crystal decisions ™

Introduction to Seagate Analysis

The Report and Query Tabs in Seagate Analysis

Overview

This document aims to provide an introduction to Crystal Decisions' new business analysis tool, Seagate Analysis.

Definition of terms, architecture, usage and features are discussed. This document should be consulted in addition to the *Seagate Analysis On-line Help*. Further information on specific topics can also be found in the *Seagate Crystal Reports Version 7 User's Guide* and the *Seagate Info Version 7 User's Guide*

Seagate Analysis embodies many features of the Seagate Crystal Reports Designer, Seagate Crystal Query Tool, the Seagate Info Cube Designer and the Seagate Worksheet. This document focuses primarily on the Query and Report tabs of Seagate Analysis.

Contents

Introduction

Seagate Analysis is a powerful, easy-to-use, desktop tool for analysis of business information.

Seagate Analysis combines the ability to perform ad-hoc database queries and navigate multi-dimensional cubes of data (OLAP). It helps the business user make sense of just about any data source imaginable - from a simple report on disk space allocation to complex web traffic analysis. Seagate Analysis also has integrated report viewing to ensure that information can be presented in a crisp, meaningful way.

Seagate Analysis integrates three existing components:

- The proven report viewing capabilities of Seagate Crystal Reports, the world standard for desktop reporting and analysis,

- Seagate Query, a tool for ad-hoc database queries,

- The proven functionality of Seagate Worksheet, a best-of-breed tool for multi-dimensional data navigation.

Seagate Analysis also delivers full integration with Microsoft Excel, to enable the user to easily drop data into Excel's familiar analytical environment.

6/25/2002 3:58 PM   Page 4

Minimum System Requirements

- Windows 95, Windows 98 or Windows NT 4

- Pentium 90 (Pentium 133 recommended)

- 32 MB of RAM (64 MB recommended)

- CD-ROM drive

- 30 MB free disk space for the install

- 100+ MB recommended for temporary working area.

- Microsoft Internet Explorer 4 or Netscape Navigator 4

NOTE	Cube building with large data sets may require more disk space.

Opening a sample Seagate Analysis document

Seagate Analysis comes complete with many sample Analysis documents that illustrate the power of this analytical tool, such as the Web traffic reports and the Back Office reports.

To open a sample Analysis document

From the Windows Explorer:

- Right-click an .rpt file and choose **Open with Seagate Analysis** from the shortcut menu.

If Seagate Crystal Reports is not installed on your computer, double-clicking an .rpt file automatically starts Seagate Analysis.

From the Windows Desktop:

1. Start Seagate Analysis.

2. In the **Welcome to Seagate Analysis** dialog box, select **Open an Analysis document from your Favorites list,** then click **OK**.

3. In the **Open** dialog box, choose the document and click **Open**.

NOTE	The sample Analysis documents are supplied with saved data. See **Sample Documents** in the On-line Help file for full details about using these documents to analyze your own data.

When you open a report or an Analysis document, Seagate Analysis creates a temporary backup of the file in the same directory as the .rpt file. This file stores the changes that you make until you save the Analysis document.

Seagate Analysis saves the Query, and, if you have already created them, the Report and Cube definition to an .rpt file. When you open a file that contains a formatted report, Seagate Analysis opens in Report view. When you open a sample document that contains a cube definition but no formatted report, Seagate Analysis opens in Cube view. Otherwise, it opens in Query view.

- To open a different sample document, choose a document from the **Favorites** menu.

- To open an existing Analysis document or a Crystal report, choose the **Open** command from the **File** menu, or click the **Open** button on the toolbar.

- To create an Analysis document from Seagate Analysis, choose the **New** command from the **File** menu, or click the **New** button on the toolbar.

To open a recently used Analysis document

You can choose it from the **File** menu, which contains a list of the five most recently opened documents, or:

1. Open the **Help** menu, and choose **Welcome to Seagate Analysis**.

2. In the **Welcome to Seagate Analysis** dialog box, click **Open an Analysis document from your History list**.

3. In the **Open** dialog box, choose the document you want and click **Open**.

Opening an existing document

Seagate Analysis allows you to open any document created in Seagate Crystal Reports and view the query that was used to produce it. You can create a Crystal report in Seagate Crystal Reports and in Seagate Info.

To open a Crystal report or Analysis document

From the Windows Explorer:

- Right-click an .rpt file and choose **Open with Seagate Analysis** from the shortcut menu.
  If you do not have Seagate Crystal Reports installed, double-clicking an .rpt file starts Seagate Analysis.

From the Windows Desktop:

1. Start Seagate Analysis.

2. In the **Welcome to Seagate Analysis** dialog box, select **Open an existing Analysis document**

3. Click **OK**.

4. In the **Open** dialog box, choose a document and click **Open**.

When you open a Crystal report or an Analysis document, Seagate Analysis creates a temporary backup of the file in the same directory as the ..rpt file. This file stores the changes that you make until you save the Analysis document.

Seagate Analysis saves the query, and if you have already created them, the report and cube definition to an .rpt file. If you open an .rpt file that contains a cube definition, Seagate Analysis opens in Cube view. If you open a file that contains a formatted report, Seagate Analysis opens in Report view. Otherwise, it opens in Query view.

- To open one of the samples, choose a file from the **Samples** subdirectory.

- To open a different document from Seagate Analysis, choose the **Open** command from the **File** menu, or click the **Open** button on the toolbar.

- To open a sample document from Seagate Analysis, choose a document from the **Favorites** menu.

- To create a new document in Seagate Analysis, choose **File** | **New**, or click the **New** button on the toolbar.

To open a recently used Analysis document:

Either choose it from the list of the five most recently opened documents on the **File** menu, or:

1. From the **Help** menu, choose **Welcome to Seagate Analysis**.

2. In the **Welcome to Seagate Analysis** dialog box, click **Open an Analysis document from your History list.**

3. In the dialog box, choose a document and click **Open**.

The Analysis Explorer

The **Analysis Explorer** shows you all the fields that you have added to the query, as well as all the data sources to which you can connect. A data source can be an ODBC data source, a SQL data source, an existing document, or one of your favorites. The Analysis Explorer lists all the tables and fields that you can use as part of the query. You use the Analysis Explorer to add tables and fields to the query. You can also create new fields, such as formulas, SQL Expressions and parameters.

The Analysis Explorer can be invoked in three ways:

- by default when you create a new Analysis document.

- when you choose **Data|Data Sources** in Query view

- when you choose **View|Query Content**.

The Analysis Explorer consists of two tabs:

- the **Data Sources** tab, and

- the **Query Contents** tab.

Data Sources tab

The **Data Sources** tab of the Analysis Explorer displays all the available data sources and their corresponding tables and views.

Seagate Analysis currently supports ODBC data sources. If you run Seagate Analysis from with Seagate Info, you can also use the database connections in Info Desktop objects as data sources. If you run Seagate Analysis as a stand-alone application, you can use the database connections in the supplied samples as data sources.

