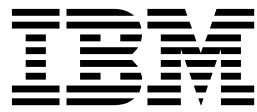


*IBM SPSS Statistics 25 Core System
User's Guide*



Note

Before using this information and the product it supports, read the information in "Notices" on page 289.

Product Information

This edition applies to version 25, release 0, modification 0 of IBM SPSS Statistics and to all subsequent releases and modifications until otherwise indicated in new editions.

Contents

Chapter 1. Overview	1
Windows	1
Designated window versus active window	1
Variable names and variable labels in dialog box lists	2
Data type, measurement level, and variable list icons	2
Statistics Coach	2
Finding out more	3
Chapter 2. Getting Help	5
Chapter 3. Data files	7
Opening data files	7
To open data files	7
Data file types	7
Reading Excel Files	8
Reading Older Excel Files and Other Spreadsheets	9
Reading dBASE files	9
Reading Stata files	9
Reading CSV Files	10
Text Wizard	10
Reading Database Files	14
Reading Cognos BI data	19
Reading Cognos TM1 data	21
File information	23
Saving data files	23
To save modified data files	23
To save data files in code page character encoding	23
Saving data files in external formats	24
Saving data files in Excel format	27
Saving data files in SAS format	27
Saving data files in Stata format	28
Saving Subsets of Variables	29
Encrypting data files	29
Exporting to a Database	30
Exporting to Data Collection	36
Exporting to Cognos TM1	36
Comparing datasets	38
Compare Datasets: Compare tab	38
Compare Datasets: Attributes tab	39
Comparing datasets: Output tab	39
Protecting original data	40
Virtual Active File	40
Creating a Data Cache	41
Chapter 4. Distributed Analysis Mode	43
Server Login	43
Adding and Editing Server Login Settings	43
To Select, Switch, or Add Servers	44
Searching for Available Servers	45
Opening Data Files from a Remote Server	45
File Access in Local and Distributed Analysis Mode	45
Availability of Procedures in Distributed Analysis Mode	46
Absolute versus Relative Path Specifications	46
Chapter 5. Data Editor	49
Data View	49
Variable View	49
To display or define variable attributes	50
Variable names	50
Variable measurement level	51
Variable type	51
Variable labels	53
Value labels	53
Inserting line breaks in labels	53
Missing values	53
Roles	54
Column width	54
Variable alignment	54
Applying variable definition attributes to multiple variables	54
Custom Variable Attributes	55
Customizing Variable View	57
Spell checking	58
Entering data	58
To enter numeric data	58
To enter non-numeric data	59
To use value labels for data entry	59
Data value restrictions in the data editor	59
Editing data	59
Replacing or modifying data values	59
Cutting, copying, and pasting data values	59
Inserting new cases	60
Inserting new variables	60
To change data type	61
Finding cases, variables, or imputations	61
Finding and replacing data and attribute values	62
Obtaining Descriptive Statistics for Selected Variables	62
Case selection status in the Data Editor	63
Data Editor display options	63
Data Editor printing	64
To print Data Editor contents	64
Chapter 6. Working with Multiple Data Sources	65
Basic Handling of Multiple Data Sources	65
Working with Multiple Datasets in Command Syntax	65
Copying and Pasting Information between Datasets	65
Renaming Datasets	66
Suppressing Multiple Datasets	66
Chapter 7. Data preparation	67
Variable properties	67
Defining Variable Properties	67
To Define Variable Properties	68
Defining Value Labels and Other Variable Properties	68
Assigning the Measurement Level	69

Custom Variable Attributes	70
Copying Variable Properties	70
Setting measurement level for variables with unknown measurement level	70
Multiple Response Sets	71
Defining Multiple Response Sets	71
Copy Data Properties	73
Copying Data Properties	73
Identifying Duplicate Cases	76
Visual Binning	77
To Bin Variables	77
Binning Variables	78
Automatically Generating Binned Categories	79
Copying Binned Categories	80
User-Missing Values in Visual Binning	81
Chapter 8. Data Transformations	83
Data Transformations	83
Computing Variables	83
Compute Variable: If Cases	83
Compute Variable: Type and Label	84
Functions	84
Missing Values in Functions	84
Random Number Generators	85
Count Occurrences of Values within Cases	85
Count Values within Cases: Values to Count	85
Count Occurrences: If Cases	85
Shift Values	86
Recoding Values	86
Recode into Same Variables	86
Recode into Same Variables: Old and New Values	87
Recode into Different Variables	88
Recode into Different Variables: Old and New Values	88
Automatic Recode	89
Rank Cases	90
Rank Cases: Types	91
Rank Cases: Ties	91
Date and Time Wizard	92
Dates and Times in IBM SPSS Statistics	92
Create a Date/Time Variable from a String	93
Create a Date/Time Variable from a Set of Variables	93
Add or Subtract Values from Date/Time Variables	94
Extract Part of a Date/Time Variable	96
Time Series Data Transformations	96
Define Dates	96
Create Time Series	97
Replace Missing Values	99
Chapter 9. File handling and file transformations	101
File handling and file transformations	101
Sort cases	101
Sort variables	102
Transpose	103
Merging Data Files	103
Add Cases	103
Add Variables	104
Aggregate Data	107
Aggregate Data: Aggregate Function	108
Aggregate Data: Variable Name and Label	108
Split file	109
Select cases	109
Select cases: If	110
Select cases: Random sample	110
Select cases: Range	111
Weight cases	111
Restructuring Data	111
To Restructure Data	112
Restructure Data Wizard: Select Type	112
Restructure Data Wizard (Variables to Cases): Number of Variable Groups	114
Restructure Data Wizard (Variables to Cases): Select Variables	115
Restructure Data Wizard (Variables to Cases): Create Index Variables	116
Restructure Data Wizard (Variables to Cases): Create One Index Variable	117
Restructure Data Wizard (Variables to Cases): Create Multiple Index Variables	117
Restructure Data Wizard (Cases to Variables): Options	118
Restructure Data Wizard (Cases to Variables): Select Variables	118
Restructure Data Wizard (Cases to Variables): Sort Data	119
Restructure Data Wizard (Cases to Variables): Options	119
Restructure Data Wizard: Finish	120
Chapter 10. Working with output	121
Working with output	121
Viewer	121
Showing and hiding results	121
Moving, deleting, and copying output	121
Changing initial alignment	122
Changing alignment of output items	122
Viewer outline	122
Adding items to the Viewer	123
Finding and replacing information in the Viewer	123
Closing output items	124
Copying output into other applications	124
Interactive output	126
Export output	126
HTML options	127
Web report options	128
Word/RTF options	129
Excel options	129
PowerPoint options	130
PDF options	130
Text options	131
Graphics only options	132
Graphics format options	132
Viewer printing	133
To print output and charts	133
Print Preview	133
Page Attributes: Headers and Footers	133
Page Attributes: Options	134
Saving output	134

To save a Viewer document	135
Chapter 11. Pivot tables	137
Pivot tables	137
Manipulating a pivot table	137
Activating a pivot table	137
Pivoting a table.	137
Changing display order of elements within a dimension	137
Moving rows and columns within a dimension element	137
Transposing rows and columns	138
Grouping rows or columns	138
Ungrouping rows or columns	138
Rotating row or column labels.	138
Sorting rows.	138
Inserting rows and columns	139
Controlling display of variable and value labels	139
Changing the output language	139
Navigating large tables	140
Undoing changes	140
Working with layers	140
Creating and displaying layers	140
Go to layer category	141
Showing and hiding items	141
Hiding rows and columns in a table.	141
Showing hidden rows and columns in a table	141
Hiding and showing dimension labels	141
Hiding and showing table titles	141
TableLooks	141
To apply a TableLook.	142
To edit or create a TableLook	142
Table properties	142
To change pivot table properties	142
Table properties: general.	142
Table properties: notes	143
Table properties: cell formats	144
Table properties: borders	144
Table properties: printing	144
Cell properties	145
Font and background.	145
Format value	145
Alignment and margins	145
Footnotes and captions	145
Adding footnotes and captions	145
To hide or show a caption	146
To hide or show a footnote in a table	146
Footnote marker	146
Renumbering footnotes	147
Editing footnotes in legacy tables.	147
Data cell widths	147
Changing column width.	148
Displaying hidden borders in a pivot table	148
Selecting rows, columns, and cells in a pivot table	148
Printing pivot tables	148
Controlling table breaks for wide and long tables	148
Creating a chart from a pivot table	149
Legacy tables	149
Chapter 12. Models	151
Interacting with a model	151
Working with the Model Viewer	151
Printing a model	152
Exporting a model.	153
Saving fields used in the model to a new dataset	153
Saving predictors to a new dataset based on importance	153
Ensemble Viewer	153
Models for Ensembles	153
Split Model Viewer	155
Chapter 13. Automated Output Modification	157
Style Output: Select	157
Style Output	158
Style Output: Labels and Text	160
Style Output: Indexing	160
Style Output: TableLooks	161
Style Output: Size	161
Table Style	161
Table Style: Condition	162
Table Style: Format	162
Chapter 14. Working with Command Syntax	165
Syntax Rules	165
Pasting Syntax from Dialog Boxes	166
To Paste Syntax from Dialog Boxes	166
Copying Syntax from the Output Log	166
To Copy Syntax from the Output Log	167
Using the Syntax Editor	167
Syntax Editor Window	167
Terminology.	168
Auto-Completion	169
Color Coding	169
Breakpoints	170
Bookmarks	171
Commenting or Uncommenting Text	172
Formatting Syntax.	172
Running Command Syntax.	176
Character Set Encoding in Syntax Files	176
Multiple Execute Commands	177
Character Set Encoding in Syntax Files	177
Multiple Execute Commands	178
Encrypting syntax files	178
Chapter 15. Overview of the chart facility	181
Building and editing a chart	181
Building Charts	181
Editing Charts	182
Chart definition options	183
Adding and Editing Titles and Footnotes	183
Setting General Options	184
Chapter 16. Scoring data with predictive models	185
Scoring Wizard	185

Matching model fields to dataset fields	186
Selecting scoring functions	187
Scoring the active dataset	188
Merging model and transformation XML files	188
Chapter 17. Utilities	191
Utilities	191
Variable information	191
Data file comments	191
Variable sets	192
Defining variable sets	192
Using variable sets to show and hide variables	192
Reordering target variable lists	193
Chapter 18. Options	195
Options	195
General options	195
Viewer options	196
Data Options	197
Changing the default variable view	199
Language options	199
Currency options	200
To create custom currency formats	200
Output options	200
Chart options	201
Data Element Colors	201
Data Element Lines	201
Data Element Markers	202
Data Element Fills	202
Pivot table options	203
File locations options	204
Script options	205
Syntax editor options	205
Multiple imputations options	206
Chapter 19. Customizing Menus and Toolbars	209
Customizing Menus and Toolbars	209
Menu Editor	209
Customizing Toolbars	209
Show Toolbars	209
To Customize Toolbars	209
Toolbar Properties	210
Edit Toolbar	210
Create New Tool	210
Chapter 20. Extensions	211
Extension Hub	211
Explore tab	212
Installed tab	212
Settings	213
Extension Details	213
Installing local extension bundles	214
Installation locations for extensions	214
Required R packages	215
Batch installation of extension bundles	216
Creating and managing custom dialogs	216
Custom Dialog Builder layout	217
Building a custom dialog	217
Dialog Properties	218
Specifying the Menu Location for a Custom Dialog	219
Laying out controls on the canvas	220
Building the Syntax Template	220
Previewing a custom dialog	222
Control types	222
Extension Properties	243
Managing custom dialogs	246
Custom Dialogs for Extension Commands	250
Creating Localized Versions of Custom Dialogs	250
Creating and editing extension bundles	252
Chapter 21. Production jobs	255
Syntax files	256
Output	256
HTML options	257
PowerPoint options	257
PDF options	257
Text options	258
Production jobs with OUTPUT commands	258
Runtime values	258
Run options	259
Server login	259
Adding and Editing Server Login Settings	259
User prompts	260
Background job status	260
Running production jobs from a command line	260
Converting Production Facility files	261
Chapter 22. Output Management System	263
Output object types	264
Command identifiers and table subtypes	265
Labels	266
OMS options	266
Logging	268
Excluding output display from the viewer	269
Routing output to IBM SPSS Statistics data files	269
Data files created from multiple tables	269
Controlling column elements to control variables in the data file	270
Variable names in OMS-generated data files	270
OXML table structure	270
OMS identifiers	273
Copying OMS identifiers from the viewer outline	273
Chapter 23. Scripting Facility	275
Autoscripts	276
Creating Autoscripts	276
Associating Existing Scripts with Viewer Objects	277
Scripting with the Python Programming Language	277
Running Python Scripts and Python programs	278
Script Editor for the Python Programming Language	279
Scripting in Basic	279
Compatibility with Versions Prior to 16.0	279
The scriptContext Object	281
Startup Scripts	282

Chapter 24. TABLES and IGRAPH Command Syntax Converter	285	Index	293
Chapter 25. Encrypting data files, output documents, and syntax files . .		287	
Notices	289		
Trademarks	291		

Chapter 1. Overview

Windows

There are a number of different types of windows in IBM® SPSS® Statistics:

Data Editor. The Data Editor displays the contents of the data file. You can create new data files or modify existing data files with the Data Editor. If you have more than one data file open, there is a separate Data Editor window for each data file.

Viewer. All statistical results, tables, and charts are displayed in the Viewer. You can edit the output and save it for later use. A Viewer window opens automatically the first time you run a procedure that generates output.

Pivot Table Editor. Output that is displayed in pivot tables can be modified in many ways with the Pivot Table Editor. You can edit text, swap data in rows and columns, add color, create multidimensional tables, and selectively hide and show results.

Chart Editor. You can modify high-resolution charts and plots in chart windows. You can change the colors, select different type fonts or sizes, switch the horizontal and vertical axes, rotate 3-D scatterplots, and even change the chart type.

Text Output Editor. Text output that is not displayed in pivot tables can be modified with the Text Output Editor. You can edit the output and change font characteristics (type, style, color, size).

Syntax Editor. You can paste your dialog box choices into a syntax window, where your selections appear in the form of command syntax. You can then edit the command syntax to use special features that are not available through dialog boxes. You can save these commands in a file for use in subsequent sessions.

Designated window versus active window

If you have more than one open Viewer window, output is routed to the **designated** Viewer window. If you have more than one open Syntax Editor window, command syntax is pasted into the designated Syntax Editor window. The designated windows are indicated by a plus sign in the icon in the title bar. You can change the designated windows at any time.

The designated window should not be confused with the **active** window, which is the currently selected window. If you have overlapping windows, the active window appears in the foreground. If you open a window, that window automatically becomes the active window and the designated window.

Changing the designated window

1. Make the window that you want to designate the active window (click anywhere in the window).
2. From the menus choose:

Utilities > Designate Window

Note: For Data Editor windows, the active Data Editor window determines the dataset that is used in subsequent calculations or analyses. There is no "designated" Data Editor window. See the topic "Basic Handling of Multiple Data Sources" on page 65 for more information.

Variable names and variable labels in dialog box lists

You can display either variable names or variable labels in dialog box lists, and you can control the sort order of variables in source variable lists. To control the default display attributes of variables in source lists, choose **Options** on the Edit menu. See the topic “General options” on page 195 for more information.

You can also change the variable list display attributes within dialogs. The method for changing the display attributes depends on the dialog:

- If the dialog provides sorting and display controls above the source variable list, use those controls to change the display attributes.
- If the dialog does not contain sorting controls above the source variable list, right-click any variable in the source list and select the display attributes from the pop-up menu.

You can display either variable names or variable labels (names are displayed for any variables without defined labels), and you can sort the source list by file order, alphabetical order, or measurement level. (In dialogs with sorting controls above the source variable list, the default selection of **None** sorts the list in file order.)

Data type, measurement level, and variable list icons

The icons that are displayed next to variables in dialog box lists provide information about the variable type and measurement level.

Table 1. Measurement level icons

	Numeric	String	Date	Time
Scale (Continuous)		n/a		
Ordinal				
Nominal				

- For more information on measurement level, see “Variable measurement level” on page 51.
- For more information on numeric, string, date, and time data types, see “Variable type” on page 51.

Statistics Coach

If you are unfamiliar with IBM SPSS Statistics or with the available statistical procedures, the Statistics Coach can help you get started by prompting you with simple questions, nontechnical language, and visual examples that help you select the basic statistical and charting features that are best suited for your data.

To use the Statistics Coach, from the menus in any IBM SPSS Statistics window choose:

Help > Statistics Coach

The Statistics Coach covers only a selected subset of procedures. It is designed to provide general assistance for many of the basic, commonly used statistical techniques.

Finding out more

For a comprehensive overview of the basics, see the online tutorial. From any IBM SPSS Statistics menu choose:

Help > Tutorial

Chapter 2. Getting Help

The Help system contains a number of different sections.

Help Information on the user interface. There is a separate section for each optional module.

Reference

Reference information for the command language and the GPL graphics language. The reference material for the command language is also available in PDF form: **Help>Command Syntax Reference**.

Tutorial

Step-by-step instructions on how to use many of the basic features.

Case Studies

Hands-on examples of how to create various types of statistical analyses and how to interpret the results.

Statistics Coach

Guides you through the process of finding the procedure that you want to use.

Integration Plug-ins

Separate sections for the Python and R plug-ins.

Context-sensitive help

In many places in the user interface, you can get context-sensitive help.

- Help buttons in dialog boxes take you directly to the help topic for that dialog.
- Right-click on terms in an activated pivot table in the Viewer and choose **What's This?** from the pop-up menu to display definitions of the terms.
- In a command syntax window, position the cursor anywhere within a syntax block for a command and press F1 on the keyboard. The help for that command is displayed

Other resources

Answers to many common problems can be found at <http://www.ibm.com/support> .

If you're a student using a student, academic or grad pack version of any IBM SPSS software product, please see our special online Solutions for Education pages for students. If you're a student using a university-supplied copy of the IBM SPSS software, please contact the IBM SPSS product coordinator at your university.

The IBM SPSS Predictive Analytics community has resources for all levels of users and application developers. Download utilities, graphics examples, new statistical modules, and articles. Visit the IBM SPSS Predictive Analytics community at <https://developer.ibm.com/predictiveanalytics/>.

Documentation in PDF format for statistical algorithms and command syntax is available at <http://www.ibm.com/support/docview.wss?uid=swg27049428>.

Chapter 3. Data files

Data files come in a wide variety of formats, and this software is designed to handle many of them, including:

- Excel spreadsheets
- Database tables from many database sources, including Oracle, SQLServer, DB2, and others
- Tab-delimited, CSV, and other types of simple text files
- SAS data files
- Stata data files

Opening data files

In addition to files saved in IBM SPSS Statistics format, you can open Excel, SAS, Stata, tab-delimited, and other files without converting the files to an intermediate format or entering data definition information.

- Opening a data file makes it the active dataset. If you already have one or more open data files, they remain open and available for subsequent use in the session. Clicking anywhere in the Data Editor window for an open data file will make it the active dataset. See the topic Chapter 6, “Working with Multiple Data Sources,” on page 65 for more information.
- In distributed analysis mode using a remote server to process commands and run procedures, the available data files, folders, and drives are dependent on what is available on or from the remote server. The current server name is indicated at the top of the dialog box. You will not have access to data files on your local computer unless you specify the drive as a shared device and the folders containing your data files as shared folders. See the topic Chapter 4, “Distributed Analysis Mode,” on page 43 for more information.

To open data files

1. From the menus choose:
File > Open > Data...
2. In the Open Data dialog box, select the file that you want to open.
3. Click **Open**.

Optionally, you can:

- Automatically set the width of each string variable to the longest observed value for that variable using **Minimize string widths based on observed values**. This is particularly useful when reading code page data files in Unicode mode. See the topic “General options” on page 195 for more information.
- Read variable names from the first row of spreadsheet files.
- Specify a range of cells to read from spreadsheet files.
- Specify a worksheet within an Excel file to read (Excel 95 or later).

For information on reading data from databases, see “Reading Database Files” on page 14. For information on reading data from text data files, see “Text Wizard” on page 10. For information on reading IBM Cognos® data, see “Reading Cognos BI data” on page 19.

Data file types

SPSS Statistics. Opens data files that are saved in IBM SPSS Statistics format and also the DOS product SPSS/PC+.

SPSS Statistics Compressed. Opens data files that are saved in IBM SPSS Statistics compressed format.

SPSS/PC+. Opens SPSS/PC+ data files. This option is available only on Windows operating systems.

Portable. Opens data files that are saved in portable format. Saving a file in portable format takes considerably longer than saving the file in IBM SPSS Statistics format.

Excel. Opens Excel files.

Lotus 1-2-3. Opens data files that are saved in 1-2-3 format for release 3.0, 2.0, or 1A of Lotus.

SYLK. Opens data files that are saved in SYLK (symbolic link) format, a format that is used by some spreadsheet applications.

dBASE. Opens dBASE-format files for either dBASE IV, dBASE III or III PLUS, or dBASE II. Each case is a record. Variable and value labels and missing-value specifications are lost when you save a file in this format.

SAS. SAS versions 6–9 and SAS transport files.

Stata. Stata versions 4–14 (Stata 14 supports Unicode data).

Reading Excel Files

This topic applies to Excel 95 and later files. To read Excel 4 or earlier versions, see the topic “Reading Older Excel Files and Other Spreadsheets” on page 9.

Worksheet

Excel files can contain multiple worksheets. By default, the Data Editor reads the first worksheet. To read a different worksheet, select the worksheet from the list.

Range You can also read a range of cells. Use the same method for specifying cell ranges as you would in Excel. For example: A1:D10.

Read variable names from first row of data

You can read variable names from the first row of the file or the first row of the defined range. Values that don't conform to variable naming rules are converted to valid variable names, and the original names are used as variable labels.

Percentage of values that determine data type

The data type for each variable is determined by the percentage of values that conform to the same format.

- The value must be greater than 50.
- The denominator used to determine the percentage is the number of non-blank values for each variable.
- If no consistent format is used by the specified percentage of values, the variable is assigned the string data type.
- For variables that are assigned a numeric format (including date and time formats) based on the percentage value, values that do not conform to that format are assigned the system-missing value.

Ignore hidden rows and columns

Hidden rows and columns in the Excel file are not included. This option is available only for Excel 2007 and later files (XLSX, XLSM).

Remove leading spaces from string values

Any blank spaces at the beginning of string values are removed.

Remove trailing spaces from string values

Blank spaces at the end of the string values are removed. This setting affects the calculation of the defined width of string variables.

Reading Older Excel Files and Other Spreadsheets

This topic applies to reading Excel 4 or earlier files, Lotus 1-2-3 files and SYLK format spreadsheet files. For information on reading Excel 95 or later files, see the topic "Reading Excel Files" on page 8.

Read variable names. For spreadsheets, you can read variable names from the first row of the file or the first row of the defined range. The values are converted as necessary to create valid variable names, including converting spaces to underscores.

Range. For spreadsheet data files, you can also read a range of cells. Use the same method for specifying cell ranges as you would with the spreadsheet application.

How Spreadsheets are Read

- The data type and width for each variable are determined by the column width and data type of the first data cell in the column. Values of other types are converted to the system-missing value. If the first data cell in the column is blank, the global default data type for the spreadsheet (usually numeric) is used.
- For numeric variables, blank cells are converted to the system-missing value, indicated by a period. For string variables, a blank is a valid string value, and blank cells are treated as valid string values.
- If you do not read variable names from the spreadsheet, the column letters (*A, B, C, ...*) are used for variable names for Excel and Lotus files. For SYLK files and Excel files saved in R1C1 display format, the software uses the column number preceded by the letter *C* for variable names (*C1, C2, C3, ...*).

Reading dBASE files

Database files are logically very similar to IBM SPSS Statistics data files. The following general rules apply to dBASE files:

- Field names are converted to valid variable names.
- Colons used in dBASE field names are translated to underscores.
- Records marked for deletion but not actually purged are included. The software creates a new string variable, *D_R*, which contains an asterisk for cases marked for deletion.

Reading Stata files

The following general rules apply to Stata data files:

- **Variable names.** Stata variable names are converted to IBM SPSS Statistics variable names in case-sensitive form. Stata variable names that are identical except for case are converted to valid variable names by appending an underscore and a sequential letter (*_A, _B, _C, ..., _Z, _AA, _AB, ...*, and so forth).
- **Variable labels.** Stata variable labels are converted to IBM SPSS Statistics variable labels.
- **Value labels.** Stata value labels are converted to IBM SPSS Statistics value labels, except for Stata value labels assigned to "extended" missing values. Value labels longer than 120 bytes are truncated.
- **String variables.** Stata strl variables are converted to string variables. Values longer than 32K bytes are truncated. Stata strl values that contain blobs (binary large objects) are converted to blank strings.
- **Missing values.** Stata "extended" missing values are converted to system-missing values.
- **Date conversion.** Stata date format values are converted to IBM SPSS Statistics DATE format (d-m-y) values. Stata "time-series" date format values (weeks, months, quarters, and so on) are converted to simple numeric (F) format, preserving the original, internal integer value, which is the number of weeks, months, quarters, and so on, since the start of 1960.

Reading CSV Files

To read CSV files, from the menus choose: **File > Import Data > CSV**

The Read CSV File dialog reads CSV format text data files that use a comma, a semicolon, or a tab as the delimiter between values.

If the text file uses a different delimiter, contains text at the beginning of the file that is not variable names or data values, or has other special considerations, use the Text Wizard to read the files.

First line contains variable names

The first non-blank line in the file contains label text that is used as variable names. Values that are invalid as variable names are automatically converted to valid variable names.

Remove leading spaces from string values

Any blank spaces at the beginning of string values are removed.

Remove trailing spaces from string values

Blank spaces at the end of the string values are removed. This setting affects the calculation of the defined width of string variables.

Delimiter between values

The delimiter can be a comma, a semicolon, or a tab. If the delimiter is any other character or a blank space, use the Text Wizard to read the file.

Decimal symbol

The symbol that is used to indicate decimals in the text data file. The symbol can be a period or a comma.

Text Qualifier

Character that is used to enclose values that contain the delimiter character. The qualifier appears at the start and the end of the value. The qualifier can be double quotation mark, single quotation mark, or none.

Percentage of values that determine data type

The data type for each variable is determined by the percentage of values that conform to the same format.

- The value must be greater than 50.
- If no consistent format is used by the specified percentage of values, the variable is assigned the string data type.
- For variables that are assigned a numeric format (including date and time formats) based on the percentage value, values that do not conform to that format are assigned the system-missing value.

Cache data locally

A data cache is a complete copy of the data file that is stored in temporary disk space. Caching the data file can improve performance.

Text Wizard

The Text Wizard can read text data files formatted in a variety of ways:

- Tab-delimited files
- Space-delimited files
- Comma-delimited files
- Fixed-field format files

For delimited files, you can also specify other characters as delimiters between values, and you can specify multiple delimiters.

To Read Text Data Files

1. From the menus choose:
File > Import Data > Text Data...
2. Select the text file in the Open Data dialog box.
3. If necessary, select the encoding of the file.
4. Follow the steps in the Text Wizard to define how to read the data file.

Encoding

The encoding of a file affects the way character data are read. Unicode data files typically contain a byte order mark that identifies the character encoding. Some applications create Unicode files without a byte order mark, and code page data files do not contain any encoding identifier.

- **Unicode (UTF-8).** Reads the file as Unicode UTF-8.
- **Unicode (UTF-16).** Reads the file as Unicode UTF-16 in the endianness of the operating system.
- **Unicode (UTF-16BE).** Reads the file as Unicode UTF-16, big endian.
- **Unicode (UTF-16LE).** Reads the file as Unicode UTF-16, little endian.
- **Local Encoding.** Reads the file in current locale code page character encoding.

If a file contains a Unicode byte order mark, it is read in that Unicode encoding, regardless of the encoding you select. If a file does not contain a Unicode byte order mark, by default the encoding is assumed to be the current locale code page character encoding, unless you select one of the Unicode encodings.

To change the current locale for data files in a different code page character encoding, select Edit>Options from the menus, and change the locale on the Language tab.

Text Wizard: Step 1

The text file is displayed in a preview window. You can apply a predefined format (previously saved from the Text Wizard) or follow the steps in the Text Wizard to specify how the data should be read.

Text Wizard: Step 2

This step provides information about variables. A variable is similar to a field in a database. For example, each item in a questionnaire is a variable.

How are your variables arranged?

The arrangement of variables defines the method that is used to differentiate one variable from the next.

Delimited

Spaces, commas, tabs, or other characters are used to separate variables. The variables are recorded in the same order for each case but not necessarily in the same column locations.

Fixed width

Each variable is recorded in the same column location on the same record (line) for each case in the data file. No delimiter is required between variables. The column location determines which variable is being read.

Note: The Text Wizard cannot read fixed-width Unicode text files. You can use the DATA LIST command to read fixed-width Unicode files.

Are variable names included at the top of your file?

The values on the specified line number are used to create variable names. Values that don't conform to variable naming rules are converted to valid variable names.

What is the decimal symbol?

The character that indicates decimal values can be either a period or a comma.

Text Wizard: Step 3 (Delimited Files)

This step provides information about cases. A case is similar to a record in a database. For example, each respondent to a questionnaire is a case.

The first case of data begins on which line number? Indicates the first line of the data file that contains data values. If the top line(s) of the data file contain descriptive labels or other text that does not represent data values, this will *not* be line 1.

How are your cases represented? Controls how the Text Wizard determines where each case ends and the next one begins.

- **Each line represents a case.** Each line contains only one case. It is fairly common for each case to be contained on a single line (row), even though this can be a very long line for data files with a large number of variables. If not all lines contain the same number of data values, the number of variables for each case is determined by the line with the greatest number of data values. Cases with fewer data values are assigned missing values for the additional variables.
- **A specific number of variables represents a case.** The specified number of variables for each case tells the Text Wizard where to stop reading one case and start reading the next. Multiple cases can be contained on the same line, and cases can start in the middle of one line and be continued on the next line. The Text Wizard determines the end of each case based on the number of values read, regardless of the number of lines. Each case must contain data values (or missing values indicated by delimiters) for all variables, or the data file will be read incorrectly.

How many cases do you want to import? You can import all cases in the data file, the first n cases (n is a number you specify), or a random sample of a specified percentage. Since the random sampling routine makes an independent pseudo-random decision for each case, the percentage of cases selected can only approximate the specified percentage. The more cases there are in the data file, the closer the percentage of cases selected is to the specified percentage.

Text Wizard: Step 3 (Fixed-Width Files)

This step provides information about cases. A case is similar to a record in a database. For example, each respondent to questionnaire is a case.

The first case of data begins on which line number? Indicates the first line of the data file that contains data values. If the top line(s) of the data file contain descriptive labels or other text that does not represent data values, this will *not* be line 1.

How many lines represent a case? Controls how the Text Wizard determines where each case ends and the next one begins. Each variable is defined by its line number within the case and its column location. You need to specify the number of lines for each case to read the data correctly.

How many cases do you want to import? You can import all cases in the data file, the first n cases (n is a number you specify), or a random sample of a specified percentage. Since the random sampling routine makes an independent pseudo-random decision for each case, the percentage of cases selected can only approximate the specified percentage. The more cases there are in the data file, the closer the percentage of cases selected is to the specified percentage.

Text Wizard: Step 4 (Delimited Files)

This step specifies delimiters and text qualifiers that are used in the text data file. You can also specify the treatment of leading and trailing spaces in string values.

Which delimiters appear between variables?

The characters or symbols that separate data values. You can select any combination of spaces, commas, semicolons, tabs, or other characters. Multiple, consecutive delimiters without intervening data values are treated as missing values.

What is the text qualifier?

Characters that are used to enclose values that contain delimiter characters. The text qualifier appears at both the beginning and the end of the value, enclosing the entire value.

Leading and Trailing Spaces

Controls the treatment of leading and trailing blank spaces in string values.

Remove leading spaces from string values

Any blank spaces at the beginning of string values are removed.

Remove trailing spaces from string values

Blank spaces at the end of a value are ignored when the defined width of string variables is calculated. If **Space** is selected as a delimiter, multiple consecutive blank spaces are not treated as multiple delimiters.

Text Wizard: Step 4 (Fixed-Width Files)

This step displays the Text Wizard's best guess on how to read the data file and allows you to modify how the Text Wizard will read variables from the data file. Vertical lines in the preview window indicate where the Text Wizard currently thinks each variable begins in the file.

Insert, move, and delete variable break lines as necessary to separate variables. If multiple lines are used for each case, the data will be displayed as one line for each case, with subsequent lines appended to the end of the line.

Notes:

For computer-generated data files that produce a continuous stream of data values with no intervening spaces or other distinguishing characteristics, it may be difficult to determine where each variable begins. Such data files usually rely on a data definition file or some other written description that specifies the line and column location for each variable.

Text Wizard: Step 5

This step controls the variable name and the data format that is used to read each variable. You can also specify variables to exclude.

Variable name

You can overwrite the default variable names with your own variable names. If you read variable names from the data file, names that do not conform to variable naming rules are automatically modified. Select a variable in the preview window and then enter a variable name.

Data format

Select a variable in the preview window and then select a format from the list.

- **Automatic** determines the data format based on an evaluation of all the data values.
- To exclude a variable, select **Do Not Import**.

Percentage of values that determine Automatic data format

For automatic format, the data format for each variable is determined by the percentage of values that conform to the same format.

- The value must be greater than 50.
- The denominator used to determine the percentage is the number of non-blank values for each variable.
- If no consistent format is used by the specified percentage of values, the variable is assigned the string data type.
- For variables that are assigned a numeric format (including date and time formats) based on the percentage value, values that do not conform to that format are assigned the system-missing value.

