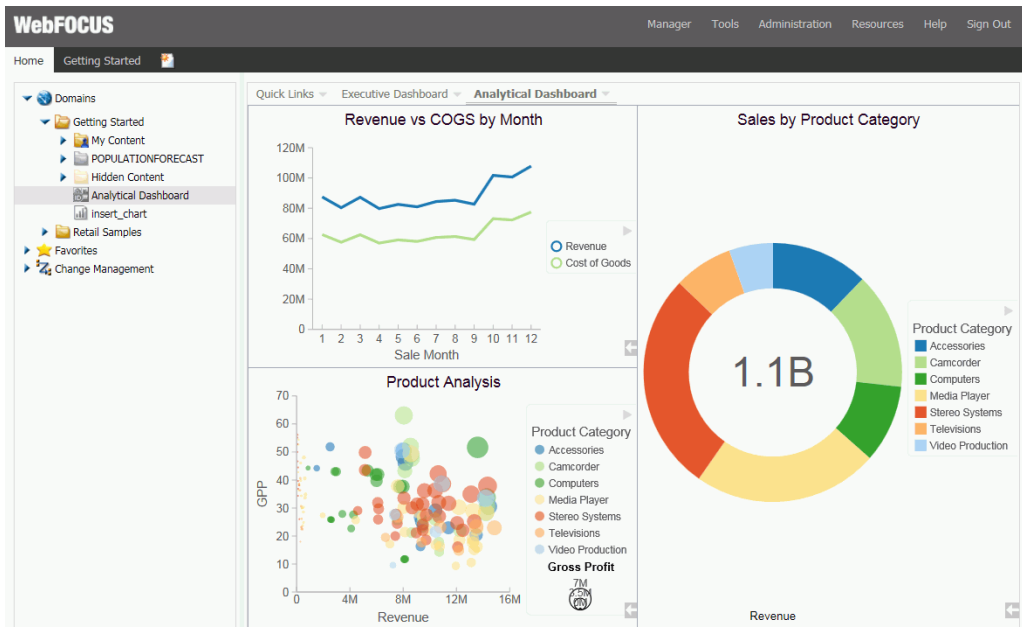
Managers and developers have the ability to publish items. Once the item is published, it becomes visible to all users.

Using Visualizations at Run-Time

In InfoAssist+, you can run visualizations that are either saved or run dynamically at run time. In this case, the visualization opens in a separate browser window, as shown in the following image.



In the WebFOCUS BUE portal, all users can run stored visualizations that are available to them. In this case, the visualization opens in a new page, as shown in the following image.

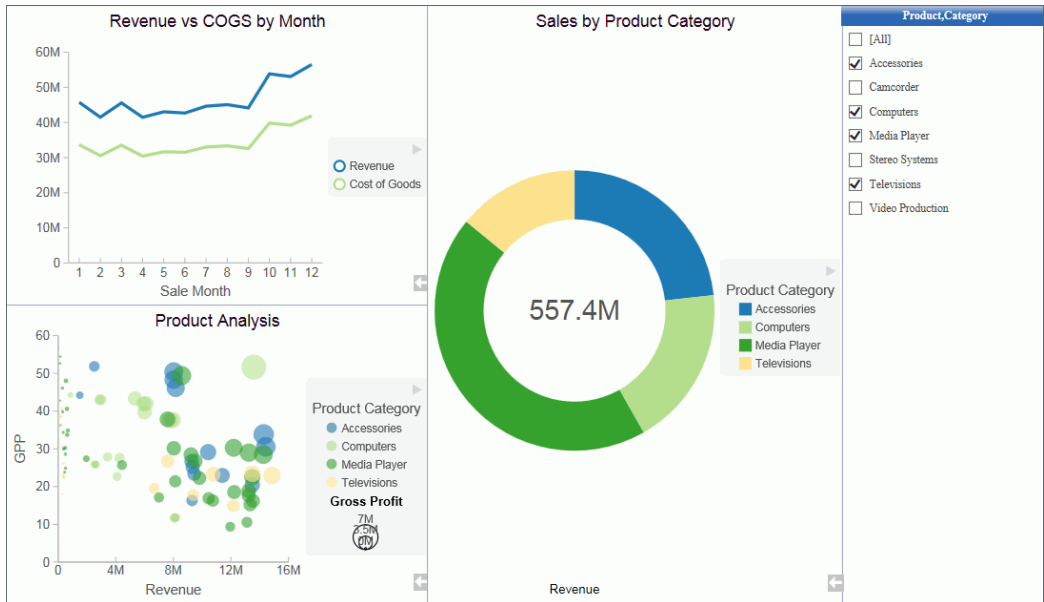


Working With Filters at Run Time

If you have added a filter to your visualization, you can use the Prompts panel to control the display of information. When you run your visualization, the Prompts panel appears. This enables you to select additional components by which to filter the visualization.

Note: The filters that you apply at run time can be removed using the Remove Filter option in the Run-Time toolbar. This does not apply to filters that were added prior to running the visualization.

The following image shows an example of a visualization with an open Prompts panel.

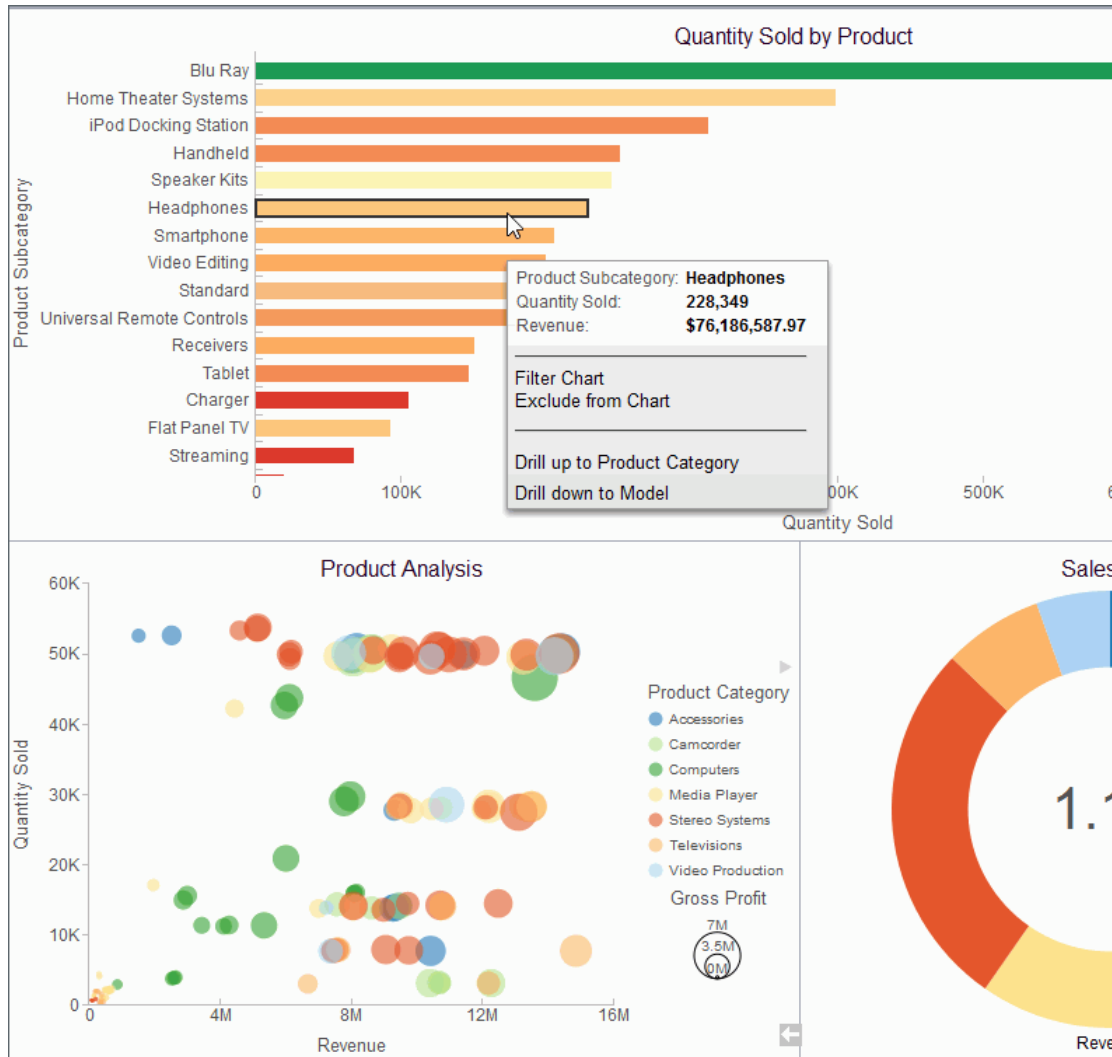


Note: If you select (or lasso) components of your visualization, the Prompts panel does not automatically update to reflect your selections.

Procedure: How to Filter Data With Tooltips at Run Time

1. Run a visualization from one of the following locations:
 - ☐ From the Resources tree in the WebFOCUS BUE portal.
 - ☐ From the Application Main Menu or Quick Access Toolbar in InfoAssist+.
2. Click, point to, or lasso data in your visualization.

A tooltip opens, as shown in the following image.



The tooltip includes the following options:

- ☐ **Filter Chart.** Filters the visualization by the data values that you have selected.
- ☐ **Exclude from Chart.** Excludes the selected data values from the visualization.
- ☐ **Remove Filter.** Removes the filter and returns the visualization to its original state.


Note: The same run-time options are available when you run your visualization on a mobile device.

Viewing Data at Run Time

When viewing data at run time, you can access the data behind each visual in your visualization. Using the Show Data option, which is available from the Run-Time toolbar, you can view your data in summary format. This grid display includes totals for the categories in your visual, based on the data fields that you selected. All InfoAssist+ users can use this option with the visualizations that are available to them.

Procedure: How to View Data at Run Time

- 1. Run a visual or visualization from one of the following locations:
 - ❑ From the Application Main Menu or Quick Access Toolbar in InfoAssist+.
 - ❑ From the Resources tree in the WebFOCUS BUE portal.
- 2. In the bottom-right corner of the visual cell, click the arrow.
The Run-Time toolbar opens.

- 3. Click the *Show Data* button .
The visual changes to the grid view, displaying the underlying data for the selected visual.
Click the *Show Data* button again to return to your visual.




Note: The same run-time options are available when you run your visualization on a mobile device.

Using the Run-Time Toolbar Options

You can access the Run-Time toolbar by clicking the arrow button, which is located in the bottom-right corner of each visual cell at run time. The Run-Time toolbar is shown in the following image.



The Run-Time toolbar options are described in the following table:

Icon	Name	Description
	Show Data	Alternates between the visual and the grid view, which shows the underlying data. To return to the visual view, click the Show Data button again.
	Reset to Default	Resets all changes and returns the visualization to its original state.
	Remove Filter	Removes the filter and returns the visualization to its original state. This option is only available if a filter is applied at run time.

Note: The same run-time options are available when you run your visualization on a mobile device.