Click the (+) to expand the Data Sources directory tree. You'll see that the tree is divided into a subset of the following subdirectories:[SS1]sections

- Data Sources

- File Directory (if you run Seagate Analysis as a stand-alone application)

- Seagate Info (if you run Seagate Analysis from within Seagate Info)

- My Favorites, if any exist.

The **Data Sources** folder contains all of the data sources available to the user.

This folder contains the following subdirectories:

- Microsoft Exchange Admin

- NT Event Log

- ODBC

- OLE DB

A list of other available data sources is also provided. This list allows you create reports off your system. For example, clicking on Microsoft Outlook will allow you to report off your email folders.

If you run Seagate Analysis as a stand-alone application, you can use the database connections in the sample reports as data sources for new reports. The **File Directory** folder contains these connections.

If you run Seagate Analysis from within Seagate Info, you can use the database connections in **Info Desktop** objects as data sources for new reports. The **Seagate Info** folder tree contains all the database connections on the Info APS. To use an existing report as a database connection, on the Info Desktop set the property **Serve as database connection** for the report. See the Seagate Info documentation for details.

The **My Favorites** subdirectory will only become accessible through the Analysis Explorer window once you have added your own data sources to this folder. When SA is first installed, the **My Favorites** subdirectory can only be accessed through the menu and contains sample reports that can also be used as data connections. This means that if you select one of these template files, you will be re-using the connection information stored with the particular report. Add data sources and report files you use frequently to the **My Favorites** directory.

For more information on the **Favorites** list, refer to *Working with the favorites list* section below and in the On-line Help file.

Once you have selected your data source, right-click a table and choose **Insert Table** to add the table to the **Tables** panel of the Query designer.

Working with the favorites list

The favorites list is used in two places in Seagate Analysis.

- The list of favorite *data sources* is displayed on the **Data Sources** tab of the Analysis Explorer.

- The list of favorite *documents* is displayed on the **Favorites** menu.

Adding a data source to the list of favorite data sources:

1. In the Analysis Explorer window, right-click a database connection name (this can be an entire database file or a single database table).

2. Choose **Add to Favorites** to add the database connection to your favorites list.

3. Choose the folder for this document, or click the **New Favorite folder** button on the toolbar to create a new folder. The **New Favorite folder** button remains unavailable until you select a folder.

4. Click **OK**.

Adding a document to the list of favorite documents:

The list of favorite documents is available on the **Favorites** menu.

If you run Seagate Analysis as a stand-alone application you can add the current document to your list of favorite documents. You can also add an Analysis document or Crystal report on the local computer.

If you run Seagate Analysis from within Seagate Info, you can also add a document from the Info APS.

To add the current document:

1. From the **Favorites** menu, choose **Add to Favorites**.

2. Choose the folder for this document, or click the **New Favorite folder** button on the toolbar to create a new folder. The **New Favorite folder** button remains unavailable until you select a folder.

3. Click **OK**.

To add a document from the local computer:

1. From the **Favorites** menu, choose **Organize Favorites**.

2. Click the **Add more Favorites from** button on the toolbar.

3. Choose **Available Local Documents** from the list on the left.

4. Type in the name of the file including the path, or click **Browse**.

5. When the name of the file is displayed in the **File** box, drag the blue flag icon to a folder in the favorites tree on the right.

6. When you have finished adding documents, click **OK**.

To add a document from the Info APS:

1. From the **Favorites** menu, choose **Organize Favorites**.

2. Click the **Add more Favorites from** button on the toolbar.

3. Choose `Available Remote Documents` from the list on the left.

4. In the tree on the left, click the name of the Info APS. Expand the tree until you find the document you want to add.

5. Select the document and drag it to a folder in the favorites tree on the right.

6. When you have finished adding documents, click `OK`.

Query Contents tab

The `Query Contents` tab lists the contents of the current query, which includes:

*   the database fields used in the query,

*   the database tables used in the query,

*   summary fields,

*   formula fields,

*   SQL Expressions, and

*   parameter fields.

Adding Fields to a Query from the Query Contents tab

To add database fields to a specific part of your query:

1. Right-click the field

2. Select `Add To` on the shortcut menu

3. Specify where you want to add the field by choosing the corresponding panel in the submenu.

To add Formula fields, SQL Expressions and Parameter fields to the query:

1. Right-click the field

2. Select **Insert** on the shortcut menu.

3. Choose the type of field to you want to add from the submenu.

4. Specify the details of the field in the `Edit Formula Field` dialog box, the `Edit SQL Expression Field` dialog box, or the `Parameter Field` dialog box, as appropriate.

See *Adding Formula Fields*, *Adding SQL Expressions*, and *Adding Parameters*.

To add Summary fields to the query:

1. Right-click on any selected field in the Explorer and choose `Create Summary Field` from the shortcut menu.

2. Specify the properties of the field in the `Create Summary Field` dialog box.

When a summary field is created, it is automatically added to the **`Fields`** panel.

To view the contents of a database field in the Query Contents tab:

1. Right-click the field.

2. Choose **`Browse Field`** from the shortcut menu.

The Query tab

A query is a request for specific information from a database. Once you have created a query, you can keep refining it until you retrieve exactly the information you want. You can view the information in a report, or use it as the basis of a cube.

If you want to create a cube, you need to design the query to suit the cube you want to build. In general, you need to retrieve a larger volume of information for this type of query, and it should be in a more summarized form. See *Hints on Cube Design* in the On-line Help file for more details on cube building.

Creating a simple query

To get the data you want for the query, add the tables that contain the data to the **Tables** panel. This panel gives you a clear picture of the sources of data you are using and relationships between them. You can use it to organize the tables you are using, as well as to define and maintain the links between them.

Once you have added and linked the tables you want to use for the query, it may be easier to hide the **Tables** panel and use the Analysis Explorer to add fields to the query.

To hide the **Tables** panel**:**

- Click the **Show/Hide Tables** toolbar button, or

- Open the **View** menu, and choose **Tables**. (Repeat the process if you wish to redisplay the **Tables** panel.)

Adding tables

There are three ways to add a table to the Tables panel:

Select the table in the **Data Sources** tab of the Analysis Explorer and:

- Double-click the table, or

- Right-click the table and choose **Insert Table** from the shortcut menu, or

- Drag the table to the **Tables** panel.

To remove a table from the **Tables** panel:

- Right-click the table and choose **Remove Table** from the shortcut menu.

NOTE	If the database you are accessing is protected by a password, Seagate Analysis prompts you for a user name and password when you select the corresponding ODBC data source in the Analysis Explorer.
	See *Logging on to a database* in the Seagate Analysis Online Help file.
	If the location of the database changes after you have created a report, use the **Set Database** command to change the query so that it accesses the database in the new location.
	See *Changing the data source accessed by a report* in the Seagate Analysis Online Help file.