Text Wizard Formatting Options: Formatting options for reading variables with the Text Wizard include:

Automatic. The format is determined based on an evaluation of all the data values.

Do not import. Omit the selected variable(s) from the imported data file.

Numeric. Valid values include numbers, a leading plus or minus sign, and a decimal indicator.

String. Valid values include virtually any keyboard characters and embedded blanks. For delimited files, you can specify the number of characters in the value, up to a maximum of 32,767. By default, the Text Wizard sets the number of characters to the longest string value encountered for the selected variable(s) in the first 250 rows of the file. For fixed-width files, the number of characters in string values is defined by the placement of variable break lines in step 4.

Date/Time. Valid values include dates of the general format *dd-mm-yyyy*, *mm/dd/yyyy*, *dd.mm.yyyy*, *yyyy/mm/dd*, *hh:mm:ss*, and a variety of other date and time formats. Months can be represented in digits, Roman numerals, or three-letter abbreviations, or they can be fully spelled out. Select a date format from the list.

Dollar. Valid values are numbers with an optional leading dollar sign and optional commas as thousands separators.

Comma. Valid values include numbers that use a period as a decimal indicator and commas as thousands separators.

Dot. Valid values include numbers that use a comma as a decimal indicator and periods as thousands separators.

Note: Values that contain invalid characters for the selected format will be treated as missing. Values that contain any of the specified delimiters will be treated as multiple values.

Text Wizard: Step 6

This is the final step of the Text Wizard. You can save your specifications in a file for use when importing similar text data files. You can also paste the syntax generated by the Text Wizard into a syntax window. You can then customize and/or save the syntax for use in other sessions or in production jobs.

Cache data locally. A data cache is a complete copy of the data file, stored in temporary disk space. Caching the data file can improve performance.

Reading Database Files

You can read data from any database format for which you have a database driver. In local analysis mode, the necessary drivers must be installed on your local computer. In distributed analysis mode (available with IBM SPSS Statistics Server), the drivers must be installed on the remote server. See the topic Chapter 4, “Distributed Analysis Mode,” on page 43 for more information.

Note: If you are running the Windows 64-bit version of IBM SPSS Statistics, you cannot read Excel, Access, or dBASE database sources, even though they may appear on the list of available database sources. The 32-bit ODBC drivers for these products are not compatible.

To Read Database Files

1. From the menus choose:
File > Import Data > Database > New Query...
2. Select the data source.

3. If necessary (depending on the data source), select the database file and/or enter a login name, password, and other information.
4. Select the table(s) and fields. For OLE DB data sources (available only on Windows operating systems), you can only select one table.
5. Specify any relationships between your tables.
6. Optionally:
 - Specify any selection criteria for your data.
 - Add a prompt for user input to create a parameter query.
 - Save your constructed query before running it.

Connection Pooling

If you access the same database source multiple times in the same session or job, you can improve performance with connection pooling.

1. In the last step of the wizard, paste the command syntax into a syntax window.
2. At the end of the quoted CONNECT string, add `Pooling=true`.

To Edit Saved Database Queries

1. From the menus choose:
File > Import Data > Database > Edit Query...
2. Select the query file (*.spq) that you want to edit.
3. Follow the instructions for creating a new query.

To Read Database Files with Saved Queries

1. From the menus choose:
File > Import Data > Database > Run Query...
2. Select the query file (*.spq) that you want to run.
3. If necessary (depending on the database file), enter a login name and password.
4. If the query has an embedded prompt, enter other information if necessary (for example, the quarter for which you want to retrieve sales figures).

Selecting a Data Source

Use the first screen of the Database Wizard to select the type of data source to read.

ODBC Data Sources

If you do not have any ODBC data sources configured, or if you want to add a new data source, click **Add ODBC Data Source**.

- On Linux operating systems, this button is not available. ODBC data sources are specified in *odbc.ini*, and the *ODBCINI* environment variables must be set to the location of that file. For more information, see the documentation for your database drivers.
- In distributed analysis mode (available with IBM SPSS Statistics Server), this button is not available. To add data sources in distributed analysis mode, see your system administrator.

An ODBC data source consists of two essential pieces of information: the driver that will be used to access the data and the location of the database you want to access. To specify data sources, you must have the appropriate drivers installed. Drivers for a variety of database formats are included with the installation media.

Selecting Data Fields

The Select Data step controls which tables and fields are read. Database fields (columns) are read as variables.

If a table has any field(s) selected, all of its fields will be visible in the following Database Wizard windows, but only fields that are selected in this step will be imported as variables. This enables you to create table joins and to specify criteria by using fields that you are not importing.

Displaying field names. To list the fields in a table, click the plus sign (+) to the left of a table name. To hide the fields, click the minus sign (-) to the left of a table name.

To add a field. Double-click any field in the Available Tables list, or drag it to the Retrieve Fields In This Order list. Fields can be reordered by dragging and dropping them within the fields list.

To remove a field. Double-click any field in the Retrieve Fields In This Order list, or drag it to the Available Tables list.

Sort field names. If this check box is selected, the Database Wizard will display your available fields in alphabetical order.

By default, the list of available tables displays only standard database tables. You can control the type of items that are displayed in the list:

- **Tables.** Standard database tables.
- **Views.** Views are virtual or dynamic "tables" defined by queries. These can include joins of multiple tables and/or fields derived from calculations based on the values of other fields.
- **Synonyms.** A synonym is an alias for a table or view, typically defined in a query.
- **System tables.** System tables define database properties. In some cases, standard database tables may be classified as system tables and will only be displayed if you select this option. Access to real system tables is often restricted to database administrators.

Note: For OLE DB data sources (available only on Windows operating systems), you can select fields only from a single table. Multiple table joins are not supported for OLE DB data sources.

Creating a Relationship between Tables

The Specify Relationships step allows you to define the relationships between the tables for ODBC data sources. If fields from more than one table are selected, you must define at least one join.

Establishing relationships. To create a relationship, drag a field from any table onto the field to which you want to join it. The Database Wizard will draw a **join line** between the two fields, indicating their relationship. These fields must be of the same data type.

Auto Join Tables. Attempts to automatically join tables based on primary/foreign keys or matching field names and data type.

Join Type. If outer joins are supported by your driver, you can specify inner joins, left outer joins, or right outer joins.

- **Inner joins.** An inner join includes only rows where the related fields are equal. In this example, all rows with matching ID values in the two tables will be included.
- **Outer joins.** In addition to one-to-one matching with inner joins, you can also use outer joins to merge tables with a one-to-many matching scheme. For example, you could match a table in which there are only a few records representing data values and associated descriptive labels with values in a table containing hundreds or thousands of records representing survey respondents. A left outer join includes all records from the table on the left and, from the table on the right, includes only those

records in which the related fields are equal. In a right outer join, the join imports all records from the table on the right and, from the table on the left, imports only those records in which the related fields are equal.

Computing New Fields

If you are in distributed mode, connected to a remote server (available with IBM SPSS Statistics Server), you can compute new fields before you read the data into IBM SPSS Statistics.

You can also compute new fields after you read the data into IBM SPSS Statistics, but computing new fields in the database can save time for large data sources.

New Field Name. The name must comply with IBM SPSS Statistics variable name rules.

Expression. Enter the expression to compute the new field. You can drag existing field names from the **Fields** list and functions from the **Functions** list.

Limiting Retrieved Cases

The Limit Retrieved Cases step allows you to specify the criteria to select subsets of cases (rows). Limiting cases generally consists of filling the criteria grid with criteria. Criteria consist of two expressions and some relation between them. The expressions return a value of *true*, *false*, or *missing* for each case.

- If the result is *true*, the case is selected.
- If the result is *false* or *missing*, the case is not selected.
- Most criteria use one or more of the six relational operators (<, >, <=, >=, =, and <>).
- Expressions can include field names, constants, arithmetic operators, numeric and other functions, and logical variables. You can use fields that you do not plan to import as variables.

To build your criteria, you need at least two expressions and a relation to connect the expressions.

1. To build an expression, choose one of the following methods:

- In an Expression cell, type field names, constants, arithmetic operators, numeric and other functions, or logical variables.
- Double-click the field in the Fields list.
- Drag the field from the Fields list to an Expression cell.
- Choose a field from the drop-down menu in any active Expression cell.

2. To choose the relational operator (such as = or >), put your cursor in the Relation cell and either type the operator or choose it from the drop-down menu.

If the SQL contains WHERE clauses with expressions for case selection, dates and times in expressions need to be specified in a special manner (including the curly braces shown in the examples):

- Date literals should be specified using the general form {d 'yyyy-mm-dd'}.
- Time literals should be specified using the general form {t 'hh:mm:ss'}.
- Date/time literals (timestamps) should be specified using the general form {ts 'yyyy-mm-dd hh:mm:ss'}.
- The entire date and/or time value must be enclosed in single quotes. Years must be expressed in four-digit form, and dates and times must contain two digits for each portion of the value. For example January 1, 2005, 1:05 AM would be expressed as:
`{ts '2005-01-01 01:05:00'}`

Functions. A selection of built-in arithmetic, logical, string, date, and time SQL functions is provided. You can drag a function from the list into the expression, or you can enter any valid SQL function. See your database documentation for valid SQL functions.

Use Random Sampling. This option selects a random sample of cases from the data source. For large data sources, you may want to limit the number of cases to a small, representative sample, which can significantly reduce the time that it takes to run procedures. Native random sampling, if available for

the data source, is faster than IBM SPSS Statistics random sampling, because IBM SPSS Statistics random sampling must still read the entire data source to extract a random sample.

- *Approximately*. Generates a random sample of approximately the specified percentage of cases. Since this routine makes an independent pseudorandom decision for each case, the percentage of cases selected can only approximate the specified percentage. The more cases there are in the data file, the closer the percentage of cases selected is to the specified percentage.
- *Exactly*. Selects a random sample of the specified number of cases from the specified total number of cases. If the total number of cases specified exceeds the total number of cases in the data file, the sample will contain proportionally fewer cases than the requested number.

Note: If you use random sampling, aggregation (available in distributed mode with IBM SPSS Statistics Server) is not available.

Prompt For Value. You can embed a prompt in your query to create a **parameter query**. When users run the query, they will be asked to enter information (based on what is specified here). You might want to do this if you need to see different views of the same data. For example, you may want to run the same query to see sales figures for different fiscal quarters.

3. Place your cursor in any Expression cell, and click **Prompt For Value** to create a prompt.

Creating a Parameter Query

Use the Prompt for Value step to create a dialog box that solicits information from users each time someone runs your query. This feature is useful if you want to query the same data source by using different criteria.

To build a prompt, enter a prompt string and a default value. The prompt string is displayed each time a user runs your query. The string should specify the kind of information to enter. If the user is not selecting from a list, the string should give hints about how the input should be formatted. An example is as follows: **Enter a Quarter (Q1, Q2, Q3, ...)**.

Allow user to select value from list. If this check box is selected, you can limit the user to the values that you place here. Ensure that your values are separated by returns.

Data type. Choose the data type here (**Number**, **String**, or **Date**).

Date and time values must be entered in special manner:

- Date values must use the general form yyyy-mm-dd.
- Time values must use the general form: hh:mm:ss.
- Date/time values (timestamps) must use the general form yyyy-mm-dd hh:mm:ss.

Aggregating Data

If you are in distributed mode, connected to a remote server (available with IBM SPSS Statistics Server), you can aggregate the data before reading it into IBM SPSS Statistics.

You can also aggregate data after reading it into IBM SPSS Statistics, but preaggregating may save time for large data sources.

1. To create aggregated data, select one or more break variables that define how cases are grouped.
2. Select one or more aggregated variables.
3. Select an aggregate function for each aggregate variable.
4. Optionally, create a variable that contains the number of cases in each break group.

Note: If you use IBM SPSS Statistics random sampling, aggregation is not available.

Defining Variables

Variable names and labels. The complete database field (column) name is used as the variable label. Unless you modify the variable name, the Database Wizard assigns variable names to each column from the database in one of two ways:

- If the name of the database field forms a valid, unique variable name, the name is used as the variable name.
- If the name of the database field does not form a valid, unique variable name, a new, unique name is automatically generated.

Click any cell to edit the variable name.

Converting strings to numeric values. Select the **Recode to Numeric** box for a string variable if you want to automatically convert it to a numeric variable. String values are converted to consecutive integer values based on alphabetical order of the original values. The original values are retained as value labels for the new variables.

Width for variable-width string fields. This option controls the width of variable-width string values. By default, the width is 255 bytes, and only the first 255 bytes (typically 255 characters in single-byte languages) will be read. The width can be up to 32,767 bytes. Although you probably don't want to truncate string values, you also don't want to specify an unnecessarily large value, which will cause processing to be inefficient.

Minimize string widths based on observed values. Automatically set the width of each string variable to the longest observed value.

Sorting Cases

If you are in distributed mode, connected to a remote server (available with IBM SPSS Statistics Server), you can sort the data before reading it into IBM SPSS Statistics.

You can also sort data after reading it into IBM SPSS Statistics, but presorting may save time for large data sources.

Results

The Results step displays the SQL Select statement for your query.

- You can edit the SQL Select statement before you run the query, but if you click the **Back** button to make changes in previous steps, the changes to the Select statement will be lost.
- To save the query for future use, use the **Save query to file** section.
- To paste complete GET DATA syntax into a syntax window, select **Paste it into the syntax editor for further modification**. Copying and pasting the Select statement from the Results window will not paste the necessary command syntax.

Note: The pasted syntax contains a blank space before the closing quote on each line of SQL that is generated by the wizard. These blanks are not superfluous. When the command is processed, all lines of the SQL statement are merged together in a very literal fashion. Without the space, there would be no space between the last character on one line and first character on the next line.

Reading Cognos BI data

If you have access to a IBM Cognos Business Intelligence server, you can read IBM Cognos Business Intelligence data packages and list reports into IBM SPSS Statistics.

To read IBM Cognos Business Intelligence data:

1. From the menus choose:
File > Import Data > Cognos Business Intelligence

2. Specify the URL for the IBM Cognos Business Intelligence server connection.
3. Specify the location of the data package or report.
4. Select the data fields or report that you want to read.

Optionally, you can:

- Select filters for data packages.
- Import aggregated data instead of raw data.
- Specify parameter values.

Mode. Specifies the type of information you want to read: **Data** or **Report**. The only type of report that can be read is a list report.

Connection. The URL of the Cognos Business Intelligence server. Click the **Edit** button to define the details of a new Cognos connection from which to import data or reports. See the topic “Cognos connections” for more information.

Location. The location of the package or report that you want to read. Click the **Edit** button to display a list of available sources from which to import content. See the topic “Cognos location” on page 21 for more information.

Content. For data, displays the available data packages and filters. For reports, display the available reports.

Fields to import. For data packages, select the fields you want to include and move them to this list.

Report to import. For reports, select the list report you want to import. The report must be a list report.

Filters to apply. For data packages, select the filters you want to apply and move them to this list.

Parameters. If this button is enabled, the selected object has parameters defined. You can use parameters to make adjustments (for example, perform a parameterized calculation) before importing the data. If parameters are defined but no default is provided, the button displays a warning triangle.

Aggregate data before performing import. For data packages, if aggregation is defined in the package, you can import the aggregated data instead of the raw data.

Cognos connections

The Cognos Connections dialog specifies the IBM Cognos Business Intelligence server URL and any required additional credentials.

Cognos server URL. The URL of the IBM Cognos Business Intelligence server. This is the value of the "external dispatcher URI" environment property of IBM Cognos Configuration on the server. Contact your system administrator for more information

Mode. Select **Set Credentials** if you need to log in with a specific namespace, username and password (for example, as an administrator). Select **Use Anonymous connection** to log in with no user credentials, in which case you do not fill in the other fields. Select **Stored Credentials** to use the login information from a stored credential. To use a stored credential, you must be connected to the IBM SPSS Collaboration and Deployment Services Repository that contains the credential. After you are connected to the repository, click **Browse** to see the list of available credentials.

Namespace ID. The security authentication provider used to log on to the server. The authentication provider is used to define and maintain users, groups, and roles, and to control the authentication process.

User name. Enter the user name with which to log on to the server.

Password. Enter the password associated with the specified user name.

Save as Default. Saves these settings as your default, to avoid having to re-enter them each time.

Cognos location

The Specify Location dialog box enables you to select a package from which to import data, or a package or folder from which to import reports. It displays the public folders that are available to you. If you select **Data** in the main dialog, the list will display folders containing data packages. If you select **Report** in the main dialog, the list will display folders containing list reports. Select the location you want by navigating through the folder structure.

Specifying parameters for data or reports

If parameters have been defined, either for a data object or a report, you can specify values for these parameters before importing the data or report. An example of parameters for a report would be start and end dates for the report contents.

Name. The parameter name as it is specified in the IBM Cognos Analytics database.

Type. A description of the parameter.

Value. The value to assign to the parameter. To enter or edit a value, double-click its cell in the table. Values are not validated here; any invalid values are detected at run time.

Automatically remove invalid parameters from table. This option is selected by default and will remove any invalid parameters found within the data object or report.

Changing variable names

For IBM Cognos Business Intelligence data packages, package field names are automatically converted to valid variable names. You can use the Fields tab of the Read Cognos Data dialog to override the default names. Names must be unique and must conform to variable naming rules. See the topic “Variable names” on page 50 for more information.

Reading Cognos TM1 data

If you have access to an IBM Cognos TM1® database, you can import TM1 data from a specified view into IBM SPSS Statistics. The multidimensional OLAP cube data from TM1 is flattened when read into SPSS Statistics.

Important: To enable the exchange of data between SPSS Statistics and TM1, you must copy the following three processes from SPSS Statistics to the TM1 server: `ExportToSPSS.pro`, `ImportFromSPSS.pro`, and `SPSSCreateNewMeasures.pro`. To add these processes to the TM1 server, you must copy them to the data directory of the TM1 server and restart the TM1 server. These files are available from the `common/scripts/TM1` directory under the SPSS Statistics installation directory.

Restriction:

- The TM1 view from which you import must include one or more elements from a measure dimension.
- The data to be imported from TM1 must be in UTF-8 format.

All of the data in the specified TM1 view are imported. It is therefore best to limit the view to the data that are required for the analysis. Any necessary filtering of the data is best done in TM1, for example with the TM1 Subset Editor.

To read TM1 data:

1. From the menus, choose:

File > Import Data > Cognos TM1

2. Connect to the TM1 Performance Management system.
3. Log in to the TM1 server.
4. Select a TM1 cube and select the view that you want to import.

Optionally, you can override the default names of the SPSS Statistics variables that are created from the names of the TM1 dimensions and measures.

PM System

The URL of the Performance Management system that contains the TM1 server to which you want to connect. The Performance Management system is defined as a single URL for all TM1 servers. From this URL, all TM1 servers that are installed and running on your environment can be discovered and accessed. Enter the URL and click **Connect**.

TM1 Server

When the connection to the Performance Management system is established, select the server that contains the data you want to import and click **Login**. If you did not previously connect to this server, you are prompted to log in.

Username and password

Select this option to log in with a specific username and password. If the server uses authentication mode 5 (IBM Cognos security), then select the namespace that identifies the security authentication provider from the available list.

Stored credential

Select this option to use the login information from a stored credential. To use a stored credential, you must be connected to the IBM SPSS Collaboration and Deployment Services Repository that contains the credential. After you are connected to the repository, click **Browse** to see the list of available credentials.

Select a TM1 cube view to import

Lists the names of the cubes within the TM1 server from which you can import data. Double-click a cube to display a list of the views that you can import. Select a view and click the right arrow to move it into the **View to import** field.

Column dimension(s)

Lists the names of the column dimensions in the selected view.

Row dimension(s)

Lists the names of the row dimensions in the selected view.

Context dimension(s)

Lists the names of the context dimensions in the selected view.

Note:

- When the data are imported, a separate SPSS Statistics variable is created for each regular dimension and for each element in the measure dimension.
- Empty cells and cells with a value of zero in TM1 are converted to the system-missing value.
- Cells with string values that cannot be converted to a numeric value are converted to the system-missing value.

Changing variable names

By default, valid IBM SPSS Statistics variable names are automatically generated from the dimension names and names of elements in the measure dimension from the selected IBM Cognos TM1 cube view. You can use the Fields tab of the Import from TM1 dialog to override the default names. Names must be unique and must conform to variable naming rules.

File information

A data file contains much more than raw data. It also contains any variable definition information, including:

- Variable names
- Variable formats
- Descriptive variable and value labels

This information is stored in the dictionary portion of the data file. The Data Editor provides one way to view the variable definition information. You can also display complete dictionary information for the active dataset or any other data file.

To Display Data File Information

1. From the menus in the Data Editor window choose:

File > Display Data File Information

2. For the currently open data file, choose **Working File**.
3. For other data files, choose **External File**, and then select the data file.

The data file information is displayed in the Viewer.

Saving data files

In addition to saving data files in IBM SPSS Statistics format, you can save data in a wide variety of external formats, including:

- Excel and other spreadsheet formats
- Tab-delimited and CSV text files
- SAS
- Stata
- Database tables

To save modified data files

1. Make the Data Editor the active window (click anywhere in the window to make it active).
2. From the menus choose:
File > Save

The modified data file is saved, overwriting the previous version of the file.

To save data files in code page character encoding

Unicode data files cannot be read by IBM SPSS Statistics versions prior to version 16.0. In Unicode mode, to save a data file in code page character encoding:

1. Make the Data Editor the active window (click anywhere in the window to make it active).
2. From the menus choose:
File > Save As
3. From the **Save as type** drop-down list in the Save Data dialog, select **SPSS Statistics Local Encoding**.
4. Enter a name for the new data file.

The modified data file is saved in the current locale code page character encoding. This action has no effect on the active dataset. The encoding of the active dataset is not changed. Saving a file in code page character encoding is similar to saving a file in an external format, such as tab-delimited text or Excel.

Saving data files in external formats

1. Make the Data Editor the active window (click anywhere in the window to make it active).
2. From the menus choose:
 File > Save As...
3. Select a file type from the drop-down list.
4. Enter a file name for the new data file.

Options

Depending on the file type, additional options are available.

Encoding

Available for SAS files and text data formats: tab-delimited, comma-delimited, and fixed ASCII text.

Write variable names to file

Available for Excel, tab-delimited, comma-delimited, 1-2-3, and SYLK. For Excel 97 and later versions, you can write either variable names or labels. For variables without defined variable labels, the variable name is used.

Sheet name

For Excel 2007 and later versions, you can specify a sheet name. You can also append a sheet to an existing file.

Save value labels to a .sas file

SAS 6 and later versions.

For information on exporting data to database tables, see “Exporting to a Database” on page 30.

Saving data: Data file types

You can save data in the following formats:

SPSS Statistics (*.sav)

IBM SPSS Statistics format.

- Data files saved in IBM SPSS Statistics format cannot be read by versions of the software prior to version 7.5. Data files saved in Unicode encoding cannot be read by releases of IBM SPSS Statistics prior to version 16.0
- When using data files with variable names longer than eight bytes in version 10.x or 11.x, unique, eight-byte versions of variable names are used—but the original variable names are preserved for use in release 12.0 or later. In releases prior to 10.0, the original long variable names are lost if you save the data file.
- When using data files with string variables longer than 255 bytes in versions prior to release 13.0, those string variables are broken up into multiple 255-byte string variables.

SPSS Statistics Compressed (*.zsav)

Compressed IBM SPSS Statistics format.

- ZSAV files have the same features as SAV files, but they take up less disk space.
- ZSAV files may take more or less time to open and save, depending on the file size and system configuration. Extra time is needed to de-compress and compress ZSAV files. However, because ZSAV files are smaller on disk, they reduce the time needed to read and write from disk. As the file size gets larger, this time savings surpasses the extra time needed to de-compress and compress the files.
- Only IBM SPSS Statistics version 21 or higher can open ZSAV files.
- The option to save the data file with your local code page encoding is not available for ZSAV files. These files are always saved in UTF-8 encoding.

SPSS Statistics Local Encoding (*.sav)

In Unicode mode, this option saves the data file in the current locale code page character encoding. This option is not available in code page mode.

SPSS 7.0 (*.sav). Version 7.0 format. Data files saved in version 7.0 format can be read by version 7.0 and earlier versions but do not include defined multiple response sets or Data Entry for Windows information.

SPSS/PC+ (*.sys). SPSS/PC+ format. If the data file contains more than 500 variables, only the first 500 will be saved. For variables with more than one defined user-missing value, additional user-missing values will be recoded into the first defined user-missing value. This format is available only on Windows operating systems.

Portable (*.por). Portable format that can be read by other versions of IBM SPSS Statistics and versions on other operating systems. Variable names are limited to eight bytes and are automatically converted to unique eight-byte names if necessary. In most cases, saving data in portable format is no longer necessary, since IBM SPSS Statistics data files should be platform/operating system independent. You cannot save data files in portable file in Unicode mode. See the topic "General options" on page 195 for more information.

Tab-delimited (*.dat). Text files with values separated by tabs. (*Note:* Tab characters embedded in string values are preserved as tab characters in the tab-delimited file. No distinction is made between tab characters embedded in values and tab characters that separate values.) You can save files in Unicode encoding or local code page encoding.

Comma-delimited (*.csv). Text files with values separated by commas or semicolons. If the current IBM SPSS Statistics decimal indicator is a period, values are separated by commas. If the current decimal indicator is a comma, values are separated by semicolons. You can save files in Unicode encoding or local code page encoding.

Fixed ASCII (*.dat). Text file in fixed format, using the default write formats for all variables. There are no tabs or spaces between variable fields. You can save files in Unicode encoding or local code page encoding.

Excel 2007 (*.xlsx). Microsoft Excel 2007 XLSX-format workbook. The maximum number of variables is 16,000; any additional variables beyond the first 16,000 are dropped. If the dataset contains more than one million cases, multiple sheets are created in the workbook.

Excel 97 through 2003 (*.xls). Microsoft Excel 97 workbook. The maximum number of variables is 256; any additional variables beyond the first 256 are dropped. If the dataset contains more than 65,356 cases, multiple sheets are created in the workbook.

Excel 2.1 (*.xls). Microsoft Excel 2.1 spreadsheet file. The maximum number of variables is 256, and the maximum number of rows is 16,384.

1-2-3 Release 3.0 (*.wk3). Lotus 1-2-3 spreadsheet file, release 3.0. The maximum number of variables that you can save is 256.

1-2-3 Release 2.0 (*.wk1). Lotus 1-2-3 spreadsheet file, release 2.0. The maximum number of variables that you can save is 256.

1-2-3 Release 1.0 (*.wks). Lotus 1-2-3 spreadsheet file, release 1A. The maximum number of variables that you can save is 256.

SYLK (*.slk). Symbolic link format for Microsoft Excel and Multiplan spreadsheet files. The maximum number of variables that you can save is 256.

dBASE IV (*.dbf). dBASE IV format.

dBASE III (*.dbf). dBASE III format.

dBASE II (*.dbf). dBASE II format.

SAS v9+ Windows (*.sas7bdat). SAS versions 9 for Windows. You can save files in Unicode (UTF-8) or local code page encoding.

SAS v9+ UNIX (*.sas7bdat). SAS versions 9 for UNIX. You can save files in Unicode (UTF-8) or local code page encoding.

SAS v7-8 Windows short extension (*.sd7). SAS versions 7–8 for Windows short filename format.

SAS v7-8 Windows long extension (*.sas7bdat). SAS versions 7–8 for Windows long filename format.

SAS v7-8 for UNIX (*.sas7bdat). SAS v8 for UNIX.

SAS v6 for Windows (*.sd2). SAS v6 file format for Windows/OS2.

SAS v6 for UNIX (*.ssd01). SAS v6 file format for UNIX (Sun, HP, IBM).

SAS v6 for Alpha/OSF (*.ssd04). SAS v6 file format for Alpha/OSF (DEC UNIX).

SAS Transport (*.xpt). SAS transport file.

Stata Version 14 Intercooled (*.dta). Stata 14 supports Unicode data

Stata Version 14 SE (*.dta). Stata 14 supports Unicode data

Stata Version 13 Intercooled (*.dta).

Stata Version 13 SE (*.dta).

Stata Version 12 Intercooled (*.dta).

Stata Version 12 SE (*.dta).

Stata Version 11 Intercooled (*.dta).

Stata Version 11 SE (*.dta).

Stata Version 10 Intercooled (*.dta).

Stata Version 10 SE (*.dta).

Stata Version 9 Intercooled (*.dta).

Stata Version 9 SE (*.dta).

Stata Version 8 Intercooled (*.dta).

Stata Version 8 SE (*.dta).

Stata Version 7 Intercooled (*.dta).

Stata Version 7 SE (*.dta).

Stata Version 6 (*.dta).

Stata Versions 4–5 (*.dta).

Note: SAS data file names can be up to 32 characters in length. Blank spaces and non-alphanumeric characters other than the underscore ("_") are not allowed and names have to start with a letter or an underscore, numbers can follow.

Saving data files in Excel format

You can save your data in one of three Microsoft Excel file formats. Excel 2.1, Excel 97, and Excel 2007.

- Excel 2.1 and Excel 97 are limited to 256 columns; so only the first 256 variables are included.
- Excel 2007 is limited to 16,000 columns; so only the first 16,000 variables are included.
- Excel 2.1 is limited to 16,384 rows; so only the first 16,384 cases are included.
- Excel 97 and Excel 2007 also have limits on the number of rows per sheet, but workbooks can have multiple sheets, and multiple sheets are created if the single-sheet maximum is exceeded.

Options

- For all versions of Excel, you can include variable names as the first row of the Excel file.
- For Excel 97 and later versions, you can write either variable names or labels. For variables without defined variable labels, the variable name is used.
- For Excel 2007 and later versions, you can specify a sheet name. You can also append a sheet to an existing file.

Variable types

The following table shows the variable type matching between the original data in IBM SPSS Statistics and the exported data in Excel.

Table 2. How Excel data formats map to IBM SPSS Statistics variable types and formats

IBM SPSS Statistics Variable Type	Excel Data Format
Numeric	0.00; #,##0.00; ...
Comma	0.00; #,##0.00; ...
Dollar	\$#,##0_); ...
Date	d-mmm-yyyy
Time	hh:mm:ss
String	General

Saving data files in SAS format

Special handling is given to various aspects of your data when saved as a SAS file. These cases include:

- Certain characters that are allowed in IBM SPSS Statistics variable names are not valid in SAS, such as @, #, and \$. These illegal characters are replaced with an underscore when the data are exported.
- IBM SPSS Statistics variable names that contain multibyte characters (for example, Japanese or Chinese characters) are converted to variables names of the general form *Vnnn*, where *nnn* is an integer value.
- IBM SPSS Statistics variable labels containing more than 40 characters are truncated when exported to a SAS v6 file.
- Where they exist, IBM SPSS Statistics variable labels are mapped to the SAS variable labels. If no variable label exists in the IBM SPSS Statistics data, the variable name is mapped to the SAS variable label.
- SAS allows only one value for system-missing, whereas IBM SPSS Statistics allows numerous user-missing values in addition to system-missing. As a result, all user-missing values in IBM SPSS Statistics are mapped to a single system-missing value in the SAS file.

- SAS 6-8 data files are saved in the current IBM SPSS Statistics locale encoding, regardless of current mode (Unicode or code page). In Unicode mode, SAS 9 files are saved in UTF-8 format. In code page mode, SAS 9 files are saved in the current locale encoding.
- A maximum of 32,767 variables can be saved to SAS 6-8.
- SAS data file names can be up to 32 characters in length. Blank spaces and non-alphanumeric characters other than the underscore ("_") are not allowed and names have to start with a letter or an underscore, numbers can follow.

Save value labels

You have the option of saving the values and value labels associated with your data file to a SAS syntax file. This syntax file contains proc format and proc datasets commands that can be run in SAS to create a SAS format catalog file.

This feature is not supported for the SAS transport file.

Variable types

The following table shows the variable type matching between the original data in IBM SPSS Statistics and the exported data in SAS.

Table 3. How SAS variable types and formats map to IBM SPSS Statistics types and formats

IBM SPSS Statistics Variable Type	SAS Variable Type	SAS Data Format
Numeric	Numeric	12
Comma	Numeric	12
Dot	Numeric	12
Scientific Notation	Numeric	12
Date	Numeric	(Date) for example, MMDDYY10, ...
Date (Time)	Numeric	Time18
Dollar	Numeric	12
Custom Currency	Numeric	12
String	Character	\$8

Saving data files in Stata format

- Data can be written in Stata 5–14 format and in both Intercooled and SE format (version 7 or later).
- Data files that are saved in Stata 5 format can be read by Stata 4.
- The first 80 bytes of variable labels are saved as Stata variable labels.
- For Stata releases 4–8, the first 80 bytes of value labels for numeric variables are saved as Stata value labels. For Stata release 9 or later, the complete value labels for numeric variables are saved. Value labels are dropped for string variables, non-integer numeric values, and numeric values greater than an absolute value of 2,147,483,647.
- For versions 7 and later, the first 32 bytes of variable names in case-sensitive form are saved as Stata variable names. For earlier versions, the first eight bytes of variable names are saved as Stata variable names. Any characters other than letters, numbers, and underscores are converted to underscores.
- IBM SPSS Statistics variable names that contain multi-byte characters (for example, Japanese or Chinese characters) are converted to generic single-byte variable names. This only applies to Stata 4–13 data. IBM SPSS Statistics can save multi-byte variable names to Stata 14 data (because Stata 14 data is UTF-8 encoded).

- For versions 5–6 and Intercooled versions 7 and later, the first 80 bytes of string values are saved. For Stata SE 7–12, the first 244 bytes of string values are saved. For Stata SE 13 or later, complete string values are saved, regardless of length.
- For versions 5–6 and Intercooled versions 7 and later, only the first 2,047 variables are saved. For Stata SE 7 or later, only the first 32,767 variables are saved.
- Stata 14 supports Unicode data.