Using the Deferred Report Status Interface

The following topics provide an overview of the Deferred Report Status Interface including a detailed description of its appearance and functions. Specific procedures guide you through viewing, saving and deleting reports, deleting deferred reports that are being processed but are not yet complete, and reviewing parameters for reports containing amper variables.

The Deferred Report Status Interface enables you to obtain information about deferred reports. From this Interface, you can perform the following actions on a deferred report:

- ☐ Sort deferred report output by date, description, domain, and server ID.
- ☐ View deferred report output.
- ☐ Delete a deferred report from the WebFOCUS Reporting Server.
- ☐ Save the report output as private content.
- ☐ Review or change parameters associated with a deferred report.
- ☐ View the number of days remaining prior to expiration (deletion) on the server.
- ☐ Terminate a deferred request that is in the deferred report queue.
- ☐ Terminate a deferred report that is running.
- ☐ Delete all expired, unknown, completed, running, and queued tickets.

You can access the Deferred Status Report Interface from the BUE portal, by clicking *Tools*, in the Menu bar, and then clicking *Deferred Status*.

Deferred Report Status Interface Features


The Deferred Report Status Interface includes:

- ☐ A banner at the top of the window that lists the date and time of the request.
- ☐ A gray toolbar below the banner that contains Refresh and Help options, a Sort By drop-down list to select sort values, a sort order button to toggle between ascending and descending order, and a Delete drop-down list. The Delete drop-down list has options to delete All, All Completed, All Running, All Queued, All Expired, or All Unknown reports, depending on which report status types exist in the Interface.
- ☐ The status of each report within the Interface.

Sort Controls for the Deferred Report Status Interface

The sorting feature pertains to the entire report. When the default sort value (Date/Time Submitted) is changed, the new primary sort becomes your choice, but the secondary sort is always fixed as Date/Time Submitted. To resort the list, select the Sort by option:

- ☐ Date (default)
- ☐ Description
- ☐ Domain

You can optionally change the sort order (ascending or descending) by clicking the Reverse Sort Order  button, which toggles between A to Z and Z to A.

Note: When the sort value is Date, the sort order option A to Z means from new to old and not alphabetical from A to Z.

To see the results of the new sort options, click *Refresh*.

Deferred Report Status


The status of deferred requests are organized under the following sections within the Interface:

- ☐ **Completed.** Indicates that the Deferred Receipt request has finished processing.
- ☐ **Running.** Indicates that the Deferred Receipt request is processing.


- ❑ **Queued.** Indicates that the Deferred Receipt request is queued for processing.
- ❑ **Unknown.** Indicates that the Deferred Receipt request cannot be identified. This can occur when the file containing the deferred report results cannot be found. For more information, see [Deferred Report Expiration Setting](#) on page 756.


The following image shows a sample Deferred Report Status window with one completed report and two queued repots.

Deferred Report Status as of 4/11/2016, 6:23:00 PM

Refresh

Sort By

Date

Delete

Help

Refresh every seconds. (min. 5 seconds) Enable Refresh: ☐

Completed

Date/Time Submitted	Domains	Description	Expires In	Options
4/11/2016, 6:22:56 PM	Retail Samples	Margin by Product Category	29 days	<div>Delete</div> <div>View</div> <div>Save</div> <div>Run</div>
4/11/2016, 6:22:38 PM	Retail Samples	Regional Analysis	29 days	<div>Delete</div> <div>View</div> <div>Save</div> <div>Run</div>
4/11/2016, 6:22:21 PM	Retail Samples	Choropleth Map - Sales by State	29 days	<div>Delete</div> <div>View</div> <div>Save</div> <div>Run</div>
4/11/2016, 6:21:57 PM	Retail Samples	Bar - Highest Margin Products	29 days	<div>Delete</div> <div>View</div> <div>Save</div> <div>Run</div>

Column headings provide information about the published content including the date and time the published content was submitted, the domain of origin, a description of the report (the report name), an expiration indicator, and an Options heading for options within the Deferred Report Status Interface.

When you select the Deferred Status option, the status for all the deferred requests submitted by your Managed Reporting user ID is retrieved. Depending on how Managed Reporting is configured, deferred status may be coming from multiple WebFOCUS Reporting Servers on various platforms. If credentials are required for the connections, you are prompted by the WebFOCUS Dynamic Server System Signon feature. You can view the status of all the deferred requests submitted by your Managed Reporting user ID, but can only delete, view, save, stop, or review parameters for deferred requests submitted with an identical WebFOCUS Reporting Server user ID.

The options available in the Deferred Report Status Interface are based upon the status of the report request and security validation. You can perform various functions by clicking the buttons under the following options.

For more information on configuring Deferred Report Status options and buttons, see *Technical Memo 4719: Managed Reporting - Deferred Status Interface How to Hide Deferred Report Options (Buttons)*.

☐ **Delete.** Available for all report status types. The Delete option deletes the deferred request according to the report status, as follows:

- ☐ **Queued.** When a deferred request is listed in the Queued tab, the Delete option removes the deferred report from the WebFOCUS Reporting Server and deletes the deferred request ticket from the WebFOCUS Repository.
- ☐ **Unknown.** When a deferred request is listed in the Unknown tab, the Delete option deletes the deferred request ticket from the WebFOCUS Repository.
- ☐ **Completed.** When a deferred request is listed in the Completed tab, the Delete option removes the report from the window, deletes the deferred report results from the WebFOCUS Reporting Server, and deletes the deferred request ticket from the WebFOCUS Repository.
- ☐ **Running.** When a deferred request is listed in the Running tab, the Delete option deletes the deferred request ticket from the WebFOCUS Repository and cancels the job on the WebFOCUS Reporting Server.

Note: The Delete drop-down list in the toolbar at the top of the Interface provides options to delete All, All Completed, All Running, All Queued, All Expired, or All Unknown reports, depending on which report status types exist in the Interface.

☐ **View.** Available when the report status is Completed.

The View option displays the completed report in a new browser session, or the report format may result in the opening of a Windows dialog box that prompts you to save the report to disk or open the report within an application (such as Microsoft(R) Excel(R), Microsoft(R) Word, or Adobe(R) Acrobat(R)).

Note: Auto Drill functionality is not available for reports that are run deferred because Auto Drill uses live data, in an interactive session, for data drilling. Data values and totals may not be the same if the data has changed since the last deferred execution. Mixing past data with current data could impact the data analysis.

☐ **Save.** Available when the report status is Completed.

The Save option allows you to save Deferred Receipt reports to the WebFOCUS Repository, if you are authorized to save deferred reports and to create private content. You can save the report output to your My Content folder or a folder you are authorized to create content in and write to. When your deferred report is saved to the WebFOCUS Repository, it is removed from the Deferred Report Status Interface.

Note: This option appears for users authorized to save deferred reports.

- ☐ **Run.** Available for reports without parameters when the report status is completed or queued. The Run option runs the report deferred again.

Note: Auto Drill functionality is not available for reports that are run deferred because Auto Drill uses live data, in an interactive session, for data drilling. Data values and totals may not be the same if the data has changed since the last deferred execution. Mixing past data with current data could impact the data analysis.

- ☐ **Parameters.** Available for reports with parameters when the report status is completed or queued. The Parameters option allows you to review or change report variables. Changing report variables generates a new report that does not overwrite the original request.




Note: Deferred reports run from within any report development tool do not have an option to view or change parameter variable values in the Deferred Status Interface window.

Under certain circumstances, WebFOCUS is unable to submit the request to run in deferred mode. This can occur, for example, when the WebFOCUS Reporting Server is unavailable. When WebFOCUS is unable to submit a deferred request, a Deferred Receipt Notification window opens, notifying you of the failure.

Deferred Report Expiration Setting

The number of days until expiration appears next to each report. On the last day, the value Today appears.

The following image shows the results of a deferred status request, run on the afternoon of Tuesday, March 15. (The current date appears in the status bar at the top of the page.) Each report is listed with the time remaining before it is deleted from the WebFOCUS Reporting Server. The time remaining is based on 24-hour intervals (rather than whole days), beginning with the time that the report was submitted. For example, the last report shown on the list will be deleted shortly after 3:51 pm on March 14, not at midnight on March 13.

Deferred Report Status as of 3/15/2016, 4:07:35 PM				
Refresh 	Sort By  Date	Delete 	Help	
Refresh every <input type="text"/> seconds. (min. 5 seconds) Enable Refresh: <input type="checkbox"/>				
Completed				
Date/Time Submitted	Domains	Description	Expires In	Options
3/15/2016, 3:51:52 PM	Retail Samples	Margin by Product Category	29 days	Delete View Save Run
3/15/2016, 3:51:34 PM	Retail Samples	Choropleth Map - Sales by State	29 days	Delete View Save Run

If a deferred report is not saved or deleted prior to its expiration, the output is automatically deleted from the WebFOCUS Reporting Server dfm_dir directory and the deferred report is moved to the Unknown status tab in the Deferred Report Status Interface. From here, you can only delete the orphaned report.

If deferred output expiration is not configured on your WebFOCUS Reporting Server, then the value never appears next to each report under the Expires In column.

Note: This setting does not affect deferred output saved to your private content area.

Saved Deferred Output Subject to Temporary Expiration

Saved Deferred Reports that utilize WebFOCUS features that create temporary files, such as OLAP, On-demand paging, and redirected formats are subject to expiration as defined by the WebFOCUS Client parameter, EXPIRE_REPORTS (located in cgivars.wfs).

Setting the Automatic Refresh Interval

You can set the automatic refresh interval to any value. The default is 5 seconds and there is no maximum value.

Procedure: **How to Set the Automatic Refresh Interval**

1. Enter a time interval (in seconds) in the input box below the gray toolbar.
The default value is 5 seconds. There is no maximum value.

2. Check the box to enable automatic refresh.

Viewing Deferred Reports

You must access the Deferred Report Status Interface to view deferred reports.

Procedure: How to View a Deferred Status Report

1. Open the Deferred Report Status Interface.
2. To view the output of a deferred report:
 - a. Locate the report description under the Completed tab.
 - b. Click *View*, under the Options column, to view the report.

The output appears in a new window.
3. The Deferred Report Status Interface remains open until closed.
 - a. To return to the Deferred Report Status Interface, close or minimize the report output window.
 - b. To return to your reporting environment, close or minimize the report output window, then close the Deferred Report Status Interface.
4. Click *Refresh* to obtain the most current status of deferred requests.

Reviewing Deferred Report Parameters

The Deferred Report Status Interface enables you to retrieve parameters submitted with a deferred request. You access parameters by opening the Deferred Report Status Interface and clicking the parameters button for the report of your choice. The parameters button is not available when the deferred request is submitted from within a report development tool, such as InfoAssist.

You can also change the parameters associated with a report and submit the report to run deferred with the new parameters you specified. WebFOCUS generates your report again using the new parameters and does not overwrite your original report request.

Procedure: How to Retrieve Deferred Request Parameters

1. Open the Deferred Report Status Interface.
2. In the Completed or Unknown tabs, identify the report containing the parameters to review.

3. Click *Parameter* under the Options column heading.

An intermediate window (HTML form) opens.

- a. To review and accept the original parameters, close the browser window.
- b. To change the parameters, enter a new value in the input box.