Browsing tables

If you want to determine the contents of a field or a table, you can browse a subset of the records.

To browse a table

- Right-click the table and choose **Browse Table** from the shortcut menu. A dialog box appears, displaying records for all the columns in the table.

To browse a field

- Right-click the field and choose **Browse Field** from the shortcut menu. A dialog box appears, displaying records for the selected field in the table.

Setting the browse data size

You can control how many records are returned when you browse a table or field by going into the **Tools** menu, and choosing **Options**. Specifying a browse records size prevents Seagate Analysis from fetching the whole data set each time you browse. For example, if you set the browse data size to 100, Seagate Analysis returns the first 100 records, but removes the duplicate entries. For more information, see *Setting the browse data size* in the Seagate Analysis Online Help file.

Linking tables

When the query takes data from more than one table, it is necessary to link the tables together. You link tables so records from one table match the related records from another. For example, if the query uses an Orders table and a Customers table, you link the tables so that each order (from the Orders table) can be matched up with the customer (from the Customer table) who placed the order.

When linking tables, you link on a field that is common to both tables. Seagate Analysis uses this linking field to match up records from one table with those from the other.

When creating the link, it is important to link *from* one table *to* another table. The *from* table is used as a 'primary' table, while the *to* table acts as a 'lookup' table where records are looked up by the primary table. In a simple link, Seagate Analysis examines the first record in the primary table and finds all matching records in the lookup table. Once all matches have been found in the lookup table for the first record in the primary table, all matches in the lookup table for the next record in the primary table are found.

When linking SQL tables, you can specify the type of join you want to use in Seagate Analysis. An SQL join indicates how linked fields in two SQL tables are compared when records are read. When you create an SQL link, it is set to an equal join by default. The result set from an equal join includes all the records where the linked field value in both tables is an *exact match*. Specify the

join type using the **Link Properties** command, which is described below. For full details of the different join types, refer to the *Seagate Analysis On-Line Help.*

The easiest way to link database tables is to use the **Smart Linking** command. Smart Linking automatically chooses links for your tables based on common fields in tables.

To link tables automatically:

- Right-click anywhere in the **Tables** panel and choose **Smart Linking** from the shortcut menu. Alternatively, you can choose **Smart Linking** from the **Data** menu.

To link two tables manually:

- Click the field that you want to link in the *from* table

- Drag it to the field in the *to* table.

The link appears as an arrow between the two fields. Use the **Link Properties** command to change the join type.

To set the properties for a link:

1. Right-click the link and choose **Link Properties** from the shortcut menu.

2. In the Link Properties dialog box, set the type of link and click **OK**.

If you are linking tables from the same ODBC data source, choose an option in the **Join Type** area. For full details of the different join types, see the *Seagate Analysis On-Line Help.*

If you are accessing a non-ODBC data source and linking three tables in an A to B, A to C manner, you can select a **Lookup Type** in addition to a join type. The lookup type determines the order in which Seagate Analysis looks up the records in the tables B and C. In this case Seagate Analysis uses a native driver to connect to the database, rather than an ODBC driver.

There are three lookup types available:

- **Parallel lookup type** This option is also referred to as 'look up both at the same time'. For each record in Table A, this option looks for a matching record in Table B and a matching record in Table C. Then it looks for the next matching record in Table B and the next matching record in the Table C, and so on. Once it finds all the matching records, it repeats the process with the next record in Table A, and so on.

- **Series lookup type** This option is also referred to as 'look up all of one, then all of others (A to B, A to C)'. For each record in Table A, this option first looks for all the matching records in Table B and then looks for all the matching records in Table C. Once it finds all the matching records, it repeats the process with the next record in Table A, and so on.

- **Product lookup type** This option is also referred to as look up all the combinations of the two files. For each record in Table A, this option looks for a matching record in Table B, and then finds all the matching records in Table C. Once it finds all the matching records in Table C, it repeats the process with the next record in Table B, and so on. Once it comes to the end of the matching records in Table B, it repeats the process with the next record in Table A, and so on.

NOTE	If you have more than one link between two tables, all the links must be of the same type.

To delete a link:

- Right-click the link and choose **Delete Link** from the shortcut menu.

To remove all links between two tables:

- Right-click one of the links and choose **Delete All Links** from the shortcut menu.

Finding tables

If the query contains many tables, you may not be able to see a particular table in the **Tables** panel. Use the **Locate Table** command to scroll quickly to the selected table in the panel.

To find a specific table:

1. Right-click anywhere in the **Tables** panel and choose **Locate Table** from the shortcut menu.

2. In the Locate Table dialog box, choose the name of the table you want from the list. The **Tables** panel scrolls to show the table you have selected.

The query design panels

Design your query in the **Design** panels, which is located beneath the **Tables** panel. You can add fields to the **Design** panels from the Analysis Explorer or from the tables displayed in the **Tables** panel. Adding and deleting fields is described in more detail in the following section.

The **Design** section is structured like a SQL query, with a panel representing each SQL clause. You can build the query by placing fields in the appropriate panels.

- **Fields** panel supplies the SELECT clause

- **Filters** panel supplies the WHERE clause

- **Groups** panel supplies the GROUP BY clause

- **Sorts** panel supplies the ORDER BY clause

Seagate Analysis, by default:

- displays the **Fields** and **Filters** panels

- hides the **Groups**, **Sorts**, **Group Filters** and **Analysis Fields** panels

You can choose which panels you want to display or hide by going to **View** | **Design Panels**, then selecting or de-selecting the appropriate panels from the submenu.

You can hide all the design panels by clicking the **Design** button on the toolbar. This is useful if you want to concentrate on linking tables, or on the query results. The **Design** button toggles between showing or hiding the Design panels.

The **Result Set** panel, located beneath the **Design** panels, displays the record set that results from running the query.

Adding and deleting fields

There are three ways to insert a field into the query:
- drag the field from the **Tables** panel to the appropriate panel,

- drag the field from the **Query Contents** tab of Analysis Explorer to the appropriate panel, or

- right-click the field from the **Query Contents** tab of Analysis Explorer. Select **Add To** in the shortcut menu. From the submenu, choose the panel to which you want to add the field.

To delete a field:

- Right-click the field and choose **Delete** from the shortcut menu.

The Fields Panel: Selecting the fields to use in the query

Specify the fields to include in the query by adding them into the **Fields** panel. You can add either individual fields or all the fields in a table at once.

The **Fields** panel supplies the SELECT clause of the SQL query.

To add a field to the Fields panel:

- Drag a field from a table in the **Tables** panel to the **Fields** panel.

To add all the fields in a table to the Fields panel:

- Right-click a field in the chosen table in the **Tables** panel and choose **Add All Fields** from the shortcut menu.

The Sorts Panel: Sorting the query

By default, the query results appear in the order they are returned from the data source. However, you may want to specify your own order, for example, to sort sales transactions according to the amount spent by each customer. You do this by adding fields to the **Sorts** panel.