Table 4. How Stata variable type and format map to IBM SPSS Statistics type and format

IBM SPSS Statistics Variable Type	Stata Variable Type	Stata Data Format
Numeric	Numeric	g
Comma	Numeric	g
Dot	Numeric	g
Scientific Notation	Numeric	g
Date*, Datetime	Numeric	D_m_Y
Time, DTime	Numeric	g (number of seconds)
Wkday	Numeric	g (1–7)
Month	Numeric	g (1–12)
Dollar	Numeric	g
Custom Currency	Numeric	g
String	String	s

*Date, Adate, Edate, SDate, Jdate, Qyr, Moyr, Wkyr

Saving Subsets of Variables

The Save Data As Variables dialog box allows you to select the variables that you want saved in the new data file. By default, all variables will be saved. Deselect the variables that you don't want to save, or click **Drop All** and then select the variables that you want to save.

Visible Only. Selects only variables in variable sets currently in use. See the topic “Using variable sets to show and hide variables” on page 192 for more information.

To Save a Subset of Variables

1. Make the Data Editor the active window (click anywhere in the window to make it active).
2. From the menus choose:
File > Save As...
3. Click **Variables**.
4. Select the variables that you want to save.

Encrypting data files

For IBM SPSS Statistics data files, you can protect confidential information stored in a data file by encrypting the file with a password. Once encrypted, the file can only be opened by providing the password.

1. Make the Data Editor the active window (click anywhere in the window to make it active).
2. From the menus choose:
File > Save As...
3. Select **Encrypt file with password** in the Save Data As dialog box.

4. Click **Save**.
5. In the Encrypt File dialog box, provide a password and re-enter it in the Confirm password text box. Passwords are limited to 10 characters and are case-sensitive.

Warning: Passwords cannot be recovered if they are lost. If the password is lost the file cannot be opened.

Creating strong passwords

- Use eight or more characters.
- Include numbers, symbols and even punctuation in your password.
- Avoid sequences of numbers or characters, such as "123" and "abc", and avoid repetition, such as "111aaa".
- Do not create passwords that use personal information such as birthdays or nicknames.
- Periodically change the password.

Note: Storing encrypted files to an IBM SPSS Collaboration and Deployment Services Repository is not supported.

Modifying encrypted files

- If you open an encrypted file, make modifications to it and choose File > Save, the modified file will be saved with the same password.
- You can change the password on an encrypted file by opening the file, repeating the steps for encrypting it, and specifying a different password in the Encrypt File dialog box.
- You can save an unencrypted version of an encrypted file by opening the file, choosing File > Save As and deselecting **Encrypt file with password** in the Save Data As dialog box.

Note: Encrypted data files and output documents cannot be opened in versions of IBM SPSS Statistics prior to version 21. Encrypted syntax files cannot be opened in versions prior to version 22.

Exporting to a Database

You can use the Export to Database Wizard to:

- Replace values in existing database table fields (columns) or add new fields to a table.
- Append new records (rows) to a database table.
- Completely replace a database table or create a new table.

To export data to a database:

1. From the menus in the Data Editor window for the dataset that contains the data you want to export, choose:
File > Export > Database
2. Select the database source.
3. Follow the instructions in the export wizard to export the data.

Creating Database Fields from IBM SPSS Statistics Variables

When creating new fields (adding fields to an existing database table, creating a new table, replacing a table), you can specify field names, data type, and width (where applicable).

Field name. The default field names are the same as the IBM SPSS Statistics variable names. You can change the field names to any names allowed by the database format. For example, many databases allow characters in field names that aren't allowed in variable names, including spaces. Therefore, a variable name like *CallWaiting* could be changed to the field name *Call Waiting*.

Type. The export wizard makes initial data type assignments based on the standard ODBC data types or data types allowed by the selected database format that most closely matches the defined IBM SPSS Statistics data format—but databases can make type distinctions that have no direct equivalent in IBM SPSS Statistics, and vice versa. For example, most numeric values in IBM SPSS Statistics are stored as double-precision floating-point values, whereas database numeric data types include float (double), integer, real, and so on. In addition, many databases don't have equivalents to IBM SPSS Statistics time formats. You can change the data type to any type available in the drop-down list.

As a general rule, the basic data type (string or numeric) for the variable should match the basic data type of the database field. If there is a data type mismatch that cannot be resolved by the database, an error results and no data are exported to the database. For example, if you export a string variable to a database field with a numeric data type, an error will result if any values of the string variable contain non-numeric characters.

Width. You can change the defined width for string (char, varchar) field types. Numeric field widths are defined by the data type.

By default, IBM SPSS Statistics variable formats are mapped to database field types based on the following general scheme. Actual database field types may vary, depending on the database.

Table 5. Format conversion for databases

IBM SPSS Statistics Variable Format	Database Field Type
Numeric	Float or Double
Comma	Float or Double
Dot	Float or Double
Scientific Notation	Float or Double
Date	Date or Datetime or Timestamp
Datetime	Datetime or Timestamp
Time, DTime	Float or Double (number of seconds)
Wkday	Integer (1–7)
Month	Integer (1–12)
Dollar	Float or Double
Custom Currency	Float or Double
String	Char or Varchar

User-Missing Values

There are two options for the treatment of user-missing values when data from variables are exported to database fields:

- **Export as valid values.** User-missing values are treated as regular, valid, nonmissing values.
- **Export numeric user-missing as nulls and export string user-missing values as blank spaces.** Numeric user-missing values are treated the same as system-missing values. String user-missing values are converted to blank spaces (strings cannot be system-missing).

Selecting a Data Source

In the first panel of the Export to Database Wizard, you select the data source to which you want to export data.

You can export data to any database source for which you have the appropriate ODBC driver. (Note: Exporting data to OLE DB data sources is not supported.)

If you do not have any ODBC data sources configured, or if you want to add a new data source, click **Add ODBC Data Source**.

- On Linux operating systems, this button is not available. ODBC data sources are specified in *odbc.ini*, and the *ODBCINI* environment variables must be set to the location of that file. For more information, see the documentation for your database drivers.
- In distributed analysis mode (available with IBM SPSS Statistics Server), this button is not available. To add data sources in distributed analysis mode, see your system administrator.

An ODBC data source consists of two essential pieces of information: the driver that will be used to access the data and the location of the database you want to access. To specify data sources, you must have the appropriate drivers installed. Drivers for a variety of database formats are included with the installation media.

Some data sources may require a login ID and password before you can proceed to the next step.

Choosing How to Export the Data

After you select the data source, you indicate the manner in which you want to export the data.

The following choices are available for exporting data to a database:

- **Replace values in existing fields.** Replaces values of selected fields in an existing table with values from the selected variables in the active dataset. See the topic “Replacing Values in Existing Fields” on page 33 for more information.
- **Add new fields to an existing table.** Creates new fields in an existing table that contain the values of selected variables in the active dataset. See the topic “Adding New Fields” on page 34 for more information. This option is not available for Excel files.
- **Append new records to an existing table.** Adds new records (rows) to an existing table containing the values from cases in the active dataset. See the topic “Appending New Records (Cases)” on page 34 for more information.
- **Drop an existing table and create a new table of the same name.** Deletes the specified table and creates a new table of the same name that contains selected variables from the active dataset. All information from the original table, including definitions of field properties (for example, primary keys, data types) is lost. See the topic “Creating a New Table or Replacing a Table” on page 34 for more information.
- **Create a new table.** Creates a new table in the database containing data from selected variables in the active dataset. The name can be any value that is allowed as a table name by the data source. The name cannot duplicate the name of an existing table or view in the database. See the topic “Creating a New Table or Replacing a Table” on page 34 for more information.

Selecting a Table

When modifying or replacing a table in the database, you need to select the table to modify or replace. This panel in the Export to Database Wizard displays a list of tables and views in the selected database.

By default, the list displays only standard database tables. You can control the type of items that are displayed in the list:

- **Tables.** Standard database tables.
- **Views.** Views are virtual or dynamic "tables" defined by queries. These can include joins of multiple tables and/or fields derived from calculations based on the values of other fields. You can append records or replace values of existing fields in views, but the fields that you can modify may be restricted, depending on how the view is structured. For example, you cannot modify a derived field, add fields to a view, or replace a view.
- **Synonyms.** A synonym is an alias for a table or view, typically defined in a query.

- **System tables.** System tables define database properties. In some cases, standard database tables may be classified as system tables and will be displayed only if you select this option. Access to real system tables is often restricted to database administrators.

Selecting Cases to Export

Case selection in the Export to Database Wizard is limited either to all cases or to cases selected using a previously defined filter condition. If no case filtering is in effect, this panel will not appear, and all cases in the active dataset will be exported.

For information on defining a filter condition for case selection, see “Select cases” on page 109.

Matching Cases to Records

When adding fields (columns) to an existing table or replacing the values of existing fields, you need to make sure that each case (row) in the active dataset is correctly matched to the corresponding record in the database.

- In the database, the field or set of fields that uniquely identifies each record is often designated as the **primary key**.
- You need to identify which variable(s) correspond to the primary key field(s) or other fields that uniquely identify each record.
- The fields don't have to be the primary key in the database, but the field value or combination of field values must be unique for each case.

To match variables with fields in the database that uniquely identify each record:

1. Drag and drop the variable(s) onto the corresponding database fields.
or
2. Select a variable from the list of variables, select the corresponding field in the database table, and click **Connect**.
To delete a connection line:
3. Select the connection line and press the Delete key.

Note: The variable names and database field names may not be identical (since database field names may contain characters not allowed in IBM SPSS Statistics variable names), but if the active dataset was created from the database table you are modifying, either the variable names or the variable labels will usually be at least similar to the database field names.

Replacing Values in Existing Fields

To replace values of existing fields in a database:

1. In the **Choose how to export the data** panel of the Export to Database Wizard, select **Replace values in existing fields**.
 2. In the **Select a table or view** panel, select the database table.
 3. In the **Match cases to records** panel, match the variables that uniquely identify each case to the corresponding database field names.
 4. For each field for which you want to replace values, drag and drop the variable that contains the new values into the **Source of values** column, next to the corresponding database field name.
- As a general rule, the basic data type (string or numeric) for the variable should match the basic data type of the database field. If there is a data type mismatch that cannot be resolved by the database, an error results and no data is exported to the database. For example, if you export a string variable to a database field with a numeric data type (for example, double, real, integer), an error will result if any values of the string variable contain non-numeric characters. The letter *a* in the icon next to a variable denotes a string variable.
 - You cannot modify the field name, type, or width. The original database field attributes are preserved; only the values are replaced.

Adding New Fields

To add new fields to an existing database table:

1. In the **Choose how to export the data** panel of the Export to Database Wizard, select **Add new fields to an existing table**.
2. In the **Select a table or view** panel, select the database table.
3. In the **Match cases to records** panel, match the variables that uniquely identify each case to the corresponding database field names.
4. Drag and drop the variables that you want to add as new fields to the **Source of values** column.

For information on field names and data types, see the section on creating database fields from IBM SPSS Statistics variables in “Exporting to a Database” on page 30.

Value Labels. If a variable has defined variable labels, export value label text instead of values. For values that do not have a defined value label, the data value is exported as a text string. This option is not available for date format variables or variables that do not have any defined value labels.

Show existing fields. Select this option to display a list of existing fields. You cannot use this panel in the Export to Database Wizard to replace existing fields, but it may be helpful to know what fields are already present in the table. If you want to replace the values of existing fields, see “Replacing Values in Existing Fields” on page 33.

Appending New Records (Cases)

To append new records (cases) to a database table:

1. In the **Choose how to export the data** panel of the Export to Database Wizard, select **Append new records to an existing table**.
2. In the **Select a table or view** panel, select the database table.
3. Match variables in the active dataset to table fields by dragging and dropping variables to the **Source of values** column.

The Export to Database Wizard will automatically select all variables that match existing fields, based on information about the original database table stored in the active dataset (if available) and/or variable names that are the same as field names. This initial automatic matching is intended only as a guide and does not prevent you from changing the way in which variables are matched with database fields.

When adding new records to an existing table, the following basic rules/limitations apply:

- All cases (or all selected cases) in the active dataset are added to the table. If any of these cases duplicate existing records in the database, an error may result if a duplicate key value is encountered. For information on exporting only selected cases, see “Selecting Cases to Export” on page 33.
- You can use the values of new variables created in the session as the values for existing fields, but you cannot add new fields or change the names of existing fields. To add new fields to a database table, see “Adding New Fields.”
- Any excluded database fields or fields not matched to a variable will have no values for the added records in the database table. (If the **Source of values** cell is empty, there is no variable matched to the field.)

Creating a New Table or Replacing a Table

To create a new database table or replace an existing database table:

1. In the **Choose how to export the data** panel of the export wizard, select **Drop an existing table and create a new table of the same name** or select **Create a new table** and enter a name for the new table. If the table name contains any characters other than letters, numbers, or an underscore, the name must be enclosed in double quotes.
2. If you are replacing an existing table, in the **Select a table or view** panel, select the database table.
3. Drag and drop variables into the **Variables to save** column.

4. Optionally, you can designate variables/fields that define the primary key, change field names, and change the data type.

Primary key. To designate variables as the primary key in the database table, select the box in the column identified with the key icon.

- All values of the primary key must be unique or an error will result.
- If you select a single variable as the primary key, every record (case) must have a unique value for that variable.
- If you select multiple variables as the primary key, this defines a composite primary key, and the *combination* of values for the selected variables must be unique for each case.

For information on field names and data types, see the section on creating database fields from IBM SPSS Statistics variables in “Exporting to a Database” on page 30.

Value Labels. If a variable has defined variable labels, export value label text instead of values. For values that do not have a defined value label, the data value is exported as a text string. This option is not available for date format variables or variables that do not have any defined value labels.

Completing the Database Export Wizard

The last step of the Export to Database Wizard provides a summary of the export specifications.

Summary

- **Dataset.** The IBM SPSS Statistics session name for the dataset that is used to export data. This information is primarily useful if you have multiple open data sources. A data source opened with command syntax has a dataset name only if one is explicitly assigned.
- **Table.** The name of the table to be modified or created.
- **Cases to Export.** Either all cases are exported or cases that are selected by a previously defined filter condition are exported.
- **Action.** Indicates how the database is modified (for example, create a new table, add fields or records to an existing table).
- **User-Missing Values.** User-missing values can be exported as valid values or treated the same as system-missing for numeric variables and converted to blank spaces for string variables. This setting is controlled in the panel in which you select the variables to export.

Bulk Loading

Bulk load. Submits data to the database in batches instead of one record at a time. This action can make the operation much faster, particularly for large data files.

- **Batch size.** Specifies the number records to submit in each batch.
- **Batch commit.** Commits records to the database in the specified batch size.
- **ODBC binding.** Uses the ODBC binding method to commit records in the specified batch size. This option is only available if the database supports ODBC binding. This option is not available on Mac OS.
 - **Row-wise binding.** Row-wise binding typically improves speed compared to the use of parameterized inserts that insert data on a record-by-record basis.
 - **Column-wise binding.** Column-wise binding improves performance by binding each database column to an array of n values.

What would you like to do?

- **Export the data.** Exports the data to the database.
- **Paste the syntax.** Pastes the command syntax for exporting the data to a syntax window. You can modify and save the pasted command syntax.

Exporting to Data Collection

The Export to Data Collection dialog box creates IBM SPSS Statistics data files and Data Collection metadata files that you can use to read the data into Data Collection applications. This is particularly useful when "roundtripping" data between IBM SPSS Statistics and Data Collection applications.

Note: There may be instances where the data cannot be re-loaded to the Data Collection database in the correct format due to the complexity in the structure.

To export data for use in Data Collection applications:

1. From the menus in the Data Editor window that contains the data you want to export, choose:
File > Export > Data Collection
2. Click **Data File** to specify the name and location of the IBM SPSS Statistics data file.
3. Click **Metadata File** to specify the name and location of the Data Collection metadata file.

For new variables and datasets not created from Data Collection data sources, IBM SPSS Statistics variable attributes are mapped to Data Collection metadata attributes.

If the active dataset was created from a Data Collection data source:

- The new metadata file is created by merging the original metadata attributes with metadata attributes for any new variables, plus any changes to original variables that might affect their metadata attributes (for example, addition of, or changes to, value labels).
- For original variables read from the Data Collection data source, any metadata attributes not recognized by IBM SPSS Statistics are preserved in their original state. For example, IBM SPSS Statistics converts grid variables to regular IBM SPSS Statistics variables, but the metadata that defines these grid variables is preserved when you save the new metadata file.
- If any Data Collection variables were automatically renamed to conform to IBM SPSS Statistics variable naming rules, the metadata file maps the converted names back to the original Data Collection variable names.

The presence or absence of value labels can affect the metadata attributes of variables and consequently the way those variables are read by Data Collection applications. If value labels have been defined for any nonmissing values of a variable, they should be defined for all nonmissing values of that variable; otherwise, the unlabeled values will be dropped when the data file is read by Data Collection.

This feature is only available with IBM SPSS Statistics installed on Microsoft Windows operating systems, and is only available in local analysis mode. This feature is not available in distributed analysis mode using IBM SPSS Statistics Server.

To write Data Collection metadata files, you must have the following items installed:

- .NET framework. To obtain the most recent version of the .NET framework, go to <http://www.microsoft.com/net>.
- Data Collection Survey Reporter Developer Kit.

Exporting to Cognos TM1

If you have access to an IBM Cognos TM1 database, you can export data from IBM SPSS Statistics to TM1. This feature is particularly useful when you import data from TM1, transform or score the data in SPSS Statistics, and then want to export the results back to TM1.

Important: To enable the exchange of data between SPSS Statistics and TM1, you must copy the following three processes from SPSS Statistics to the TM1 server: `ExportToSPSS.pro`, `ImportFromSPSS.pro`, and `SPSSCreateNewMeasures.pro`. To add these processes to the TM1 server, you must copy them to the data directory of the TM1 server and restart the TM1 server. These files are available from the `common/scripts/TM1` directory under the SPSS Statistics installation directory.

To export data to TM1:

1. From the menus, choose:
File > Export > Cognos TM1
2. Connect to the TM1 Performance Management system.
3. Log in to the TM1 server.
4. Select the TM1 cube where you want to export the data.
5. Specify the mappings from the fields in the active dataset to the dimensions and measures in the TM1 cube.

PM System

The URL of the Performance Management system that contains the TM1 server to which you want to connect. The Performance Management system is defined as a single URL for all TM1 servers. From this URL, all TM1 servers that are installed and running on your environment can be discovered and accessed. Enter the URL and click **Connect**.

TM1 Server

When the connection to the Performance Management system is established, select the server that contains the cube to which you want to export the data and click **Login**. If you did not previously connect to this server, you are prompted to enter your user name and password. If the server uses authentication mode 5 (IBM Cognos security), then select the namespace that identifies the security authentication provider from the available list.

Select a TM1 cube to export

Lists the names of the cubes within the TM1 server into which you can export data. Select a cube and click the right arrow to move it into the **Export to cube** field.

Note:

- System-missing and user-missing values of fields that are mapped to elements in the measure dimension of the TM1 cube are ignored on export. The associated cells in the TM1 cube are unchanged.
- Fields with a value of zero, that are mapped to elements in the measure dimension, are exported as a valid value.

Mapping fields to TM1 dimensions

Use the Mapping tab in the Export to TM1 dialog box to map SPSS Statistics fields to the associated IBM Cognos TM1 dimensions and measures. You can map to existing elements in the measure dimension or you can create new elements in the measure dimension of the TM1 cube.

- For each regular dimension in the specified TM1 cube, you must either map a field in the active dataset to the dimension or specify a slice for the dimension. A slice specifies a single leaf element of a dimension so that all exported cases are associated with the specified leaf element.
- For a field that is mapped to a regular dimension, cases with field values that do not match a leaf element in the specified dimension are not exported. In that regard, you can export only to leaf elements.
- Only string fields in the active dataset can be mapped to regular dimensions. Only numeric fields in the active dataset can be mapped to elements in the measure dimension of the cube.
- Values that are exported to an existing element in the measure dimension overwrite the associated cells in the TM1 cube.

To map an SPSS Statistics field to a regular TM1 dimension or to an existing element in the measure dimension:

1. Select the SPSS Statistics field from the Fields list.
2. Select the associated TM1 dimension or measure from the TM1 Dimensions list.
3. Click **Map**.

To map an SPSS Statistics field to a new element in the measure dimension:

1. Select the SPSS Statistics field from the Fields list.
2. Select the item for the measure dimension from the TM1 Dimensions list.
3. Click **Create New**, enter the name of the measure element in the TM1 Measure Name dialog and click **OK**.

To specify a slice for a regular dimension:

1. Select the dimension from the TM1 Dimensions list.
2. Click **Slice on**.
3. In the Select Leaf Member dialog, select the element that specifies the slice and then click **OK**. You can search for a specific element by entering a search string in the Find text box and clicking **Find Next**. A match is found if any part of an element matches the search string.
 - Spaces included in the search string are included in the search.
 - Searches are not case-sensitive.
 - The asterisk (*) is treated as any other character and does not indicate a wildcard search.

You can remove a mapping definition by selecting the mapped item from the TM1 Dimensions list and clicking **Unmap**. You can delete the specification of a new measure by selecting the measure from the TM1 Dimensions list and clicking **Delete**.

Comparing datasets

Compare Datasets compares the active dataset to another dataset in the current session or an external file in IBM SPSS Statistics format.

To compare datasets

1. Open a data file and make sure it is the active dataset. (You can make a dataset the active dataset by clicking on the Data Editor window for that dataset.)
2. From the menus choose:
Data > Compare Datasets
3. Select the open dataset or IBM SPSS Statistics data file that you want to compare to the active dataset.
4. Select one or more fields (variables) that you want to compare.

Optionally, you can:

- Match cases (records) based on one or more case ID values.
- Compare data dictionary properties (field and value labels, user-missing values, measurement level, etc.).
- Create a flag field in the active dataset that identifies mismatched cases.
- Create new datasets that contain only matched cases or only mismatched cases.

Compare Datasets: Compare tab

The Matched fields list displays a list of fields that have the same name and same basic type (string or numeric) in both datasets.

1. Select one or more fields (variables) to compare. The comparison of the two datasets is based on the selected fields only.
2. To see a list of fields that either do not have matching names or are not the same basic type in both datasets, click **Unmatched fields**. Unmatched fields are excluded from the comparison of the two datasets.
3. Optionally, select one or more case (record) ID fields that identify each case.
 - If you specify multiple case ID fields, each unique combination of values identifies a case.

- Both files must be sorted in ascending order of the case ID fields. If the datasets are not already sorted, select (check) **Sort Cases** to sort both datasets in case ID order.
- If you do not include any case ID fields, cases are compared in file order. That is, the first case (record) in the active dataset is compared to the first case in the other dataset, and so on.

Compare Datasets: Unmatched Fields

The Unmatched Fields dialog displays a list of fields (variables) that are considered unmatched in the two datasets. An **unmatched** field is a field that either is missing from one of the datasets or is not the same basic type (string or numeric) in both files. Unmatched fields are excluded from the comparison of the two datasets.

Compare Datasets: Attributes tab

By default, only data values are compared, and field attributes (data dictionary properties) such as value labels, user-missing values, and measurement level, are not compared. To compare field attributes:

- In the Compare Datasets dialog, click the **Attributes** tab.
- Click Compare the Data Dictionaries.
- Select the attributes you want to compare.
 - Width.** For numeric fields, the maximum number of characters displayed (digits plus formatting characters, such as currency symbols, grouping symbols, and decimal indicator). For string fields, the maximum number of bytes allowed.
 - Label.** Descriptive field label.
 - Value Label.** Descriptive value labels.
 - Missing.** Defined user-missing values.
 - Columns.** Column width in Data view of the Data Editor.
 - Align.** Alignment in Data view of the Data Editor.
 - Measure.** Measurement level.
 - Role.** Field role.
 - Attributes.** User-defined custom field attributes.

Comparing datasets: Output tab

By default, Compare Datasets creates a new field in the active dataset that identifies mismatches and produces a table that provides details for the first 100 mismatches. You can use the Output tab to change the output options.

Flag mismatches in a new field. A new field that identifies mismatches is created in the active dataset.

- The value of the new field is 1 if there are differences and 0 if all the values are the same. If there are cases (records) in the active dataset that are not present in the other dataset, the value is -1.
- The default name of the new field is *CasesCompare*. You can specify a different field name. The name must conform to field (variable) naming rules. See the topic “Variable names” on page 50 for more information.

Copy matched cases to a new dataset. Creates a new dataset that contains only cases (records) from the active dataset that have matching values in the other dataset. The dataset name must conform to field (variable) naming rules. If the dataset already exists, it will be overwritten.

Copy mismatched cases to a new dataset. Creates a new dataset that contains only cases from the active dataset that have different values in the other dataset. The dataset name must conform to field (variable) naming rules. If the dataset already exists, it will be overwritten.

Limit the case-by-case table. For cases (records) in the active dataset that also exist in the other dataset and also have the same basic type (numeric or string) in both datasets, the case-by-case table provides

details on the mismatched values for each case. By default, the table is limited to the first 100 mismatches. You can specify a different value or deselect (clear) this item to include all mismatches.

Protecting original data

To prevent the accidental modification or deletion of your original data, you can mark the file as read-only.

1. From the Data Editor menus choose:

File > Mark File Read Only

If you make subsequent modifications to the data and then try to save the data file, you can save the data only with a different filename, so the original data are not affected.

You can change the file permissions back to read-write by choosing **Mark File Read Write** from the File menu.

Virtual Active File

The virtual active file enables you to work with large data files without requiring equally large (or larger) amounts of temporary disk space. For most analysis and charting procedures, the original data source is reread each time you run a different procedure. Procedures that modify the data require a certain amount of temporary disk space to keep track of the changes, and some actions always require enough disk space for at least one entire copy of the data file.

Actions that don't require any temporary disk space include:

- Reading IBM SPSS Statistics data files
- Merging two or more IBM SPSS Statistics data files
- Reading database tables with the Database Wizard
- Merging IBM SPSS Statistics data files with database tables
- Running procedures that read data (for example, Frequencies, Crosstabs, Explore)

Actions that create one or more columns of data in temporary disk space include:

- Computing new variables
- Recoding existing variables
- Running procedures that create or modify variables (for example, saving predicted values in Linear Regression)

Actions that create an entire copy of the data file in temporary disk space include:

- Reading Excel files
- Running procedures that sort data (for example, Sort Cases, Split File)
- Reading data with GET TRANSLATE or DATA LIST commands
- Using the Cache Data facility or the CACHE command
- Launching other applications from IBM SPSS Statistics that read the data file (for example, AnswerTree, DecisionTime)

Note: The GET DATA command provides functionality comparable to DATA LIST without creating an entire copy of the data file in temporary disk space. The SPLIT FILE command in command syntax does not sort the data file and therefore does not create a copy of the data file. This command, however, requires sorted data for proper operation, and the dialog box interface for this procedure will automatically sort the data file, resulting in a complete copy of the data file. (Command syntax is not available with the Student Version.)

Actions that create an entire copy of the data file by default:

- Reading databases with the Database Wizard
- Reading text files with the Text Wizard

The Text Wizard provides an optional setting to automatically cache the data. By default, this option is selected. You can turn it off by deselecting **Cache data locally**. For the Database Wizard, you can paste the generated command syntax and delete the CACHE command.

Creating a Data Cache

Although the virtual active file can vastly reduce the amount of temporary disk space required, the absence of a temporary copy of the "active" file means that the original data source has to be reread for each procedure. For large data files read from an external source, creating a temporary copy of the data may improve performance. For example, for data tables read from a database source, the SQL query that reads the information from the database must be reexecuted for any command or procedure that needs to read the data. Since virtually all statistical analysis procedures and charting procedures need to read the data, the SQL query is reexecuted for each procedure you run, which can result in a significant increase in processing time if you run a large number of procedures.

If you have sufficient disk space on the computer performing the analysis (either your local computer or a remote server), you can eliminate multiple SQL queries and improve processing time by creating a data cache of the active file. The data cache is a temporary copy of the complete data.

Note: By default, the Database Wizard automatically creates a data cache, but if you use the GET DATA command in command syntax to read a database, a data cache is not automatically created. (Command syntax is not available with the Student Version.)

To Create a Data Cache

1. From the menus choose:
File > Cache Data...
2. Click **OK** or **Cache Now**.

OK creates a data cache the next time the program reads the data (for example, the next time you run a statistical procedure), which is usually what you want because it doesn't require an extra data pass.

Cache Now creates a data cache immediately, which shouldn't be necessary under most circumstances.

Cache Now is useful primarily for two reasons:

- A data source is "locked" and can't be updated by anyone until you end your session, open a different data source, or cache the data.
- For large data sources, scrolling through the contents of the Data View tab in the Data Editor will be much faster if you cache the data.

To Cache Data Automatically

You can use the SET command to automatically create a data cache after a specified number of changes in the active data file. By default, the active data file is automatically cached after 20 changes in the active data file.

1. From the menus choose:
File > New > Syntax
2. In the syntax window, type SET CACHE *n* (where *n* represents the number of changes in the active data file before the data file is cached).
3. From the menus in the syntax window choose:
Run > All

Note: The cache setting is not persistent across sessions. Each time you start a new session, the value is reset to the default of 20.

Chapter 4. Distributed Analysis Mode

Distributed analysis mode allows you to use a computer other than your local (or desktop) computer for memory-intensive work. Because remote servers that are used for distributed analysis are typically more powerful and faster than your local computer, distributed analysis mode can significantly reduce computer processing time. Distributed analysis with a remote server can be useful if your work involves:

- Large data files, particularly data read from database sources.
- Memory-intensive tasks. Any task that takes a long time in local analysis mode may be a good candidate for distributed analysis.

Distributed analysis affects only data-related tasks, such as reading data, transforming data, computing new variables, and calculating statistics. Distributed analysis has no effect on tasks related to editing output, such as manipulating pivot tables or modifying charts.

Note: Distributed analysis is available only if you have both a local version and access to a licensed server version of the software that is installed on a remote server.

Server Login

The Server Login dialog box allows you to select the computer that processes commands and runs procedures. You can select your local computer or a remote server.

You can add, modify, or delete remote servers in the list. Remote servers usually require a user ID and password, and a domain name may also be necessary. If you are licensed to use the Statistics Adapter and your site is running IBM SPSS Collaboration and Deployment Services you may be able to connect to a remote server using single sign-on. Single sign-on allows users to connect to a remote server without explicitly providing a user ID and password. The necessary authentication is performed with the user's existing credentials on the current machine--obtained, for example, from Windows Active Directory. Contact your system administrator for information about available servers, a user ID and password, domain names, and other connection information, including whether single sign-on is supported at your site.

You can select a default server and save the user ID, domain name, and password that are associated with any server. You are automatically connected to the default server when you start a new session.

Important: You can connect to a server that is not at the same release level as the client. The server can be one or two releases newer or older compared to the client. However, it is not recommended to maintain this configuration for an extended time. If the server is newer than the client, the server may create output that cannot be read by the client. If the client is newer than the server, syntax submitted by the client may not be recognized by the server. Therefore, you should talk to your administrator about connecting to a server that is at the same release level as the client.

If you are licensed to use the Statistics Adapter and your site is running IBM SPSS Collaboration and Deployment Services 3.5 or later, you can click **Search...** to view a list of servers that are available on your network. If you are not logged on to a IBM SPSS Collaboration and Deployment Services Repository, you will be prompted to enter connection information before you can view the list of servers.

Adding and Editing Server Login Settings

Use the Server Login Settings dialog box to add or edit connection information for remote servers for use in distributed analysis mode.

Contact your system administrator for a list of available servers, port numbers for the servers, and additional connection information. Do not use the Secure Socket Layer unless instructed to do so by your administrator.

Server Name. A server "name" can be an alphanumeric name that is assigned to a computer (for example, NetworkServer) or a unique IP address that is assigned to a computer (for example, 202.123.456.78).

Port Number. The port number is the port that the server software uses for communications.

Description. You can enter an optional description to display in the servers list.

Connect with Secure Socket Layer. Secure Socket Layer (SSL) encrypts requests for distributed analysis when they are sent to the remote server. Before you use SSL, check with your administrator. For this option to be enabled, SSL must be configured on your desktop computer and the server.

To Select, Switch, or Add Servers

1. From the menus choose:

File > Switch Server...

To select a default server:

2. In the server list, select the box next to the server that you want to use.

3. If the server is configured for single sign-on then simply ensure that **Set Credentials** is unchecked. Otherwise, select **Set Credentials** and enter the user ID, domain name, and password that were provided by your administrator.

Note: You are automatically connected to the default server when you start a new session.

To switch to another server:

4. Select the server from the list.

5. If the server is configured for single sign-on then simply ensure that **Set Credentials** is unchecked. Otherwise, select **Set Credentials** and enter the user ID, domain name, and password (if necessary).

Note: When you switch servers during a session, all open windows are closed. You will be prompted to save changes before the windows are closed.

To add a server:

6. Get the server connection information from your administrator.

7. Click **Add** to open the Server Login Settings dialog box.

8. Enter the connection information and optional settings, and then click **OK**.

To edit a server:

9. Get the revised connection information from your administrator.

10. Click **Edit** to open the Server Login Settings dialog box.

11. Enter the changes and click **OK**.

To search for available servers:

Note: The ability to search for available servers is available only if you are licensed to use the Statistics Adapter and your site is running IBM SPSS Collaboration and Deployment Services 3.5 or later.