The original request runs in addition to the newly submitted request.

4. Click *Submit*.

The Deferred Report Notification window opens.

5. Close the Deferred Report Notification window to return to the Deferred Report Status Interface.

Saving Deferred Reports

You can save Deferred Receipt reports to the WebFOCUS Repository, if you are authorized to save deferred reports. You can save the report output to your My Content folder or a folder you are authorized to create content in and write to. When your deferred report is saved to the WebFOCUS Repository, it is removed from the Deferred Report Status Interface. .

Procedure: How to Save a Deferred Report

1. Open the Deferred Report Status Interface.
2. Under the Completed tab, locate the report you want to save.
3. Under the Options column, click *Save*, which is located to the right of the deferred report description.

The Save File Content dialog box opens.

4. Navigate to your My Content folder or another folder you are permitted to create content in and write to.
5. Click *Save* to save the deferred report results.

To return to your reporting environment, close the Deferred Report Status Interface.

Deferred Status Delete Confirmation Messages

The Deferred Status Interface presents the user with a delete confirmation message before deleting a deferred report that is in Completed, Running, or Queued states. (A confirmation message is already displayed for deferred reports in Unknown status.)

When you click the delete button from the Deferred Status page, you are prompted to confirm the delete before the deferred report is actually deleted. A similar confirmation message is used for all deferred reports, but the message varies depending on the conditions.

The following are the confirmation messages and the associated conditions:

- ❑ If the report is expired or was deleted from the server, the message recommends deletion and indicates that there is no report output on the specific WebFOCUS server.
- ❑ If the WebFOCUS server is unavailable, the message indicates there is an error attaching to the specific WebFOCUS server.
- ❑ If there is no entry for the server in the WebFOCUS client configuration, the message recommends deletion and indicates that the specific WebFOCUS server is not defined in the WebFOCUS client configuration file.

Each of the deletion confirmation messages also displays the date and time the deferred report was submitted, and the description that is displayed in the Deferred Status Interface.

Troubleshooting

Learn how to troubleshoot all areas of WebFOCUS Business User Edition.

In this chapter:

- ☐ [Troubleshooting WebFOCUS Business User Edition](#)
 - ☐ [Monitoring WebFOCUS Sessions](#)
 - ☐ [Exporting the Session Monitor Log](#)
 - ☐ [Viewing WebFOCUS Sessions](#)
 - ☐ [Working With Log Files](#)
 - ☐ [Understanding Warning Messages in InfoAssist+](#)
-

Troubleshooting WebFOCUS Business User Edition

This topic describes how to troubleshoot the WebFOCUS BUE installation.

Starting the Reporting Server Service

If you receive a message during the installation process related to failure to start the Reporting Server service, perform either of the following steps after the installation completes:

- ☐ Start the *WebFOCUS BUE 82 Reporting Server* service from the Windows Services.
- or
- ☐ Use the *Start WebFOCUS_BUE Services* program shortcut.

Starting WebFOCUS BUE

If you are unable to launch WebFOCUS BUE, ensure that all services are running. If they are not, do the following:

- ☐ Stop all services.
- ☐ Restart the services, and launch WebFOCUS BUE.

If this does not work, stop the services again. Before you restart the services, ensure that the Hyperstage processes, *ibengine.exe* and *postgres.exe*, are also stopped. Restart your machine if you are still unable to restart the services.

Port Assignment

By default, ports in the range of 26000 to 26040 are checked for availability. If the installation program detects that no ports in that range are available, it then increases the range by 10 and checks again for availability. For example, if ports 26000 to 26040 are unavailable, ports 26010 to 26050 are then checked.

Default Port Assignment.

- ☐ Ports 26000 to 26003 are used by the application server.
- ☐ Port 26010 is used by the repository server.
- ☐ Ports 26020 to 26023 and port 26040 are used by the Reporting Server
- ☐ Port 26030 is used by the Distribution Server.

WebFOCUS Business User Edition Log Files

WebFOCUS BUE creates a log file in the following location for Windows 7:

drive:\Users\user_id\WebFOCUS_BUE82_Install_date_time.log

where:

user_id

Is your Windows user ID.

date_time

Is the date and time the log file was created. This log file provides information about the WebFOCUS BUE installation. If you contact Customer Support Services with an installation problem, have this file available.

Troubleshooting the Uninstall Process

If you choose to uninstall WebFOCUS BUE, and the uninstall process fails, you can follow one of the procedures in this section to clean up your machine before reinstalling the product.

Before performing one of the following procedures, ensure that you have uninstalled WebFOCUS BUE.

Procedure: How to Manually Uninstall WebFOCUS Business User Edition Components Using the Cleanup Utility

If the WebFOCUS BUE uninstall process fails, you can run the `idis_cleanup.bat` file to clean a damaged installation on your machine.

1. Navigate to the following location on your machine:

`drive:\ibi\WebFOCUS_BUE82\WebFOCUS\utilities\install\`

2. Copy the `idis_cleanup.bat` file from the install directory to a different directory (for example, `drive:\ibi\`).
3. Right-click `idis_cleanup.bat`, and then click *Run as administrator*.
4. When prompted, type the location of the install directory to proceed.

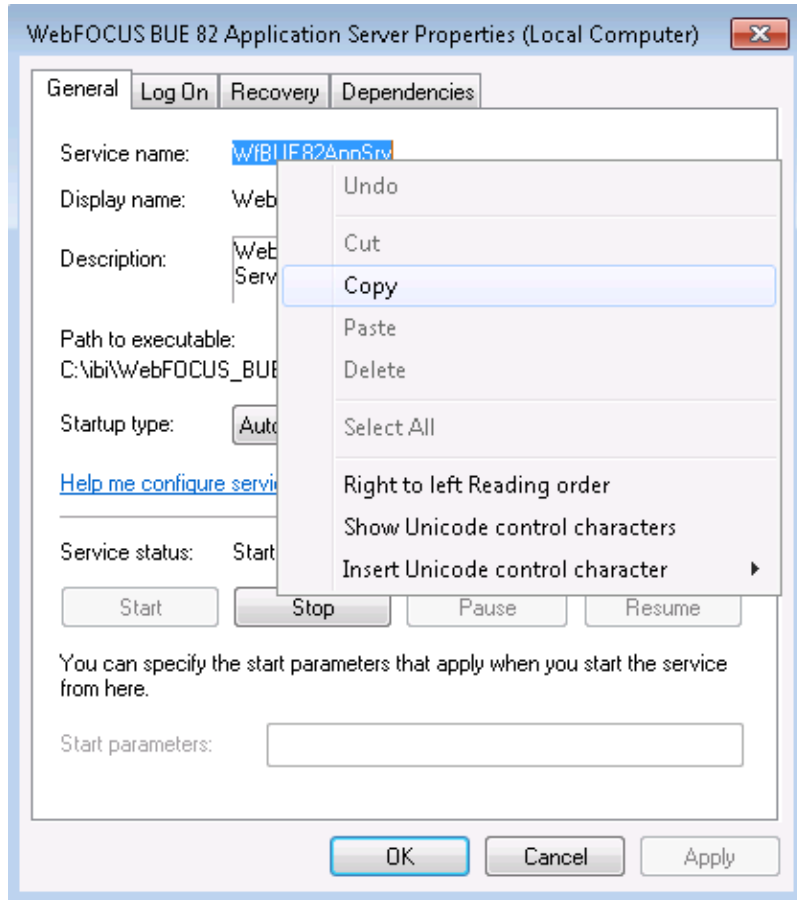
Note: By default, the utility will back up the Reporting Server apps and the Derby database.

Procedure: How to Manually Uninstall WebFOCUS Business User Edition Components

If the WebFOCUS BUE uninstall process fails, you can do the following to manually cleanup the remaining WebFOCUS BUE components on your machine.

1. Stop any running WebFOCUS BUE services on the system

2. Remove any remaining services from your machine.
 - a. Locate the existing WebFOCUS BUE services. To get the service name, right-click the service in the Windows Services program and click *Properties*. The Properties dialog box opens, where you can right-click the service name and click *Copy*, as shown in the following image.



- b. To remove a service, open the Command Prompt as an Administrator and issue the following command:

```
sc delete "service name"
```

where:

service name

Is the name of the service you are manually deleting. For example:

```
C:\sc delete "WfBUE82AppSrv"
```



Note: The service name must be enclosed in double quotation marks.

Repeat this for the remaining services.

3. Open Task Manager and ensure that the process *ibengine.exe* is not running.
4. Remove files from disk, except for the Reporting Server application folders.
For example, if WebFOCUS BUE is installed on the C:\ drive, remove all folders under C:\ibi\WebFOCUS_BUE82\ except for the C:\ibi\WebFOCUS_BUE82\data\ folder.
5. Click *Start*, point to *All Programs*, and expand the *Information Builders* folder.
6. Right-click the *WebFOCUS Business User Edition 82* folder and click *Delete*.
7. Launch the Windows Registry Editor, and remove the following registry keys.
 - ☐ HKEY_LOCAL_MACHINE\SOFTWARE\WebFOCUS_BUE82
 - ☐ HKEY_LOCAL_MACHINE\SOFTWARE\Information Builders\Derby\WfBUE82DbSrv
 - ☐ HKEY_LOCAL_MACHINE\SOFTWARE\Wow6432Node\Apache Software Foundation\Procrun 2.0\WfBUE82AppSrv
 - ☐ HKEY_LOCAL_MACHINE\SOFTWARE\Information Builders\WebFOCUS BUE 82 Reporting Server
 - ☐ HKEY_LOCAL_MACHINE\SOFTWARE\Wow6432Node\Information Builders\ReportCaster\WfBUE82DistSrv

Monitoring WebFOCUS Sessions

The Session Monitor enables Managers to track all client sessions, as well as connections and activity on the Reporting Server. The Session Monitor displays information about connected users, report requests, and Reporting Server nodes, as shown in the following image.