The **Sorts** panel supplies the ORDER BY clause of the SQL query.

If you sort by more than one field, the sorts are applied in the order in which they appear in the **Sorts** panel. You can also sort by fields that are not included in the query, although they exist in one of your tables.

When sorting on multiple fields, Seagate Analysis first sorts the records based on the values in the first field selected, putting them in ascending or descending order as specified. When two or more records have the same field value in the first sort field, Seagate Analysis then sorts those records based on the value in the second sort field. For example, in a sort on last name and then first name (in ascending order), Smith, Bob would be returned before Smith, John no matter which way the fields are listed in the database. The process is similar for three field sorts, four field sorts, and so on.

All sorts can be ascending or descending; the default is ascending. The data type of a field determines the method in which the data from that field is sorted. An ascending sort means that the numbers 0-9 (in that order) come before A-Z.

To sort the query results by the values of a field:

- Add the field to the **Sorts** panel.

To change the sort order:

1. Either double-click the field, or right-click it and choose **Edit** from the shortcut menu.

2. In the **Edit Sort Order** dialog box, choose **ascending order** or **descending order**.

    If the field you are sorting by is a summary field, you can also choose to sort:

    - all of the values,

    - only the top N values, or

    - only the bottom N values.

3. Click **OK**.

4. Click the **Preview** button on the toolbar to display the sorted the data.

The Groups Panel: Grouping the query

The **Groups** panel enables you to group the query results into categories and supplies the GROUP BY clause of the SQL query. These categories are then displayed in the result set and in the report. For example, you might need to show the orders made by each customer in the report. To do this, you must have the fields Order ID and Customer in the **Fields** panel and Customer in the **Groups** panel. When you group the data, it is easier to navigate the data in both the result set and in the Report view. You can drill down on the areas of interest using the group tree.

See Viewing grouped data in the result set in the Seagate Analysis On-Line Help.

You can group the query results based on database fields, formula fields, or specify your own grouping. Note that you can use a formula based on a parameter as a grouping field.

See *Adding Formula Fields and Adding Parameters*.

Once you have specified a grouping, you can add a group total or a different summary field to the query. In this way, you can show a group total at the end of each group in the report, or a grand total at the end of the report. For example, you could create a summary field that totals up all the orders for each customer.

See *Creating Summary Fields* below, or in the On-Line Help

If the query uses grouping, you can choose to perform the majority of the processing on the database server and push only the result to your computer.

See *Performing the grouping on the server* in the Seagate Analysis On-Line Help.

NOTE	If you specify a field to group by, Seagate Analysis needs to sort by that field or a summary of it. Adding a field to the **Groups** panel automatically adds the same field to the **Sorts** panel. Only delete this sort field from the query if you replace it with a summary field.

Using custom grouping

Most of the time, data is sorted based on the values from a field in the report. For example, if you have a customer list and you want to group by state, Seagate Analysis first sorts the list by state. Then it breaks the list into state groups whenever the value in the State field changes.

Sometimes, however, you may not want to group based on the values found in one of the fields on your report.

For example:

- The report may not contain the field you want to group on. Your report may contain a City field but not a State field, although you need to group by State.   In this case, you could build for each state, and manually assign the city values that you want to be in each group.

- The report may contain the field you want to group on, but you may not be happy with the grouping based on the values in that field. For example, you may have a Color field on your report that includes specific color names, such as Navy Blue, but you want all shades of each color to appear as a single group, such as Blues. In this case you can build custom groups and manually assign the records you want to be in each group.

- The report may contain the field you want to group on, but you only want to include specific values or ranges of values for each group. For example, you might want one group to contain records where gross sales are less than a certain value, a second group where gross sales are greater than a certain value, and a final group where gross sales fall between two values. In this case, you can build your groups using the same range of selection facilities that are available to you for building record selection queries.

Custom grouping enables you to create both the customized groups that will appear on your report and the records that each group will contain. Your only real limitation is that a record can be assigned to only one group.

To create custom grouping:

1. Right-click on the grouping field and choose **Change to Custom Grouping** from the shortcut menu.

2. In the **Edit Specifying Group** dialog box, choose a grouping that you want to change from the list.

3. In the tabbed dialog pane, specify the members of the group.

4. Repeat steps 2 and 3 for all the groupings you want to change.

5. If you want to create a new grouping, click the **new** tab.

6. Specify how the remaining records are to be grouped using the **Group Others** area.

7. Click **OK**.

Creating summary fields

If you want to insert a summary of the data, such as an average, sum, or a count for a group, you can add a Summary Field to the query. For example, you might need the total number of orders made by each customer. To do this, you must have the Order ID in the **Fields** panel and the Customer field in the **Groups** panel. To create a summary field that adds together all the orders for each customer, select the Order ID field in the **Fields** panel and choose **Create Summary Field** from the shortcut menu, choosing 'Count' as the summary operation.

To create a summary field for one group level:

1. Right-click the selected field in the **Fields** panel and choose **Create Summary Field** from the shortcut menu.

2. In the dialog box, choose the appropriate summary group from the list.

3. Choose the summary operation you want to use. The choices available depend on whether the field contains numeric, or non-numeric data.

4. Click **OK**.

To create a summary field on all groups in the report:

*   Right-click the selected field in the **Fields** panel and choose **Create Summary Field on All Groups** from the shortcut menu.

This creates a summary at each group level, and creates a grand total for the entire report.

To edit a summary field:

*   Double-click the field, or

*   Right-click the field and choose **Edit** from the shortcut menu.

The Filters Panel: Restricting the data

Often you'll want to restrict the data returned by the query by specifying selection criteria that a record must satisfy for it to be included in the query's resulting data set. You do this using the adding the fields on which you want to put a restriction into the **Filters** panel.

The **Filters** panel supplies the WHERE clause of the SQL query.

To specify a selection condition:

1. Add the field whose value you want to restrict to the **Filters** panel.

2. Right-click the field and choose **Edit** from the shortcut menu.

3. In the **Edit Field Range** dialog box, choose a type of condition from the list. If a value is required, type it in the list that appears below, or choose a value from the list. You can also use a parameter as the value.

4. If you want to add any other conditions to the query, add the fields to the **Filters** panel and repeat steps 2 and 3. By default, new conditions are added to the existing ones with an AND operator between them. Double-click the AND operator to switch it to an OR operator.

If you add multiple conditions, specify how they are applied by changing the logical operators between them and the way they are grouped. Refer *to Adding operators to conditions* (below) for more details.

5. Click **OK**.

Modifying selection conditions

To modify a condition in the Filters or Group Filters panel:

1. Double-click the field, or right-click the field and choose **Edit** from the shortcut menu.

2. Make your changes in the **Edit Field Range** dialog box.

Adding operators to conditions

When you add multiple conditions in the **Filters** and **Group Filters** panels, combine them using operators and brackets.