12. Click **Search...** to open the Search for Servers dialog box. If you are not logged on to a IBM SPSS Collaboration and Deployment Services Repository, you will be prompted for connection information.

13. Select one or more available servers and click **OK**. The servers will now appear in the Server Login dialog box.

14. To connect to one of the servers, follow the instructions "To switch to another server."

Searching for Available Servers

Use the Search for Servers dialog box to select one or more servers that are available on your network. This dialog box appears when you click **Search...** on the Server Login dialog box.

Select one or more servers and click **OK** to add them to the Server Login dialog box. Although you can manually add servers in the Server Login dialog box, searching for available servers lets you connect to servers without requiring that you know the correct server name and port number. This information is automatically provided. However, you still need the correct logon information, such as user name, domain, and password.

Opening Data Files from a Remote Server

In distributed analysis mode, the Open Remote File dialog box replaces the standard Open File dialog box.

- The contents of the list of available files, folders, and drives depends on what is available on or from the remote server. The current server name is indicated at the top of the dialog box.
- In distributed analysis mode, you will not have access to files on your local computer unless you specify the drive as a shared device or specify the folders containing your data files as shared folders. Consult the documentation for your operating system for information on how to "share" folders on your local computer with the server network.
- If the server is running a different operating system (for example, you are running Windows and the server is running UNIX), you probably won't have access to local data files in distributed analysis mode even if they are in shared folders.

File Access in Local and Distributed Analysis Mode

The view of data folders (directories) and drives for both your local computer and the network is based on the computer that you are currently using to process commands and run procedures—which is not necessarily the computer in front of you.

Local analysis mode. When you use your local computer as your "server," the view of data files, folders, and drives in the file access dialog box (for opening data files) is similar to what you see in other applications or in Windows Explorer. You can see all of the data files and folders on your computer and any files and folders on mounted network drives.

Distributed analysis mode. When you use another computer as a "remote server" to run commands and procedures, the view of data files, folders, and drives represents the view from the remote server computer. Although you may see familiar folder names (such as *Program Files*) and drives (such as C), these items are *not* the folders and drives on your computer; they are the folders and drives on the remote server.

In distributed analysis mode, you will not have access to data files on your local computer unless you specify the drive as a shared device or specify the folders containing your data files as shared folders. If the server is running a different operating system (for example, you are running Windows and the server is running UNIX), you probably won't have access to local data files in distributed analysis mode even if they are in shared folders.

Distributed analysis mode is not the same as accessing data files that reside on another computer on your network. You can access data files on other network devices in local analysis mode or in distributed analysis mode. In local mode, you access other devices from your local computer. In distributed mode, you access other network devices from the remote server.

If you're not sure if you're using local analysis mode or distributed analysis mode, look at the title bar in the dialog box for accessing data files. If the title of the dialog box contains the word *Remote* (as in **Open Remote File**), or if the text **Remote Server: [server name]** appears at the top of the dialog box, you're using distributed analysis mode.

Note: This situation affects only dialog boxes for accessing data files (for example, Open Data, Save Data, Open Database, and Apply Data Dictionary). For all other file types (for example, Viewer files, syntax files, and script files), the local view is always used.

Availability of Procedures in Distributed Analysis Mode

In distributed analysis mode, procedures are available for use only if they are installed on both your local version and the version on the remote server.

If you have optional components installed locally that are not available on the remote server and you switch from your local computer to a remote server, the affected procedures will be removed from the menus and the corresponding command syntax will result in errors. Switching back to local mode will restore all affected procedures.

Absolute versus Relative Path Specifications

In distributed analysis mode, relative path specifications for data files and command syntax files are relative to the current server, not relative to your local computer. A relative path specification such as */mydocs/mydata.sav* does not point to a directory and file on your local drive; it points to a directory and file on the remote server's hard drive.

Windows UNC Path Specifications

If you are using a Windows server version, you can use universal naming convention (UNC) specifications when accessing data and syntax files with command syntax. The general form of a UNC specification is:

`\servername\sharename\path\filename`

- *Servername* is the name of the computer that contains the data file.
- *Sharename* is the folder (directory) on that computer that is designated as a shared folder.
- *Path* is any additional folder (subdirectory) path below the shared folder.
- *Filename* is the name of the data file.

An example is as follows:

```
GET FILE='\\hqdev001\\public\\july\\sales.sav'.
```

If the computer does not have a name assigned to it, you can use its IP address, as in:

```
GET FILE='\\204.125.125.53\\public\\july\\sales.sav'.
```

Even with UNC path specifications, you can access data and syntax files only from devices and folders that are designated as shared. When you use distributed analysis mode, this situation includes data and syntax files on your local computer.

UNIX Absolute Path Specifications

For UNIX server versions, there is no equivalent to the UNC path, and all directory paths must be absolute paths that start at the root of the server; relative paths are not allowed. For example, if the data file is located in */bin/data* and the current directory is also */bin/data*, GET FILE='sales.sav' is not valid; you must specify the entire path, as in:

```
GET FILE='/bin/sales.sav'.
INSERT FILE='/bin/salesjob.sps'.
```

Chapter 5. Data Editor

The Data Editor provides a convenient, spreadsheet-like method for creating and editing data files. The Data Editor window opens automatically when you start a session.

The Data Editor provides two views of your data:

- **Data View.** This view displays the actual data values or defined value labels.
- **Variable View.** This view displays variable definition information, including defined variable and value labels, data type (for example, string, date, or numeric), measurement level (nominal, ordinal, or scale), and user-defined missing values.

In both views, you can add, change, and delete information that is contained in the data file.

Data View

Many of the features of Data View are similar to the features that are found in spreadsheet applications. There are, however, several important distinctions:

- Rows are cases. Each row represents a case or an observation. For example, each individual respondent to a questionnaire is a case.
- Columns are variables. Each column represents a variable or characteristic that is being measured. For example, each item on a questionnaire is a variable.
- Cells contain values. Each cell contains a single value of a variable for a case. The cell is where the case and the variable intersect. Cells contain only data values. Unlike spreadsheet programs, cells in the Data Editor cannot contain formulas.
- The data file is rectangular. The dimensions of the data file are determined by the number of cases and variables. You can enter data in any cell. If you enter data in a cell outside the boundaries of the defined data file, the data rectangle is extended to include any rows and/or columns between that cell and the file boundaries. There are no "empty" cells within the boundaries of the data file. For numeric variables, blank cells are converted to the system-missing value. For string variables, a blank is considered a valid value.

Variable View

Variable View contains descriptions of the attributes of each variable in the data file. In Variable View:

- Rows are variables.
- Columns are variable attributes.

You can add or delete variables and modify attributes of variables, including the following attributes:

- Variable name
- Data type
- Number of digits or characters
- Number of decimal places
- Descriptive variable and value labels
- User-defined missing values
- Column width
- Measurement level

All of these attributes are saved when you save the data file.

In addition to defining variable properties in Variable View, there are two other methods for defining variable properties:

- The Copy Data Properties Wizard provides the ability to use an external IBM SPSS Statistics data file or another dataset that is available in the current session as a template for defining file and variable properties in the active dataset. You can also use variables in the active dataset as templates for other variables in the active dataset. Copy Data Properties is available on the Data menu in the Data Editor window.
- Define Variable Properties (also available on the Data menu in the Data Editor window) scans your data and lists all unique data values for any selected variables, identifies unlabeled values, and provides an auto-label feature. This method is particularly useful for categorical variables that use numeric codes to represent categories--for example, 0 = *Male*, 1 = *Female*.

To display or define variable attributes

1. Make the Data Editor the active window.
2. Double-click a variable name at the top of the column in Data View, or click the **Variable View** tab.
3. To define new variables, enter a variable name in any blank row.
4. Select the attribute(s) that you want to define or modify.

Variable names

The following rules apply to variable names:

- Each variable name must be unique; duplication is not allowed.
- Variable names can be up to 64 bytes long, and the first character must be a letter or one of the characters @, #, or \$. Subsequent characters can be any combination of letters, numbers, nonpunctuation characters, and a period (.). In code page mode, sixty-four bytes typically means 64 characters in single-byte languages (for example, English, French, German, Spanish, Italian, Hebrew, Russian, Greek, Arabic, and Thai) and 32 characters in double-byte languages (for example, Japanese, Chinese, and Korean). Many string characters that only take one byte in code page mode take two or more bytes in Unicode mode. For example, é is one byte in code page format but is two bytes in Unicode format; so *résumé* is six bytes in a code page file and eight bytes in Unicode mode.

Note: Letters include any nonpunctuation characters used in writing ordinary words in the languages supported in the platform's character set.

- Variable names cannot contain spaces.
- A # character in the first position of a variable name defines a scratch variable. You can only create scratch variables with command syntax. You cannot specify a # as the first character of a variable in dialog boxes that create new variables.
- A \$ sign in the first position indicates that the variable is a system variable. The \$ sign is not allowed as the initial character of a user-defined variable.
- The period, the underscore, and the characters \$, #, and @ can be used within variable names. For example, *A._\$@#1* is a valid variable name.
- Variable names ending with a period should be avoided, since the period may be interpreted as a command terminator. You can only create variables that end with a period in command syntax. You cannot create variables that end with a period in dialog boxes that create new variables.
- Variable names ending in underscores should be avoided, since such names may conflict with names of variables automatically created by commands and procedures.
- Reserved keywords cannot be used as variable names. Reserved keywords are ALL, AND, BY, EQ, GE, GT, LE, LT, NE, NOT, OR, TO, and WITH.
- Variable names can be defined with any mixture of uppercase and lowercase characters, and case is preserved for display purposes.
- When long variable names need to wrap onto multiple lines in output, lines are broken at underscores, periods, and points where content changes from lower case to upper case.

Variable measurement level

You can specify the level of measurement as scale (numeric data on an interval or ratio scale), ordinal, or nominal. Nominal and ordinal data can be either string (alphanumeric) or numeric.

- *Nominal*. A variable can be treated as nominal when its values represent categories with no intrinsic ranking (for example, the department of the company in which an employee works). Examples of nominal variables include region, postal code, and religious affiliation.
- *Ordinal*. A variable can be treated as ordinal when its values represent categories with some intrinsic ranking (for example, levels of service satisfaction from highly dissatisfied to highly satisfied). Examples of ordinal variables include attitude scores representing degree of satisfaction or confidence and preference rating scores.
- *Scale*. A variable can be treated as scale (continuous) when its values represent ordered categories with a meaningful metric, so that distance comparisons between values are appropriate. Examples of scale variables include age in years and income in thousands of dollars.

Note: For ordinal string variables, the alphabetic order of string values is assumed to reflect the true order of the categories. For example, for a string variable with the values of *low*, *medium*, *high*, the order of the categories is interpreted as *high*, *low*, *medium*, which is not the correct order. In general, it is more reliable to use numeric codes to represent ordinal data.

For new numeric variables created with transformations, data from external sources, and IBM SPSS Statistics data files created prior to version 8, default measurement level is determined by the conditions in the following table. Conditions are evaluated in the order listed in the table . The measurement level for the first condition that matches the data is applied.

Table 6. Rules for determining measurement level

Condition	Measurement Level
All values of a variable are missing	Nominal
Format is dollar or custom-currency	Continuous
Format is date or time (excluding Month and Wkday)	Continuous
Variable contains at least one non-integer value	Continuous
Variable contains at least one negative value	Continuous
Variable contains no valid values less than 10,000	Continuous
Variable has <i>N</i> or more valid, unique values*	Continuous
Variable has no valid values less than 10	Continuous
Variable has less than <i>N</i> valid, unique values*	Nominal

* *N* is a user-specified cut-off value. The default is 24.

- You can change the cutoff value in the Options dialog box. See the topic “Data Options” on page 197 for more information.
- The Define Variable Properties dialog box, available from the Data menu, can help you assign the correct measurement level. See the topic “Assigning the Measurement Level” on page 69 for more information.

Variable type

Variable Type specifies the data type for each variable. By default, all new variables are assumed to be numeric. You can use Variable Type to change the data type. The contents of the Variable Type dialog box depend on the selected data type. For some data types, there are text boxes for width and number of decimals; for other data types, you can simply select a format from a scrollable list of examples.

The available data types are as follows:

Numeric. A variable whose values are numbers. Values are displayed in standard numeric format. The Data Editor accepts numeric values in standard format or in scientific notation.

Comma. A numeric variable whose values are displayed with commas delimiting every three places and displayed with the period as a decimal delimiter. The Data Editor accepts numeric values for comma variables with or without commas or in scientific notation. Values cannot contain commas to the right of the decimal indicator.

Dot. A numeric variable whose values are displayed with periods delimiting every three places and with the comma as a decimal delimiter. The Data Editor accepts numeric values for dot variables with or without periods or in scientific notation. Values cannot contain periods to the right of the decimal indicator.

Scientific notation. A numeric variable whose values are displayed with an embedded E and a signed power-of-10 exponent. The Data Editor accepts numeric values for such variables with or without an exponent. The exponent can be preceded by E or D with an optional sign or by the sign alone--for example, 123, 1.23E2, 1.23D2, 1.23E+2, and 1.23+2.

Date. A numeric variable whose values are displayed in one of several calendar-date or clock-time formats. Select a format from the list. You can enter dates with slashes, hyphens, periods, commas, or blank spaces as delimiters. The century range for two-digit year values is determined by your Options settings (from the Edit menu, choose **Options**, and then click the **Data** tab).

Dollar. A numeric variable displayed with a leading dollar sign (\$), commas delimiting every three places, and a period as the decimal delimiter. You can enter data values with or without the leading dollar sign.

Custom currency. A numeric variable whose values are displayed in one of the custom currency formats that you have defined on the Currency tab of the Options dialog box. Defined custom currency characters cannot be used in data entry but are displayed in the Data Editor.

String. A variable whose values are not numeric and therefore are not used in calculations. The values can contain any characters up to the defined length. Uppercase and lowercase letters are considered distinct. This type is also known as an alphanumeric variable.

Restricted numeric. A variable whose values are restricted to non-negative integers. Values are displayed with leading zeros padded to the maximum width of the variable. Values can be entered in scientific notation.

To define variable type

1. Click the button in the *Type* cell for the variable that you want to define.
2. Select the data type in the Variable Type dialog box.
3. Click **OK**.

Input versus display formats

Depending on the format, the display of values in Data View may differ from the actual value as entered and stored internally. Following are some general guidelines:

- For numeric, comma, and dot formats, you can enter values with any number of decimal positions (up to 16), and the entire value is stored internally. The Data View displays only the defined number of decimal places and rounds values with more decimals. However, the complete value is used in all computations.
- For string variables, all values are right-padded to the maximum width. For a string variable with a maximum width of three, a value of *No* is stored internally as '*No*' and is not equivalent to '*No*'.
- For date formats, you can use slashes, dashes, spaces, commas, or periods as delimiters between day, month, and year values, and you can enter numbers, three-letter abbreviations, or complete names for

month values. Dates of the general format *dd-mmm-yy* are displayed with dashes as delimiters and three-letter abbreviations for the month. Dates of the general format *dd/mm/yy* and *mm/dd/yy* are displayed with slashes for delimiters and numbers for the month. Internally, dates are stored as the number of seconds from October 14, 1582. The century range for dates with two-digit years is determined by your Options settings (from the Edit menu, choose **Options**, and then click the **Data** tab).

- For time formats, you can use colons, periods, or spaces as delimiters between hours, minutes, and seconds. Times are displayed with colons as delimiters. Internally, times are stored as a number of seconds that represents a time interval. For example, 10:00:00 is stored internally as 36000, which is 60 (seconds per minute) × 60 (minutes per hour) × 10 (hours).

Variable labels

You can assign descriptive variable labels up to 256 characters (128 characters in double-byte languages). Variable labels can contain spaces and reserved characters that are not allowed in variable names.

To specify variable labels

1. Make the Data Editor the active window.
2. Double-click a variable name at the top of the column in Data View, or click the **Variable View** tab.
3. In the *Label* cell for the variable, enter the descriptive variable label.

Value labels

You can assign descriptive value labels for each value of a variable. This process is particularly useful if your data file uses numeric codes to represent non-numeric categories (for example, codes of 1 and 2 for *male* and *female*).

To specify value labels

1. Click the button in the *Values* cell for the variable that you want to define.
2. For each value, enter the value and a label.
3. Click **Add** to enter the value label.
4. Click **OK**.

Inserting line breaks in labels

Variable labels and value labels automatically wrap to multiple lines in pivot tables and charts if the cell or area isn't wide enough to display the entire label on one line, and you can edit results to insert manual line breaks if you want the label to wrap at a different point. You can also create variable labels and value labels that will *always* wrap at specified points and be displayed on multiple lines.

1. For variable labels, select the *Label* cell for the variable in Variable View in the Data Editor.
2. For value labels, select the *Values* cell for the variable in Variable View in the Data Editor, click the  button in the cell, and select the label that you want to modify in the Value Labels dialog box.
3. At the place in the label where you want the label to wrap, type `\n`.

The `\n` is not displayed in pivot tables or charts; it is interpreted as a line break character.

Missing values

Missing Values defines specified data values as **user-missing**. For example, you might want to distinguish between data that are missing because a respondent refused to answer and data that are missing because the question didn't apply to that respondent. Data values that are specified as user-missing are flagged for special treatment and are excluded from most calculations.

To define missing values

1. Click the button in the *Missing* cell for the variable that you want to define.
2. Enter the values or range of values that represent missing data.

Roles

Some dialogs support predefined roles that can be used to pre-select variables for analysis. When you open one of these dialogs, variables that meet the role requirements will be automatically displayed in the destination list(s). Available roles are:

Input. The variable will be used as an input (e.g., predictor, independent variable).

Target. The variable will be used as an output or target (e.g., dependent variable).

Both. The variable will be used as both input and output.

None. The variable has no role assignment.

Partition. The variable will be used to partition the data into separate samples for training, testing, and validation.

Split. Included for round-trip compatibility with IBM SPSS Modeler. Variables with this role are *not* used as split-file variables in IBM SPSS Statistics.

- By default, all variables are assigned the *Input* role. This includes data from external file formats and data files from versions of IBM SPSS Statistics prior to version 18.
- Role assignment only affects dialogs that support role assignment. It has no effect on command syntax.

To assign roles

1. Select the role from the list in the *Role* cell for the variable.

Column width

You can specify a number of characters for the column width. Column widths can also be changed in Data View by clicking and dragging the column borders.

- Column width for proportional fonts is based on average character width. Depending on the characters used in the value, more or fewer characters may be displayed in the specified width.
- Column width affect only the display of values in the Data Editor. Changing the column width does not change the defined width of a variable.

Variable alignment

Alignment controls the display of data values and/or value labels in Data View. The default alignment is right for numeric variables and left for string variables. This setting affects only the display in Data View.

Applying variable definition attributes to multiple variables

After you have defined variable definition attributes for a variable, you can copy one or more attributes and apply them to one or more variables.

Basic copy and paste operations are used to apply variable definition attributes. You can:

- Copy a single attribute (for example, value labels) and paste it to the same attribute cell(s) for one or more variables.
- Copy all attributes from one variable and paste them to one or more other variables.
- Create multiple new variables with all the attributes of a copied variable.

Applying variable definition attributes to other variables

To Apply Individual Attributes from a Defined Variable

1. In Variable View, select the attribute cell that you want to apply to other variables.
2. From the menus choose:
Edit > Copy
3. Select the attribute cell(s) to which you want to apply the attribute. (You can select multiple target variables.)
4. From the menus choose:
Edit > Paste

If you paste the attribute to blank rows, new variables are created with default attributes for all attributes except the selected attribute.

To apply all attributes from a defined variable

1. In Variable View, select the row number for the variable with the attributes that you want to use. (The entire row is highlighted.)
2. From the menus choose:
Edit > Copy
3. Select the row number(s) for the variable(s) to which you want to apply the attributes. (You can select multiple target variables.)
4. From the menus choose:
Edit > Paste

Generating multiple new variables with the same attributes

1. In Variable View, click the row number for the variable that has the attributes that you want to use for the new variable. (The entire row is highlighted.)
2. From the menus choose:
Edit > Copy
3. Click the empty row number beneath the last defined variable in the data file.
4. From the menus choose:
Edit > Paste Variables...
5. In the Paste Variables dialog box, enter the number of variables that you want to create.
6. Enter a prefix and starting number for the new variables.
7. Click **OK**.

The new variable names will consist of the specified prefix plus a sequential number starting with the specified number.

Custom Variable Attributes

In addition to the standard variable attributes (for example, value labels, missing values, measurement level), you can create your own custom variable attributes. Like standard variable attributes, these custom attributes are saved with IBM SPSS Statistics data files. Therefore, you could create a variable attribute that identifies the type of response for survey questions (for example, single selection, multiple selection, fill-in-the-blank) or the formulas used for computed variables.

Creating Custom Variable Attributes

To create new custom attributes:

1. In Variable View, from the menus choose:
Data > New Custom Attribute...

2. Drag and drop the variables to which you want to assign the new attribute to the Selected Variables list.
3. Enter a name for the attribute. Attribute names must follow the same rules as variable names. See the topic “Variable names” on page 50 for more information.
4. Enter an optional value for the attribute. If you select multiple variables, the value is assigned to all selected variables. You can leave this blank and then enter values for each variable in Variable View.

Display attribute in the Data Editor. Displays the attribute in Variable View of the Data Editor. For information on controlling the display of custom attributes, see “Displaying and Editing Custom Variable Attributes” below.

Display Defined List of Attributes. Displays a list of custom attributes already defined for the dataset. Attribute names that begin with a dollar sign (\$) are reserved attributes that cannot be modified.

Displaying and Editing Custom Variable Attributes

Custom variable attributes can be displayed and edited in the Data Editor in Variable View.

- Custom variable attribute names are enclosed in square brackets.
- Attribute names that begin with a dollar sign are reserved and cannot be modified.
- A blank cell indicates that the attribute does not exist for that variable; the text *Empty* displayed in a cell indicates that the attribute exists for that variable but no value has been assigned to the attribute for that variable. Once you enter text in the cell, the attribute exists for that variable with the value you enter.
- The text *Array...* displayed in a cell indicates that this is an attribute array--an attribute that contains multiple values. Click the button in the cell to display the list of values.

To Display and Edit Custom Variable Attributes

1. In Variable View, from the menus choose:
View > Customize Variable View...
2. Select (check) the custom variable attributes you want to display. (The custom variable attributes are the ones enclosed in square brackets.)

Once the attributes are displayed in Variable View, you can edit them directly in the Data Editor.

Variable Attribute Arrays: The text *Array...* (displayed in a cell for a custom variable attribute in Variable View or in the Custom Variable Properties dialog box in Define Variable Properties) indicates that this is an **attribute array**, an attribute that contains multiple values. For example, you could have an attribute array that identifies all of the source variables used to compute a derived variable. Click the button in the cell to display and edit the list of values.

Customizing Variable View

You can use Customize Variable View to control which attributes are displayed in Variable View (for example, name, type, label) and the order in which they are displayed.

- Any custom variable attributes associated with the dataset are enclosed in square brackets. See the topic “Creating Custom Variable Attributes” on page 55 for more information.
- Customized display settings are saved with IBM SPSS Statistics data files.
- You can also control the default display and order of attributes in Variable View. See the topic “Changing the default variable view” on page 199 for more information.

To customize Variable View

1. In Variable View, from the menus choose:
View > Customize Variable View...
2. Select (check) the variable attributes you want to display.

3. Use the up and down arrow buttons to change the display order of the attributes.

Restore Defaults. Apply the default display and order settings.

Spell checking

variable and value labels

To check the spelling of variable labels and value labels:

1. Select the Variable View tab in the Data Editor window.
2. Right-click the *Labels* or *Values* column and from the pop-up menu choose:

Spelling

or

3. In Variable View, from the menus choose:

Utilities > Spelling

or

4. In the Value Labels dialog box, click **Spelling**. (This limits the spell checking to the value labels for a particular variable.)

Spell checking is limited to variable labels and value labels in Variable View of the Data Editor.

String data values

To check the spelling of string data values:

1. Select the Data View tab of the Data Editor.
2. Optionally, select one or more variables (columns) to check. To select a variable, click the variable name at the top of the column.
3. From the menus choose:
Utilities > Spelling
 - If there are no selected variables in Data View, all string variables will be checked.
 - If there are no string variables in the dataset or the none of the selected variables is a string variable, the Spelling option on the Utilities menu is disabled.

Customizing Variable View

You can use Customize Variable View to control which attributes are displayed in Variable View (for example, name, type, label) and the order in which they are displayed.

- Any custom variable attributes associated with the dataset are enclosed in square brackets. See the topic “Creating Custom Variable Attributes” on page 55 for more information.
- Customized display settings are saved with IBM SPSS Statistics data files.
- You can also control the default display and order of attributes in Variable View. See the topic “Changing the default variable view” on page 199 for more information.

To customize Variable View

1. In Variable View, from the menus choose:
View > Customize Variable View...
2. Select (check) the variable attributes you want to display.
3. Use the up and down arrow buttons to change the display order of the attributes.

Restore Defaults. Apply the default display and order settings.

Spell checking

variable and value labels

To check the spelling of variable labels and value labels:

1. Select the Variable View tab in the Data Editor window.
2. Right-click the *Labels* or *Values* column and from the pop-up menu choose:
Spelling
or
3. In Variable View, from the menus choose:
Utilities > Spelling
or
4. In the Value Labels dialog box, click **Spelling**. (This limits the spell checking to the value labels for a particular variable.)

Spell checking is limited to variable labels and value labels in Variable View of the Data Editor.

String data values

To check the spelling of string data values:

1. Select the Data View tab of the Data Editor.
2. Optionally, select one or more variables (columns) to check. To select a variable, click the variable name at the top of the column.
3. From the menus choose:
Utilities > Spelling
 - If there are no selected variables in Data View, all string variables will be checked.
 - If there are no string variables in the dataset or the none of the selected variables is a string variable, the Spelling option on the Utilities menu is disabled.

Entering data

In Data View, you can enter data directly in the Data Editor. You can enter data in any order. You can enter data by case or by variable, for selected areas or for individual cells.

- The active cell is highlighted.
- The variable name and row number of the active cell are displayed in the top left corner of the Data Editor.
- When you select a cell and enter a data value, the value is displayed in the cell editor at the top of the Data Editor.
- Data values are not recorded until you press Enter or select another cell.
- To enter anything other than simple numeric data, you must define the variable type first.

If you enter a value in an empty column, the Data Editor automatically creates a new variable and assigns a variable name.

To enter numeric data

1. Select a cell in Data View.
2. Enter the data value. (The value is displayed in the cell editor at the top of the Data Editor.)
3. To record the value, press Enter or select another cell.

To enter non-numeric data

1. Double-click a variable name at the top of the column in Data View or click the **Variable View** tab.
2. Click the button in the *Type* cell for the variable.
3. Select the data type in the Variable Type dialog box.
4. Click **OK**.
5. Double-click the row number or click the **Data View** tab.
6. Enter the data in the column for the newly defined variable.

To use value labels for data entry

1. If value labels aren't currently displayed in Data View, from the menus choose:
View > Value Labels
2. Click the cell in which you want to enter the value.
3. Choose a value label from the drop-down list.

The value is entered, and the value label is displayed in the cell.

Note: This process works only if you have defined value labels for the variable.

Data value restrictions in the data editor

The defined variable type and width determine the type of value that can be entered in the cell in Data View.

- If you type a character that is not allowed by the defined variable type, the character is not entered.
- For string variables, characters beyond the defined width are not allowed.
- For numeric variables, integer values that exceed the defined width can be entered, but the Data Editor displays either scientific notation or a portion of the value followed by an ellipsis (...) to indicate that the value is wider than the defined width. To display the value in the cell, change the defined width of the variable.

Note: Changing the column width does not affect the variable width.

Editing data

With the Data Editor, you can modify data values in Data View in many ways. You can:

- Change data values
- Cut, copy, and paste data values
- Add and delete cases
- Add and delete variables
- Change the order of variables

Replacing or modifying data values

To Delete the Old Value and Enter a New Value

1. In Data View, double-click the cell. (The cell value is displayed in the cell editor.)
2. Edit the value directly in the cell or in the cell editor.
3. Press Enter or select another cell to record the new value.

Cutting, copying, and pasting data values

You can cut, copy, and paste individual cell values or groups of values in the Data Editor. You can:

- Move or copy a single cell value to another cell
- Move or copy a single cell value to a group of cells

- Move or copy the values for a single case (row) to multiple cases
- Move or copy the values for a single variable (column) to multiple variables
- Move or copy a group of cell values to another group of cells
- Copy and paste variable names and labels. The options are available through the Data Editor's right-click menu.

Data conversion for pasted values in the data editor

If the defined variable types of the source and target cells are not the same, the Data Editor attempts to convert the value. If no conversion is possible, the system-missing value is inserted in the target cell.

Converting numeric or date into string. Numeric (for example, numeric, dollar, dot, or comma) and date formats are converted to strings if they are pasted into a string variable cell. The string value is the numeric value as displayed in the cell. For example, for a dollar format variable, the displayed dollar sign becomes part of the string value. Values that exceed the defined string variable width are truncated.

Converting string into numeric or date. String values that contain acceptable characters for the numeric or date format of the target cell are converted to the equivalent numeric or date value. For example, a string value of 25/12/91 is converted to a valid date if the format type of the target cell is one of the day-month-year formats, but the value is converted to system-missing if the format type of the target cell is one of the month-day-year formats.

Converting date into numeric. Date and time values are converted to a number of seconds if the target cell is one of the numeric formats (for example, numeric, dollar, dot, or comma). Because dates are stored internally as the number of seconds since October 14, 1582, converting dates to numeric values can yield some extremely large numbers. For example, the date 10/29/91 is converted to a numeric value of 12,908,073,600.

Converting numeric into date or time. Numeric values are converted to dates or times if the value represents a number of seconds that can produce a valid date or time. For dates, numeric values that are less than 86,400 are converted to the system-missing value.

Inserting new cases

Entering data in a cell in a blank row automatically creates a new case. The Data Editor inserts the system-missing value for all other variables for that case. If there are any blank rows between the new case and the existing cases, the blank rows become new cases with the system-missing value for all variables. You can also insert new cases between existing cases.

To insert new cases between existing cases

1. In Data View, select any cell in the case (row) below the position where you want to insert the new case.
2. From the menus choose:
Edit > Insert Cases

A new row is inserted for the case, and all variables receive the system-missing value.

Inserting new variables

Entering data in an empty column in Data View or in an empty row in Variable View automatically creates a new variable with a default variable name (the prefix *var* and a sequential number) and a default data format type (numeric). The Data Editor inserts the system-missing value for all cases for the new variable. If there are any empty columns in Data View or empty rows in Variable View between the new variable and the existing variables, these rows or columns also become new variables with the system-missing value for all cases. You can also insert new variables between existing variables.

To insert new variables between existing variables

1. Select any cell in the variable to the right of (Data View) or below (Variable View) the position where you want to insert the new variable.
2. From the menus choose:

Edit > Insert Variable

A new variable is inserted with the system-missing value for all cases.

To move variables

1. To select the variable, click the variable name in Data View or the row number for the variable in Variable View.
2. Drag and drop the variable to the new location.
3. If you want to place the variable between two existing variables: In Data View, drop the variable on the variable column to the right of where you want to place the variable, or in Variable View, drop the variable on the variable row below where you want to place the variable.

To change data type

You can change the data type for a variable at any time by using the Variable Type dialog box in Variable View. The Data Editor will attempt to convert existing values to the new type. If no conversion is possible, the system-missing value is assigned. The conversion rules are the same as the rules for pasting data values to a variable with a different format type. If the change in data format may result in the loss of missing-value specifications or value labels, the Data Editor displays an alert box and asks whether you want to proceed with the change or cancel it.

Finding cases, variables, or imputations

The Go To dialog box finds the specified case (row) number or variable name in the Data Editor.

Cases

1. For cases, from the menus choose:
Edit > Go to Case...
2. Enter an integer value that represents the current row number in Data View.

Note: The current row number for a particular case can change due to sorting and other actions.

Variables

1. For variables, from the menus choose:
Edit > Go to Variable...
2. Enter the variable name or select the variable from the drop-down list.

Imputations

1. From the menus choose:
Edit > Go to Imputation...
2. Select the imputation (or Original data) from the drop-down list.

Alternatively, you can select the imputation from the drop-down list in the edit bar in Data View of the Data Editor.

Relative case position is preserved when selecting imputations. For example, if there are 1000 cases in the original dataset, case 1034, the 34th case in the first imputation, displays at the top of the grid. If you select imputation 2 in the dropdown, case 2034, the 34th case in imputation 2, would display at the top of

the grid. If you select **Original data** in the dropdown, case 34 would display at the top of the grid. Column position is also preserved when navigating between imputations, so that it is easy to compare values between imputations.

Finding and replacing data and attribute values

To find and/or replace data values in Data View or attribute values in Variable View:

1. Click a cell in the column you want to search. (Finding and replacing values is restricted to a single column.)
2. From the menus choose:
Edit > Find
or
Edit > Replace

Data View

- You cannot search up in Data View. The search direction is always down.
- For dates and times, the formatted values as displayed in Data View are searched. For example, a date displayed as 10/28/2007 will not be found by a search for a date of 10-28-2007.
- For other numeric variables, **Contains**, **Begins with**, and **Ends with** search formatted values. For example, with the **Begins with** option, a search value of \$123 for a Dollar format variable will find both \$123.00 and \$123.40 but not \$1,234. With the **Entire cell** option, the search value can be formatted or unformatted (simple F numeric format), but only exact numeric values (to the precision displayed in the Data Editor) are matched.
- The numeric system-missing value is represented by a single period (.). To find system-missing values, enter a single period as the search value and select **Entire cell**.
- If value labels are displayed for the selected variable column, the label text is searched, not the underlying data value, and you cannot replace the label text.