Session Monitor  Refresh interval: 10 second(s)								
URL Logging Level is currently set to None								
Current Session Number: 22								
IP address	Mode	Client User	URL Logging	Trace Control	Trace FEX	URL # / AVG / Max	Server # / AVG / Max	W/ DBMS # / AVG / Max
172.30.234.147	WEB	admin	OFF	Off		357 / 0.031 / 2.073	0 / 0.000 / 0.000	0 / 0.000 / 0.000
172.30.238.163	WSRV	admin	OFF	Off		168 / 0.048 / 0.894	37 / 0.126 / 0.601	10 / 0.001 / 0.001
Last req:EDASERVE/ (10.39.47) -> Run:Fex=METAQUERY-APPLIST								
172.30.244.180	WEB	admin	OFF	Off		561 / 0.048 / 2.858	8 / 0.578 / 1.288	2 / 0.001 / 0.001
Last req:EDASERVE/ (10.38.37) -> Run:Domain=Peter/:SubAct=MR_STD_REPORT:Fex=visuali								
172.30.234.52	WEB	admin	OFF	Off		211 / 0.011 / 0.211	0 / 0.000 / 0.000	0 / 0.000 / 0.000
172.19.21.75	WEB	admin	OFF	Off		306 / 0.017 / 0.775	1 / 0.231 / 0.231	1 / 0.001 / 0.001
Last req:EDASERVE/ (09.58.37) -> Run:Fex=METAQUERY-APPLIST								
172.30.234.78	WEB	admin	OFF	Off		120 / 0.020 / 0.288	0 / 0.000 / 0.000	0 / 0.000 / 0.000
172.30.238.84	WEB	admin	OFF	Off		705 / 0.074 / 5.482	42 / 0.781 / 5.413	13 / 0.001 / 0.005
Last req:EDASERVE/ (09.52.16) -> Run:Domain=MithuD/:SubAct=MR_STD_REPORT:Fex=population_density								
172.30.238.74	WEB	admin	OFF	Off		731 / 0.044 / 3.908	28 / 0.769 / 3.835	12 / 0.001 / 0.001
Last req:EDASERVE/ (09.49.36) -> Run:Fex=c1che002								
172.30.234.175	WEB	admin	OFF	Off		93 / 0.015 / 0.180	0 / 0.000 / 0.000	0 / 0.000 / 0.000
172.30.238.83	WEB	admin	OFF	Off		872 / 0.013 / 1.143	11 / 0.442 / 1.034	3 / 0.001 / 0.001
Last req:EDASERVE/ (09.34.42) -> Run:Domain=Public/:SubAct=MR_OLAP:Fex=ADHOCRCQ								
172.19.23.135	WEB	admin	OFF	Off		701 / 0.017 / 1.442	11 / 0.517 / 1.371	5 / 0.001 / 0.001
Last req:EDASERVE/ (09.25.49) -> Run:Domain=samantha/:SubAct=MR_OLAP:Fex=ADHOCRCQ								
172.30.238.81	WEB	admin	OFF	Off		93 / 0.017 / 0.224	0 / 0.000 / 0.000	0 / 0.000 / 0.000
172.19.21.160	WEB	admin	OFF	Off		200 / 0.020 / 0.280	0 / 0.000 / 0.000	0 / 0.000 / 0.000
172.19.21.111	WEB	public	OFF	Off		1 / 0.008 / 0.008	0 / 0.000 / 0.000	0 / 0.000 / 0.000
172.19.21.111	WEB	public	OFF	Off		1 / 0.058 / 0.058	1 / 0.042 / 0.042	0 / 0.000 / 0.000
Last req:EDASERVE/ (09.01.39) -> Run:Fex=carinst								
172.19.23.26	WEB	admin	OFF	Off		807 / 0.009 / 0.512	19 / 0.092 / 0.167	1 / 0.001 / 0.001
Last req:BIGSCMRH/admin (09.00.34) -> Run:Domain=Public/:SubAct=MR_OLAP:Fex=ADHOCRCQ								
172.19.22.117	WEB	admin	OFF	Off		1,056 / 0.054 / 7.857	23 / 1.995 / 7.653	15 / 0.726 / 2.815
Last req:EDASERVE/ (08.48.55) -> Run:Fex=c1che006								

To refresh the information displayed, click the *Refresh* icon.

Managers can enable or disable logging for all current sessions by clicking *All/None/Selective* next to *URL Logging Level is currently set to*. To enable or disable logging for individual sessions, click *Selective*, and then *On* or *Off* under the URL Logging column for the individual sessions. By default, all log information is located in the *drive:/ibi/WebFOCUS_BUE82/WebFOCUS/logs* directory.

For each session, the following information is available:

IP Address

The numerical label assigned to the computer or other device that initiated the session.

Using this address, you can identify the user assigned to the computer or other device that initiated the session.

Mode

Identifies the product component that started the session and provides information about all active requests. The product component values are as follows:

WEB

Specifies the Client.

WSRV

Specifies the Reporting Server.

Client User

Specifies the user ID that started the client session. A value of null indicates that it is a request from a self-service application.

URL Logging

Enables or disables logging for an individual session or a current user.

Trace Control

Enables or disables tracing for a specific IP Address, that is, user.

Trace FEX

Enables or disables WFServlet, Client Connector, and Reporting Server traces for each session. If tracing is enabled, click the *View Trace* icon to see the trace.

URL

Specifies the number, average duration, and maximum duration of dynamic URLs sent in HTTP requests. Duration is measured in seconds, calculated to the millisecond. Not all URLs in HTTP requests are forwarded to the server, and not all requests forwarded to the server are then forwarded to the DBMS.

Server


Specifies the number, average duration, and maximum duration of dynamic URLs that run reports on the Reporting Server. Duration is measured in seconds, calculated to the millisecond. Not all URLs in HTTP requests are forwarded to the server, and not all requests forwarded to the server are then forwarded to the DBMS.

W/DBMS

Specifies the number, average duration, and maximum duration of dynamic URLs that run reports against an external database. Duration is measured in seconds, calculated to the millisecond. Not all URLs in HTTP requests are forwarded to the server, and not all requests forwarded to the server are then forwarded to the DBMS.

Exporting the Session Monitor Log

You may need to export a session monitor log for troubleshooting.

- 1. In the Administration Console, click the *Diagnostics* tab, then click *Session Monitor*.
Current sessions appear in the right pane.
- 2. Set the Trace Control option for the chosen session to Details.
Information icons now appear in the Trace FEX column.
- 3. If necessary, run a request for logging, then return to the Administration Console.
- 4. Click the *View Trace* icon .

The Session Viewer appears, as shown in the following image.

manager Session Viewer 160317/manager_110210

User Name	Tracing Level	Fex Echo Setting	SQL Tracing	Viewing File	Session Started	Last Activity	URL Count	Average Response	Server Requests	Average Server Time	Average DBMS Time
manager	Details ▾	<input checked="" type="radio"/> Off <input type="radio"/> On <input type="radio"/> All	<input checked="" type="radio"/> Off <input type="radio"/> On	11.07-* New File	11.02.10	11.07.55	63	0.037	1	0.127	0.000

[Refresh](#) [Previous](#) [Next](#) [Last](#)

View: Server Requests ▾ In groups of: 5 ▾ HighLight:

[ZIP Session Traces](#)

No trace lines for:Server Requests

Date	Existing session traces
160317	manager_101320 manager_110210

- 5. Click on one of the *Zip Session Traces* links and save the zip file.
- 6. Close the Session Viewer and return to the Session Monitor pane.
- 7. Set Trace Control to its previous value. Typically, this is OFF.

WebFOCUS BUE confirms that tracing has been changed or stopped in a message above the Session Monitor table.

Viewing WebFOCUS Sessions

The Session Viewer enables you to review traces of 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.

The Session Viewer complements the Session Monitor page by extending the range of sessions under review, from those that are currently active, to those that occurred in the past. The parameter Days Until Traces Are Deleted (IBI_TRACE_RETAIN_DAYS) defines the number of days that the Session Viewer retains information about sessions. It also focuses your review by limiting the range of available sessions to those created by you and by those users whose session activities you have permission to review.

The Session Viewer is only available to Managers. To open the Session Viewer, sign in as a Manager, and in the Portal Menu bar, click *Tools*, and then click *Session Viewer*. If you are signed in as a Manager, you can also open the Session Viewer from the View Trace icons in the Session Monitor page.

You can view sessions you started and sessions started by other users.

Reviewing the Session Viewer Main Page

The main page of the Session Viewer displays information about your current work session. It also lists entries for all recently completed sessions that you have permission to review, as shown in the following image.

The screenshot displays the 'manager Session Viewer 160317/manager_102754' interface. At the top, there is a summary table with the following data:

User	Tracing Level	File Echo Setting	SQL Tracing	Viewing File	Session Started	Last Activity	URL Count	Average Response	Server Requests	Average Server Time	Average DBMS Time
manager	Off	On	Off	On	10:28:54	10:29:39	112	0.183	3	1.008	0.000

Below the summary table, there are tabs for 'Start', 'End', 'Previous', 'Next', 'Last', and 'View Work URLs'. The 'View Work URLs' tab is selected, showing a list of session events. The list includes details such as the user, the action performed, the file path, and the session ID. For example, one entry shows a user performing a 'GET' action on a file named 'IBI_TRACE_RETAIN_DAYS'.

At the bottom of the page, there is a section titled 'Click two time bars to view the timeline between these transactions. Click one time bar to return to the default view'. This section contains a table with columns for 'User Agent', 'Host/OS', 'Window NT', 'Browser', 'Trident', 'Date', and 'File'. The table lists several sessions, including one from 'Mozilla/5.0 (Windows NT 6.1; WOW64; Trident/7.0; ...)' and another from 'Label-HEAD Gen/Number:10801 Gen/Date:Wed Mar 18 22:04:41 EDT 2016'.

To open the main page of the Session Viewer, sign in as a Manager, and in the Portal Menu bar, click *Tools*, and then click *Session Viewer*. When the main page opens, your sign-in information and the ID of the session you are viewing appear at the top of the screen. The session ID follows the format YYMMDD/username_HHMMSS. It contains the date, time, and username of the session on display. For each session, the following information is available:

User Name

The name of the user that signed in to this work session.

Tracing Level

The level of traces captured by the current session. The default value for this field is Off, but you can choose another value from the drop-down list. The Session Viewer saves this selection when you close the viewer, and uses it as the default setting for your next session.

The four tracing levels are cumulative, meaning each higher level includes the traces of all of the levels below it. These levels include:

- ☐ **Basic.** Generates a trace for each URL, which includes IBFS traces and procedure traces.
- ☐ **Outputs.** Includes Basic level traces and output from URLs that run requests on the Reporting Server. This level of tracing affects the amount of disk space required to capture output traces, but does not affect system performance.
- ☐ **Debug.** Includes Outputs level traces and log4j debug level written to the Session Viewer output.
- ☐ **Details.** Includes Debug level traces and legacy WFServlet traces. This level of tracing affects session performance.
- ☐ **Server.** Includes Details level traces and generates traces for the Reporting Server activity for the current work session.

Fex Echo Setting

The level of echo traces captured from the execution of FEX file commands. In a FEX file, the &ECHO variable displays command lines as they execute in order to test and debug procedures. These levels include:

- ☐ **On.** Displays WebFOCUS commands that are expanded and stacked for execution in its traces.
- ☐ **All.** Displays Dialogue Manager commands and WebFOCUS commands that are expanded and stacked for execution in its traces.

- ☐ **Off.** Suppresses the display of both stacked commands and Dialogue Manager commands in its traces. This value is the default.
- ☐ **None.** Prevents procedure code from being displayed (echoed). Once the value of &ECHO has been set to NONE, it cannot be changed during the session or connection.

SQL Tracing

The level of traces captured from SQL events. These levels include:

- ☐ **On.** Displays traces of all SQL request and response events. Even if you select this setting, however, the Session Viewer will not display SQL event traces if there are no requests issued to an SQL database.
- ☐ **Off.** Suppresses the display of traces of SQL request and response events.

Viewing File

The name of the Viewing File. File names are identified by their start time and end time in HH.MM format. If an asterisk (*) is displayed as the end time, current traces are being routed to that file.