To add operators or brackets to a field:

1. Right-click the field, point to **Add Operator** in the shortcut menu and choose from the submenu.

2. The operator appears before the selected field.

3. Double-click an operator to switch it between AND and OR.

The Group Filters panel: Restricting summarized data

If you have added a summary field on one of your groups and want to restrict which groups appear in the query results by the values in the summary field, you need to add the restriction to the **Group Filters** panel.

The **Group Filters** panel supplies the HAVING clause of the SQL query.

NOTE	The **Group Filters** panel can only contain summary fields.

To restrict summarized data:

1. If necessary, choose the **Group Filters Panel** option from the **View | Design Panels** shortcut menu to display the **Group Filters** panel.

2. Add the summary field whose value you want to restrict to the **Group Filters** panel.

3. Right-click the field and choose **Edit** from the shortcut menu.

4. In the **Edit Field Range** dialog box, choose a type of condition from the list. If a value is required, type it in the list that appears below.

NOTE	You can also use a parameter to specify the value. See *Creating parameter fields* and *Creating Complex Queries*.

5. If you want to add any other conditions to the query, add the fields to the **Group Filters** panel and repeat steps 2 and 3. By default, new conditions are added to the existing ones with an AND operator between them. Double-click the AND operator to switch it to an OR operator.

   You can specify the order Seagate Analysis applies multiple selection conditions by changing:

   - the logical operators between them, and

   - the order they appear in the **Group Filters** panel.

6. Click **OK**.

The Analysis Fields Panel

Parameters, formulas and SQL expression fields are added to the **Analysis Fields** panel in the query. To use one of these fields, drag it from the **Analysis Fields** panel to the relevant query design panel. You can also see a list of these fields in the **Query Contents** tab of the Analysis Explorer.

To edit a field in the **Analysis Fields** panel:

- Double-click the field, or

- select the field, right-click it and choose **Edit** from the shortcut menu.

NOTE	If you remove a field from the **Analysis Fields** panel, the field is removed from the entire query.

Copying data to Microsoft Excel

Seagate Analysis makes it easy to move data into Microsoft Excel, so that you can take advantage of its powerful analytical features.

To copy data to Excel:

1. In the grid view of the result set, drag the mouse to select the cells you want to copy.

2. To select ranges of cells, hold down the SHIFT key while you click a cell.

   To select nonadjacent cells, hold down the CTRL key while you click a cell.

3. Go to **Edit | Copy to Excel**. This launches Microsoft Excel, and shows the data you selected. The rows and columns in the result set are mapped to corresponding rows and columns in Excel.

NOTE	You can only copy data *from* Analysis *to* Excel; you cannot copy from Excel into Analysis.

Saving a query

The **Save** command saves the whole Analysis document, which includes your query, as well as the report, worksheet, and cube definition if you have already created them. The document is saved to a *.rpt* file.

To save the document:

- Click the **Save** button on the toolbar, or choose **File | Save**.

- If you want to save the current data in the report, check the **Save with Data** option in the **File** menu before saving**.** Otherwise, the report may display different data the next time you open it.

- To save the document to a different filename, use the **Save As** command.

Creating complex queries

You can create powerful and flexible queries that give exactly the information you want. Formulas and SQL expressions add calculations to the query. Parameter fields allow you to create a single query that you can modify whenever your needs change.

Performing calculations on the data

Calculations allow you to create new fields for the query, based on existing database fields. For example, you can add a calculation to find the difference between order and delivery times, using the order date and ship date fields. There are two types of calculations that can be added to a query:

- Formula, and

- SQL expression

NOTE	SQL expression fields are only available when you use an SQL-capable database or an ODBC data source.

Adding formulas

Formulas use the calculation capabilities of Seagate Analysis, and have a much wider range of calculations available than SQL expressions.

To display data that is a calculated value, you need to create a formula field and place that formula in the query. For example, if the database only stores the order dates and ship dates for orders but you need to display the number of days it takes to ship the order, you must create a formula that will calculate the number of days between ordering and shipping. This is just one example of the use of formulas.

Go to Favorites | Web Server, and choose the sample report 'Most Popular Directories (Top 10)– NCSA' for an example of using formulas.

To add a formula:

1. Go to **Insert | Formula**. Make sure the 'Perform on server' box is unchecked.

2. Click **OK**.

3. In the **New Name** dialog box, type a name for the field in the **Name** box.

4. In the **Edit Formula Field** dialog box, you can double-click or drag a field, function, or operator to add it to the formula.

[RK2]

You must enter the various components in the correct syntax, for example:

- enclose text in quotation marks: "This is a text string"

- enclose function arguments in parentheses, where applicable: ToNumber ({Orders. Order ID})

- reference formulas with a leading @ symbol: {@formula}

- reference parameters with a leading ? symbol: {?parameter}

  For more details about formula syntax, refer to the *Edit Formula Field dialog box* section in the On-line Help file.

5. Edit the formula, if necessary. To make the formula more readable, press **ENTER** to insert a line break.

6. Click the **Check Formula** button on the toolbar to check for errors, then fix any syntax errors that are identified.

7. Click **OK** to close the dialog box, or **Apply** to save the formula without closing the window.

8. The new formula field will appear in the **Analysis Fields** panel, prefixed by @ symbol. To use the field in the query, drag it from the **Analysis Fields** panel to the appropriate panel. The new field will also appear in the **Query Contents** tab of the Analysis Explorer.

NOTE	There is no need to specify record or group selection conditions using a formula. To restrict data, specify a condition in the **Filters** or **Group Filters** panel.

Adding SQL expression fields

SQL expressions are like formulas, except that they use SQL (Structured Query Language) syntax, rather than the Seagate Analysis formula language. You can use SQL expressions in much the same way as database fields. SQL expressions are evaluated on the database server as long as the conditions for server-side processing are satisfied. See *Processing data on the database server* in the On-line Help for more details.

NOTE	You must have some familiarity with the SQL language and SQL expressions before trying to add an SQL expression field to your query.

To add a SQL expression field:

1. Go to **Insert | Formula.**

2. In the **New Name** dialog box, type a name for the field in the **Name** box.

3. Make sure the 'Perform on server' option is checked on.

4. In the **Edit SQL Expression Field** dialog box, double-click or drag a field or function to add it to the expression.

5. Edit the expression, if necessary. To make the expression more readable, press **ENTER** to insert a line break.

6. Click the **Check formula** button on the toolbar to check the expression for errors; then fix any syntax errors that are identified.

7. Click **OK** to close the dialog box, or **Apply** to save the formula without closing.

The new SQL expression field appears in the **Analysis Fields** panel, prefixed by the @ symbol. To use the field in the query, drag it from the **Analysis Fields** panel to the appropriate panel. The new field also appears in the **Query Contents** tab of the Analysis Explorer.

Creating parameter fields

Parameters are fields that prompt the user to specify a value each time the data is refreshed. When a value is supplied, the program runs the query using that value in either the selection criteria or a formula. By using parameters, you can create a single query that will return different data sets, depending on the your needs.