Variable View

- Find is only available for the *Name*, *Label*, *Values*, *Missing*, and custom variable attribute columns.
- Replace is only available for the *Label*, *Values*, and custom attribute columns.
- In the *Values* (value labels) column, the search string can match either the data value or a value label.
Note: Replacing the data value will delete any previous value label associated with that value.

Obtaining Descriptive Statistics for Selected Variables

To obtain descriptive statistics for selected variables:

1. Right-click the selected variables in either Data View or Variable View.
2. From the pop-up menu select **Descriptive Statistics**.

By default, frequency tables (tables of counts) are displayed for all variables with 24 or fewer unique values. Summary statistics are determined by variable measurement level and data type (numeric or string):

- **String.** No summary statistics are calculated for string variables.
- **Numeric, nominal or unknown measurement level.** Range, minimum, maximum, mode.
- **Numeric, ordinal measurement level.** Range, minimum, maximum, mode, mean, median.
- **Numeric, continuous (scale) measurement level.** Range, minimum, maximum, mode, mean, median, standard deviation.

You can also obtain bar charts for nominal and ordinal variables, histograms for continuous (scale) variables, and change the cut-off value that determines when to display frequency tables. See the topic "Output options" on page 200 for more information.

Case selection status in the Data Editor

If you have selected a subset of cases but have not discarded unselected cases, unselected cases are marked in the Data Editor with a diagonal line (slash) through the row number.

1
2
3
4
5
6
7
8
9
10
11

Figure 1. Filtered cases in the Data Editor

Data Editor display options

The View menu provides several display options for the Data Editor:

Fonts. This option controls the font characteristics of the data display.

Grid Lines. This option toggles the display of grid lines.

Value Labels. This option toggles between the display of actual data values and user-defined descriptive value labels. This option is available only in Data View.

Using Multiple Views

In Data View, you can create multiple views (panes) by using the splitters that are located below the horizontal scroll bar and to the right of the vertical scroll bar.

You can also use the Window menu to insert and remove pane splitters. To insert splitters:

1. In Data View, from the menus choose:

Window > Split

Splitters are inserted above and to the left of the selected cell.

- If the top left cell is selected, splitters are inserted to divide the current view approximately in half, both horizontally and vertically.
- If any cell other than the top cell in the first column is selected, a horizontal pane splitter is inserted above the selected cell.
- If any cell other than the first cell in the top row is selected, a vertical pane splitter is inserted to the left of the selected cell.

Data Editor printing

A data file is printed as it appears on the screen.

- The information in the currently displayed view is printed. In Data View, the data are printed. In Variable View, data definition information is printed.
- Grid lines are printed if they are currently displayed in the selected view.
- Value labels are printed in Data View if they are currently displayed. Otherwise, the actual data values are printed.

Use the View menu in the Data Editor window to display or hide grid lines and toggle between the display of data values and value labels.

To print Data Editor contents

1. Make the Data Editor the active window.
2. Click the tab for the view that you want to print.
3. From the menus choose:
File > Print...

Chapter 6. Working with Multiple Data Sources

Starting with version 14.0, multiple data sources can be open at the same time, making it easier to:

- Switch back and forth between data sources.
- Compare the contents of different data sources.
- Copy and paste data between data sources.
- Create multiple subsets of cases and/or variables for analysis.
- Merge multiple data sources from various data formats (for example, spreadsheet, database, text data) without saving each data source first.

Basic Handling of Multiple Data Sources

By default, each data source that you open is displayed in a new Data Editor window. (See “General options” on page 195 for information on changing the default behavior to only display one dataset at a time, in a single Data Editor window.)

- Any previously open data sources remain open and available for further use.
- When you first open a data source, it automatically becomes the **active dataset**.
- You can change the active dataset simply by clicking anywhere in the Data Editor window of the data source that you want to use or by selecting the Data Editor window for that data source from the Window menu.
- Only the variables in the active dataset are available for analysis.
- You cannot change the active dataset when any dialog box that accesses the data is open (including all dialog boxes that display variable lists).
- At least one Data Editor window must be open during a session. When you close the last open Data Editor window, IBM SPSS Statistics automatically shuts down, prompting you to save changes first.

Working with Multiple Datasets in Command Syntax

If you use command syntax to open data sources (for example, GET FILE, GET DATA), you need to use the DATASET NAME command to name each dataset explicitly in order to have more than one data source open at the same time.

When working with command syntax, the active dataset name is displayed on the toolbar of the syntax window. All of the following actions can change the active dataset:

- Use the DATASET ACTIVATE command.
- Click anywhere in the Data Editor window of a dataset.
- Select a dataset name from the **Active** drop-down list on the toolbar in the syntax window.

Copying and Pasting Information between Datasets

You can copy both data and variable definition attributes from one dataset to another dataset in basically the same way that you copy and paste information within a single data file.

- Copying and pasting selected data cells in Data View pastes only the data values, with no variable definition attributes.
- Copying and pasting an entire variable in Data View by selecting the variable name at the top of the column pastes all of the data and all of the variable definition attributes for that variable.
- Copying and pasting variable definition attributes or entire variables in Variable View pastes the selected attributes (or the entire variable definition) but does not paste any data values.

Renaming Datasets

When you open a data source through the menus and dialog boxes, each data source is automatically assigned a dataset name of *DataSetn*, where *n* is a sequential integer value, and when you open a data source using command syntax, no dataset name is assigned unless you explicitly specify one with **DATASET NAME**. To provide more descriptive dataset names:

1. From the menus in the Data Editor window for the dataset whose name you want to change choose:
File > Rename Dataset...
 2. Enter a new dataset name that conforms to variable naming rules. See the topic “Variable names” on page 50 for more information.
-

Suppressing Multiple Datasets

If you prefer to have only one dataset available at a time and want to suppress the multiple dataset feature:

1. From the menus choose:
Edit > Options...
2. Click the **General** tab.

Select (check) **Open only one dataset at a time**.

See the topic “General options” on page 195 for more information.

Chapter 7. Data preparation

Once you've opened a data file or entered data in the Data Editor, you can start creating reports, charts, and analyses without any additional preliminary work. However, there are some additional data preparation features that you may find useful, including the ability to:

- Assign variable properties that describe the data and determine how certain values should be treated.
- Identify cases that may contain duplicate information and exclude those cases from analyses or delete them from the data file.
- Create new variables with a few distinct categories that represent ranges of values from variables with a large number of possible values.

Variable properties

Data entered in the Data Editor in Data View or read from an external file format (such as an Excel spreadsheet or a text data file) lack certain variable properties that you may find very useful, including:

- Definition of descriptive value labels for numeric codes (for example, 0 = *Male* and 1 = *Female*).
- Identification of missing values codes (for example, 99 = *Not applicable*).
- Assignment of measurement level (nominal, ordinal, or scale).

All of these variable properties (and others) can be assigned in Variable View in the Data Editor. There are also several utilities that can assist you in this process:

- **Define Variable Properties** can help you define descriptive value labels and missing values. This is particularly useful for categorical data with numeric codes used for category values. See the topic "Defining Variable Properties" for more information.
- **Set Measurement Level for Unknown** identifies variables (fields) that do not have a defined measurement level and provides the ability to set the measurement level for those variables. This is important for procedures in which measurement level can affect the results or determines which features are available. See the topic "Setting measurement level for variables with unknown measurement level" on page 70 for more information.
- **Copy Data Properties** provides the ability to use an existing IBM SPSS Statistics data file as a template for file and variable properties in the current data file. This is particularly useful if you frequently use external-format data files that contain similar content (such as monthly reports in Excel format). See the topic "Copying Data Properties" on page 73 for more information.

Defining Variable Properties

Define Variable Properties is designed to assist you in the process of assigning attributes to variables, including creating descriptive value labels for categorical (nominal, ordinal) variables. Define Variable Properties:

- Scans the actual data values and lists all unique data values for each selected variable.
- Identifies unlabeled values and provides an "auto-label" feature.
- Provides the ability to copy defined value labels and other attributes from another variable to the selected variable or from the selected variable to multiple additional variables.

Note: To use Define Variable Properties without first scanning cases, enter 0 for the number of cases to scan.

To Define Variable Properties

1. From the menus choose:
Data > Define Variable Properties...
2. Select the numeric or string variables for which you want to create value labels or define or change other variable properties, such as missing values or descriptive variable labels.
3. Specify the number of cases to scan to generate the list of unique values. This is particularly useful for data files with a large number of cases for which a scan of the complete data file might take a significant amount of time.
4. Specify an upper limit for the number of unique values to display. This is primarily useful to prevent listing hundreds, thousands, or even millions of values for scale (continuous interval, ratio) variables.
5. Click **Continue** to open the main Define Variable Properties dialog box.
6. Select a variable for which you want to create value labels or define or change other variable properties.
7. Enter the label text for any unlabeled values that are displayed in the Value Label grid.
8. If there are values for which you want to create value labels but those values are not displayed, you can enter values in the *Value* column below the last scanned value.
9. Repeat this process for each listed variable for which you want to create value labels.
10. Click **OK** to apply the value labels and other variable properties.

Defining Value Labels and Other Variable Properties

The Define Variable Properties main dialog box provides the following information for the scanned variables:

Scanned Variable List. For each scanned variable, a check mark in the *Unlabeled (U.)* column indicates that the variable contains values without assigned value labels.

To sort the variable list to display all variables with unlabeled values at the top of the list:

1. Click the *Unlabeled* column heading under Scanned Variable List.

You can also sort by variable name or measurement level by clicking the corresponding column heading under Scanned Variable List.

Value Label Grid

- **Label.** Displays any value labels that have already been defined. You can add or change labels in this column.
- **Value.** Unique values for each selected variable. This list of unique values is based on the number of scanned cases. For example, if you scanned only the first 100 cases in the data file, then the list reflects only the unique values present in those cases. If the data file has already been sorted by the variable for which you want to assign value labels, the list may display far fewer unique values than are actually present in the data.
- **Count.** The number of times each value occurs in the scanned cases.
- **Missing.** Values defined as representing missing data. You can change the missing values designation of the category by clicking the check box. A check indicates that the category is defined as a user-missing category. If a variable already has a range of values defined as user-missing (for example, 90-99), you cannot add or delete missing values categories for that variable with Define Variable Properties. You can use Variable View in the Data Editor to modify the missing values categories for variables with missing values ranges. See the topic "Missing values" on page 53 for more information.
- **Changed.** Indicates that you have added or changed a value label.

Note: If you specified 0 for the number of cases to scan in the initial dialog box, the Value Label grid will initially be blank, except for any preexisting value labels and/or defined missing values categories for the selected variable. In addition, the **Suggest** button for the measurement level will be disabled.

Measurement Level. Value labels are primarily useful for categorical (nominal and ordinal) variables, and some procedures treat categorical and scale variables differently; so it is sometimes important to assign the correct measurement level. However, by default, all new numeric variables are assigned the scale measurement level. Thus, many variables that are in fact categorical may initially be displayed as scale.

If you are unsure of what measurement level to assign to a variable, click **Suggest**.

Role. Some dialogs support the ability to pre-select variables for analysis based on defined roles. See the topic “Roles” on page 54 for more information.

Copy Properties. You can copy value labels and other variable properties from another variable to the currently selected variable or from the currently selected variable to one or more other variables.

Unlabeled Values. To create labels for unlabeled values automatically, click **Automatic Labels**.

Variable Label and Display Format

You can change the descriptive variable label and the display format.

- You cannot change the variable's fundamental type (string or numeric).
- For string variables, you can change only the variable label, not the display format.
- For numeric variables, you can change the numeric type (such as numeric, date, dollar, or custom currency), width (maximum number of digits, including any decimal and/or grouping indicators), and number of decimal positions.
- For numeric date format, you can select a specific date format (such as dd-mm-yyyy, mm/dd/yy, and yyyyddd)
- For numeric custom format, you can select one of five custom currency formats (CCA through CCE). See the topic “Currency options” on page 200 for more information.
- An asterisk is displayed in the *Value* column if the specified width is less than the width of the scanned values or the displayed values for preexisting defined value labels or missing values categories.
- A period (.) is displayed if the scanned values or the displayed values for preexisting defined value labels or missing values categories are invalid for the selected display format type. For example, an internal numeric value of less than 86,400 is invalid for a date format variable.

Assigning the Measurement Level

When you click **Suggest** for the measurement level in the Define Variable Properties main dialog box, the current variable is evaluated based on the scanned cases and defined value labels, and a measurement level is suggested in the Suggest Measurement Level dialog box that opens. The Explanation area provides a brief description of the criteria used to provide the suggested measurement level.

Note: Values defined as representing missing values are not included in the evaluation for measurement level. For example, the explanation for the suggested measurement level may indicate that the suggestion is in part based on the fact that the variable contains no negative values, whereas it may in fact contain negative values--but those values are already defined as missing values.

1. Click **Continue** to accept the suggested level of measurement or **Cancel** to leave the measurement level unchanged.

Custom Variable Attributes

The **Attributes** button in Define Variable Properties opens the Custom Variable Attributes dialog box. In addition to the standard variable attributes, such as value labels, missing values, and measurement level, you can create your own custom variable attributes. Like standard variable attributes, these custom attributes are saved with IBM SPSS Statistics data files.

Name. Attribute names must follow the same rules as variable names. See the topic “Variable names” on page 50 for more information.

Value. The value assigned to the attribute for the selected variable.

- Attribute names that begin with a dollar sign are reserved and cannot be modified. You can view the contents of a reserved attribute by clicking the button in the desired cell.
- The text *Array...*, displayed in a Value cell, indicates that this is an **attribute array**, an attribute that contains multiple values. Click the button in the cell to display the list of values.

Copying Variable Properties

The Apply Labels and Level dialog box is displayed when you click **From Another Variable** or **To Other Variables** in the Define Variable Properties main dialog box. It displays all of the scanned variables that match the current variable's type (numeric or string). For string variables, the defined width must also match.

- Select a single variable from which to copy value labels and other variable properties (except variable label).
- or
- Select one or more variables to which to copy value labels and other variable properties.
 - Click **Copy** to copy the value labels and the measurement level.
 - Existing value labels and missing value categories for target variable(s) are not replaced.
 - Value labels and missing value categories for values not already defined for the target variable(s) are added to the set of value labels and missing value categories for the target variable(s).
 - The measurement level for the target variable is always replaced.
 - The role for the target variable is always replaced.
 - If either the source or target variable has a defined range of missing values, missing values definitions are not copied.

Setting measurement level for variables with unknown measurement level

For some procedures, measurement level can affect the results or determine which features are available, and you cannot access the dialogs for these procedures until all variables have a defined measurement level. The Set Measurement Level for Unknown dialog allows you to define measurement level for any variables with an unknown measurement level without performing a data pass (which may be time-consuming for large data files).

Under certain conditions, the measurement level for some or all numeric variables (fields) in a file may be unknown. These conditions include:

- Numeric variables from Excel 95 or later files, text data files, or data base sources prior to the first data pass.
- New numeric variables created with transformation commands prior to the first data pass after creation of those variables.

These conditions apply primarily to reading data or creating new variables via command syntax. Dialogs for reading data and creating new transformed variables automatically perform a data pass that sets the measurement level, based on the default measurement level rules.

To set the measurement level for variables with an unknown measurement level

1. In the alert dialog that appears for the procedure, click **Assign Manually**.
or
2. From the menus, choose:
Data > Set Measurement Level for Unknown
3. Move variables (fields) from the source list to the appropriate measurement level destination list.
 - *Nominal*. A variable can be treated as nominal when its values represent categories with no intrinsic ranking (for example, the department of the company in which an employee works). Examples of nominal variables include region, postal code, and religious affiliation.
 - *Ordinal*. A variable can be treated as ordinal when its values represent categories with some intrinsic ranking (for example, levels of service satisfaction from highly dissatisfied to highly satisfied). Examples of ordinal variables include attitude scores representing degree of satisfaction or confidence and preference rating scores.
 - *Continuous*. A variable can be treated as scale (continuous) when its values represent ordered categories with a meaningful metric, so that distance comparisons between values are appropriate. Examples of scale variables include age in years and income in thousands of dollars.

Multiple Response Sets

Custom Tables and the Chart Builder support a special kind of "variable" called a **multiple response set**. Multiple response sets aren't really "variables" in the normal sense. You can't see them in the Data Editor, and other procedures don't recognize them. Multiple response sets use multiple variables to record responses to questions where the respondent can give more than one answer. Multiple response sets are treated like categorical variables, and most of the things you can do with categorical variables, you can also do with multiple response sets.

Multiple response sets are constructed from multiple variables in the data file. A multiple response set is a special construct within a data file. You can define and save multiple response sets in IBM SPSS Statistics data files, but you cannot import or export multiple response sets from/to other file formats. You can copy multiple response sets from other IBM SPSS Statistics data files using Copy Data Properties, which is accessed from the Data menu in the Data Editor window.

Defining Multiple Response Sets

To define multiple response sets:

1. From the menus, choose:
Data > Define Multiple Response Sets...
2. Select two or more variables. If your variables are coded as dichotomies, indicate which value you want to have counted.
3. Enter a unique name for each multiple response set. The name can be up to 63 bytes long. A dollar sign is automatically added to the beginning of the set name.
4. Enter a descriptive label for the set. (This is optional.)
5. Click **Add** to add the multiple response set to the list of defined sets.

Dichotomies

A multiple dichotomy set typically consists of multiple dichotomous variables: variables with only two possible values of a yes/no, present/absent, checked/not checked nature. Although the variables may not be strictly dichotomous, all of the variables in the set are coded the same way, and the Counted Value represents the positive/present/checked condition.

For example, a survey asks the question, "Which of the following sources do you rely on for news?" and provides five possible responses. The respondent can indicate multiple choices by checking a box next to each choice. The five responses become five variables in the data file, coded 0 for *No* (not checked) and 1 for *Yes* (checked). In the multiple dichotomy set, the Counted Value is 1.

The sample data file *survey_sample.sav* already has three defined multiple response sets. *\$mltnews* is a multiple dichotomy set.

1. Select (click) *\$mltnews* in the Mult. Response Sets list.

This displays the variables and settings used to define this multiple response set.

- The Variables in Set list displays the five variables used to construct the multiple response set.
- The Variable Coding group indicates that the variables are dichotomous.
- The Counted Value is 1.

2. Select (click) one of the variables in the Variables in Set list.

3. Right-click the variable and select **Variable Information** from the pop-up pop-up menu.

4. In the Variable Information window, click the arrow on the Value Labels drop-down list to display the entire list of defined value labels.

The value labels indicate that the variable is a dichotomy with values of 0 and 1, representing *No* and *Yes*, respectively. All five variables in the list are coded the same way, and the value of 1 (the code for *Yes*) is the counted value for the multiple dichotomy set.

Categories

A multiple category set consists of multiple variables, all coded the same way, often with many possible response categories. For example, a survey item states, "Name up to three nationalities that best describe your ethnic heritage." There may be hundreds of possible responses, but for coding purposes the list is limited to the 40 most common nationalities, with everything else relegated to an "other" category. In the data file, the three choices become three variables, each with 41 categories (40 coded nationalities and one "other" category).

In the sample data file, *\$ethmult* and *\$mltcars* are multiple category sets.

Category Label Source

For multiple dichotomies, you can control how sets are labeled.

- **Variable labels.** Uses the defined variable labels (or variable names for variables without defined variable labels) as the set category labels. For example, if all of the variables in the set have the same value label (or no defined value labels) for the counted value (for example, *Yes*), then you should use the variable labels as the set category labels.
- **Labels of counted values.** Uses the defined value labels of the counted values as set category labels. Select this option only if all variables have a defined value label for the counted value and the value label for the counted value is different for each variable.
- **Use variable label as set label.** If you select **Label of counted values**, you can also use the variable label for the first variable in the set with a defined variable label as the set label. If none of the variables in the set have defined variable labels, the name of the first variable in the set is used as the set label.

Copy Data Properties

Copying Data Properties

The Copy Data Properties Wizard provides the ability to use an external IBM SPSS Statistics data file as a template for defining file and variable properties in the active dataset. You can also use variables in the active dataset as templates for other variables in the active dataset. You can:

- Copy selected file properties from an external data file or open dataset to the active dataset. File properties include documents, file labels, multiple response sets, variable sets, and weighting.
- Copy selected variable properties from an external data file or open dataset to matching variables in the active dataset. Variable properties include value labels, missing values, level of measurement, variable labels, print and write formats, alignment, and column width (in the Data Editor).
- Copy selected variable properties from one variable in either an external data file, open dataset, or the active dataset to many variables in the active dataset.
- Create new variables in the active dataset based on selected variables in an external data file or open dataset.

When copying data properties, the following general rules apply:

- If you use an external data file as the source data file, it must be a data file in IBM SPSS Statistics format.
- If you use the active dataset as the source data file, it must contain at least one variable. You cannot use a completely blank active dataset as the source data file.
- Undefined (empty) properties in the source dataset do not overwrite defined properties in the active dataset.
- Variable properties are copied from the source variable only to target variables of a matching type--string (alphanumeric) or numeric (including numeric, date, and currency).

Note: Copy Data Properties replaces Apply Data Dictionary, formerly available on the File menu.

To Copy Data Properties

1. From the menus in the Data Editor window choose:
Data > Copy Data Properties...
2. Select the data file with the file and/or variable properties that you want to copy. This can be a currently open dataset, an external IBM SPSS Statistics data file, or the active dataset.
3. Follow the step-by-step instructions in the Copy Data Properties Wizard.

Selecting Source and Target Variables

In this step, you can specify the source variables containing the variable properties that you want to copy and the target variables that will receive those variable properties.

Apply properties from selected source dataset variables to matching active dataset variables. Variable properties are copied from one or more selected source variables to matching variables in the active dataset. Variables "match" if both the variable name and type (string or numeric) are the same. For string variables, the defined length must also be the same. By default, only matching variables are displayed in the two variable lists.

- **Create matching variables in the active dataset if they do not already exist.** This updates the source list to display all variables in the source data file. If you select source variables that do not exist in the active dataset (based on variable name), new variables will be created in the active dataset with the variable names and properties from the source data file.

If the active dataset contains no variables (a blank, new dataset), all variables in the source data file are displayed and new variables based on the selected source variables are automatically created in the active dataset.

Apply properties from a single source variable to selected active dataset variables of the same type.

Variable properties from a single selected variable in the source list can be applied to one or more selected variables in the active dataset list. Only variables of the same type (numeric or string) as the selected variable in the source list are displayed in the active dataset list. For string variables, only strings of the same defined length as the source variable are displayed. This option is not available if the active dataset contains no variables.

Note: You cannot create new variables in the active dataset with this option.

Apply dataset properties only--no variable selection. Only file properties (for example, documents, file label, weight) will be applied to the active dataset. No variable properties will be applied. This option is not available if the active dataset is also the source data file.

Choosing Variable Properties to Copy

You can copy selected variable properties from the source variables to the target variables. Undefined (empty) properties in the source variables do not overwrite defined properties in the target variables.

Value Labels. Value labels are descriptive labels associated with data values. Value labels are often used when numeric data values are used to represent non-numeric categories (for example, codes of 1 and 2 for *Male* and *Female*). You can replace or merge value labels in the target variables.

- **Replace** deletes any defined value labels for the target variable and replaces them with the defined value labels from the source variable.
- **Merge** merges the defined value labels from the source variable with any existing defined value label for the target variable. If the same value has a defined value label in both the source and target variables, the value label in the target variable is unchanged.

Custom Attributes. User-defined custom variable attributes. See the topic “Custom Variable Attributes” on page 55 for more information.

- **Replace** deletes any custom attributes for the target variable and replaces them with the defined attributes from the source variable.
- **Merge** merges the defined attributes from the source variable with any existing defined attributes for the target variable.

Missing Values. Missing values are values identified as representing missing data (for example, 98 for *Do not know* and 99 for *Not applicable*). Typically, these values also have defined value labels that describe what the missing value codes stand for. Any existing defined missing values for the target variable are deleted and replaced with the defined missing values from the source variable.

Variable Label. Descriptive variable labels can contain spaces and reserved characters not allowed in variable names. If you're copying variable properties from a single source variable to multiple target variables, you might want to think twice before selecting this option.

Measurement Level. The measurement level can be nominal, ordinal, or scale.

Role. Some dialogs support the ability to pre-select variables for analysis based on defined roles. See the topic “Roles” on page 54 for more information.

Formats. For numeric variables, this controls numeric type (such as numeric, date, or currency), width (total number of displayed characters, including leading and trailing characters and decimal indicator), and number of decimal places displayed. This option is ignored for string variables.

Alignment. This affects only alignment (left, right, center) in Data View in the Data Editor.

Data Editor Column Width. This affects only column width in Data View in the Data Editor.

Copying Dataset (File) Properties

You can apply selected, global dataset properties from the source data file to the active dataset. (This is not available if the active dataset is the source data file.)

Multiple Response Sets. Applies multiple response set definitions from the source data file to the active dataset.

- Multiple response sets that contain no variables in the active dataset are ignored unless those variables will be created based on specifications in step 2 (Selecting Source and Target Variables) in the Copy Data Properties Wizard.
- **Replace** deletes all multiple response sets in the active dataset and replaces them with the multiple response sets from the source data file.
- **Merge** adds multiple response sets from the source data file to the collection of multiple response sets in the active dataset. If a set with the same name exists in both files, the existing set in the active dataset is unchanged.

Variable Sets. Variable sets are used to control the list of variables that are displayed in dialog boxes. Variable sets are defined by selecting Define Variable Sets from the Utilities menu.

- Sets in the source data file that contain variables that do not exist in the active dataset are ignored unless those variables will be created based on specifications in step 2 (Selecting Source and Target Variables) in the Copy Data Properties Wizard.
- **Replace** deletes any existing variable sets in the active dataset, replacing them with variable sets from the source data file.
- **Merge** adds variable sets from the source data file to the collection of variable sets in the active dataset. If a set with the same name exists in both files, the existing set in the active dataset is unchanged.

Documents. Notes appended to the data file via the DOCUMENT command.

- **Replace** deletes any existing documents in the active dataset, replacing them with the documents from the source data file.
- **Merge** combines documents from the source and active dataset. Unique documents in the source file that do not exist in the active dataset are added to the active dataset. All documents are then sorted by date.

Custom Attributes. Custom data file attributes, typically created with the DATAFILE ATTRIBUTE command in command syntax.

- **Replace** deletes any existing custom data file attributes in the active dataset, replacing them with the data file attributes from the source data file.
- **Merge** combines data file attributes from the source and active dataset. Unique attribute names in the source file that do not exist in the active dataset are added to the active dataset. If the same attribute name exists in both data files, the named attribute in the active dataset is unchanged.

Weight Specification. Weights cases by the current weight variable in the source data file if there is a matching variable in the active dataset. This overrides any weighting currently in effect in the active dataset.

File Label. Descriptive label applied to a data file with the FILE LABEL command.

Results

The last step in the Copy Data Properties Wizard provides information on the number of variables for which variable properties will be copied from the source data file, the number of new variables that will be created, and the number of dataset (file) properties that will be copied.

You can also choose to paste the generated command syntax into a syntax window and save the syntax for later use.

Identifying Duplicate Cases

"Duplicate" cases may occur in your data for many reasons, including:

- Data entry errors in which the same case is accidentally entered more than once.
- Multiple cases share a common primary ID value but have different secondary ID values, such as family members who all live in the same house.
- Multiple cases represent the same case but with different values for variables other than those that identify the case, such as multiple purchases made by the same person or company for different products or at different times.

Identify Duplicate Cases allows you to define *duplicate* almost any way that you want and provides some control over the automatic determination of primary versus duplicate cases.

To Identify and Flag Duplicate Cases

1. From the menus choose:

Data > Identify Duplicate Cases...

2. Select one or more variables that identify matching cases.
3. Select one or more of the options in the Variables to Create group.

Optionally, you can:

4. Select one or more variables to sort cases within groups defined by the selected matching cases variables. The sort order defined by these variables determines the "first" and "last" case in each group. Otherwise, the original file order is used.
5. Automatically filter duplicate cases so that they won't be included in reports, charts, or calculations of statistics.

Define matching cases by. Cases are considered duplicates if their values match for *all* selected variables. If you want to identify only cases that are a 100% match in all respects, select all of the variables.

Sort within matching groups by. Cases are automatically sorted by the variables that define matching cases. You can select additional sorting variables that will determine the sequential order of cases in each matching group.

- For each sort variable, you can sort in ascending or descending order.
- If you select multiple sort variables, cases are sorted by each variable within categories of the preceding variable in the list. For example, if you select *date* as the first sorting variable and *amount* as the second sorting variable, cases will be sorted by amount within each date.
- Use the up and down arrow buttons to the right of the list to change the sort order of the variables.
- The sort order determines the "first" and "last" case within each matching group, which determines the value of the optional primary indicator variable. For example, if you want to filter out all but the most recent case in each matching group, you could sort cases within the group in ascending order of a date variable, which would make the most recent date the last date in the group.

Indicator of primary cases. Creates a variable with a value of 1 for all unique cases and the case identified as the primary case in each group of matching cases and a value of 0 for the nonprimary duplicates in each group.

- The primary case can be either the last or first case in each matching group, as determined by the sort order within the matching group. If you don't specify any sort variables, the original file order determines the order of cases within each group.
- You can use the indicator variable as a filter variable to exclude non-primary duplicates from reports and analyses without deleting those cases from the data file.

Sequential count of matching cases in each group. Creates a variable with a sequential value from 1 to *n* for cases in each matching group. The sequence is based on the current order of cases in each group, which is either the original file order or the order determined by any specified sort variables.

Move matching cases to the top. Sorts the data file so that all groups of matching cases are at the top of the data file, making it easy to visually inspect the matching cases in the Data Editor.

Display frequencies for created variables. Frequency tables containing counts for each value of the created variables. For example, for the primary indicator variable, the table would show the number of cases with a value 0 for that variable, which indicates the number of duplicates, and the number of cases with a value of 1 for that variable, which indicates the number of unique and primary cases.

Missing values

For numeric variables, the system-missing value is treated like any other value—cases with the system-missing value for an identifier variable are treated as having matching values for that variable. For string variables, cases with no value for an identifier variable are treated as having matching values for that variable.

Filtered cases

Filter conditions are ignored. Filtered cases are included in the evaluation of duplicate cases. If you want to exclude cases, define selection rules with **Data > Select Cases** and select **Delete unselected cases**.

Visual Binning

Visual Binning is designed to assist you in the process of creating new variables based on grouping contiguous values of existing variables into a limited number of distinct categories. You can use Visual Binning to:

- Create categorical variables from continuous scale variables. For example, you could use a scale income variable to create a new categorical variable that contains income ranges.
- Collapse a large number of ordinal categories into a smaller set of categories. For example, you could collapse a rating scale of nine down to three categories representing low, medium, and high.

In the first step, you:

1. Select the numeric scale and/or ordinal variables for which you want to create new categorical (binned) variables.

Optionally, you can limit the number of cases to scan. For data files with a large number of cases, limiting the number of cases scanned can save time, but you should avoid this if possible because it will affect the distribution of values used in subsequent calculations in Visual Binning.

Note: String variables and nominal numeric variables are not displayed in the source variable list. Visual Binning requires numeric variables, measured on either a scale or ordinal level, since it assumes that the data values represent some logical order that can be used to group values in a meaningful fashion. You can change the defined measurement level of a variable in Variable View in the Data Editor. See the topic “Variable measurement level” on page 51 for more information.

To Bin Variables

1. From the menus in the Data Editor window choose:
Transform > Visual Binning...
2. Select the numeric scale and/or ordinal variables for which you want to create new categorical (binned) variables.
3. Select a variable in the Scanned Variable List.

4. Enter a name for the new binned variable. Variable names must be unique and must follow variable naming rules. See the topic “Variable names” on page 50 for more information.
5. Define the binning criteria for the new variable. See the topic “Binning Variables” for more information.
6. Click **OK**.

Binning Variables

The Visual Binning main dialog box provides the following information for the scanned variables:

Scanned Variable List. Displays the variables you selected in the initial dialog box. You can sort the list by measurement level (scale or ordinal) or by variable label or name by clicking on the column headings.

Cases Scanned. Indicates the number of cases scanned. All scanned cases without user-missing or system-missing values for the selected variable are used to generate the distribution of values used in calculations in Visual Binning, including the histogram displayed in the main dialog box and cutpoints based on percentiles or standard deviation units.

Missing Values. Indicates the number of scanned cases with user-missing or system-missing values. Missing values are not included in any of the binned categories. See the topic “User-Missing Values in Visual Binning” on page 81 for more information.

Current Variable. The name and variable label (if any) for the currently selected variable that will be used as the basis for the new, binned variable.

Binned Variable. Name and optional variable label for the new, binned variable.

- **Name.** You must enter a name for the new variable. Variable names must be unique and must follow variable naming rules. See the topic “Variable names” on page 50 for more information.
- **Label.** You can enter a descriptive variable label up to 255 characters long. The default variable label is the variable label (if any) or variable name of the source variable with (*Binned*) appended to the end of the label.

Minimum and Maximum. Minimum and maximum values for the currently selected variable, based on the scanned cases and not including values defined as user-missing.

Nonmissing Values. The histogram displays the distribution of nonmissing values for the currently selected variable, based on the scanned cases.

- After you define bins for the new variable, vertical lines on the histogram are displayed to indicate the cutpoints that define bins.
- You can click and drag the cutpoint lines to different locations on the histogram, changing the bin ranges.
- You can remove bins by dragging cutpoint lines off the histogram.