You can click New File, to capture a new set of traces, which allows users to capture a set of URLs to be reviewed. When you click this link, the Session Viewer automatically creates a new file and assigns all subsequent traces to it. You can review prior traces by clicking a file containing completed traces in the Viewing File list.

Session Started

The time that your active session started, in HH:MM:SS format.

Last Activity

The start time of the most recent activity in your active session, in HH:MM:SS format.

URL Count

The total number of URLs issued for the session that you are viewing.

Average Response

The average response time for all URLs issued for the session that you are viewing.

Server Requests

The number of requests made to your Reporting Server during your active session.

Average Server Time

The average time (in seconds) that it takes the Reporting Server to respond to a request.

Average DBMS Time

The average time (in seconds) that it takes the Reporting Server to respond to a request directed to a non-WebFOCUS or RDBMS database.

If no current session file is available, the section below the status bar displays the following text:

Session file does not exist.

If a current session file is available, the section below the status bar lists traces for that file.

You can also view a table containing links representing recently completed sessions. If multiple viewable sessions occurred on a specific date, they are listed from left to right in that table in the order in which they occurred, earliest to latest.

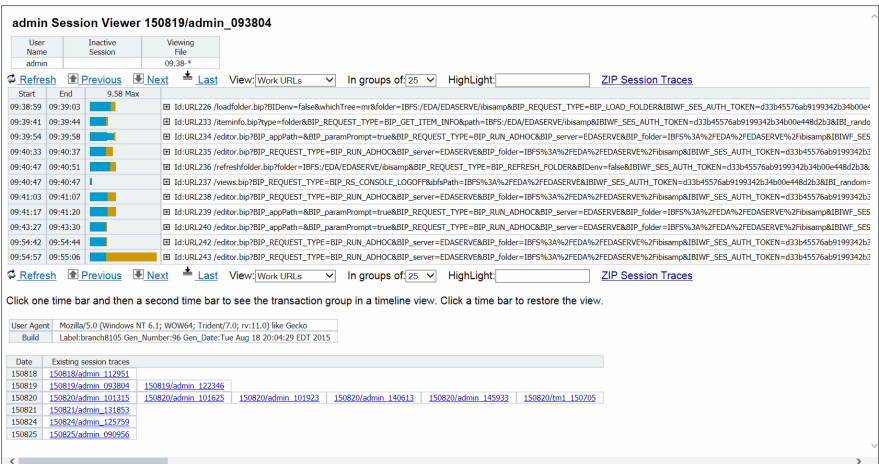
Information from completed sessions remains available for the period defined in the setting, Days Until Traces Are Deleted (IBI_TRACE_RETAIN_DAYS).

To view a different session, click a session link on the main page or the session details page. A new page displaying traces for your selected session opens.

Note: The session information links connect to completed sessions only. To view a current session, open the *Session Monitor* page from the Administration Console, and click an Information icon, if one appears.

Reviewing the Session Details Page

To open the session details page, click a session link in the Existing session traces column of the main page. The session details page opens, as shown in the following image:



This page displays a group of features that enables you to review relevant details about your selected session, review summary versions of the traces it created, and move on to other sessions.

When the review of your selected session is complete, close the session details page to return to the main page.

When you open the session details page, your sign-in information and the ID of your selected session appear at the top of the screen.

A table underneath the Session ID lists additional details identifying the session under review. The User Name entry identifies the name of the user who initiated the session on display.

The Inactive Session entry identifies the session as being inactive or active. If this entry is blank, the session on display is inactive. If a value appears in this entry, the session on display is currently open.

The Viewing File entry identifies the range of trace entries on display, as defined by start time. By default, this value displays the entire range of trace entries from the start time to an undefined end time. If a drop-down button appears, you can select a different time range from the drop-down menu.

You can use the following options to change the display of trace information that you want to view.

- ☐ **Refresh.** Adds any traces to the list that were generated after you opened an active session. This option is not available to previously completed sessions.
- ☐ **Previous.** Moves the display back to view an earlier set of traces.
- ☐ **Next.** Moves the display forward to view a later set of traces.
- ☐ **Last.** Moves the display to view the final set of traces captured right before the end of the session.
- ☐ **View.** Limits the list of traces by type.
 - ☐ **All URLs.** Displays URLs that return static content, such as .css files, .html files, .js files, and dynamic URLs that perform a Client action, as well as URLs that perform an action on the Reporting Server.
 - ☐ **Work URLs.** Limits the display to dynamic URLs, as well as Reporting Server requests. This is the default setting.
 - ☐ **Server Requests.** Limits the display to URLs that access the Reporting Server.

☐ **In groups of.** Determines the number of trace entries that appear on a single page. You can select 1, 5, 10, 25, 50, 100, or 200. Your selection in this field impacts the use of the Previous, Next, and Last options. The larger the value you select, the fewer times you will be required to move to the previous or next page.

☐ **HighLight.** Assigns a yellow highlight to the start time field of all trace entries that contain the search term that you type in this field.

For example, if you type the term *short*, a highlight appears in the Start Time field for any trace entry that contains this term, such as:

[IBFS checkPolicy Success](#)
[IBFS:/EDA/ACTWIN7/ibisamp/short.mas](#)

Note: After your search results produce a match, the highlights from that match will not clear until you close the session details window. Therefore, to prevent false matches to any subsequent search, you must close and reopen the session details window.

☐ **ZIP Session Traces.** Saves all of the traces from the session to a single.zip file. When you click this link, you are prompted to open or save the file. Click *Save As*, browse to a storage location for the.zip file, and then click *Save*.

The default name for this .zip file is *sessionmonitor*, followed by the number of trace and log files it contains.

These options also appear below the trace information table.



The trace information table enables you to review individual session traces in more detail. It displays one summary entry for each trace captured during the session. You can expand these entries to review the detailed event messages captured by the trace.

The table located below the trace information table and options identify the User Agent and Build that started the session. Details identifying the User Agent include the browser, operating systems, and supporting applications. Details identifying the Build include, the version number, build number, and generation date of the version of WebFOCUS BUE to which this session was connected.

A list of recently completed sessions appears at the bottom of the page. This list is a duplicate of the session list on the main page and appears here to enable you to move on to another session without having to leave the session details page.

Reviewing Trace Entries

Each entry in the list of traces on the session details page represents the record for a single system activity, as shown in the following image.

Start	End	0.0900 Max	Details
11:31:32	11:31:32		 Id:URL114 /run.bip?BIP_REQUEST_TYPE=BIP_RUN&BIP_item=simplecar.few&BIP_folder=1BF5%3A%2FEDA%2FEDASERVE%2Fbaseapp&IBI_random=2060.358382295817

One activity can include multiple events, and these events become visible when you expand the icon next to a trace to view its full detail.

For each trace, the following information is available:

Start

The time, in hours, minutes, and seconds, that an event in the trace began. Hours are expressed in twenty-four hour time notation.

End




The time, in hours, minutes, and seconds, that an event in the trace ended. Hours are expressed in twenty-four hour time notation.

Number of Seconds Max

The number in the header of this column represents the maximum number of seconds that were required to complete the longest trace in the list.

Entries in this column contain a (time) bar that represents the relative duration of the events in the trace, as shown in the following image.



-  The darkest blue section of the bar represents the number of Web CPU seconds that were required to process the events in this trace. It also identifies the trace as containing a Work URL component.
-  The lightest blue section of the bar represents the number of Web wait seconds that were required to retrieve a response from a database. It also identifies the trace as containing a Work URL component.
-  The brown section of the bar represents the number of Reporting Server seconds that were required to process the events represented in this trace. It also identifies the trace as containing a Server Request component.

You can view tooltips that identify the exact number of seconds that each section of a bar represents by pointing to that section with your mouse. If you are reviewing an active trace in a current session, the bar appears green and occupies the entire column entry.

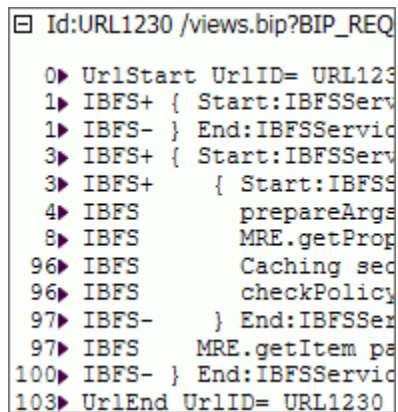
Details

The ID of the trace. This is the URL of the destination of the request message that launched the trace events. The first term in the URL identifies the servlet or other application that launched the request. Each trace ID is unique.

When the URL ID number is highlighted in orange, events in the trace associated with it include one or more error messages. Within the detail trace display, events that contain error messages are also highlighted in orange to help you identify when the errors occurred.

Reviewing Expanded URL Details

When you expand an individual Trace Details list entry, a nested list of system-generated messages opens. These messages identify the events captured by that trace and the time, in milliseconds, at which those events took place. Events include request and response messages exchanged between the Client and the Reporting Server or between the Reporting Server and the application server. They also include error messages, informational messages, and system status messages generated by application programs as they execute commands. Entries representing repetitive or subordinate events are nested to help you identify them more quickly, as shown in the following image.



```

Id:URL1230 /views.bip?BIP_REQ
0> UrlStart UrlID= URL1230
1> IBFS+ { Start:IBFSServ
1> IBFS- } End:IBFSServ
3> IBFS+ { Start:IBFSServ
3> IBFS+   { Start:IBFSS
4> IBFS   prepareArgs
8> IBFS   MRE.getProp
96> IBFS   Caching sec
96> IBFS   checkPolicy
97> IBFS-   } End:IBFSServ
97> IBFS   MRE.getItem pa
100> IBFS- } End:IBFSServ
103> UrlEnd UrlID= URL1230
  
```

A trace entry begins with the event start time and the number of milliseconds after the trace start time at which the event took place. This value helps you distinguish between individual events, and places them in sequence within an individual trace.

The IBFS status code for the trace event follows the event start time.

This column contains one of the following symbols:

Symbol	Description
IBFS+	The starting event of a program or exchange of data between programs or applications.
IBFS-	The ending or final event of a program or exchange of data between programs or applications.
IBFX*	An error message.
IBFSX	An administrative or informational message.

The text of the message generated by the application or program that describes the event appears next. The type of text displayed in this section varies with the type of trace you have selected from the View drop-down list.

- ☐ If you select *All URLs* or *Work URLs*, an expanded URL entry displays the status and error messages that were generated as the program ran.

Note: If your entry includes a Server Request message, the underlined Request ID term of the trace entry links you to full details of the Reporting Server Request trace, and the underlined Response ID entry links you to full details of the Reporting Server Response trace.

- ☐ If you select *Server Request*, an expanded URL entry displays the Reporting Server Request procedure, followed by a list of status or error messages generated during that procedure. (This is the same display that appears when you open a Server Request link from a Session Monitor Information icon.)