For example, if you have a database that contains information about the sales of products across the United States, you could create a query that displays all the customers in a particular state. By defining the state as a parameter, when you refresh the data you can specify which state value Seagate Analysis will retrieve from the data source. If you refresh the query again, you can add data for another state to the result set.

There are two sorts of parameters: **single-value** prompts and **multi-value** prompts. A multi-value prompt allows you to enter more than one value for the parameter. In the previous example, if you enter the state as a multi-value prompt, you can run the query for one or more states at a time.

To see an example of using a parameter, go to **Favorites | Desktop Admin**, and open the *File Extensions* report.

To add a [RK3]parameter to the query:

1. From the **Fields** panel, right-click the field whose value you want to specify as a parameter, and choose **Create Prompting Parameter Field** on the shortcut menu.

2. Choose **Create Single-value Prompt** or, if you want the parameter to accept multiple values, select **Create Multi-value Prompt**.

3. The program creates a simple parameter, which it adds to the **Analysis Fields** panel, and adds a simple condition based on the parameter to the **Filters** panel. By default, Seagate Analysis will make the parameter name the same as the selected database field, prefixed by the ? symbol.

4. When the query is refreshed, only those records that satisfy the condition in the **Filters** panel will be returned. By default, the condition is set to database field 'is equal to' parameter value. For example, state = ?state. If you specified "New York" when prompted, Seagate Analysis would only return the records where "New York" was the value for the state field.

To change the selection condition:

1. Right-click the condition in the **Filters** panel, and choose **Edit** from the shortcut menu.

2. In the **Edit Field Range** dialog box, choose a type of condition from the list. The value is already set to the parameter.

3. Click **OK**.

To change the parameter properties:

1. Right-click the parameter in the **Analysis Fields** panel, and choose **Edit** from the shortcut menu.

2. In the Parameter Field dialog box, change the name for the parameter in the **Field Name** box, if necessary.

3. From the **Field Type** list, choose a data type.

4. If you want the parameter to accept multiple values, select the **Multiple Values** check box.

5. Enter the text you want to be prompted by in the **Prompting text** box. This is the message that appears when the program prompts the user for a parameter value. The prompting text can have up to 255 alphanumeric characters.

6. Type the default value in the **Default Values** box and press **ENTER**. Alternatively, you can select a field from the list of fields and then choose a default value from the list of values.

The default values are the values that you can choose from when you run the query. If you want a choice of values, you must select them one at a time from the list.

7. Click **OK**.

The new field appears in the **Analysis Fields** panel, prefixed by the ? symbol.

When you create a parameter by the method above, it is automatically added to the **Filters** panel.

You can also create a parameter by choosing the **Insert | Parameter** command. To use the field in this case, drag it from the **Analysis Fields** panel to the relevant panel. The new field also appears in the **Query Contents** tab of the Analysis Explorer.

To enter the parameter value(s):

1. Refresh the query.

2. In the Parameter Field Prompts dialog box, choose a value for the selected parameter.

3. You can type the value, or choose from the list of default values that you set for the parameter. For a multi-value prompt, choose more than one value from the list.

4. If the query contains more than one parameter, select the next parameter in the list and specify a value for it, and so on.

5. When you have set the values for all the parameters in the query, click **OK** to run the query.

Changing table logon information

You can change the logon information (user name and password) for each table involved in the query. Use this feature to prevent a logon dialog box appearing when you refresh the data in the query.

To change the logon information for a table:

1. From the **Data** menu, choose **Logon to Data Server**.

2. In the Logon to Data Server dialog box, choose the table from the list on the left.

3. Type a new user name in the **User Name** box.

4. Type a new password in the **Password** box.

5. If you want the new logon information to apply to all the tables, click **Apply to all tables**.

6. To change the logon information for a second table, choose the table in the list, and repeat steps 3 through 5.

7. Click **OK**.

NOTE	Before you using an ODBC data source in Seagate Analysis, make sure to configure the ODBC DSN in the Windows Control panel. For more information about configuring ODBC DSNs, see the Seagate Analysis Online Help files.

The Report tab

Viewing a report

Once you have defined a query using the **Query** tab in Seagate Analysis, you can view it as a report.

To view a query as a report

- Click the **Report** tab, or go to **View** | **Report**.

If you haven't already run the query, Seagate Analysis queries the data source when you click the Report tab.

If the query retrieves a large amount of data, you can cancel the request by clicking the **Stop** button on the toolbar, or by pressing the **ESC** key.

The report displays the data fetched by the query in a default format. If you have grouped the results in the query, this grouping is also shown in the report.

The Group Tree is shown on the left of the report. By expanding and clicking the entries in the Group Tree, you can move directly to the detailed information for a particular group. This is known as **drilling down**. (See below.)

You can apply one of a standard set of styles to the report to obtain sophisticated formatting quickly. (See *Formatting a report… Setting a report style*.)

When you initially view a report, the first page is displayed at a magnification of 100%. You can move to a different page in the report, or change the magnification.

Changing pages

You can move between pages to view any part of the report you want. Use the buttons on the toolbar to move to the first, last, next, or previous page.

To move to a specific page:

1. Choose **Edit|Go To**.

2. In the **Go To Page** dialog box, enter the page number you want to view.

3. Click **Go To**.

Drilling down on a group

If you have grouped the query results, this grouping is also shown in the report. The Group Tree is shown on the left of the report. By clicking the entries in the Group Tree, or clicking group headers or footers, you can move directly to the detailed information for a particular group. This is known as 'drilling down'. Each time you drill down, Analysis queries the database for the necessary data.

If you drill down in the Group Tree, Seagate Analysis does not open a separate view.

If you drill down in the report itself, the details are shown in a drill-down view. To show detailed information in a separate view, double-click a group header or footer. The separate view enables you to switch between different parts of the report.

To close the drill-down view, choose **View |Close Drill-Down**, or click the **Close Drill-Down** button on the toolbar.

Using the **Preview** command also closes any drill-down views.

Zooming in and out

To zoom in or out, select a value from the drop-down list on the toolbar. You can choose any magnification between 25% and 400%, or make the report fit the page.

Formatting a report

Report Sections

You can format the report to improve its legibility and to obtain the look you want.

Reports are divided into a number of horizontal bands, called sections. A basic report has five sections:

Report Header	This section is used for the report title and other information you want to appear only once, at the beginning of the report. It can also be used for charts that include data for the entire report.
Page Header	This section is used for information that you want to appear at the top of each page. This can include the name of the document. It can also display the field titles above the fields in a report.
Details	This section is used for the body of the report, and is printed once per record. The bulk of the report data generally appears in this section.
Page Footer	This section usually contains the page number and any other information you want to appear at the bottom of each page.
Report Footer	This section is used for information you want to appear only once in a report, such as grand totals. It can also be used for charts that include data for the entire report.