Note: The histogram (displaying nonmissing values), the minimum, and the maximum are based on the scanned values. If you do not include all cases in the scan, the true distribution may not be accurately reflected, particularly if the data file has been sorted by the selected variable. If you scan zero cases, no information about the distribution of values is available.

Grid. Displays the values that define the upper endpoints of each bin and optional value labels for each bin.

- **Value.** The values that define the upper endpoints of each bin. You can enter values or use **Make Cutpoints** to automatically create bins based on selected criteria. By default, a cutpoint with a value of *HIGH* is automatically included. This bin will contain any nonmissing values above the other

cutpoints. The bin defined by the lowest cutpoint will include all nonmissing values lower than or equal to that value (or simply lower than that value, depending on how you define upper endpoints).

- **Label.** Optional, descriptive labels for the values of the new, binned variable. Since the values of the new variable will simply be sequential integers from 1 to n , labels that describe what the values represent can be very useful. You can enter labels or use **Make Labels** to automatically create value labels.

To Delete a Bin from the Grid

1. Right-click either the *Value* or *Label* cell for the bin.
2. From the pop-up menu, select **Delete Row**.

Note: If you delete the *HIGH* bin, any cases with values higher than the last specified cutpoint value will be assigned the system-missing value for the new variable.

To Delete All Labels or Delete All Defined Bins

1. Right-click anywhere in the grid.
2. From the pop-up menu select either **Delete All Labels** or **Delete All Cutpoints**.

Upper Endpoints. Controls treatment of upper endpoint values entered in the *Value* column of the grid.

- **Included (\leq).** Cases with the value specified in the *Value* cell are included in the binned category. For example, if you specify values of 25, 50, and 75, cases with a value of exactly 25 will go in the first bin, since this will include all cases with values less than or equal to 25.
- **Excluded (<).** Cases with the value specified in the *Value* cell are not included in the binned category. Instead, they are included in the next bin. For example, if you specify values of 25, 50, and 75, cases with a value of exactly 25 will go in the second bin rather than the first, since the first bin will contain only cases with values less than 25.

Make Cutpoints. Generates binned categories automatically for equal width intervals, intervals with the same number of cases, or intervals based on standard deviations. This is not available if you scanned zero cases. See the topic “Automatically Generating Binned Categories” for more information.

Make Labels. Generates descriptive labels for the sequential integer values of the new, binned variable, based on the values in the grid and the specified treatment of upper endpoints (included or excluded).

Reverse scale. By default, values of the new, binned variable are ascending sequential integers from 1 to n . Reversing the scale makes the values descending sequential integers from n to 1.

Copy Bins. You can copy the binning specifications from another variable to the currently selected variable or from the selected variable to multiple other variables. See the topic “Copying Binned Categories” on page 80 for more information.

Automatically Generating Binned Categories

The Make Cutpoints dialog box allows you to auto-generate binned categories based on selected criteria.

To Use the Make Cutpoints Dialog Box

1. Select (click) a variable in the Scanned Variable List.
2. Click **Make Cutpoints**.
3. Select the criteria for generating cutpoints that will define the binned categories.
4. Click **Apply**.

Note: The Make Cutpoints dialog box is not available if you scanned zero cases.

Equal Width Intervals. Generates binned categories of equal width (for example, 1–10, 11–20, and 21–30) based on any two of the following three criteria:

- **First Cutpoint Location.** The value that defines the upper end of the lowest binned category (for example, a value of 10 indicates a range that includes all values up to 10).
- **Number of Cutpoints.** The number of binned categories is the number of cutpoints *plus one*. For example, 9 cutpoints generate 10 binned categories.
- **Width.** The width of each interval. For example, a value of 10 would bin *age in years* into 10-year intervals.

Equal Percentiles Based on Scanned Cases. Generates binned categories with an equal number of cases in each bin (using the **aempirical algorithm** for percentiles), based on either of the following criteria:

- **Number of Cutpoints.** The number of binned categories is the number of cutpoints *plus one*. For example, three cutpoints generate four percentile bins (quartiles), each containing 25% of the cases.
- **Width (%).** Width of each interval, expressed as a percentage of the total number of cases. For example, a value of 33.3 would produce three binned categories (two cutpoints), each containing 33.3% of the cases.

If the source variable contains a relatively small number of distinct values or a large number of cases with the same value, you may get fewer bins than requested. If there are multiple identical values at a cutpoint, they will all go into the same interval; so the actual percentages may not always be exactly equal.

Cutpoints at Mean and Selected Standard Deviations Based on Scanned Cases. Generates binned categories based on the values of the mean and standard deviation of the distribution of the variable.

- If you don't select any of the standard deviation intervals, two binned categories will be created, with the mean as the cutpoint dividing the bins.
- You can select any combination of standard deviation intervals based on one, two, and/or three standard deviations. For example, selecting all three would result in eight binned categories--six bins in one standard deviation intervals and two bins for cases more than three standard deviations above and below the mean.

In a normal distribution, 68% of the cases fall within one standard deviation of the mean; 95%, within two standard deviations; and 99%, within three standard deviations. Creating binned categories based on standard deviations may result in some defined bins outside of the actual data range and even outside of the range of possible data values (for example, a negative salary range).

Note: Calculations of percentiles and standard deviations are based on the scanned cases. If you limit the number of cases scanned, the resulting bins may not contain the proportion of cases that you wanted in those bins, particularly if the data file is sorted by the source variable. For example, if you limit the scan to the first 100 cases of a data file with 1000 cases and the data file is sorted in ascending order of age of respondent, instead of four percentile age bins each containing 25% of the cases, you may find that the first three bins each contain only about 3.3% of the cases, and the last bin contains 90% of the cases.

Copying Binned Categories

When creating binned categories for one or more variables, you can copy the binning specifications from another variable to the currently selected variable or from the selected variable to multiple other variables.

To Copy Binning Specifications

1. Define binned categories for at least one variable--but do *not* click **OK** or **Paste**.
2. Select (click) a variable in the Scanned Variable List for which you have defined binned categories.
3. Click **To Other Variables**.
4. Select the variables for which you want to create new variables with the same binned categories.

5. Click **Copy**.
- or
6. Select (click) a variable in the Scanned Variable List to which you want to copy defined binned categories.
7. Click **From Another Variable**.
8. Select the variable with the defined binned categories that you want to copy.
9. Click **Copy**.

If you have specified value labels for the variable from which you are copying the binning specifications, those are also copied.

Note: Once you click **OK** in the Visual Binning main dialog box to create new binned variables (or close the dialog box in any other way), you cannot use Visual Binning to copy those binned categories to other variables.

User-Missing Values in Visual Binning

Values defined as user-missing (values identified as codes for missing data) for the source variable are not included in the binned categories for the new variable. User-missing values for the source variables are copied as user-missing values for the new variable, and any defined value labels for missing value codes are also copied.

If a missing value code conflicts with one of the binned category values for the new variable, the missing value code for the new variable is recoded to a nonconflicting value by adding 100 to the highest binned category value. For example, if a value of 1 is defined as user-missing for the source variable and the new variable will have six binned categories, any cases with a value of 1 for the source variable will have a value of 106 for the new variable, and 106 will be defined as user-missing. If the user-missing value for the source variable had a defined value label, that label will be retained as the value label for the recoded value of the new variable.

Note: If the source variable has a defined range of user-missing values of the form $LO-n$, where n is a positive number, the corresponding user-missing values for the new variable will be negative numbers.

Chapter 8. Data Transformations

Data Transformations

In an ideal situation, your raw data are perfectly suitable for the type of analysis you want to perform, and any relationships between variables are either conveniently linear or neatly orthogonal. Unfortunately, this is rarely the case. Preliminary analysis may reveal inconvenient coding schemes or coding errors, or data transformations may be required in order to expose the true relationship between variables.

You can perform data transformations ranging from simple tasks, such as collapsing categories for analysis, to more advanced tasks, such as creating new variables based on complex equations and conditional statements.

Computing Variables

Use the Compute dialog box to compute values for a variable based on numeric transformations of other variables.

- You can compute values for numeric or string (alphanumeric) variables.
- You can create new variables or replace the values of existing variables. For new variables, you can also specify the variable type and label.
- You can compute values selectively for subsets of data based on logical conditions.
- You can use a large variety of built-in functions, including arithmetic functions, statistical functions, distribution functions, and string functions.

To Compute Variables

1. From the menus choose:
Transform > Compute Variable...
2. Type the name of a single target variable. It can be an existing variable or a new variable to be added to the active dataset.
3. To build an expression, either paste components into the Expression field or type directly in the Expression field.
 - You can paste functions or commonly used system variables by selecting a group from the Function group list and double-clicking the function or variable in the Functions and Special Variables list (or select the function or variable and click the arrow adjacent to the Function group list). Fill in any parameters indicated by question marks (only applies to functions). The function group labeled **All** provides a listing of all available functions and system variables. A brief description of the currently selected function or variable is displayed in a reserved area in the dialog box.
 - String constants must be enclosed in quotation marks or apostrophes.
 - If values contain decimals, a period (.) must be used as the decimal indicator.
 - For new string variables, you must also select **Type & Label** to specify the data type.

Compute Variable: If Cases

The If Cases dialog box allows you to apply data transformations to selected subsets of cases, using conditional expressions. A conditional expression returns a value of *true*, *false*, or *missing* for each case.

- If the result of a conditional expression is *true*, the case is included in the selected subset.
- If the result of a conditional expression is *false* or *missing*, the case is not included in the selected subset.

- Most conditional expressions use one or more of the six relational operators (<, >, <=, >=, =, and ~=) on the calculator pad.
- Conditional expressions can include variable names, constants, arithmetic operators, numeric (and other) functions, logical variables, and relational operators.

Compute Variable: Type and Label

By default, new computed variables are numeric. To compute a new string variable, you must specify the data type and width.

Label. Optional, descriptive variable label up to 255 bytes long. You can enter a label or use the first 110 characters of the compute expression as the label.

Type. Computed variables can be numeric or string (alphanumeric). String variables cannot be used in calculations.

Functions

Many types of functions are supported, including:

- Arithmetic functions
- Statistical functions
- String functions
- Date and time functions
- Distribution functions
- Random variable functions
- Missing value functions
- Scoring functions

For more information and a detailed description of each function, type **functions** on the Index tab of the Help system.

Missing Values in Functions

Functions and simple arithmetic expressions treat missing values in different ways. In the expression:

`(var1+var2+var3)/3`

the result is missing if a case has a missing value for any of the three variables.

In the expression:

`MEAN(var1, var2, var3)`

the result is missing only if the case has missing values for all three variables.

For statistical functions, you can specify the minimum number of arguments that must have nonmissing values. To do so, type a period and the minimum number after the function name, as in:

`MEAN.2(var1, var2, var3)`

Random Number Generators

The Random Number Generators dialog box allows you to select the random number generator and set the starting sequence value so you can reproduce a sequence of random numbers.

Active Generator. Two different random number generators are available:

- *Version 12 Compatible.* The random number generator used in version 12 and previous releases. If you need to reproduce randomized results generated in previous releases based on a specified seed value, use this random number generator.
- *Mersenne Twister.* A newer random number generator that is more reliable for simulation purposes. If reproducing randomized results from version 12 or earlier is not an issue, use this random number generator.

Active Generator Initialization. The random number seed changes each time a random number is generated for use in transformations (such as random distribution functions), random sampling, or case weighting. To replicate a sequence of random numbers, set the initialization starting point value prior to each analysis that uses the random numbers. The value must be a positive integer.

Some procedures, such as Linear Models, have internal random number generators.

To select the random number generator and/or set the initialization value:

1. From the menus choose:

Transform > Random Number Generators

Count Occurrences of Values within Cases

This dialog box creates a variable that counts the occurrences of the same value(s) in a list of variables for each case. For example, a survey might contain a list of magazines with *yes/no* check boxes to indicate which magazines each respondent reads. You could count the number of *yes* responses for each respondent to create a new variable that contains the total number of magazines read.

To Count Occurrences of Values within Cases

1. From the menus choose:
Transform > Count Values within Cases...
2. Enter a target variable name.
3. Select two or more variables of the same type (numeric or string).
4. Click **Define Values** and specify which value or values should be counted.

Optionally, you can define a subset of cases for which to count occurrences of values.

Count Values within Cases: Values to Count

The value of the target variable (on the main dialog box) is incremented by 1 each time one of the selected variables matches a specification in the Values to Count list here. If a case matches several specifications for any variable, the target variable is incremented several times for that variable.

Value specifications can include individual values, missing or system-missing values, and ranges. Ranges include their endpoints and any user-missing values that fall within the range.

Count Occurrences: If Cases

The If Cases dialog box allows you to count occurrences of values for a selected subset of cases, using conditional expressions. A conditional expression returns a value of *true*, *false*, or *missing* for each case.

Shift Values

Shift Values creates new variables that contain the values of existing variables from preceding or subsequent cases.

Name. Name for the new variable. This must be a name that does not already exist in the active dataset.

Get value from earlier case (lag). Get the value from a previous case in the active dataset. For example, with the default number of cases value of 1, each case for the new variable has the value of the original variable from the case that immediately precedes it.

Get value from following case (lead). Get the value from a subsequent case in the active dataset. For example, with the default number of cases value of 1, each case for the new variable has the value of the original variable from the next case.

Number of cases to shift. Get the value from the n th preceding or subsequent case, where n is the value specified. The value must be a non-negative integer.

- If split file processing is on, the scope of the shift is limited to each split group. A shift value cannot be obtained from a case in a preceding or subsequent split group.
- Filter status is ignored.
- The value of the result variable is set to system-missing for the first or last n cases in the dataset or split group, where n is the value specified for **Number of cases to shift**. For example, using the Lag method with a value of 1 would set the result variable to system-missing for the first case in the dataset (or first case in each split group).
- User-missing values are preserved.
- Dictionary information from the original variable, including defined value labels and user-missing value assignments, is applied to the new variable. (Note: Custom variable attributes are not included.)
- A variable label is automatically generated for the new variable that describes the shift operation that created the variable.

To Create a New Variable with Shifted Values

1. From the menus, choose:
Transform > Shift Values
2. Select the variable to use as the source of values for the new variable.
3. Enter a name for the new variable.
4. Select the shift method (lag or lead) and the number of cases to shift.
5. Click **Change**.
6. Repeat for each new variable you want to create.

Recoding Values

You can modify data values by recoding them. This is particularly useful for collapsing or combining categories. You can recode the values within existing variables, or you can create new variables based on the recoded values of existing variables.

Recode into Same Variables

The Recode into Same Variables dialog box allows you to reassign the values of existing variables or collapse ranges of existing values into new values. For example, you could collapse salaries into salary range categories.

You can recode numeric and string variables. If you select multiple variables, they must all be the same type. You cannot recode numeric and string variables together.

To Recode Values of a Variable

1. From the menus choose:
Transform > Recode into Same Variables...
2. Select the variables you want to recode. If you select multiple variables, they must be the same type (numeric or string).
3. Click **Old and New Values** and specify how to recode values.

Optionally, you can define a subset of cases to recode. The If Cases dialog box for doing this is the same as the one described for Count Occurrences.

Recode into Same Variables: Old and New Values

You can define values to recode in this dialog box. All value specifications must be the same data type (numeric or string) as the variables selected in the main dialog box.

Old Value. The value(s) to be recoded. You can recode single values, ranges of values, and missing values. System-missing values and ranges cannot be selected for string variables because neither concept applies to string variables. Ranges include their endpoints and any user-missing values that fall within the range.

- *Value.* Individual old value to be recoded into a new value. The value must be the same data type (numeric or string) as the variable(s) being recoded.
- *System-missing.* Values assigned by the program when values in your data are undefined according to the format type you have specified, when a numeric field is blank, or when a value resulting from a transformation command is undefined. Numeric system-missing values are displayed as periods. String variables cannot have system-missing values, since any character is legal in a string variable.
- *System- or user-missing.* Observations with values that either have been defined as user-missing values or are unknown and have been assigned the system-missing value, which is indicated with a period (.).
- *Range.* Inclusive range of values. Not available for string variables. Any user-missing values within the range are included.
- *All other values.* Any remaining values not included in one of the specifications on the Old-New list. This appears as ELSE on the Old-New list.

New Value. The single value into which each old value or range of values is recoded. You can enter a value or assign the system-missing value.

- *Value.* Value into which one or more old values will be recoded. The value must be the same data type (numeric or string) as the old value.
- *System-missing.* Recodes specified old values into the system-missing value. The system-missing value is not used in calculations, and cases with the system-missing value are excluded from many procedures. Not available for string variables.

Old->New. The list of specifications that will be used to recode the variable(s). You can add, change, and remove specifications from the list. The list is automatically sorted, based on the old value specification, using the following order: single values, missing values, ranges, and all other values. If you change a recode specification on the list, the procedure automatically re-sorts the list, if necessary, to maintain this order.

Recode into Different Variables

The Recode into Different Variables dialog box allows you to reassign the values of existing variables or collapse ranges of existing values into new values for a new variable. For example, you could collapse salaries into a new variable containing salary-range categories.

- You can recode numeric and string variables.
- You can recode numeric variables into string variables and vice versa.
- If you select multiple variables, they must all be the same type. You cannot recode numeric and string variables together.

To Recode Values of a Variable into a New Variable

1. From the menus choose:
Transform > Recode into Different Variables...
2. Select the variables you want to recode. If you select multiple variables, they must be the same type (numeric or string).
3. Enter an output (new) variable name for each new variable and click **Change**.
4. Click **Old and New Values** and specify how to recode values.

Optionally, you can define a subset of cases to recode. The If Cases dialog box for doing this is the same as the one described for Count Occurrences.

Recode into Different Variables: Old and New Values

You can define values to recode in this dialog box.

Old Value. The value(s) to be recoded. You can recode single values, ranges of values, and missing values. System-missing values and ranges cannot be selected for string variables because neither concept applies to string variables. Old values must be the same data type (numeric or string) as the original variable. Ranges include their endpoints and any user-missing values that fall within the range.

- *Value.* Individual old value to be recoded into a new value. The value must be the same data type (numeric or string) as the variable(s) being recoded.
- *System-missing.* Values assigned by the program when values in your data are undefined according to the format type you have specified, when a numeric field is blank, or when a value resulting from a transformation command is undefined. Numeric system-missing values are displayed as periods. String variables cannot have system-missing values, since any character is legal in a string variable.
- *System- or user-missing.* Observations with values that either have been defined as user-missing values or are unknown and have been assigned the system-missing value, which is indicated with a period (.).
- *Range.* Inclusive range of values. Not available for string variables. Any user-missing values within the range are included.
- *All other values.* Any remaining values not included in one of the specifications on the Old-New list. This appears as ELSE on the Old-New list.

New Value. The single value into which each old value or range of values is recoded. New values can be numeric or string.

- *Value.* Value into which one or more old values will be recoded. The value must be the same data type (numeric or string) as the old value.
- *System-missing.* Recodes specified old values into the system-missing value. The system-missing value is not used in calculations, and cases with the system-missing value are excluded from many procedures. Not available for string variables.
- *Copy old values.* Retains the old value. If some values don't require recoding, use this to include the old values. Any old values that are not specified are not included in the new variable, and cases with those values will be assigned the system-missing value for the new variable.

Output variables are strings. Defines the new, recoded variable as a string (alphanumeric) variable. The old variable can be numeric or string.

Convert numeric strings to numbers. Converts string values containing numbers to numeric values. Strings containing anything other than numbers and an optional sign (+ or -) are assigned the system-missing value.

Old->New. The list of specifications that will be used to recode the variable(s). You can add, change, and remove specifications from the list. The list is automatically sorted, based on the old value specification, using the following order: single values, missing values, ranges, and all other values. If you change a recode specification on the list, the procedure automatically re-sorts the list, if necessary, to maintain this order.

Automatic Recode

The Automatic Recode dialog box allows you to convert string and numeric values into consecutive integers. When category codes are not sequential, the resulting empty cells reduce performance and increase memory requirements for many procedures. Additionally, some procedures cannot use string variables, and some require consecutive integer values for factor levels.

- The new variable(s) created by Automatic Recode retain any defined variable and value labels from the old variable. For any values without a defined value label, the original value is used as the label for the recoded value. A table displays the old and new values and value labels.
- String values are recoded in alphabetical order, with uppercase letters preceding their lowercase counterparts.
- Missing values are recoded into missing values higher than any nonmissing values, with their order preserved. For example, if the original variable has 10 nonmissing values, the lowest missing value would be recoded to 11, and the value 11 would be a missing value for the new variable.

Use the same recoding scheme for all variables. This option allows you to apply a single autorecoding scheme to all the selected variables, yielding a consistent coding scheme for all the new variables.

If you select this option, the following rules and limitations apply:

- All variables must be of the same type (numeric or string).
- All observed values for all selected variables are used to create a sorted order of values to recode into sequential integers.
- User-missing values for the new variables are based on the first variable in the list with defined user-missing values. All other values from other original variables, except for system-missing, are treated as valid.

Treat blank string values as user-missing. For string variables, blank or null values are not treated as *system-missing*. This option will autorecode blank strings into a *user-missing* value higher than the highest nonmissing value.

Templates

You can save the autorecoding scheme in a template file and then apply it to other variables and other data files.

For example, you may have a large number of alphanumeric product codes that you autorecode into integers every month, but some months new product codes are added that change the original autorecoding scheme. If you save the original scheme in a template and then apply it to the new data that contain the new set of codes, any new codes encountered in the data are autorecoded into values higher than the last value in the template, preserving the original autorecode scheme of the original product codes.

Save template as. Saves the autorecode scheme for the selected variables in an external template file.

- The template contains information that maps the original nonmissing values to the recoded values.
- Only information for nonmissing values is saved in the template. User-missing value information is not retained.
- If you have selected multiple variables for recoding but you have not selected to use the same autorecoding scheme for all variables or you are not applying an existing template as part of the autorecoding, the template will be based on the first variable in the list.
- If you have selected multiple variables for recoding and you have also selected **Use the same recoding scheme for all variables** and/or you have selected **Apply template**, then the template will contain the combined autorecoding scheme for all variables.

Apply template from. Applies a previously saved autorecode template to variables selected for recoding, appending any additional values found in the variables to the end of the scheme and preserving the relationship between the original and autorecoded values stored in the saved scheme.

- All variables selected for recoding must be the same type (numeric or string), and that type must match the type defined in the template.
- Templates do not contain any information on user-missing values. User-missing values for the target variables are based on the first variable in the original variable list with defined user-missing values. All other values from other original variables, except for system-missing, are treated as valid.
- Value mappings from the template are applied first. All remaining values are recoded into values higher than the last value in the template, with user-missing values (based on the first variable in the list with defined user-missing values) recoded into values higher than the last valid value.
- If you have selected multiple variables for autorecoding, the template is applied first, followed by a common, combined autorecoding for all additional values found in the selected variables, resulting in a single, common autorecoding scheme for all selected variables.

To Recode String or Numeric Values into Consecutive Integers

1. From the menus choose:
Transform > Automatic Recode...
2. Select one or more variables to recode.
3. For each selected variable, enter a name for the new variable and click **New Name**.

Rank Cases

The Rank Cases dialog box allows you to create new variables containing ranks, normal and Savage scores, and percentile values for numeric variables.

New variable names and descriptive variable labels are automatically generated, based on the original variable name and the selected measure(s). A summary table lists the original variables, the new variables, and the variable labels. (Note: The automatically generated new variable names are limited to a maximum length of 8 bytes.)

Optionally, you can:

- Rank cases in ascending or descending order.
- Organize rankings into subgroups by selecting one or more grouping variables for the By list. Ranks are computed within each group. Groups are defined by the combination of values of the grouping variables. For example, if you select *gender* and *minority* as grouping variables, ranks are computed for each combination of *gender* and *minority*.

To Rank Cases

1. From the menus choose:
Transform > Rank Cases...

- Select one or more variables to rank. You can rank only numeric variables.

Optionally, you can rank cases in ascending or descending order and organize ranks into subgroups.

Rank Cases: Types

You can select multiple ranking methods. A separate ranking variable is created for each method. Ranking methods include simple ranks, Savage scores, fractional ranks, and percentiles. You can also create rankings based on proportion estimates and normal scores.

Rank. Simple rank. The value of the new variable equals its rank.

Savage score. The new variable contains Savage scores based on an exponential distribution.

Fractional rank. The value of the new variable equals rank divided by the sum of the weights of the nonmissing cases.

Fractional rank as percent. Each rank is divided by the number of cases with valid values and multiplied by 100.

Sum of case weights. The value of the new variable equals the sum of case weights. The new variable is a constant for all cases in the same group.

Ntiles. Ranks are based on percentile groups, with each group containing approximately the same number of cases. For example, 4 Ntiles would assign a rank of 1 to cases below the 25th percentile, 2 to cases between the 25th and 50th percentile, 3 to cases between the 50th and 75th percentile, and 4 to cases above the 75th percentile.

Proportion estimates. Estimates of the cumulative proportion of the distribution corresponding to a particular rank.

Normal scores. The z scores corresponding to the estimated cumulative proportion.

Proportion Estimation Formula. For proportion estimates and normal scores, you can select the proportion estimation formula: **Blom**, **Tukey**, **Rankit**, or **Van der Waerden**.

- Blom.* Creates new ranking variable based on proportion estimates that uses the formula $(r-3/8) / (w+1/4)$, where w is the sum of the case weights and r is the rank.
- Tukey.* Uses the formula $(r-1/3) / (w+1/3)$, where r is the rank and w is the sum of the case weights.
- Rankit.* Uses the formula $(r-1/2) / w$, where w is the number of observations and r is the rank, ranging from 1 to w.
- Van der Waerden.* Van der Waerden's transformation, defined by the formula $r/(w+1)$, where w is the sum of the case weights and r is the rank, ranging from 1 to w.

Rank Cases: Ties

This dialog box controls the method for assigning rankings to cases with the same value on the original variable.

The following table shows how the different methods assign ranks to tied values:

Table 7. Ranking methods and results

Value	Mean	Low	High	Sequential
10	1	1	1	1
15	3	2	4	2
15	3	2	4	2

Table 7. Ranking methods and results (continued)

Value	Mean	Low	High	Sequential
15	3	2	4	2
16	5	5	5	3
20	6	6	6	4

Date and Time Wizard

The Date and Time Wizard simplifies a number of common tasks associated with date and time variables.

To Use the Date and Time Wizard

1. From the menus choose:

Transform > Date and Time Wizard...

2. Select the task you want to accomplish and follow the steps to define the task.
 - **Learn how dates and times are represented.** This choice leads to a screen that provides a brief overview of date/time variables in IBM SPSS Statistics. By clicking on the **Help** button, it also provides a link to more detailed information.
 - **Create a date/time variable from a string containing a date or time.** Use this option to create a date/time variable from a string variable. For example, you have a string variable representing dates in the form mm/dd/yyyy and want to create a date/time variable from this.
 - **Create a date/time variable from variables holding parts of dates or times.** This choice allows you to construct a date/time variable from a set of existing variables. For example, you have a variable that represents the month (as an integer), a second that represents the day of the month, and a third that represents the year. You can combine these three variables into a single date/time variable.
 - **Calculate with dates and times.** Use this option to add or subtract values from date/time variables. For example, you can calculate the duration of a process by subtracting a variable representing the start time of the process from another variable representing the end time of the process.
 - **Extract a part of a date or time variable.** This option allows you to extract part of a date/time variable, such as the day of the month from a date/time variable, which has the form mm/dd/yyyy.
 - **Assign periodicity to a dataset.** This choice takes you to the Define Dates dialog box, used to create date/time variables that consist of a set of sequential dates. This feature is typically used to associate dates with time series data.

Note: Tasks are disabled when the dataset lacks the types of variables required to accomplish the task. For instance, if the dataset contains no string variables, then the task to create a date/time variable from a string does not apply and is disabled.

Dates and Times in IBM SPSS Statistics

Variables that represent dates and times in IBM SPSS Statistics have a variable type of numeric, with display formats that correspond to the specific date/time formats. These variables are generally referred to as date/time variables. Date/time variables that actually represent dates are distinguished from those that represent a time duration that is independent of any date, such as 20 hours, 10 minutes, and 15 seconds. The latter are referred to as duration variables and the former as date or date/time variables. For a complete list of display formats, see "Date and Time" in the "Universals" section of the *Command Syntax Reference*.

Date and date/time variables. Date variables have a format representing a date, such as mm/dd/yyyy. Date/time variables have a format representing a date and time, such as dd-mmm-yyyy hh:mm:ss. Internally, date and date/time variables are stored as the number of seconds from October 14, 1582. Date and date/time variables are sometimes referred to as date-format variables.

- Both two- and four-digit year specifications are recognized. By default, two-digit years represent a range beginning 69 years prior to the current date and ending 30 years after the current date. This range is determined by your Options settings and is configurable (from the Edit menu, choose **Options** and click the **Data** tab).
- Dashes, periods, commas, slashes, or blanks can be used as delimiters in day-month-year formats.
- Months can be represented in digits, Roman numerals, or three-character abbreviations, and they can be fully spelled out. Three-letter abbreviations and fully spelled-out month names must be in English; month names in other languages are not recognized.

Duration variables. Duration variables have a format representing a time duration, such as hh:mm. They are stored internally as seconds without reference to a particular date.

- In time specifications (applies to date/time and duration variables), colons can be used as delimiters between hours, minutes, and seconds. Hours and minutes are required, but seconds are optional. A period is required to separate seconds from fractional seconds. Hours can be of unlimited magnitude, but the maximum value for minutes is 59 and for seconds is 59.999....

Current date and time. The system variable \$TIME holds the current date and time. It represents the number of seconds from October 14, 1582, to the date and time when the transformation command that uses it is executed.

Create a Date/Time Variable from a String

To create a date/time variable from a string variable:

- Select **Create a date/time variable from a string containing a date or time** on the introduction screen of the Date and Time Wizard.

Select String Variable to Convert to Date/Time Variable

- Select the string variable to convert in the Variables list. Note that the list displays only string variables.
- Select the pattern from the Patterns list that matches how dates are represented by the string variable. The Sample Values list displays actual values of the selected variable in the data file. Values of the string variable that do not fit the selected pattern result in a value of system-missing for the new variable.

Specify Result of Converting String Variable to Date/Time Variable

- Enter a name for the Result Variable. This cannot be the name of an existing variable.

Optionally, you can:

- Select a date/time format for the new variable from the Output Format list.
- Assign a descriptive variable label to the new variable.

Create a Date/Time Variable from a Set of Variables

To merge a set of existing variables into a single date/time variable:

- Select **Create a date/time variable from variables holding parts of dates or times** on the introduction screen of the Date and Time Wizard.

Select Variables to Merge into Single Date/Time Variable

- Select the variables that represent the different parts of the date/time.
 - Some combinations of selections are not allowed. For instance, creating a date/time variable from Year and Day of Month is invalid because once Year is chosen, a full date is required.
 - You cannot use an existing date/time variable as one of the parts of the final date/time variable you're creating. Variables that make up the parts of the new date/time variable must be integers. The

exception is the allowed use of an existing date/time variable as the Seconds part of the new variable. Since fractional seconds are allowed, the variable used for Seconds is not required to be an integer.

- Values, for any part of the new variable, that are not within the allowed range result in a value of system-missing for the new variable. For instance, if you inadvertently use a variable representing day of month for Month, any cases with day of month values in the range 14–31 will be assigned the system-missing value for the new variable since the valid range for months in IBM SPSS Statistics is 1–13.

Specify Date/Time Variable Created by Merging Variables

1. Enter a name for the Result Variable. This cannot be the name of an existing variable.
2. Select a date/time format from the Output Format list.

Optionally, you can:

- Assign a descriptive variable label to the new variable.

Add or Subtract Values from Date/Time Variables

To add or subtract values from date/time variables:

1. Select **Calculate with dates and times** on the introduction screen of the Date and Time Wizard.

Select Type of Calculation to Perform with Date/Time Variables

- **Add or subtract a duration from a date.** Use this option to add to or subtract from a date-format variable. You can add or subtract durations that are fixed values, such as 10 days, or the values from a numeric variable, such as a variable that represents years.
- **Calculate the number of time units between two dates.** Use this option to obtain the difference between two dates as measured in a chosen unit. For example, you can obtain the number of years or the number of days separating two dates.
- **Subtract two durations.** Use this option to obtain the difference between two variables that have formats of durations, such as hh:mm or hh:mm:ss.

Note: Tasks are disabled when the dataset lacks the types of variables required to accomplish the task. For instance, if the dataset lacks two variables with formats of durations, then the task to subtract two durations does not apply and is disabled.

Add or Subtract a Duration from a Date

To add or subtract a duration from a date-format variable:

1. Select **Add or subtract a duration from a date** on the screen of the Date and Time Wizard labeled *Do Calculations on Dates*.

Select Date/Time Variable and Duration to Add or Subtract:

1. Select a date (or time) variable.
2. Select a duration variable or enter a value for Duration Constant. Variables used for durations cannot be date or date/time variables. They can be duration variables or simple numeric variables.
3. Select the unit that the duration represents from the drop-down list. Select **Duration** if using a variable and the variable is in the form of a duration, such as hh:mm or hh:mm:ss.

Specify Result of Adding or Subtracting a Duration from a Date/Time Variable:

1. Enter a name for Result Variable. This cannot be the name of an existing variable.

Optionally, you can:

- Assign a descriptive variable label to the new variable.

Subtract Date-Format Variables

To subtract two date-format variables:

1. Select **Calculate the number of time units between two dates** on the screen of the Date and Time Wizard labeled *Do Calculations on Dates*.

Select Date-Format Variables to Subtract:

1. Select the variables to subtract.
2. Select the unit for the result from the drop-down list.
3. Select how the result should be calculated (Result Treatment).