Reviewing Reporting Server Request Details

Traces captured from a Reporting Server Request identify the details of the query or other request operation sent from the Client to the Reporting Server during the session, as shown in the following image.

```

Id:URL85Req2 Run:Domain=Getting_Started/:App=getting_started retail_samples:SubAct=MR_STD_REPORT:Fex=WebFOCUS_Report UrlID
Plain text:---Focexec-Start--- RequestID=URL85Req2 UrlID=URL85 ReqInfo="Run:Domain=Getting_Started/:App
....:SET PCHOLD-FMT=XML
0001:EX -LINES 6 EDAPUT FOCEXEC,mrheader,C,MEM,-* mr header include start
0002:-* mr as NOT html
0003:-SET &FOCEXURL=&FOCEXURL | '&' | 'IBIMR_drill=IBFS,RUNFEX,IBIF_ex,true' | '&';
0004:SET FOCExURL='&FOCEXURL'
0005:-* mr header include end
0006:-*
0001:EX -LINES 25 EDAPUT FOCEXEC,WebFOCUS_Report,C,MEM,ENGINE INT CACHE SET ON
0002:
0003:
0004:-DEFAULTH &WF_SUMMARY='Summary';
0005:-DEFAULTH &WF_TITLE='WebFOCUS Report';
0006:TABLE FILE retail_samples/wf_retail_lite
0007:SUM WF_RETAIL_LITE.WF_RETAIL_SALES.COGS US
0008:BY WF_RETAIL_LITE.WF_RETAIL_PRODUCT.PRODUCT CATEGORY
0009:ACROSS WF_RETAIL_LITE.WF_RETAIL_TIME_SALES.TIME_YEAR
0010:ON TABLE PCHOLD FORMAT HTML
0011:ON TABLE NOTOTAL
0012:ON TABLE SET CACHELINES 100
0013:ON TABLE SET PAGE-NUM NOLEAD
0014:ON TABLE SET SQUEEZE ON
0015:ON TABLE SET HTMLCSS ON
0016:ON TABLE SET HTMLENCODE ON
0017:ON TABLE SET EMPTYREPORT ON
0018:ON TABLE SET GRWIDTH 1
0019:ON TABLE SET STYLE *
0020:-INCLUDE ENWarm
0021:TYPE=REPORT, TITLETEXT=&WF_TITLE.QUOTEDSTRING, SUMMARY=&WF_SUMMARY.QUOTEDSTRING, HFREEZE=OFF, $
0022:ENDSTYLE
0023:END
0024:
0025:-RUN

```

This information identifies the variables and commands sent during the request operation. These requests are usually TABLE requests or -HTMLFORM BEGIN/END requests that are sent from the Client to the Reporting Server.

The ID term in the first line above the procedure links it to the URL trace from which it was generated. For example, the ID, *URL85*, links the procedure to the server request event line within the activity captured in trace URL 85.

At the end of the list of variables and commands, the procedure displays a list of status messages describing the results of the query or other operation, as shown in the following image.

```

-----End----- RequestID=URL85Req2 UrlID=URL85 ReqInfo="Run:Domain=Getting_Started,
---Server---Times--- RequestID=URL85Req2 UrlID=URL85 ReqInfo="Run:Domain=Getting_Started,

```

Reviewing Reporting Server Response Details

Traces captured from a Reporting Server response identify the information returned in response to a query or other request operation sent during a work session from the Reporting Server to the Client during a work session.

To view output traces, click on the link from an underlined URL request response entry in a URL Trace entry, such as:

[URL103Req4Resp](#)

The first part of this display identifies the format variables returned to the Client during the response operation, as shown in the following image.

```
<column_desc>
  <col title="" valign="left" reference="" property="" help_message="" accept="" description="" focus_format="A8" width="8" datatype="char" alias="E01" fieldname="FOLDER" column="c0"> </col>
  <col title="" valign="left" reference="" property="" help_message="" accept="" description="" focus_format="A100" width="100" datatype="char" alias="E02" fieldname="FILENAME" column="c1"> </col>
  <col title="" valign="left" reference="" property="" help_message="" accept="" description="" focus_format="A64" width="64" datatype="char" alias="E03" fieldname="EXTENSION" column="c2"> </col>
  <col title="" valign="left" reference="" property="" help_message="" accept="" description="" focus_format="A10" width="10" datatype="char" alias="E04" fieldname="DATE" column="c3"> </col>
  <col title="" valign="left" reference="" property="" help_message="" accept="" description="" focus_format="A10" width="10" datatype="char" alias="E05" fieldname="GMTIME" column="c4"> </col>
  <col title="" valign="left" reference="" property="" help_message="" accept="" description="" focus_format="A8" width="8" datatype="char" alias="E06" fieldname="SIZE" column="c5"> </col>
  <col title="" valign="left" reference="" property="" help_message="" accept="" description="" focus_format="A512" width="512" datatype="char" alias="E08" fieldname="APPNAME" column="c7"> </col>
  <col title="" valign="left" reference="" property="" help_message="" accept="" description="" focus_format="A1" width="1" datatype="char" alias="E09" fieldname="OTYPE" column="c8"> </col>
  <col title="" valign="right" reference="" property="" help_message="" accept="" description="File Read Access" focus_format="I2" width="2" datatype="integer" alias="E10" fieldname="FREAD" column="c9"> </col>
  <col title="" valign="right" reference="" property="" help_message="" accept="" description="File Write Access" focus_format="I2" width="2" datatype="integer" alias="E11" fieldname="FWRITE" column="c10"> </col>
  <col title="" valign="right" reference="" property="" help_message="" accept="" description="File Execute Access" focus_format="I2" width="2" datatype="integer" alias="E12" fieldname="FEXEC" column="c11"> </col>
  <col title="" valign="right" reference="" property="" help_message="" accept="" description="File List Access" focus_format="I2" width="2" datatype="integer" alias="E13" fieldname="FLIST" column="c12"> </col>
  <col title="" valign="left" reference="" property="" help_message="" accept="" description="Master File Suffix" focus_format="A8" width="8" datatype="char" alias="E14" fieldname="SUFFIX" column="c13"> </col>
  <col title="" valign="left" reference="" property="" help_message="" accept="" description="File Description" focus_format="A512" width="512" datatype="char" alias="E15" fieldname="FDESCR" column="c14"> </col>
</column_desc>
```

The second part of the display identifies the data returned to the Client during the response operation, as shown in the following image.

```
<table>
  <tr>
    <tr>
      <td colspan="14">OTHER</td>
      <td colspan="1">etc</td>
    </tr>
    <td colspan="1">ETC</td>
    <td colspan="1">19/08/2015</td>
    <td colspan="1">21.17.38</td>
    <td colspan="1">1440033458</td>
    <td colspan="1">0</td>
    <td colspan="1">EDAHOME</td>
    <td colspan="1">D</td>
    <td colspan="1">1</td>
    <td colspan="1">1</td>
    <td colspan="1">1</td>
    <td colspan="1">1</td>
    <td colspan="1">1</td>
    <td colspan="1">1</td>
    <td colspan="1">1</td>
  </tr>
</table>
```

The Session Viewer can display Reporting Server response traces in XML, HTML, or a standard report format. Reporting Server responses usually contain data or status messages returned in response to SQL-based queries, updates, or other database-related operations.

Working With Log Files

The Log Files page displays links to all log files. The main grid lists log files in alphabetical order by name.

Next to each log file entry is a list of Logger Names, that is, those pages or events that contribute entries to that log. For example, the audit.log file captures events from com.ibi.uoa, com.ibi.config, com.ibi.content, and others. The Log Level field, next to each Logger, identifies the level of events captured by that contributor.

Event levels are cumulative. Events captured by a higher level are included when you select a lower level. For example, if you set the level to *Warn*, you capture Fatal level and Error level events as well as events that generate a warning.

The levels are defined below:

- ☐ **Off.** Capture no events.
- ☐ **Fatal.** Capture only events that disrupt system operations.
- ☐ **Error.** Capture events that generate error messages in addition to fatal events.
- ☐ **Warn.** Capture events that generate warning messages in addition to fatal and error events.
- ☐ **Info.** Capture events that generate informational messages in addition to warning, error, and fatal events.
- ☐ **Debug.** Capture events that generate debug messages in addition to informational, warning, error, and fatal events.
- ☐ **Trace.** Capture events that generate trace messages in addition to debug, informational, warning, error, and fatal events.

Understanding Warning Messages in InfoAssist+

This topic describes InfoAssist+ warning messages.

InfoAssist+ Warning Messages

Message	Description	OK	Cancel
Are you sure you want to switch data source? Doing so will delete your current report.	This warning message displays when a report is first saved in a different format and one or more additional reports are created and saved, and the user clicks <i>Switch</i> on the Data tab and selects a different report from the shortcut menu.	Deletes the current report.	Preserves the current report.
Are you sure you want to add a data source? Doing so will delete your current report.	This warning message displays when a report is first saved in a different format and an additional report is created and saved, and <i>Add</i> on the Data tab is selected.	Deletes the current report.	Preserves the current report.
User Selection in this request is not allowed with your configuration. Your report will be converted to default output.	This warning message displays when the user selection option is not available on restore, or when you are restoring a user selection option when it is disabled at global preference.	Converts the report to use the default output type.	No changes are made to the original procedure and it is closed.

Unsupported Syntax and Objects

This section describes the syntax and objects that are not supported.

- ❑ SUB-TOTAL syntax is not supported. If you try to open an existing procedure in InfoAssist+ from a legacy tool that contains SUB-TOTAL syntax, an unsupported syntax warning message displays. If you choose to continue, InfoAssist+ will convert the syntax to SUBTOTAL and add the converted syntax to all higher level sort breaks.
- ❑ SUMMARIZE syntax is not supported. If you try to open an existing procedure in InfoAssist+ from a legacy tool that contains SUMMARIZE syntax, an unsupported syntax warning message displays. If you choose to continue, InfoAssist+ will convert the syntax to RECOMPUTE.
- ❑ HTML FULL, FIXED, and PAGED syntax is not supported. If you try to open an existing procedure in InfoAssist+ from a legacy tool that contains any of these three options, an unsupported syntax warning message displays. If you choose to continue, InfoAssist+ will convert the procedure to regular HTML output.
- ❑ COLUMN-TOTAL syntax is not supported. If you try to open an existing procedure in InfoAssist+ from a legacy tool that contains this syntax, an unsupported syntax warning message displays. If you choose to continue, InfoAssist+ will convert it to use RECOMPUTE syntax.
- ❑ Line objects in a compound document imported from a legacy tool are not supported. If you try to open an existing procedure that includes line objects, an unsupported syntax warning message displays. If you choose to continue, InfoAssist+ will remove these line objects.