If the data in the query is grouped or includes a summary field, there are two extra sections:

Group Header	This section usually contains the group name field, and can be used to display charts that include data specific to the group. It is printed once for every value in the grouping field, at the beginning of the group.
Group Footer	This section usually contains the group name field and summary, and can be used to display charts that include data specific to the group. It is printed once for every value in the grouping field, at the end of the group. The details for the group appear between the Group Header and the Group Footer.

When you format a section, such as a group header, this changes the formatting for all the group headers in the report. Similarly, formatting the details section changes the formatting for all the details of the report.

Setting a report style

You can get a slick appearance for the report very quickly by applying one of the standard report styles to the report.

- To set a standard style, go to **Format | Report Styles** and choose the style you want from the submenu. Analysis applies that report style to the report.

- To remove a style, go to **Format | Report Styles** and then click **None**.

Changing row and column spacing

You can alter the row and column spacing in the report simply by dragging the mouse.

To change the row spacing throughout a section:

1. Click within the section to select one of the entries.

2. A gray border with sizing handles appears.

3. Drag the bottom handle of the border to change the row spacing.

To change the width of a column throughout the report:

1. Click within the column to select one of the entries.

2. A gray border with sizing handles appears.

3. Drag the right or left handles on the border to set the new column width.

Formatting columns

You can specify the format of columns in the report and their headers.

1. To format a column or header, right-click the column or header, and choose `Format Field` from the shortcut menu.

2. You can change general, border, and font properties by clicking the corresponding tab:

General tab

To choose the alignment for the text in the field, use the `Horizontal Alignment` list.
To size the field according to the value in the field, select `Tight Horizontal`.
To give the field a drop shadow, select `drop shadow`.

Border tab

To choose the style of each part of the border, use the lists on the `Border` tab.
Set the color of the field using the `Color` area.
Use the `Border` list to set the border color.
Use the `Background` list to set the background color. To select a custom color, click one of the buttons next to the `Border` or `Background` lists to call the `Color Chooser`. For more details, see `Color Chooser` dialog box in the On-line Help file.
[SS4]

Font tab

To change the font, choose from the `Font Name` list.
To change the font size, choose from the list to the right of `Font Name`.
To change the font color, choose from the `Font Color` list or click the button to call the `Color Chooser`.
Select `Bold`, `Italic`, or `Underlined` as required.

3. Once done, click `OK`

Formatting sections

You can specify the format of each section of the report. When you format a section, you can:

- choose the background color for the section,

- insert a new page before or after the section,

- keep the section together (for page breaks),

- suppress the section if it is blank

- have the section underlay the following sections

- reset the page number after the section, and

- print the section at the bottom of the page.

NOTE	Not all options are available for all section types.

To format a section:

1. Right-click within a section and choose **Format Section** from the shortcut menu.

2. In the Section Expert dialog box, choose a section to format in the **Sections** list.

3. In the Attributes area, select the formatting options you want.

4. Click **OK**.

Adding text to a report

You can use text to add extra titles or annotations to a report.

To add text to a report:

1. From the **Insert** menu, choose **Text Object**.

2. In the Insert Text Object dialog box, type the text you want to add in the **Text** box.

3. Click **OK**. An object frame appears.

4. Click where you want to position the text.

To edit the text:

1. Right-click on the text and select **Edit Text** Object from the shortcut menu.

2. In the **Insert Text Object** dialog box, make the changes to the text.

3. Click **OK**.

Adding special fields to a report

You can add text with special properties to a report, such as dates and page numbers. These fields are called special fields.

To add a special field:

1. From the **Insert** menu, choose **Special Field.**

2. In the **Insert Special Field** dialog box, choose the type of field you want to add:

Page Number	adds a page number
TotalPageCount	adds the total number of pages in the report.
Page N of M	adds the page number and total number of pages
Report Title	adds a title to the report
PrintDate	adds the date when the report was printed
PrintTime	adds the time when the report was printed.
Record Selection Formula	adds a record selection formula field
Group Selection Formula	adds a group selection formula field.
Data Date	adds the date when the data was last retrieved or refreshed
Data Time	adds the time when the data was last retrieved or refreshed.
Modification Date	adds the date when the report was last modified
Modification Time	adds the time when the report was last modified
Record Number	adds a number for each record in the details section
Group Number	adds a number for each group

3. Click **Insert**. An object frame appears.

4. Click where you want to insert the field on the report.

Showing and hiding sections

By default, all sections of a report are visible.

To specify which sections should be visible in the report:

1. Right-click anywhere in the report and choose **Show All Sections or Hide Section** from the shortcut menu.

2. To hide a report section, right-click the section and choose **Hide Section** from the shortcut menu.

3. To show all the report sections, right-click anywhere in the report and choose **Show All Sections** from the shortcut menu.

4. If you have previously hidden any of the report sections, these sections are now displayed.

Showing report data in a chart

Seagate Analysis allows you to insert charts into reports. You can use charts to organize and present your data more efficiently, and to identify trends.

Creating and modifying the chart is easy with the use of the **Chart Expert**. The Expert walks you through the creation of a chart step-by-step.

For information about adding charts to the Worksheet, refer to *Showing worksheet data in a chart* in the On-line Help.

Within a report in Analysis, you can chart on:

- a summary field, such as a sum or average, or

- a detail field.

Charting on a summary field

Many useful charts you create are based on summary fields within the report. For example, if you have a report showing sales summed by region, you can add a chart to illustrate the percentage of sales in each region. In order to create a summary chart, you must already have the summary field in the query.

To chart on a summary field:

1. Choose **Insert**|**Chart**, or click the **Chart** button on the toolbar.

2. In the **Chart Type** tab of the Chart Expert, choose a chart type from the list. Click one of the buttons on the right to select the specific style you want for that type of chart.
   For example, you can choose 'Bar Chart' as the chart type, and 'Stacked Bar' as the specific style.

   For a description of chart types, see *Chart Type Tab* in the On-line Help file.

3. Click the **Data Field** tab, and click the **Group** button to specify that the chart will be based on grouped data.

4. Specify where the chart is placed, using the **Place graph** list and the **Header** and **Footer** options. The placement of a chart determines which data is displayed and where it is shown. For example, if you place a chart in the Report Header section, the chart includes data for the entire report. If you place it in a Group header or Group footer section, it displays group specific data.

NOTE	When you summarize a field, the data is automatically grouped.

5. Choose a value from the **On change of** list to specify when a point is to be plotted. This determines the x-axis values for 2-dimensional charts.

6. Choose a value from the **Show Value(s)** list to specify what data you want to chart. This determines the y-axis values for 2-dimensional charts.

7. If you wish, you can use the **Chart Text** tab to add titles to the axes, the **Options** tab to specify the chart color and legend and the **Axes** tab to control gridlines and tick marks.

8. Click **OK**.

Charting on a detail field

You can also chart on the values that appear in the details section of the report.