Result Treatment

The following options are available for how the result is calculated:

- **Truncate to integer.** Any fractional portion of the result is ignored. For example, subtracting 10/28/2006 from 10/21/2007 returns a result of 0 for years and 11 for months.
- **Round to integer.** The result is rounded to the closest integer. For example, subtracting 10/28/2006 from 10/21/2007 returns a result of 1 for years and 12 for months.
- **Retain fractional part.** The complete value is retained; no rounding or truncation is applied. For example, subtracting 10/28/2006 from 10/21/2007 returns a result of 0.98 for years and 11.76 for months.

For rounding and fractional retention, the result for years is based on average number of days in a year (365.25), and the result for months is based on the average number of days in a month (30.4375). For example, subtracting 2/1/2007 from 3/1/2007 (m/d/y format) returns a fractional result of 0.92 months, whereas subtracting 3/1/2007 from 2/1/2007 returns a fractional difference of 1.02 months. This also affects values calculated on time spans that include leap years. For example, subtracting 2/1/2008 from 3/1/2008 returns a fractional difference of 0.95 months, compared to 0.92 for the same time span in a non-leap year.

Table 8. Date difference for years

Date 1	Date 2	Truncate	Round	Fraction
10/21/2006	10/28/2007	1	1	1.02
10/28/2006	10/21/2007	0	1	.98
2/1/2007	3/1/2007	0	0	.08
2/1/2008	3/1/2008	0	0	.08
3/1/2007	4/1/2007	0	0	.08
4/1/2007	5/1/2007	0	0	.08

Table 9. Date difference for months

Date 1	Date 2	Truncate	Round	Fraction
10/21/2006	10/28/2007	12	12	12.22
10/28/2006	10/21/2007	11	12	11.76
2/1/2007	3/1/2007	1	1	.92
2/1/2008	3/1/2008	1	1	.95
3/1/2007	4/1/2007	1	1	1.02
4/1/2007	5/1/2007	1	1	.99

Specify Result of Subtracting Two Date-Format Variables:

1. Enter a name for Result Variable. This cannot be the name of an existing variable.

Optionally, you can:

- Assign a descriptive variable label to the new variable.

Subtract Duration Variables

To subtract two duration variables:

1. Select **Subtract two durations** on the screen of the Date and Time Wizard labeled *Do Calculations on Dates*.

Select Duration Variables to Subtract:

1. Select the variables to subtract.

Specify Result of Subtracting Two Duration Variables:

1. Enter a name for Result Variable. This cannot be the name of an existing variable.
2. Select a duration format from the Output Format list.

Optionally, you can:

- Assign a descriptive variable label to the new variable.

Extract Part of a Date/Time Variable

To extract a component--such as the year--from a date/time variable:

1. Select **Extract a part of a date or time variable** on the introduction screen of the Date and Time Wizard.

Select Component to Extract from Date/Time Variable

1. Select the variable containing the date or time part to extract.
2. Select the part of the variable to extract, from the drop-down list. You can extract information from dates that is not explicitly part of the display date, such as day of the week.

Specify Result of Extracting Component from Date/Time Variable

1. Enter a name for Result Variable. This cannot be the name of an existing variable.
2. If you're extracting the date or time part of a date/time variable, then you must select a format from the Output Format list. In cases where the output format is not required, the Output Format list will be disabled.

Optionally, you can:

- Assign a descriptive variable label to the new variable.

Time Series Data Transformations

Several data transformations that are useful in time series analysis are provided:

- Generate date variables to establish periodicity and to distinguish between historical, validation, and forecasting periods.
- Create new time series variables as functions of existing time series variables.
- Replace system- and user-missing values with estimates based on one of several methods.

A **time series** is obtained by measuring a variable (or set of variables) regularly over a period of time. Time series data transformations assume a data file structure in which each case (row) represents a set of observations at a different time, and the length of time between cases is uniform.

Define Dates

The Define Dates dialog box allows you to generate date variables that can be used to establish the periodicity of a *time series* and to label output from time series analysis.

Cases Are. Defines the time interval used to generate dates.

- **Not dated** removes any previously defined date variables. Any variables with the following names are deleted: *year_*, *quarter_*, *month_*, *week_*, *day_*, *hour_*, *minute_*, *second_*, and *date_*.
- **Custom** indicates the presence of custom date variables created with command syntax (for example, a four-day workweek). This item merely reflects the current state of the active dataset. Selecting it from the list has no effect.

First Case Is. Defines the starting date value, which is assigned to the first case. Sequential values, based on the time interval, are assigned to subsequent cases.

Periodicity at higher level. Indicates the repetitive cyclical variation, such as the number of months in a year or the number of days in a week. The value displayed indicates the maximum value you can enter. For hours, minutes, and seconds the maximum is the displayed value minus one.

A new numeric variable is created for each component that is used to define the date. The new variable names end with an underscore. A descriptive string variable, *date_*, is also created from the components. For example, if you selected **Weeks**, **days**, **hours**, four new variables are created: *week_*, *day_*, *hour_*, and *date_*.

If date variables have already been defined, they are replaced when you define new date variables that will have the same names as the existing date variables.

To Define Dates for Time Series Data

1. From the menus choose:
Data > Define Dates...
2. Select a time interval from the Cases Are list.
3. Enter the value(s) that define the starting date for First Case Is, which determines the date assigned to the first case.

Date Variables versus Date Format Variables

Date variables created with Define Dates should not be confused with date format variables defined in the Variable View of the Data Editor. Date variables are used to establish periodicity for time series data. Date format variables represent dates and/or times displayed in various date/time formats. Date variables are simple integers representing the number of days, weeks, hours, and so on, from a user-specified starting point. Internally, most date format variables are stored as the number of seconds from October 14, 1582.

Create Time Series

The Create Time Series dialog box allows you to create new variables based on functions of existing numeric *time series* variables. These transformed values are useful in many time series analysis procedures.

Default new variable names are the first six characters of the existing variable used to create it, followed by an underscore and a sequential number. For example, for the variable *price*, the new variable name would be *price_1*. The new variables retain any defined value labels from the original variables.

Available functions for creating time series variables include differences, moving averages, running medians, lag, and lead functions.

To Create New Time Series Variables

1. From the menus choose:
Transform > Create Time Series...
2. Select the time series function that you want to use to transform the original variable(s).

3. Select the variable(s) from which you want to create new time series variables. Only numeric variables can be used.

Optionally, you can:

- Enter variable names to override the default new variable names.
- Change the function for a selected variable.

Time Series Transformation Functions

Difference. Nonseasonal difference between successive values in the series. The order is the number of previous values used to calculate the difference. Because one observation is lost for each order of difference, system-missing values appear at the beginning of the series. For example, if the difference order is 2, the first two cases will have the system-missing value for the new variable.

Seasonal difference. Difference between series values a constant span apart. The span is based on the currently defined periodicity. To compute seasonal differences, you must have defined date variables (Data menu, Define Dates) that include a periodic component (such as months of the year). The order is the number of seasonal periods used to compute the difference. The number of cases with the system-missing value at the beginning of the series is equal to the periodicity multiplied by the order. For example, if the current periodicity is 12 and the order is 2, the first 24 cases will have the system-missing value for the new variable.

Centered moving average. Average of a span of series values surrounding and including the current value. The span is the number of series values used to compute the average. If the span is even, the moving average is computed by averaging each pair of uncentered means. The number of cases with the system-missing value at the beginning and at the end of the series for a span of n is equal to $n/2$ for even span values and $(n-1)/2$ for odd span values. For example, if the span is 5, the number of cases with the system-missing value at the beginning and at the end of the series is 2.

Prior moving average. Average of the span of series values preceding the current value. The span is the number of preceding series values used to compute the average. The number of cases with the system-missing value at the beginning of the series is equal to the span value.

Running medians. Median of a span of series values surrounding and including the current value. The span is the number of series values used to compute the median. If the span is even, the median is computed by averaging each pair of uncentered medians. The number of cases with the system-missing value at the beginning and at the end of the series for a span of n is equal to $n/2$ for even span values and $(n-1)/2$ for odd span values. For example, if the span is 5, the number of cases with the system-missing value at the beginning and at the end of the series is 2.

Cumulative sum. Cumulative sum of series values up to and including the current value.

Lag. Value of a previous case, based on the specified lag order. The order is the number of cases prior to the current case from which the value is obtained. The number of cases with the system-missing value at the beginning of the series is equal to the order value.

Lead. Value of a subsequent case, based on the specified lead order. The order is the number of cases after the current case from which the value is obtained. The number of cases with the system-missing value at the end of the series is equal to the order value.

Smoothing. New series values based on a compound data smoother. The smoother starts with a running median of 4, which is centered by a running median of 2. It then resmothes these values by applying a running median of 5, a running median of 3, and hanning (running weighted averages). Residuals are computed by subtracting the smoothed series from the original series. This whole process is then repeated on the computed residuals. Finally, the smoothed residuals are computed by subtracting the smoothed values obtained the first time through the process. This is sometimes referred to as **T4253H smoothing**.

Replace Missing Values

Missing observations can be problematic in analysis, and some time series measures cannot be computed if there are missing values in the series. Sometimes the value for a particular observation is simply not known. In addition, missing data can result from any of the following:

- Each degree of differencing reduces the length of a series by 1.
- Each degree of seasonal differencing reduces the length of a series by one season.
- If you create new series that contain forecasts beyond the end of the existing series (by clicking a **Save** button and making suitable choices), the original series and the generated residual series will have missing data for the new observations.
- Some transformations (for example, the log transformation) produce missing data for certain values of the original series.

Missing data at the beginning or end of a series pose no particular problem; they simply shorten the useful length of the series. Gaps in the middle of a series (*embedded* missing data) can be a much more serious problem. The extent of the problem depends on the analytical procedure you are using.

The Replace Missing Values dialog box allows you to create new *time series* variables from existing ones, replacing missing values with estimates computed with one of several methods. Default new variable names are the first six characters of the existing variable used to create it, followed by an underscore and a sequential number. For example, for the variable *price*, the new variable name would be *price_1*. The new variables retain any defined value labels from the original variables.

To Replace Missing Values for Time Series Variables

1. From the menus choose:
Transform > Replace Missing Values...
2. Select the estimation method you want to use to replace missing values.
3. Select the variable(s) for which you want to replace missing values.

Optionally, you can:

- Enter variable names to override the default new variable names.
- Change the estimation method for a selected variable.

Estimation Methods for Replacing Missing Values

Series mean. Replaces missing values with the mean for the entire series.

Mean of nearby points. Replaces missing values with the mean of valid surrounding values. The span of nearby points is the number of valid values above and below the missing value used to compute the mean.

Median of nearby points. Replaces missing values with the median of valid surrounding values. The span of nearby points is the number of valid values above and below the missing value used to compute the median.

Linear interpolation. Replaces missing values using a linear interpolation. The last valid value before the missing value and the first valid value after the missing value are used for the interpolation. If the first or last case in the series has a missing value, the missing value is not replaced.

Linear trend at point. Replaces missing values with the linear trend for that point. The existing series is regressed on an index variable scaled 1 to n . Missing values are replaced with their predicted values.

Chapter 9. File handling and file transformations

File handling and file transformations

Data files are not always organized in the ideal form for your specific needs. You may want to combine data files, sort the data in a different order, select a subset of cases, or change the unit of analysis by grouping cases together. A wide range of file transformation capabilities is available, including the ability to:

Sort data. You can sort cases based on the value of one or more variables.

Transpose cases and variables. The IBM SPSS Statistics data file format reads rows as cases and columns as variables. For data files in which this order is reversed, you can switch the rows and columns and read the data in the correct format.

Merge files. You can merge two or more data files. You can combine files with the same variables but different cases or the same cases but different variables.

Select subsets of cases. You can restrict your analysis to a subset of cases or perform simultaneous analyses on different subsets.

Aggregate data. You can change the unit of analysis by aggregating cases based on the value of one or more grouping variables.

Weight data. You can weight cases for analysis based on the value of a weight variable.

Restructure data. You can restructure data to create a single case (record) from multiple cases or create multiple cases from a single case.

Sort cases

This dialog box sorts cases (rows) of the active dataset based on the values of one or more sorting variables. You can sort cases in ascending or descending order.

- If you select multiple sort variables, cases are sorted by each variable within categories of the preceding variable on the Sort list. For example, if you select *gender* as the first sorting variable and *minority* as the second sorting variable, cases will be sorted by minority classification within each gender category.
- The sort sequence is based on the locale-defined order (and is not necessarily the same as the numerical order of the character codes). The default locale is the operating system locale. You can control the locale with the Language setting on the General tab of the Options dialog box (Edit menu).

To Sort Cases

1. From the menus choose:

Data > Sort Cases...

2. Select one or more sorting variables.

Optionally, you can do the following:

Index the saved file. Indexing table lookup files can improve performance when merging data files with STAR JOIN.

Save the sorted file. You can save the sorted file, with the option of saving it as encrypted. Encryption allows you to protect confidential information stored in the file. Once encrypted, the file can only be opened by providing the password assigned to the file.

- To save the sorted file with encryption:
3. Select **Save file with sorted data** and click **File**.
 4. Select **Encrypt file with password** in the Save Sorted Data As dialog box.
 5. Click **Save**.
 6. In the Encrypt File dialog box, provide a password and re-enter it in the Confirm password text box.
Passwords are limited to 10 characters and are case-sensitive.

Warning: Passwords cannot be recovered if they are lost. If the password is lost the file cannot be opened.

Creating strong passwords

- Use eight or more characters.
- Include numbers, symbols and even punctuation in your password.
- Avoid sequences of numbers or characters, such as "123" and "abc", and avoid repetition, such as "111aaa".
- Do not create passwords that use personal information such as birthdays or nicknames.
- Periodically change the password.

Note: Storing encrypted files to an IBM SPSS Collaboration and Deployment Services Repository is not supported.

Note: Encrypted data files and output documents cannot be opened in versions of IBM SPSS Statistics prior to version 21. Encrypted syntax files cannot be opened in versions prior to version 22.

Sort variables

You can sort the variables in the active dataset based on the values of any of the variable attributes (e.g., variable name, data type, measurement level), including custom variable attributes.

- Values can be sorted in ascending or descending order.
- You can save the original (pre-sorted) variable order in a custom variable attribute.
- Sorting by values of custom variable attributes is limited to custom variable attributes that are currently visible in Variable View.

For more information on custom variable attributes, see “Custom Variable Attributes” on page 55.

To Sort Variables

In Variable View of the Data Editor:

1. Right-click the attribute column heading and from the pop-up menu choose **Sort Ascending** or **Sort Descending**.
or
2. From the menus in Variable View or Data View, choose:
Data > Sort Variables
3. Select the attribute you want to use to sort variables.
4. Select the sort order (ascending or descending).
 - The list of variable attributes matches the attribute column names displayed in Variable View of the Data Editor.
 - You can save the original (pre-sorted) variable order in a custom variable attribute. For each variable, the value of the attribute is an integer value indicating its position prior to sorting; so by sorting variables based on the value of that custom attribute you can restore the original variable order.

Transpose

Transpose creates a new data file in which the rows and columns in the original data file are transposed so that cases (rows) become variables and variables (columns) become cases. Transpose automatically creates new variable names and displays a list of the new variable names.

- A new string variable that contains the original variable name, *case_lbl*, is automatically created.
- If the active dataset contains an ID or name variable with unique values, you can use it as the name variable, and its values will be used as variable names in the transposed data file. If it is a numeric variable, the variable names start with the letter *V*, followed by the numeric value.
- User-missing values are converted to the system-missing value in the transposed data file. To retain any of these values, change the definition of missing values in the Variable View in the Data Editor.

To Transpose Variables and Cases

1. From the menus choose:
Data > Transpose...
2. Select one or more variables to transpose into cases.

Merging Data Files

You can merge data from two files in two different ways. You can:

- Merge the active dataset with another open dataset or IBM SPSS Statistics data file containing the same variables but different cases.
- Merge the active dataset with another open dataset or IBM SPSS Statistics data file containing the same cases but different variables.

To Merge Files

1. From the menus choose:
Data > Merge Files
2. Select **Add Cases** or **Add Variables**.

Add Cases

Add Cases merges the active dataset with a second dataset or external IBM SPSS Statistics data file that contains the same variables (columns) but different cases (rows). For example, you might record the same information for customers in two different sales regions and maintain the data for each region in separate files. The second dataset can be an external IBM SPSS Statistics data file or a dataset available in the current session.

Unpaired Variables. Variables to be excluded from the new, merged data file. Variables from the active dataset are identified with an asterisk (*). Variables from the other dataset are identified with a plus sign (+). By default, this list contains:

- Variables from either data file that do not match a variable name in the other file. You can create pairs from unpaired variables and include them in the new, merged file.
- Variables defined as numeric data in one file and string data in the other file. Numeric variables cannot be merged with string variables.
- String variables of unequal width. The defined width of a string variable must be the same in both data files.

Variables in New Active Dataset. Variables to be included in the new, merged data file. By default, all of the variables that match both the name and the data type (numeric or string) are included on the list.

- You can remove variables from the list if you do not want them to be included in the merged file.

- Any unpaired variables included in the merged file will contain missing data for cases from the file that does not contain that variable.

Indicate case source as variable. Indicates the source data file for each case. This variable has a value of 0 for cases from the active dataset and a value of 1 for cases from the external data file.

1. Open at least one of the data files that you want to merge. If you have multiple datasets open, make one of the datasets that you want to merge the active dataset. The cases from this file will appear first in the new, merged data file.
 2. From the menus choose:
- Data > Merge Files > Add Cases...**
3. Select the dataset or external IBM SPSS Statistics data file to merge with the active dataset.
 4. Remove any variables that you do not want from the Variables in New Active Dataset list.
 5. Add any variable pairs from the Unpaired Variables list that represent the same information recorded under different variable names in the two files. For example, date of birth might have the variable name *brthdate* in one file and *datebrth* in the other file.

To Select a Pair of Unpaired Variables

1. Click one of the variables on the Unpaired Variables list.
2. Ctrl-click the other variable on the list. (Press the Ctrl key and click the left mouse button at the same time.)
3. Click **Pair** to move the variable pair to the Variables in New Active Dataset list. (The variable name from the active dataset is used as the variable name in the merged file.)

Add Cases: Rename

You can rename variables from either the active dataset or the other dataset before moving them from the unpaired list to the list of variables to be included in the merged data file. Renaming variables enables you to:

- Use the variable name from the other dataset rather than the name from the active dataset for variable pairs.
- Include two variables with the same name but of unmatched types or different string widths. For example, to include both the numeric variable *sex* from the active dataset and the string variable *sex* from the other dataset, one of them must be renamed first.

Add Cases: Dictionary Information

Any existing dictionary information (variable and value labels, user-missing values, display formats) in the active dataset is applied to the merged data file.

- If any dictionary information for a variable is undefined in the active dataset, dictionary information from the other dataset is used.
- If the active dataset contains any defined value labels or user-missing values for a variable, any additional value labels or user-missing values for that variable in the other dataset are ignored.

Merging More Than Two Data Sources

Using command syntax, you can merge up to 50 datasets and/or data files. For more information, see the ADD FILES command in the *Command Syntax Reference* (available from the Help menu).

Add Variables

Add Variables merges the active data file with another open data file or external IBM SPSS Statistics data file that contains the same cases (rows) but different variables (columns). For example, you might want to merge a data file that contains pre-test results with one that contains post-test results.

Merge Method tab

Use the **Merge Method** tab to define the merge type.

One-to-one merge based on file order

Order of cases in the files determines how cases are matched. This is the default setting when no variables exist with the same name and basic type (string or numeric) in both files. This setting generates MATCH FILES command syntax in the following format (where [name] is either a dataset or external file specification (enclosed in quotation marks)):

```
MATCH FILES FILE=* /FILE="[name]"
```

One-to-one merge based on key values

Cases are matched based on the values of one or more key variables. This is the default setting when one or more variables have the same name and basic type (string or numeric) in both files. This setting generates MATCH FILES command syntax in the following format (where [name] is either a dataset or external file specification (enclosed in quotation marks)):

```
MATCH FILES FILE=* /FILE="[name]" /BY [key varlist]
```

One-to-many merge based on key values

One file contains case data, and one file is a lookup table. Cases from the lookup table are merged with cases with matching key values in the case data file. The same key value can occur multiple times in the case data file. One case in the lookup table can be merged with many cases in the case data file.

- All cases in the case data file are included in the merged file.
- Cases in the table lookup file that do not have cases with matching key values in the case data file not included.
- The table lookup file cannot contain duplicate key values. If the files have multiple key variables, the key value is the combination of those values.

This setting generates MATCH FILES syntax with a TABLE subcommand for the lookup table.

Select Lookup Table

The following settings are enabled only when **One-to-many merge based on key values** is selected.

[active dataset name]* - The active dataset name followed by an asterisk.

- By default, this option is not selected.
- When selected, this setting generates syntax in the following format (where [name] is either a dataset or external file specification (enclosed in quotation marks)):

```
MATCH FILES TABLE==* /FILE="[name]" /BY [key varlist]
```

[second dataset name or file name] - The name of the second dataset or file.

- This is the default setting.
- When the file is external, only the file name (not the entire path) is provided (the full path is included in the generated syntax).
- When the second file is an external file (and sorting is selected, or it has string keys that require a change in the defined length), the file must first be opened and assigned a unique name.
- When the second file is an external file that needs to be opened (based on the previous condition), and the active dataset is unnamed, the active dataset must be named before the second data file is opened.
- When the second file is opened for sorting or changing string key lengths, it remains open after the merge action. Because the file was changed, you are prompted to save changes when the file is closed.
- When selected, this setting generates syntax in the following format (where [name] is either a dataset or external file specification (enclosed in quotation marks)):

```
MATCH FILES FILE=* /TABLE=[name] /BY [key varlist]
```

Sort files by key values before merging

For key value merges, both files must be sorted in order of the key variable values.

- This setting is enabled only when one of the **key values** merge options is selected.
- If one of the files is an external file, it is opened and sorted. The sorted file is not saved unless you explicitly save the file.
- If the files are already sorted, you can save time by clearing this option.
- When selected, this setting generates SORT CASES syntax.

Key Variables

For key value merges, variables with the same name and basic data type (string or numeric) are included as key variables by default. Use the **Variables** tab to add, remove, or change the order of the key variables.

Notes:

- The selected merge always generates MATCH FILES command syntax (never STAR JOIN syntax).
- SORT CASES and ALTER TYPE command syntax takes precedence over MATCH FILES command syntax.
- DROP subcommand syntax, and optional RENAME subcommand syntax, is included based on selections that were made on the **Variables** tab.
- BY subcommand syntax is included if one of the **key values** merge options is selected.
- When string keys have different defined lengths, ALTER TYPE syntax is automatically generated to ensure equally defined lengths.

Variables tab

Use the **Variables** tab to add, remove, and rename variables to include in the merged file.

- Variables from the active data file are identified with an asterisk (*).
- Variables from the other data file are identified with a plus sign (+).

Excluded Variables

Variables to exclude from the new, merged data file.

- When the **One-to-one merge based on file order** setting is selected on the **Merge Method** tab, all variables in the second dataset whose names are the same as names in active dataset are excluded.
- When the **One-to-one merge based on key values** or **One-to-many merge based on key values** setting is selected on the **Merge Method** tab, all duplicate names variables in the second dataset (that are a different basic type (string/numeric) than the same variable names in the active dataset) are excluded.

Included Variables

Variables to include in the new, merged data file. If you want to include an excluded variable with a duplicate name in the merged data file, use **Rename** to change the name.

- When the **One-to-one merge based on file order** setting is selected on the **Merge Method** tab, all variables in the active dataset and all uniquely named variables in the second dataset are included.
- When the **One-to-one merge based on key values** or **One-to-many merge based on key values** setting is selected on the **Merge Method** tab, all uniquely named variables in both datasets are included. Variables in the active dataset, with duplicate names in second dataset that are a different basic type (string/numeric) in the second dataset, are also included.

Key Variables

For key value merges, cases are merged based on key variable values.

- The default list assignment is determined by the selected **Merge Method**. Variables are never automatically assigned to the **Key Variables** list.

- When the **One-to-one merge based on file order** setting is selected on the **Merge Method** tab, no variables are included, the **Move** control is disabled, and manually dragging and dropping variables has no effect.
- When the **One-to-one merge based on key values** or **One-to-many merge based on key values** setting is selected on the **Merge Method** tab, all variables with same name and same basic type (string/numeric) in both datasets are included.
- Each key variable must have the same name and same basic data type (string or numeric) in both files.
- If a key variable has different names in the two files, use **Rename** to change one of the names.
- If a string key variable has different defined lengths in the two files, the variable with the shorter length is automatically adjusted to the longer length. The modified version of the original file is not saved unless you explicitly save it.

Add Variables: Rename

You can rename variables from either the active dataset or the other data file before moving them to the list of variables to be included in the merged data file. This is primarily useful if you want to include two variables with the same name that contain different information in the two files or if a key variable has different names in the two files.

Merging More Than Two Data Sources

Using command syntax, you can merge more than two data files.

- Use **MATCH FILES** to merge multiple files that don't contain key variables or multiple files already sorted on key variable values.
- Use **STAR JOIN** to merge multiple files where there is one case data file and multiple table lookup files. Files do not need to be sorted in order of key variable values, and each table lookup file can use a different key variable.

Aggregate Data

Aggregate Data aggregates groups of cases in the active dataset into single cases and creates a new, aggregated file or creates new variables in the active dataset that contain aggregated data. Cases are aggregated based on the value of zero or more break (grouping) variables. If no break variables are specified, then the entire dataset is a single break group.

- If you create a new, aggregated data file, the new data file contains one case for each group defined by the break variables. For example, if there is one break variable with two values, the new data file will contain only two cases. If no break variable is specified, the new data file will contain one case.
- If you add aggregate variables to the active dataset, the data file itself is not aggregated. Each case with the same value(s) of the break variable(s) receives the same values for the new aggregate variables. For example, if *gender* is the only break variable, all males would receive the same value for a new aggregate variable that represents average age. If no break variable is specified, all cases would receive the same value for a new aggregate variable that represents average age.

Break Variable(s). Cases are grouped together based on the values of the break variables. Each unique combination of break variable values defines a group. When creating a new, aggregated data file, all break variables are saved in the new file with their existing names and dictionary information. The break variable, if specified, can be either numeric or string.

Aggregated Variables. Source variables are used with aggregate functions to create new aggregate variables. The aggregate variable name is followed by an optional variable label, the name of the aggregate function, and the source variable name in parentheses.

You can override the default aggregate variable names with new variable names, provide descriptive variable labels, and change the functions used to compute the aggregated data values. You can also create a variable that contains the number of cases in each break group.

To Aggregate a Data File

1. From the menus choose:

Data > Aggregate...

2. Optionally select break variables that define how cases are grouped to create aggregated data. If no break variables are specified, then the entire dataset is a single break group.
3. Select one or more aggregate variables.
4. Select an aggregate function for each aggregate variable.

Saving Aggregated Results

You can add aggregate variables to the active dataset or create a new, aggregated data file.

- *Add aggregated variables to active dataset.* New variables based on aggregate functions are added to the active dataset. The data file itself is not aggregated. Each case with the same value(s) of the break variable(s) receives the same values for the new aggregate variables.
- *Create a new dataset containing only the aggregated variables.* Saves aggregated data to a new dataset in the current session. The dataset includes the break variables that define the aggregated cases and all aggregate variables defined by aggregate functions. The active dataset is unaffected.
- *Write a new data file containing only the aggregated variables.* Saves aggregated data to an external data file. The file includes the break variables that define the aggregated cases and all aggregate variables defined by aggregate functions. The active dataset is unaffected.

Sorting Options for Large Data Files

For very large data files, it may be more efficient to aggregate presorted data.

File is already sorted on break variable(s). If the data have already been sorted by values of the break variables, this option enables the procedure to run more quickly and use less memory. Use this option with caution.

- Data must be sorted by values of the break variables in the same order as the break variables specified for the Aggregate Data procedure.
- If you are adding variables to the active dataset, select this option only if the data are sorted by ascending values of the break variables.

Sort file before aggregating. In very rare instances with large data files, you may find it necessary to sort the data file by values of the break variables prior to aggregating. This option is not recommended unless you encounter memory or performance problems.

Aggregate Data: Aggregate Function

This dialog box specifies the function to use to calculate aggregated data values for selected variables on the Aggregate Variables list in the Aggregate Data dialog box. Aggregate functions include:

- Summary functions for numeric variables, including mean, median, standard deviation, and sum
- Number of cases, including unweighted, weighted, nonmissing, and missing
- Percentage, fraction or count of values above or below a specified value
- Percentage, fraction, or count of values inside or outside of a specified range

Aggregate Data: Variable Name and Label

Aggregate Data assigns default variable names for the aggregated variables in the new data file. This dialog box enables you to change the variable name for the selected variable on the Aggregate Variables list and provide a descriptive variable label. See the topic “Variable names” on page 50 for more information.

Split file

Split File splits the data file into separate groups for analysis based on the values of one or more grouping variables. If you select multiple grouping variables, cases are grouped by each variable within categories of the preceding variable on the Groups Based On list. For example, if you select *gender* as the first grouping variable and *minority* as the second grouping variable, cases will be grouped by minority classification within each gender category.

- You can specify up to eight grouping variables.
- Each eight bytes of a long string variable (string variables longer than eight bytes) counts as a variable toward the limit of eight grouping variables.
- Cases should be sorted by values of the grouping variables and in the same order that variables are listed in the Groups Based On list. If the data file isn't already sorted, select **Sort the file by grouping variables**.

Compare groups. Split-file groups are presented together for comparison purposes. For pivot tables, a single pivot table is created and each split-file variable can be moved between table dimensions. For charts, a separate chart is created for each split-file group and the charts are displayed together in the Viewer.

Organize output by groups. All results from each procedure are displayed separately for each split-file group.

To Split a Data File for Analysis

1. From the menus choose:
Data > Split File...
 2. Select **Compare groups** or **Organize output by groups**.
 3. Select one or more grouping variables.
-

Select cases

Select Cases provides several methods for selecting a subgroup of cases based on criteria that include variables and complex expressions. You can also select a random sample of cases. The criteria used to define a subgroup can include:

- Variable values and ranges
- Date and time ranges
- Case (row) numbers
- Arithmetic expressions
- Logical expressions
- Functions

All cases. Turns case filtering off and uses all cases.

If condition is satisfied. Use a conditional expression to select cases. If the result of the conditional expression is true, the case is selected. If the result is false or missing, the case is not selected.

Random sample of cases. Selects a random sample based on an approximate percentage or an exact number of cases.

Based on time or case range. Selects cases based on a range of case numbers or a range of dates/times.

Use filter variable. Use the selected numeric variable from the data file as the filter variable. Cases with any value other than 0 or missing for the filter variable are selected.

Output

This section controls the treatment of unselected cases. You can choose one of the following alternatives for the treatment of unselected cases:

- **Filter out unselected cases.** Unselected cases are not included in the analysis but remain in the dataset. You can use the unselected cases later in the session if you turn filtering off. If you select a random sample or if you select cases based on a conditional expression, this generates a variable named *filter_*\$ with a value of 1 for selected cases and a value of 0 for unselected cases.
- **Copy selected cases to a new dataset.** Selected cases are copied to a new dataset, leaving the original dataset unaffected. Unselected cases are not included in the new dataset and are left in their original state in the original dataset.
- **Delete unselected cases.** Unselected cases are deleted from the dataset. Deleted cases can be recovered only by exiting from the file without saving any changes and then reopening the file. The deletion of cases is permanent if you save the changes to the data file.

Note: If you delete unselected cases and save the file, the cases cannot be recovered.

To Select a Subset of Cases

1. From the menus choose:
Data > Select Cases...
2. Select one of the methods for selecting cases.
3. Specify the criteria for selecting cases.

Select cases: If

This dialog box allows you to select subsets of cases using conditional expressions. A conditional expression returns a value of *true*, *false*, or *missing* for each case.

- If the result of a conditional expression is *true*, the case is included in the selected subset.
- If the result of a conditional expression is *false* or *missing*, the case is not included in the selected subset.
- Most conditional expressions use one or more of the six relational operators (<, >, <=, >=, =, and ~=) on the calculator pad.
- Conditional expressions can include variable names, constants, arithmetic operators, numeric (and other) functions, logical variables, and relational operators.

Select cases: Random sample

This dialog box allows you to select a random sample based on an approximate percentage or an exact number of cases. Sampling is performed without replacement; so, the same case cannot be selected more than once.

Approximately. Generates a random sample of approximately the specified percentage of cases. Since this routine makes an independent pseudo-random decision for each case, the percentage of cases selected can only approximate the specified percentage. The more cases there are in the data file, the closer the percentage of cases selected is to the specified percentage.

Exactly. A user-specified number of cases. You must also specify the number of cases from which to generate the sample. This second number should be less than or equal to the total number of cases in the data file. If the number exceeds the total number of cases in the data file, the sample will contain proportionally fewer cases than the requested number.

Select cases: Range

This dialog box selects cases based on a range of case numbers or a range of dates or times.

- Case ranges are based on row number as displayed in the Data Editor.
- Date and time ranges are available only for *time series data* with defined date variables (Data menu, Define Dates).

Note: If unselected cases are filtered (rather than deleted), subsequently sorting the dataset will turn off filtering applied by this dialog.

Weight cases

Weight Cases gives cases different weights (by simulated replication) for statistical analysis.

- The values of the weighting variable should indicate the number of observations represented by single cases in your data file.
- Cases with zero, negative, or missing values for the weighting variable are excluded from analysis.
- Fractional values are valid and some procedures, such as Frequencies, Crosstabs, and Custom Tables, will use fractional weight values. However, most procedures treat the weighting variable as a replication weight and will simply round fractional weights to the nearest integer. Some procedures ignore the weighting variable completely, and this limitation is noted in the procedure-specific documentation.