Index

3D depth

removing from a bar chart [638](#)

A

active cache [381](#)

enabling [381](#)

enabling by component [381](#)

rows retrieved [381](#)

active report [480](#)

active report Options dialog box [425](#)

Advanced tab [429](#)

Colors tab [428](#)

General tab [426](#)

Menu Options tab [426](#)

Active Technologies

for charts [430](#)

adding a Reporting Server [179](#)

Address Book access types

configuring [152](#)

Administration Console

accessing [103](#)

Client Verification [68](#)

Diagnostics [68](#)

navigating [63](#)

verification tools [68](#)

aggregation values [583](#)

amper auto-prompting in ReportCaster [701](#)

setting default variables [701](#)

annotations

displaying [642](#)

API for extensions [351](#)

application button [70](#)

application main menu [70](#)

application window [69](#)

area charts [392](#)

attribute categories

for an extension [360](#)

authentication

by Active Directory [225](#)

by LDAP [225](#)

external [225](#)

automatic refresh [756](#)

axis labels

custom format characters [610](#)

deleting [614](#)

formatting [608](#), [615](#)

horizontal axis labels [608](#)

rotating [615](#)

secondary axis labels [609](#)

staggering [615](#)

vertical axis labels [608](#)

axis titles

adding [616](#)

axis

manually setting [616](#)

setting advanced properties [617](#)

shortcut menu [613](#)

axis
 using properties [614](#)

B

bar charts [388](#)

Basic Scheduling Tool

- advanced settings [732](#)
- creating a schedule [695](#)
- delete parameters [709](#)
- Distribution Options [712](#)
- Email Distribution Option [712](#)
- notifications [718](#)
- Parameter Values [698](#)
- Printer Distribution Option [717](#)
- Quick Access Toolbar [693](#)
- Tasks [697](#)

bid-config.xml [60](#)

Blackout Dates [180](#)

- defining [185](#)
- extracting [205](#)
- importing [199](#)

blend [327](#), [328](#)

blending data [327](#), [328](#)

browser requirements [23](#)

bubble charts [398](#)

bubble maps [444](#)

Build/GEN Date

Build/GEN Number

burst reports [744](#)

burst values [690](#)

bursting [686](#)

bursting reports [690](#)
 guidelines and limitations [688](#)

C

Cache Cleaner [140](#)

cache clearing [110](#)

callback functions

- for rendering extensions [351](#)
- initCallback [351](#)
- preRenderCallback [351](#)
- properties [354](#)
- renderCallback [351](#)

canvas [70](#), [81](#), [82](#)

cell padding in reports [565](#)

change [383](#)

Change Management [234](#)

Change Management settings [121](#)

- IBI_CM_Export_WFRS_File_Extensions [121](#)

- IBI_CM_Retain_Handles [121](#)

- IBI_CM_Zip [121](#)

- IBI_CM_Zip_File_Format [121](#)

- IBI_CM_Zip_File_Verify_Signature [121](#)

chart frame and background

- formatting [617](#)

chart types

- custom [349](#)

chart

- adding a page heading and a page footing
[635](#)

chart

- controlling the rendering of a page heading and page footing [637](#)
- frame and background [618](#)
- rotating [639](#)

charts

- chart elements
- chart formatting tools [573](#)
- chart outputs [348](#)
- creating a basic chart [346](#)
- creating a combination chart [347](#)
- creating a dual-axis chart [348](#)
- creating a multi-axis chart [348](#)
- creating an advanced chart [346](#)
- custom [349](#)
- displaying annotations [642](#)
- dynamic reference lines [640](#)
- extensions [349](#)
- library of advanced charts [345](#)
- Live Preview [573](#)
- plug-ins [349](#)
- removing 3D depth [638](#)
- shortcut menu [573](#)
- static reference lines [639](#)
- using Active Technologies [430](#)

choropleth maps [444](#)

circular gauges [402](#)

cleanup utility [36](#), [763](#)

Clear Cache [110](#)

Client Verification [68](#)

Client

- license management [106](#)

color bands

- setting [605](#)

Color dialog box [567](#)

color mode

- by group [587](#)
- By Group setting [577](#)
- by series [587](#)
- By Series setting [577](#)

color-banded rows of data in reports [562](#)

COMPUTE fields [329](#)

config object for extensions [355](#)

configuration [146](#)

- change default user IDs [149](#)
- Data Server Settings [173](#)
- Distribution Server settings [149](#)
- Email Settings [158](#), [163](#)
- General Preferences [152](#)
- LDAP Setting [170](#)
- Log Settings [169](#)
- Other Schedule Defaults [168](#)
- ZIP Settings [166](#)

configuring

- InfoAssist Basic [106](#)
- InfoAssist for MR [106](#)
- Magnify license [106](#)
- Mobile Favorites [106](#)
- Quick Data [106](#)

connections

- adding [286](#)
- changing [287](#)
- configuring [285](#)

Creating Distribution Lists [679](#)

creating report queries

- in InfoAssist [493](#)

custom charts [349](#)custom interval [730](#)custom reports [758](#)**D**daily interval [725](#)Dashboard Properties [60](#)

- dialog [60](#)

data [327](#)

data labels

- associated dialog boxes [587](#)
- changing the position [593](#)
- custom format characters [589](#)
- formatting [587](#)
- shortcut menu [592](#)
- showing [592](#)
- using properties [592](#)

Data pane [87](#)

- adding fields to a report [88](#)

Data Panel group [87](#)Data Server settings [173](#)Data tab [78](#)

- Filter group [645](#)

default variable values in ReportCaster [701](#)Deferred Report Status interface [751](#), [752](#), [756](#), [758](#)

- expiration setting [756](#)
- parameters [751](#), [755](#), [757](#)
- refreshing [752](#)

Deferred Reporting settings

- IBI_Deferred_Custom_Description [121](#)
- IBI_Deferred_Notify_Submitted [121](#)
- IBI_Deferred_Ticket_Delete_Confirm [122](#)

deferred reports [751](#), [758](#)

- deleting [754](#), [756](#)
- options [752](#)
- parameters [751](#), [755](#), [757](#)
- running [754](#), [758](#)
- saving [751](#), [754](#), [758](#)
- sorting [751](#), [752](#)
- status [752](#)
- viewing [751](#), [752](#), [757](#)

DEFINE fields [329](#)delete confirmation message [758](#)delete option [758](#)deletion confirmation message [758](#)Detail (DEFINE) dialog box [330](#)Diagnostics Tab [66](#)

- Client Verification [68](#)

distribution files [690](#)Distribution Lists [679](#)

- creating [679](#)
- deleting [684](#)

Distribution Lists [679](#)

- editing [684](#)
- multiple email addresses [690](#)
- publishing [686](#)
- sharing [686](#)
- unpublish [686](#)

Distribution Server [139](#)

- configuring [149](#)
- start [143](#)
- status [140](#)
- stop [143](#)

documents

- aligning components [374](#)
- building [367](#)
- cascading prompts [442](#)
- changing the filter condition [441](#)
- creating from a single report [370](#)
- deleting a component [378](#)
- editing component [371](#)
- editing text [377](#)
- inserting a chart [369](#)
- inserting a report [368](#)
- inserting an existing report [370](#)
- inserting an existing report into a document with the Insert tab [370](#)
- inserting an image [371](#)
- inserting reports from multiple data sources [367](#)
- inserting text [371](#)
- inserting text and images [371](#)

documents

- inserting two reports from two different data sources [367](#)
- moving a component [373](#)
- resizing a component [372](#)
- setting target reports [433](#)
- styling and customizing a chart [376](#)
- styling and customizing a report [376](#)

domain templates [211](#)Dynamic Distribution lists [690](#)dynamic reference lines [640](#)**E**Edit Title dialog box [575](#)editing a schedule with unavailable options [735](#)Enterprise domain template [211](#)Excel output for reports [345](#)expiration settings for deferred reports [756](#)Export Data [481](#)extensions [349](#)

- buckets [360](#)
- build cycle [350](#)
- building [356](#)
- callback functions [351](#)
- config object [355](#)
- configuring [355](#), [357](#)
- creating [350](#), [351](#)
- data for [359](#)
- installing [363](#)
- null data [363](#)

- extensions [349](#)
 - partial data [363](#)
 - registering [356](#)
 - reinstalling [364](#)
 - structure [350](#)
 - using [365](#)
- external authentication [225](#)
- external distribution files [690](#)
- External Security settings [113](#)

F

- field formats
 - alphanumeric [673](#)
 - changing in reports [673](#)
 - date [674](#)
 - date-time [674](#)
 - numeric [673](#)
 - percent sign [675](#)
- field images [86](#)
- Field tab [80](#)
 - Filter group [645](#)
- Filter group [645](#)
- Filter pane [86](#)
- filtering with slicers [672](#)
- FOCUS format index files [487](#)
- folders [213](#)
 - changing title [216](#)
 - creating [215](#)
 - deleting [217](#)
 - duplicating [216](#)

- folders [213](#)
 - publishing [216](#)
 - viewing properties [214](#)
- Format Axis dialog box [609](#)
 - Advanced tab [611](#), [612](#)
 - General tab [612](#)
 - Labels tab [610](#), [612](#)
 - Scale tab [609](#)
 - Title tab [610](#), [612](#)
- Format Gauge dialog box [626](#)
 - Advanced tab [629](#)
 - Axis Scale & Labels tab [628](#)
 - Bands tab [629](#)
 - General Options tab [627](#)
 - Tick Marks tab [628](#)
- Format Grid Lines dialog box [598](#)
 - Color Bands tab [599](#)
 - Frames tab [600](#)
 - Major Grid Lines tab [599](#)
 - Minor Grid Lines tab [599](#)
 - Quadrant Lines tab [600](#)
- Format Labels dialog box [588](#)
 - Advanced tab [589](#)
 - Funnel Labels tab [591](#)
 - General Options tab [588](#)
 - Pie Labels tab [590](#)
 - Pie Title tab [590](#)
- Format Legend dialog box [594](#)
 - Legend Options tab [594](#)
 - Markers & Labels tab [594](#)

Format Series dialog box [574](#)

Border tab [575](#)

Fill tab [574](#)

Format tab [77](#)

Frame & Background dialog box [618](#)

Advanced tab for 3D charts [622](#)

Background tab [623](#)

Frame Edge tab for 2D charts and 2D charts with 3D effects [620](#)

Frame Edge tab for pie charts [621](#)

Frame tab for 2D charts and 2D charts with 3D effect [619](#)

Frame tab for pie charts [620](#)

shortcut menu [624](#)

Wall and Floor tabs [622](#)

frame and background

using properties [625](#)

G

gauge chart

shortcut menu [630](#)

gauge charts

Format Gauge dialog box [626](#)

gauge

using properties [630](#)

Global Updates [207](#)

Email address [208](#)

Email from [209](#)

mail server [208](#)

gridlines [598](#)

deleting [606](#)

displaying horizontal minor [602](#)

displaying vertical major [602](#)

displaying vertical minor [602](#)

formatting [598](#)

major [598](#)

minor [598](#)

setting the color, weight, and style [603](#)

shortcut menu [601](#)

using properties [601](#)

grids [386](#)

groups

adding users [223](#)

deleting users [224](#)

H

hardware requirements [22](#)

headings and footings

adding to reports [559](#)

styling in a report [561](#)

Help [70](#)

HFREEZE [713](#)

Home tab [76](#)

Report group [568](#)

horizontal major gridlines

displaying [601](#)

horizontal menu [417](#)

hourly interval [724](#)

HTTP request headers [68](#)

HTTP Request Info setting [68](#)

I

IBI_CM_Export_WFRS_File_Extensions [121](#)

IBI_CM_Retain_Handles [121](#)

IBI_CM_Zip [121](#)

IBI_CM_Zip_File_Format [121](#)

IBI_CM_Zip_File_Verify_Signature [121](#)

IBI_Deferred_Custom_Description [121](#)