To chart on a detail field, you need to specify two things:

- The condition, which indicates when to plot the point.
  For example, a chart showing last year's sales for your customers uses the Customer Name field as the condition. Each time the condition - the customer name - changes, a point is plotted.

There can be up to two conditions when charting on a detail field.

- The value, which indicates what information is to be plotted on the chart.
  For example, to create a chart showing last year's sales for your customers, use the Last Year's Sales field as the value.

You can specify multiple values.

To chart on a detail field:

1. Choose **Insert │ Chart,** or click the **Chart** button on the toolbar.

2. In the **Chart Type** tab of the Chart Expert, choose a chart type from the list. Click one of the buttons on the right to select the specific style you want for your chart type.
   For example, you can choose 'Bar Chart' as the chart type, and 'Stacked Bar' as the specific style.
   For a description of chart types, see 'Chart Type Tab' in the On-line Help file.

3. Click the **Data Field** tab, and click the **Detail** button to specify a detail chart.

4. Use the **Place Chart** list and the **Header** and **Footer** options to specify chart placement. The placement of a chart determines which data is displayed and where it is printed. For example, if you place a chart in the Report header section, the chart includes data for the entire report. If you place it in a Group header or Group footer section, it displays group specific data.

5. Select fields from the **Available Fields** tree view which specify when you want a point to be plotted and click the arrow (>) button to add them to the **On change of** list.

6. Select fields from the **Available Fields** tree view which specify the values you want to plot, then click the arrow (>) button to move them to the **Show Values** list.

7. If you want to summarize the values you have chosen, select a summary operation from the **Data summary type** list.

8. If you want, you can use the **Chart Text** tab to add titles to the axes, the **Options** tab to specify the chart color and legend and the **Axes** tab to control gridlines and tick marks.

9. Click **OK**.

Positioning a chart

If the chart is not in the correct position, you can move or resize it. You do this in the same way as positioning other report objects. See 'Changing row and column spacing'.

Formatting a chart

You can change an existing chart to alter any aspect of its formatting.

If you edit a chart in a group header or footer, editing one instance of the chart will change the format of all occurrences of the chart.

The outline steps for editing a chart in the report are:

1. Right-click the chart and choose **Format Chart** from the shortcut menu.

2. Make your changes in the **Chart Expert** (see below for how to do specific changes).

3. Click **OK**.

Changing the chart type

A wide range of chart types is available, so that you can pick the chart which best displays your data.

1. Select the **Chart Type** tab of the Chart Expert, and choose a chart type from the list. On the right, a set of buttons shows the available styles within the selected type.

2. Click one of the buttons on the right to select a particular style of chart. A short description of the type is shown in the **Chart Description** area.

For a description of chart types and how they might be used, refer to the *Chart type tab* section under the *Chart Expert dialog box* in the On-line Help.

Changing the chart axes

The chart axes are automatically scaled and given gridlines and tick marks. You can control these settings using the **Axes** tab of the Chart Expert.

Changing the range of axes

You can use the **Axes Tickers** area in the **Axes** tab of the Chart Expert to control the highest and lowest values shown and the format of labels.

To change the range of an axis:

1. In the **Axes Tickers** area, unselect **Auto Range**.

2. Enter new values in the **Maximum** and **Minimum** boxes.

3. If you want to change the format of the axis labels, choose an option from the **Number format** list.

Changing the divisions of axes

You can use the Axes Divisions area in the Axes tab of the Chart Expert to control the number of divisions in an axis

1. In the **Axes** tab of the Chart Expert, click **Manual**.

2. In the box to the right, type the number of divisions you want.

Displaying data values on the chart

You can control whether the data values are shown on the chart and the format in which they appear.

1. In the **Chart Type** tab of the Chart Expert, select one of the options in the **Data Points** area.

   - **None** does show a data value.

   - **Show Label** shows a label.

   - **Show Value** shows the value of the point.

2. If you selected **Show Value**, select a format for the numbers using the **Number format** list. You can use this formatting to add a dollar sign in front of the numbers, or a percent sign after them.

Changing the chart legend

You can control whether a legend appears, and where it is positioned.

1. In the **Options** tab of the Chart Expert, select **Show Legend** to display a chart legend.

2. Choose a position for the legend from the **Placement** list.

Changing the text on the chart

You can change the text that appears in the chart, such as the chart title or axis titles.

1. In the **Chart Text** tab of the Chart Expert, enter the text you want to appear in each box beside each item in the Titles Text area.

2. Format the text by choosing an item from the list and clicking the **Style** button.

Pulling out pie slices

You can give a pie chart more impact by separating one of the slices from the rest.

1. In the **Chart Type** tab of the Chart Expert, select **Detach Slice** in the **Chart Options** area.

2. Select which slice you want to detach, by selecting **Largest slice** or **Smallest slice**.

Saving a report

The **Save** command saves the whole Analysis document. If you have created an Analysis document, or edited an existing one, the following information is saved to a `.rpt` file:

*   the query,

*   the report,

*   the Worksheet view, and

*   the cube definition.

The saved Worksheet view includes everything displayed in the Worksheet. When you next open the document, the Worksheet looks the same as when you saved it, right down to the cell formatting, the sorting and the chart.

To save the document:

*   Click the **Save** button on the toolbar, or choose **Save** from the **File** menu.

*   If you want to save the current data in the report, check the **Save with Data** option on the **File** menu before saving. Otherwise, the report may show different data when you next open it.

*   To save the document to a different file, use the **Save As** command.

Exporting a report

To gain further benefit from the information in the report, you can export it to other commonly used formats. The formats you can use are:

*   Crystal Reports (`.rpt`)

*   Microsoft Word (`.doc`)

*   Microsoft Excel (`.xls`)

*   Rich Text (`.rtf`)

To export a report to a file:

1. Choose **File | Export**, or click the **Export** button on the toolbar.

2. In the Export dialog box, choose an export format from the **Export Format** list.

3. Enter the file name in the **Save To** box, or click **Browse** to select a file path.

4. Click **OK**.

Printing a report

To print a report:

1. Choose **File | Print**, or click the **Print** button on the toolbar.

2. In the Windows Print dialog box, specify the print options you want.

3. Click **OK**.

Finding More Information

The following reference materials were used during the creation of this Technical Support document, or are recommended for further reading on subjects covered by the Topics of this Module Section.

Seagate Analysis On-line Help

Seagate Crystal Reports User's Guide (Version 7)

Contacting Crystal Decisions for Technical Support

We recommend that you refer to the product documentation and that you visit our Technical Support web site for more resources.

Self-serve Support:

http://support.crystaldecisions.com/

Email Support:

http://support.crystaldecisions.com/support/answers.asp

Telephone Support:

http://www.crystaldecisions.com/contact/support.asp

Page: 8

[SS1]What's the fifth part?

Page: 26

[RK2] this is misleading

Page: 28

[RK3] simple = single?

Page: 33

[SS4]