IBI_Deferred_Notify_Submitted [121](#)

IBI_Deferred_Ticket_Delete_Confirm [122](#)

IBI_WF_Describe_HTML [123](#)

IBI_WFDescribe_Default [123](#)

IBIF_Describe_Null [123](#)

IBIMR_Prompting [122](#)

IBIMR_PromptingUnset [122](#)

InfoAssist

- creating report queries [493](#)

- report queries [493](#)

InfoAssist+ interface

- application button [70](#)

- application window [69](#)

- canvas [70](#)

- Help [70](#)

- main components [70](#)

- Navigation taskbar [70](#)

- Quick Access Toolbar [70](#)

- Resources panel [70](#)

- ribbon [70](#)

- status bar [70](#)

InfoAssist+

- license configuration [106](#)

InfoDiscovery Portal

- adding pages [493](#)

- pages [493](#)

InfoDiscovery

- troubleshooting [761](#)

- troubleshooting [761](#)

initCallback [351](#)

insert [406](#)

Insert tab [76](#)

inserting a chart

- [369](#)

installation [22](#)

installation steps

- Linux [40](#)

installing an extension [363](#)

interactive reports [417](#)

- horizontal menu [417](#)

- vertical menu [417](#)

J

Java environment [68](#)

Java Regular Expressions [688](#)

Job Logs [145](#)

Job Status [144](#)

Join dialog box [326](#)

JVM Property Info [69](#)

L

Layout tab [79](#)

LDAP [170](#)

configuration [170](#)

email settings [170](#)

settings [170](#)

legend

shortcut menu [595](#)

legends

formatting [594](#)

hiding [596](#)

positioning [596](#)

specifying border color [596](#)

using properties [596](#)

license code configuration [106](#)

Lightweight Mapping [345](#)

line charts [390](#)

applying a smooth effect [580](#)

Line Style dialog box [592](#)

List view [88](#)

log files [742](#)

log reports [738](#), [742](#)

considerations [743](#)

viewing [738](#), [743](#)

Logical view [88](#)

M

Magnify license [106](#)

Maintaining Distribution Lists [679](#)

major gridlines [598](#)

Manage Private Resources [225](#)

Managed Reporting [751](#)

Managed Reporting Repository [754](#), [758](#)

maps

bubble [444](#)

choropleth [444](#)

marker

changing the appearance of [581](#)

markers

connecting or hiding a series line [580](#)

mathematical equations [580](#)

matrix charts [412](#)

matrix marker charts [400](#)

maximize visualization [410](#)

[410](#)

Menu Bar [49](#), [50](#)

Metadata tool

accessing [285](#)

metadata

creating [285](#)

importing [287](#)

mime.wfs configuration file [118](#)

redirection settings [118](#)

minor gridlines [598](#)

minutes interval [723](#)

Mobile Favorites license [106](#)

mode [767](#)

monthly interval [727](#)

multi-page dashboard

creating [380](#)

multi-page document

creating [379](#)

multi-page documents [378](#)

multi-page

dashboards [378](#)

Multi-Y [482](#)

Multi-Y Axis [481](#)

multipage documents and dashboards

Page menu [380](#)

N

Navigation taskbar [70](#), [85](#)

null data for an extension [363](#)

O

Options window [239](#)

Output window [83](#)

cascade [84](#)

output format options [84](#)

output target options [83](#)

output view options [84](#)

switch output [84](#)

tile horizontally [84](#)

tile vertically [84](#)

P

Package Name

Packet Email [716](#)

page footing

adding to a chart [635](#)

page heading

adding to a chart [635](#)

page headings and page footings

formatting [635](#)

Page Options dialog box [380](#)

pages [493](#)

Parameter Prompting settings

IBI_WFDescribe_Default [123](#)

IBIF_describe_null [123](#)

IBIMR_Prompting [122](#)

IBIMR_PromptingUnset [122](#)

partial data for an extension [363](#)

pie charts [393](#)

pie slices

expanding [582](#)

hiding [582](#)

plug-ins [349](#)

API [351](#)

buckets [360](#)

build cycle [350](#)

building [356](#)

callback functions [351](#)

config object [355](#)

configuring [355](#), [357](#)

creating [350](#)

data for [359](#)

installing [363](#)

null data [363](#)

partial data [363](#)

registering [356](#)

- plug-ins [349](#)
 - reinstalling [364](#)
 - structure [350](#)
 - using [365](#)
- preRenderCallback [351](#)
- Product Release
- Properties dialog box [214](#)
- properties.json file for an extension [357](#)
- published content [753](#)
- publishing [745](#)

Q

- Query pane [82](#)
 - right-click options [95](#)
- Quick Access Toolbar [70](#), [73](#)
- Quick Data license [106](#)

R

- Redirection Settings [118](#)
- Reference Line dialog box [640](#)
- reference lines
 - COMPUTE field [641](#)
 - dynamic [640](#)
 - static [639](#)
- refresh options [752](#), [756](#), [757](#)
- registering an extension [356](#)
- Release ID
- renderCallback [351](#)
- Report group [568](#)

- report queries
 - InfoAssist [493](#)
- Report Style dialog box [566](#)
- report types
 - custom [758](#)
 - deferred [751](#)
 - Standard [753](#)
- ReportCaster configuration folders [147](#)
- ReportCaster Tab [66](#)
- ReportCaster
 - tracking schedules [738](#)
 - unavailable options [744](#)
- Reporting Server settings [178](#)
- Reporting Servers
 - adding servers [179](#)
- reports
 - Excel output [345](#)
 - bursting [686](#), [688](#)
 - cell padding [565](#)
 - color-banded rows of data [562](#)
 - deferred [752](#)
 - deleting [751](#)
 - distributing [686](#)
 - headings and footings [559](#)
 - Live Preview [573](#)
 - report-level styling [556](#)
 - Style group [558](#)
 - styling [556](#)
 - styling column titles [557](#)
 - styling data [557](#)

reports

- styling headings and footings [561](#)
- styling rows with alternate colors [562](#)
- tracking [738](#)
- traffic light conditional styling by constant [562](#)
- traffic light conditional styling by field [564](#)
- viewing deferred output [752](#)

resource templates [211](#)Resource Tree [51](#)Resources panel [70](#), [86](#)

- field images [86](#)
- using Query pane in [90](#)

reviewing parameters [757](#)ribbon [70](#)ring pie charts [395](#)run once interval [722](#)**S**SaaS Tenant domain template [211](#)scatter charts [397](#)schedule blackout dates [180](#)Schedule Distribution Method Settings [156](#)Schedule Format settings [153](#), [154](#)

schedule formats

- configuring [152](#)

schedule logs [739](#)Schedule Method setting [156](#)

schedule methods

- configuring [152](#)

schedule status

- checking [739](#)

Schedule

- multiple email addresses [690](#)

schedules

- copying [736](#)
- deleting [736](#)
- editing [735](#)
- maintaining [733](#)
- publishing [738](#)
- sharing [736](#)

secondary axes options

- Format Secondary Axis dialog box [613](#)

Security Tab [65](#)Series tab [81](#)

series

- changing the title [586](#)
- changing the type [579](#)
- color mode [587](#)
- deleting [587](#)
- dialog boxes [574](#)
- filtering values in [582](#)
- formatting [573](#)
- formatting the fill and border [579](#)
- hiding fields [583](#)
- shortcut menu [576](#)
- sorting fields [582](#)
- using properties [578](#)

Session Monitor [766](#)

- Show data [480](#)
 - [479](#)
- Size and Position dialog box [372](#), [373](#)
- Slicers tab [78](#)
- slicers
 - filtering with [672](#)
- software requirements [23](#)
- sorting deferred reports [751](#), [752](#)
- sql [284](#), [285](#)
- static reference lines [639](#)
- status bar [70](#), [86](#)
- storyboard [642](#), [643](#)
 - create [644](#)
 - creating [642](#)
- Structured view [88](#)
- Style dialog box [591](#)
- subquery filter [487](#)
- Summary Field (COMPUTE) dialog box [330](#)
- synonyms
 - creating [285](#)
 - importing [287](#)
- system requirements [22](#)

T

- text
 - adding [407](#)
 - inserting [406](#)
- ticks
 - setting [604](#)
- tracking schedules [738](#)

- Traffic Light Condition dialog box [575](#)
- traffic light conditional styling by constant in reports [562](#)
- traffic light conditional styling by field in reports [564](#)
- traffic light conditions
 - applying with drill-down to numeric measure field (by constant) [584](#)
 - applying with drill-down to numeric measure field (by field) [585](#)
- treemap charts [401](#)
- trendlines
 - adding [580](#)
- troubleshooting [35](#), [761](#)

U

- unavailable options
 - in ReportCaster [735](#)
 - log information [744](#)
 - notification [744](#)
- uninstalling
 - on Linux [45](#)
 - on Windows [33](#), [36](#), [762](#)
- uploading
 - documents [283](#)
 - images [282](#)
 - to repository [282](#), [283](#)
- user preferences
 - changing [238](#)
 - environment and styling [240](#)

user preferences

format [240](#)

layout [240](#)

view [239](#)

users [217](#)

adding to groups [223](#)

creating [220](#)

deleting [221](#)

editing details [221](#)

removing from groups [224](#)

V

verification tools [68](#)

vertical menu [417](#)

View tab [80](#)

 Data Panel group [87](#)

viewing deferred reports [751](#), [757](#)

visualizations [404](#), [407](#)

 creating [407](#)

 inserting [406](#)

 renaming [411](#)

visuals

 area charts [392](#)

 bar charts [388](#)

 bubble charts [398](#)

 bubble maps [444](#)

 changing [383](#)

 choropleth maps [444](#)

 circular gauges [402](#)

 gauges [402](#)

visuals

 grids [386](#)

 horizontal bar charts [389](#)

 inserting [406](#)

 line charts [390](#)

 maps [444](#)

 matrix marker charts [400](#)

 pie charts [393](#)

 ring pie charts [395](#)

 scatter charts [397](#)

 selecting [384](#)

 stacked area charts [392](#)

 stacked bar charts [388](#)

 treemaps [401](#)

 vertical bar charts [388](#)

W

web browsers [22](#)

web servers [22](#)

WebFOCUS [68](#)

 Diagnostics menu [68](#)

weekly interval [726](#)

Wildcard characters [688](#)

Within [556](#)

Y

yearly interval [729](#)

Z

- Zip distribution
 - configuring [166](#)
 - settings [166](#)



Feedback

Customer success is our top priority. Connect with us today!

Information Builders Technical Content Management team is comprised of many talented individuals who work together to design and deliver quality technical documentation products. Your feedback supports our ongoing efforts!

You can also preview new innovations to get an early look at new content products and services. Your participation helps us create great experiences for every customer.

To send us feedback or make a connection, contact Sarah Buccellato, Technical Editor, Technical Content Management at Sarah_Buccellato@ibi.com.

To request permission to repurpose copyrighted material, please contact Frances Gambino, Vice President, Technical Content Management at Frances_Gambino@ibi.com.

WebFOCUS 8

Online Help PDF

WebFOCUS Business User Edition Release 8.2 Version 01