Once you apply a weight variable, it remains in effect until you select another weight variable or turn off weighting. If you save a weighted data file, weighting information is saved with the data file. You can turn off weighting at any time, even after the file has been saved in weighted form.

Weights in Crosstabs. The Crosstabs procedure has several options for handling case weights.

Weights in scatterplots and histograms. Scatterplots and histograms have an option for turning case weights on and off, but this does not affect cases with a zero, negative, or missing value for the weight variable. These cases remain excluded from the chart even if you turn weighting off from within the chart.

To Weight Cases

1. From the menus choose:
Data > Weight Cases...
2. Select **Weight cases by**.
3. Select a frequency variable.

The values of the frequency variable are used as case weights. For example, a case with a value of 3 for the frequency variable will represent three cases in the weighted data file.

Restructuring Data

Use the Restructure Data Wizard to restructure your data for the procedure that you want to use. The wizard replaces the current file with a new, restructured file. The wizard can:

- Restructure selected variables into cases
- Restructure selected cases into variables
- Transpose all data

To Restructure Data

1. From the menus choose:
Data > Restructure...
2. Select the type of restructuring that you want to do.
3. Select the data to restructure.

Optionally, you can:

- Create identification variables, which allow you to trace a value in the new file back to a value in the original file
- Sort the data prior to restructuring
- Define options for the new file
- Paste the command syntax into a syntax window

Restructure Data Wizard: Select Type

Use the Restructure Data Wizard to restructure your data. In the first dialog box, select the type of restructuring that you want to do.

- **Restructure selected variables into cases.** Choose this when you have groups of related columns in your data and you want them to appear in groups of rows in the new data file. If you choose this, the wizard will display the steps for **Variables to Cases**.
- **Restructure selected cases into variables.** Choose this when you have groups of related rows in your data and you want them to appear in groups of columns in the new data file. If you choose this, the wizard will display the steps for **Cases to Variables**.
- **Transpose all data.** Choose this when you want to transpose your data. All rows will become columns and all columns will become rows in the new data. This choice closes the Restructure Data Wizard and opens the Transpose Data dialog box.

Deciding How to Restructure the Data

A **variable** contains information that you want to analyze--for example, a measurement or a score. A **case** is an observation--for example, an individual. In a *simple* data structure, each variable is a single column in your data and each case is a single row. So, for example, if you were measuring test scores for all students in a class, all score values would appear in only one column, and there would be a row for each student.

When you analyze data, you are often analyzing how a variable varies according to some condition. The condition can be a specific experimental treatment, a demographic, a point in time, or something else. In data analysis, conditions of interest are often referred to as **factors**. When you analyze factors, you have a *complex* data structure. You may have information about a variable in more than one column in your data (for example, a column for each level of a factor), or you may have information about a case in more than one row (for example, a row for each level of a factor). The Restructure Data Wizard helps you to restructure files with a complex data structure.

The structure of the current file and the structure that you want in the new file determine the choices that you make in the wizard.

How are the data arranged in the current file? The current data may be arranged so that factors are recorded in a *separate* variable (in groups of cases) or *with* the variable (in groups of variables).

- **Groups of cases.** Does the current file have variables and conditions recorded in separate columns?
For example:

Table 10. Data with variables and conditions in separate columns

var	factor
8	1
9	1
3	2
1	2

In this example, the first two rows are a **case group** because they are related. They contain data for the same factor level. In IBM SPSS Statistics data analysis, the factor is often referred to as a **grouping variable** when the data are structured this way.

- **Groups of columns.** Does the current file have variables and conditions recorded in the same column? For example:

Table 11. Data with variables and conditions in same column

var_1	var_2
8	3
9	1

In this example, the two columns are a **variable group** because they are related. They contain data for the same variable--*var_1* for factor level 1 and *var_2* for factor level 2. In IBM SPSS Statistics data analysis, the factor is often referred to as a **repeated measure** when the data are structured this way.

How should the data be arranged in the new file? This is usually determined by the procedure that you want to use to analyze your data.

- **Procedures that require groups of cases.** Your data must be structured in case groups to do analyses that require a grouping variable. Examples are *univariate*, *multivariate*, and *variance components* with General Linear Model, Mixed Models, and OLAP Cubes and *independent samples* with T Test or Nonparametric Tests. If your current data structure is variable groups and you want to do these analyses, select **Restructure selected variables into cases**.
- **Procedures that require groups of variables.** Your data must be structured in variable groups to analyze repeated measures. Examples are *repeated measures* with General Linear Model, *time-dependent covariate* analysis with Cox Regression Analysis, *paired samples* with T Test, or *related samples* with Nonparametric Tests. If your current data structure is case groups and you want to do these analyses, select **Restructure selected cases into variables**.

Example of Variables to Cases

In this example, test scores are recorded in separate columns for each factor, *A* and *B*.

Table 12. Test scores recorded in separate columns for each factor

score_a	score_b
1014	864
684	636
810	638

You want to do an independent-samples *t* test. You have a column group consisting of *score_a* and *score_b*, but you don't have the **grouping variable** that the procedure requires. Select **Restructure selected variables into cases** in the Restructure Data Wizard, restructure one variable group into a new variable named *score*, and create an index named *group*. The new data file is shown in the following figure.

Table 13. New, restructured data for variables to cases

group	score
SCORE_A	1014
SCORE_B	864
SCORE_A	684
SCORE_B	636
SCORE_A	810
SCORE_B	638

When you run the independent-samples *t* test, you can now use *group* as the grouping variable.

Example of Cases to Variables

In this example, test scores are recorded twice for each subject—before and after a treatment.

Table 14. Current data for cases to variables

id	scor	time
1	1014	bef
1	864	aft
2	684	bef
2	636	aft

You want to do a paired-samples *t* test. Your data structure is case groups, but you don't have the **repeated measures** for the paired variables that the procedure requires. Select **Restructure selected cases into variables** in the Restructure Data Wizard, use *id* to identify the row groups in the current data, and use *time* to create the variable group in the new file.

Table 15. New, restructured data for cases to variables

id	aft	bef
1	864	1014
2	636	684

When you run the paired-samples *t* test, you can now use *bef* and *aft* as the variable pair.

Restructure Data Wizard (Variables to Cases): Number of Variable Groups

Note: The wizard presents this step if you choose to restructure variable groups into rows.

In this step, decide how many variable groups in the current file that you want to restructure in the new file.

How many variable groups are in the current file? Think about how many variable groups exist in the current data. A group of related columns, called a **variable group**, records repeated measures of the same variable in separate columns. For example, if you have three columns in the current data—*w1*, *w2*, and *w3*—that record **width**, you have one variable group. If you have an additional three columns—*h1*, *h2*, and *h3*—that record **height**, you have two variable groups.

How many variable groups should be in the new file? Consider how many variable groups you want to have represented in the new data file. You do not have to restructure all variable groups into the new file.

- **One.** The wizard will create a single restructured variable in the new file from one variable group in the current file.
- **More than one.** The wizard will create multiple restructured variables in the new file. The number that you specify affects the next step, in which the wizard automatically creates the specified number of new variables.

Restructure Data Wizard (Variables to Cases): Select Variables

Note: The wizard presents this step if you choose to restructure variable groups into rows.

In this step, provide information about how the variables in the current file should be used in the new file. You can also create a variable that identifies the rows in the new file.

How should the new rows be identified? You can create a variable in the new data file that identifies the row in the current data file that was used to create a group of new rows. The identifier can be a sequential case number or it can be the values of the variable. Use the controls in Case Group Identification to define the identification variable in the new file. Click a cell to change the default variable name and provide a descriptive variable label for the identification variable.

What should be restructured in the new file? In the previous step, you told the wizard how many variable groups you want to restructure. The wizard created one new variable for each group. The values for the variable group will appear in that variable in the new file. Use the controls in Variable to be Transposed to define the restructured variable in the new file.

To Specify One Restructured Variable

1. Put the variables that make up the variable group that you want to transform into the Variable to be Transposed list. All of the variables in the group must be of the same type (numeric or string).

You can include the same variable more than once in the variable group (variables are copied rather than moved from the source variable list); its values are repeated in the new file.

To Specify Multiple Restructured Variables

1. Select the first target variable that you want to define from the Target Variable drop-down list.
 2. Put the variables that make up the variable group that you want to transform into the Variable to be Transposed list. All of the variables in the group must be of the same type (numeric or string). You can include the same variable more than once in the variable group. (A variable is copied rather than moved from the source variable list, and its values are repeated in the new file.)
 3. Select the next target variable that you want to define, and repeat the variable selection process for all available target variables.
- Although you can include the same variable more than once in the same target variable group, you cannot include the same variable in more than one target variable group.
 - Each target variable group list must contain the same number of variables. (Variables that are listed more than once are included in the count.)
 - The number of target variable groups is determined by the number of variable groups that you specified in the previous step. You can change the default variable names here, but you must return to the previous step to change the number of variable groups to restructure.
 - You must define variable groups (by selecting variables in the source list) for all available target variables before you can proceed to the next step.

What should be copied into the new file? Variables that aren't restructured can be copied into the new file. Their values will be propagated in the new rows. Move variables that you want to copy into the new file into the Fixed Variable(s) list.

Restructure Data Wizard (Variables to Cases): Create Index Variables

Note: The wizard presents this step if you choose to restructure variable groups into rows.

In this step, decide whether to create index variables. An index is a new variable that sequentially identifies a row group based on the original variable from which the new row was created.

How many index variables should be in the new file? Index variables can be used as grouping variables in procedures. In most cases, a single index variable is sufficient; however, if the variable groups in your current file reflect multiple factor levels, multiple indices may be appropriate.

- **One.** The wizard will create a single index variable.
- **More than one.** The wizard will create multiple indices and enter the number of indices that you want to create. The number that you specify affects the next step, in which the wizard automatically creates the specified number of indices.
- **None.** Select this if you do not want to create index variables in the new file.

Example of One Index for Variables to Cases

In the current data, there is one variable group, *width*, and one factor, *time*. Width was measured three times and recorded in *w1*, *w2*, and *w3*.

Table 16. Current data for one index

subject	w1	w2	w3
1	6.7	4.3	5.7
2	7.1	5.9	5.6

We'll restructure the variable group into a single variable, *width*, and create a single numeric index. The new data are shown in the following table.

Table 17. New, restructured data with one index

subject	index	width
1	1	6.7
1	2	4.3
1	3	5.7
2	1	7.1
2	2	5.9
2	3	5.6

Index starts with 1 and increments for each variable in the group. It restarts each time a new row is encountered in the original file. We can now use *index* in procedures that require a grouping variable.

Example of Two Indices for Variables to Cases

When a variable group records more than one factor, you can create more than one index; however, the current data must be arranged so that the levels of the first factor are a primary index within which the levels of subsequent factors cycle. In the current data, there is one variable group, *width*, and two factors, *A* and *B*. The data are arranged so that levels of factor *B* cycle within levels of factor *A*.

Table 18. Current data for two indices

subject	w_a1b1	w_a1b2	w_a2b1	w_a2b2
1	5.5	6.4	5.8	5.9
2	7.4	7.1	5.6	6.7

We'll restructure the variable group into a single variable, *width*, and create two indices. The new data are shown in the following table.

Table 19. New, restructured data with two indices

subject	index_a	index_b	width
1	1	1	5.5
1	1	2	6.4
1	2	1	5.8
1	2	2	5.9
2	1	1	7.4
2	1	2	7.1
2	2	1	5.6
2	2	2	6.7

Restructure Data Wizard (Variables to Cases): Create One Index Variable

Note: The wizard presents this step if you choose to restructure variable groups into rows and create one index variable.

In this step, decide what values you want for the index variable. The values can be sequential numbers or the names of the variables in an original variable group. You can also specify a name and a label for the new index variable.

See the topic "Example of One Index for Variables to Cases" on page 116 for more information.

- **Sequential numbers.** The wizard will automatically assign sequential numbers as index values.
- **Variable names.** The wizard will use the names of the selected variable group as index values. Choose a variable group from the list.
- **Names and labels.** Click a cell to change the default variable name and provide a descriptive variable label for the index variable.

Restructure Data Wizard (Variables to Cases): Create Multiple Index Variables

Note: The wizard presents this step if you choose to restructure variable groups into rows and create multiple index variables.

In this step, specify the number of levels for each index variable. You can also specify a name and a label for the new index variable.

See the topic "Example of Two Indices for Variables to Cases" on page 116 for more information.

How many levels are recorded in the current file? Consider how many factor levels are recorded in the current data. A level defines a group of cases that experienced identical conditions. If there are multiple factors, the current data must be arranged so that the levels of the first factor are a primary index within which the levels of subsequent factors cycle.

How many levels should be in the new file? Enter the number of levels for each index. The values for multiple index variables are always sequential numbers. The values start at 1 and increment for each level. The first index increments the slowest, and the last index increments the fastest.

Total combined levels. You cannot create more levels than exist in the current data. Because the restructured data will contain one row for each combination of treatments, the wizard checks the number of levels that you create. It will compare the product of the levels that you create to the number of variables in your variable groups. They must match.

Names and labels. Click a cell to change the default variable name and provide a descriptive variable label for the index variables.

Restructure Data Wizard (Variables to Cases): Options

Note: The wizard presents this step if you choose to restructure variable groups into rows.

In this step, specify options for the new, restructured file.

Drop unselected variables? In the Select Variables step (step 3), you selected variable groups to be restructured, variables to be copied, and an identification variable from the current data. The data from the selected variables will appear in the new file. If there are other variables in the current data, you can choose to discard or keep them.

Keep missing data? The wizard checks each potential new row for null values. A **null value** is a system-missing or blank value. You can choose to keep or discard rows that contain only null values.

Create a count variable? The wizard can create a **count variable** in the new file. It contains the number of new rows generated by a row in the current data. A count variable may be useful if you choose to discard null values from the new file because that makes it possible to generate a different number of new rows for a given row in the current data. Click a cell to change the default variable name and provide a descriptive variable label for the count variable.

Restructure Data Wizard (Cases to Variables): Select Variables

Note: The wizard presents this step if you choose to restructure case groups into columns.

In this step, provide information about how the variables in the current file should be used in the new file.

What identifies case groups in the current data? A **case group** is a group of rows that are related because they measure the same observational unit—for example, an individual or an institution. The wizard needs to know which variables in the current file identify the case groups so that it can consolidate each group into a single row in the new file. Move variables that identify case groups in the current file into the Identifier Variable(s) list. Variables that are used to split the current data file are automatically used to identify case groups. Each time a new combination of identification values is encountered, the wizard will create a new row, so cases in the current file should be sorted by values of the identification variables in the same order that variables are listed in the Identifier Variable(s) list. If the current data file isn't already sorted, you can sort it in the next step.

How should the new variable groups be created in the new file? In the original data, a variable appears in a single column. In the new data file, that variable will appear in multiple new columns. **Index variables** are variables in the current data that the wizard should use to create the new columns. The restructured data will contain one new variable for each unique value in these columns. Move the variables that should be used to form the new variable groups to the Index Variable(s) list. When the wizard presents options, you can also choose to order the new columns by index.

What happens to the other columns? The wizard automatically decides what to do with the variables that remain in the Current File list. It checks each variable to see if the data values vary within a case group. If they do, the wizard restructures the values into a variable group in the new file. If they don't, the wizard copies the values into the new file. When determining if a variable varies within a group, user-missing values are treated like valid values, but system-missing values are not. If the group contains

one valid or use-missing value plus the system-missing value, then it is treated as a variable that does not vary within the group, and the wizard copies the values into the new file.

Restructure Data Wizard (Cases to Variables): Sort Data

Note: The wizard presents this step if you choose to restructure case groups into columns.

In this step, decide whether to sort the current file before restructuring it. Each time the wizard encounters a new combination of identification values, a new row is created, so it is important that the data are sorted by the variables that identify case groups.

How are the rows ordered in the current file? Consider how the current data are sorted and which variables you are using to identify case groups (specified in the previous step).

- **Yes.** The wizard will automatically sort the current data by the identification variables in the same order that variables are listed in the Identifier Variable(s) list in the previous step. Choose this when the data aren't sorted by the identification variables or when you aren't sure. This choice requires a separate pass of the data, but it guarantees that the rows are correctly ordered for restructuring.
- **No.** The wizard will not sort the current data. Choose this when you are sure that the current data are sorted by the variables that identify case groups.

Restructure Data Wizard (Cases to Variables): Options

Note: The wizard presents this step if you choose to restructure case groups into columns.

In this step, specify options for the new, restructured file.

How should the new variable groups be ordered in the new file?

- **By variable.** The wizard groups the new variables created from an original variable together.
- **By index.** The wizard groups the variables according to the values of the index variables.

Example. The variables to be restructured are *w* and *h*, and the index is *month*:

w, h, month

Grouping by variable results in:

w.jan, w.feb, h.jan

Grouping by index results in:

w.jan, h.jan, w.feb

Create a count variable? The wizard can create a count variable in the new file. It contains the number of rows in the current data that were used to create a row in the new data file.

Create indicator variables? The wizard can use the index variables to create **indicator variables** in the new data file. It creates one new variable for each unique value of the index variable. The indicator variables signal the presence or absence of a value for a case. An indicator variable has the value of 1 if the case has a value; otherwise, it is 0.

Example. The index variable is *product*. It records the products that a customer purchased. The original data are:

Table 20. All products in a single variable (column)

customer	product
1	chick
1	eggs
2	eggs
3	chick

Creating an indicator variable results in one new variable for each unique value of *product*. The restructured data are:

Table 21. Separate indicator variables for each product type

customer	indchick	indeggs
1	1	1
2	0	1
3	1	0

In this example, the restructured data could be used to get frequency counts of the products that customers buy.

Restructure Data Wizard: Finish

This is the final step of the Restructure Data Wizard. Decide what to do with your specifications.

- **Restructure now.** The wizard will create the new, restructured file. Choose this if you want to replace the current file immediately.

Note: If original data are weighted, the new data will be weighted unless the variable that is used as the weight is restructured or dropped from the new file.

- **Paste syntax.** The wizard will paste the syntax it generates into a syntax window. Choose this when you are not ready to replace the current file, when you want to modify the syntax, or when you want to save it for future use.

Chapter 10. Working with output

Working with output

When you run a procedure, the results are displayed in a window called the Viewer. In this window, you can easily navigate to the output that you want to see. You can also manipulate the output and create a document that contains precisely the output that you want.

Viewer

Results are displayed in the Viewer. You can use the Viewer to:

- Browse results
- Show or hide selected tables and charts
- Change the display order of results by moving selected items
- Move items between the Viewer and other applications

The Viewer is divided into two panes:

- The left pane contains an outline view of the contents.
- The right pane contains statistical tables, charts, and text output.

You can click an item in the outline to go directly to the corresponding table or chart. You can click and drag the right border of the outline pane to change the width of the outline pane.

Showing and hiding results

In the Viewer, you can selectively show and hide individual tables or results from an entire procedure. This process is useful when you want to shorten the amount of visible output in the contents pane.

To hide tables and charts

1. Double-click the item's book icon in the outline pane of the Viewer.
or
2. Click the item to select it.
3. From the menus choose:
View > Hide
or
4. Click the closed book (Hide) icon on the Outlining toolbar.

The open book (Show) icon becomes the active icon, indicating that the item is now hidden.

To hide procedure results

1. Click the box to the left of the procedure name in the outline pane.

This hides all results from the procedure and collapses the outline view.

Moving, deleting, and copying output

You can rearrange the results by copying, moving, or deleting an item or a group of items.

To move output in the Viewer

1. Select the items in the outline or contents pane.
2. Drag and drop the selected items into a different location.

To delete output in the Viewer

1. Select the items in the outline or contents pane.
2. Press the **Delete** key.
or
3. From the menus choose:

Edit > Delete

Changing initial alignment

By default, all results are initially left-aligned. To change the initial alignment of new output items:

1. From the menus choose:
Edit > Options
2. Click the **Viewer** tab.
3. In the Initial Output State group, select the item type (for example, pivot table, chart, text output).
4. Select the alignment option you want.

Changing alignment of output items

1. In the outline or contents pane, select the items that you want to align.
2. From the menus choose:
Format > Align Left
or
Format > Center
or
Format > Align Right

Viewer outline

The outline pane provides a table of contents of the Viewer document. You can use the outline pane to navigate through your results and control the display. Most actions in the outline pane have a corresponding effect on the contents pane.

- Selecting an item in the outline pane displays the corresponding item in the contents pane.
- Moving an item in the outline pane moves the corresponding item in the contents pane.
- Collapsing the outline view hides the results from all items in the collapsed levels.

Controlling the outline display. To control the outline display, you can:

- Expand and collapse the outline view
- Change the outline level for selected items
- Change the size of items in the outline display
- Change the font that is used in the outline display

To collapse and expand the outline view

1. Click the box to the left of the outline item that you want to collapse or expand.
or
2. Click the item in the outline.
3. From the menus choose:
View > Collapse
or

View > Expand

To change the outline level

1. Click the item in the outline pane.
2. From the menus choose:
Edit > Outline > Promote
or
Edit > Outline > Demote

To change the size of outline items

1. From the menus choose:
View > Outline Size
2. Select the outline size (**Small**, **Medium**, or **Large**).

To change the font in the outline

1. From the menus choose:
View > Outline Font...
2. Select a font.

Adding items to the Viewer

In the Viewer, you can add items such as titles, new text, charts, or material from other applications.

To add a title or text

Text items that are not connected to a table or chart can be added to the Viewer.

1. Click the table, chart, or other object that will precede the title or text.
2. From the menus choose:
Insert > New Title
or
Insert > New Text
3. Double-click the new object.
4. Enter the text.

To add a text file

1. In the outline pane or contents pane of the Viewer, click the table, chart, or other object that will precede the text.
2. From the menus choose:
Insert > Text File...
3. Select a text file.

To edit the text, double-click it.

Pasting Objects into the Viewer

Objects from other applications can be pasted into the Viewer. You can use either Paste After or Paste Special. Either type of pasting puts the new object after the currently selected object in the Viewer. Use Paste Special when you want to choose the format of the pasted object.

Finding and replacing information in the Viewer

1. To find or replace information in the Viewer, from the menus choose:
Edit > Find
or
Edit > Replace

You can use Find and Replace to:

- Search the entire document or just the selected items.
- Search down or up from the current location.
- Search both panes or restrict the search to the contents or outline pane.
- Search for hidden items. These include any items hidden in the contents pane (for example, Notes tables, which are hidden by default) and hidden rows and columns in pivot tables.
- Restrict the search criteria to case-sensitive matches.
- Restrict the search criteria in pivot tables to matches of the entire cell contents.
- Restrict the search criteria in pivot tables to footnote markers only. This option is not available if the selection in the Viewer includes anything other than pivot tables.

Hidden Items and Pivot Table Layers

- Layers beneath the currently visible layer of a multidimensional pivot table are not considered hidden and will be included in the search area even when hidden items are not included in the search.
- Hidden items include hidden items in the contents pane (items with closed book icons in the outline pane or included within collapsed blocks of the outline pane) and rows and columns in pivot tables either hidden by default (for example, empty rows and columns are hidden by default) or manually hidden by editing the table and selectively hiding specific rows or columns. Hidden items are only included in the search if you explicitly select **Include hidden items**.
- In both cases, the hidden or nonvisible element that contains the search text or value is displayed when it is found, but the item is returned to its original state afterward.

Finding a range of values in pivot tables

To find values that fall within a specified range of values in pivot tables:

1. Activate a pivot table or select one or more pivot tables in the Viewer. Make sure that only pivot tables are selected. If any other objects are selected, the Range option is not available.
2. From the menus choose:
Edit > Find
3. Click the **Range** tab.
4. Select the type of range: Between, Greater than or equal to, or Less than or equal to.
5. Select the value or values that define the range.
 - If either value contains non-numeric characters, both values are treated as strings.
 - If both values are numbers, only numeric values are searched.
 - You cannot use the Range tab to replace values.

This feature is not available for legacy tables. See the topic “Legacy tables” on page 149 for more information.

Closing output items

You can close all currently open output items that were opened from a particular Viewer window.

1. Select the Viewer window that you want.
2. From the menus, choose:
Window > Close output items

Copying output into other applications

Output objects can be copied and pasted into other applications, such as a word-processing program or a spreadsheet. You can paste output in a variety of formats. Depending on the target application and the selected output object(s), some or all of the following formats may be available:

Metafile. WMF and EMF metafile format. These formats are available only on Windows operating systems.

RTF (rich text format). Multiple selected objects, text output, and pivot tables can be copied and pasted in RTF format. For pivot tables, in most applications this means that the tables are pasted as tables that can then be edited in the other application. Pivot tables that are too wide for the document width will either be wrapped, scaled down to fit the document width, or left unchanged, depending on the pivot table options settings. See the topic “Pivot table options” on page 203 for more information.

Note: Microsoft Word may not display extremely wide tables properly.

Image. JPG and PNG image formats.

BIFF. Pivot tables and text output can be pasted into a spreadsheet in BIFF format. Numbers in pivot tables retain numeric precision. This format is available only on Windows operating systems.

Text. Pivot tables and text output can be copied and pasted as text. This process can be useful for applications such as e-mail, where the application can accept or transmit only text.

Microsoft Office Graphic Object. Charts that support this format can be copied to Microsoft Office applications and edited in those applications as native Microsoft Office charts. Because of differences between SPSS Statistics/SPSS Modeler charts and Microsoft Office charts, some features of SPSS Statistics/SPSS Modeler charts are not retained in the copied version. Copying multiple selected charts in Microsoft Office Graphic Object format is not supported.

If the target application supports multiple available formats, it may have a Paste Special menu item that allows you to select the format, or it may automatically display a list of available formats.

Copying and pasting multiple output objects

The following limitations apply when pasting multiple output objects into other applications:

- **RTF format.** In most applications, pivot tables are pasted as tables that can be edited in that application. Charts, trees, and model views are pasted as images.
- **Metafile and image formats.** All the selected output objects are pasted as a single object in the other application.
- **BIFF format.** Charts, trees, and model views are excluded.

You can also use “Export output” on page 126 to export multiple output objects to other applications/formats.

Copy special

When copying and pasting large amounts of output, particularly very large pivot tables, you can improve the speed of the operation by using **Edit > Copy Special** to limit the number of formats copied to the clipboard.

You can also save the selected formats as the default set of formats to copy to the clipboard. This setting persists across sessions.

Copy as

You can right-click a selected object in the Output Viewer and select **Edit > Copy as** to copy to the most popular formats (for example, **All**, **Image**, or **Microsoft Office Graphic Object**). Selecting **Edit > Copy** copies **All**. Note that if **Copy as** is grayed out or not present for a selected object, this copy format is not available for that particular object.

Interactive output

Interactive output objects contain multiple, related output objects. The selection in one object can change what is displayed or highlighted in the other object. For example, selecting a row in a table might highlight an area in a map or display a chart for a different category.

Interactive output objects do not support editing features, such as changing text, colors, fonts, or table borders. The individual objects can be copied from the interactive object to the Viewer. Tables copied from interactive output can be edited in the pivot table editor.

Copying objects from interactive output

File>Copy to Viewer copies individual output objects to the Viewer window.

- The available options depend on the contents of the interactive output.
- **Chart and Map** create chart objects.
- **Table** creates a pivot table that can be edited in the pivot table editor.
- **Snapshot** creates an image of the current view.
- **Model** creates a copy of the current interactive output object.

Edit>Copy Object copies individual output objects to the clipboard.

- Pasting the copied object into the Viewer is equivalent to **File>Copy to Viewer**.
- Pasting the object into another application pastes the object as an image.

Zoom and Pan

For maps, you can use **View>Zoom** to zoom the view of the map. Within a zoomed map view, you can use **View>Pan** to move the view.

Print settings

File>Print Settings controls how interactive objects are printed.

- **Print visible view only.** Prints only the view that is currently displayed. This option is the default setting.
- **Print all views.** Prints all views contained in the interactive output.
- The selected option also determines the default action for exporting the output object.

Export output

Export Output saves Viewer output in HTML, text, Word/RTF, Excel, PowerPoint (requires PowerPoint 97 or later), and PDF formats. Charts can also be exported in a number of different graphics formats.

Note: Export to PowerPoint is available only on Windows operating systems and is not available in the Student Version.

To export output

1. Make the Viewer the active window (click anywhere in the window).
2. From the menus choose:
File > Export...
3. Enter a file name (or prefix for charts) and select an export format.

Objects to Export. You can export all objects in the Viewer, all visible objects, or only selected objects.

Document Type. The available options are:

- **Word/RTF.** Pivot tables are exported as Word tables with all formatting attributes intact (for example, cell borders, font styles, and background colors). Text output is exported as formatted RTF. Charts, tree diagrams, and model views are included in PNG format. Note that Microsoft Word might not display extremely wide tables properly.
- **Excel.** Pivot table rows, columns, and cells are exported as Excel rows, columns, and cells, with all formatting attributes intact (for example, cell borders, font styles, and background colors). Text output is exported with all font attributes intact. Each line in the text output is a row in the Excel file, with the entire contents of the line in a single cell. Charts, tree diagrams, and model views are included in PNG format. Output can be exported as *Excel 97-2004* or *Excel 2007 and higher*.
- **HTML.** Pivot tables are exported as HTML tables. Text output is exported as preformatted HTML. Charts, tree diagrams, and model views are embedded in the document in the selected graphic format. A browser compatible with HTML 5 is required for viewing output that is exported in HTML format.
- **Web Report.** A web report is an interactive document that is compatible with most browsers. Many of the interactive features of pivot tables available in the Viewer are also available in web reports. You can also export a web report as an IBM Cognos Active Report.
- **Portable Document Format.** All output is exported as it appears in Print Preview, with all formatting attributes intact.
- **PowerPoint file.** Pivot tables are exported as Word tables and are embedded on separate slides in the PowerPoint file, with one slide for each pivot table. All formatting attributes of the pivot table are retained (for example, cell borders, font styles, and background colors). Charts, tree diagrams, and model views are exported in TIFF format. Text output is not included.
Export to PowerPoint is available only on Windows operating systems.
- **Text.** Text output formats include plain text, UTF-8, and UTF-16. Pivot tables can be exported in tab-separated or space-separated format. All text output is exported in space-separated format. For charts, tree diagrams, and model views, a line is inserted in the text file for each graphic, indicating the image file name.
- **None (Graphics Only).** Available export formats include: EPS, JPEG, TIFF, PNG, and BMP. On Windows operating systems, EMF (enhanced metafile) format is also available.

Open the containing folder. Opens the folder that contains the files that are created by the export.

Output Management System. You can also automatically export all output or user-specified types of output as Word, Excel, PDF, HTML, text, or IBM SPSS Statistics format data files. See Chapter 22, “Output Management System,” on page 263 for more information.

HTML options

HTML export requires a browser that is compatible with HTML 5. The following options are available for exporting output in HTML format:

Layers in pivot tables. By default, inclusion or exclusion of pivot table layers is controlled by the table properties for each pivot table. You can override this setting and include all layers or exclude all but the currently visible layer. See the topic “Table properties: printing” on page 144 for more information.

Export layered tables as interactive. Layered tables are displayed as they appear in the Viewer, and you can interactively change the displayed layer in the browser. If this option is not selected, each table layer is displayed as a separate table.

Tables as HTML. This controls style information included for exported pivot tables.

- **Export with styles and fixed column width.** All pivot table style information (font styles, background colors, etc.) and column widths are preserved.
- **Export without styles.** Pivot tables are converted to default HTML tables. No style attributes are preserved. Column width is determined automatically.

Include footnotes and captions. Controls the inclusion or exclusion of all pivot table footnotes and captions.

Views of Models. By default, inclusion or exclusion of model views is controlled by the model properties for each model. You can override this setting and include all views or exclude all but the currently visible view. See the topic “Model properties” on page 152 for more information. (Note: all model views, including tables, are exported as graphics.)

Note: For HTML, you can also control the image file format for exported charts. See the topic “Graphics format options” on page 132 for more information.

To set HTML export options

1. Select **HTML** as the export format.
2. Click **Change Options**.

Web report options

A web report is an interactive document that is compatible with most browsers. Many of the interactive features of pivot tables available in the Viewer are also available in web reports.

Report Title. The title that is displayed in the header of the report. By default, the file name is used. You can specify a custom title to use instead of the file name.

Format. There are two options for report format:

- **SPSS Web Report (HTML 5).** This format requires a browser that is compatible with HTML 5.
- **Cognos Active Report (mht).** This format requires a browser that supports MHT format files or the Cognos Active Report application.

Exclude Objects. You can exclude selected object types from the report:

- **Text.** Text objects that are not logs. This option includes text objects that contain information about the active dataset.
- **Logs.** Text objects that contain a listing of the command syntax that was run. Log items also include warnings and error messages that are encountered by commands that do not produce any Viewer output.
- **Notes Tables.** Output from statistical and charting procedures includes a Notes table. This table contains information about the dataset that was used, missing values, and the command syntax that was used to run the procedure.
- **Warnings and Error Messages.** Warnings and error messages from statistical and charting procedures.

Restyle the tables and charts to match the Web Report. This option applies the standard web report style to all tables and charts. This setting overrides any fonts, colors, or other styles in the output as displayed in the Viewer. You cannot modify the standard web report style.

Web Server Connection. You can include the URL location of one or more application servers that are running the IBM SPSS Statistics Web Report Application Server. The web application server provides features to pivot tables, edit charts, and save modified web reports.

- Select **Use** for each application server that you want to include in the web report.
- If a web report contains a URL specification, the web report connects to that application server to provide the additional editing features.
- If you specify multiple URLs, the web report attempts to connect to each server in the order in which they are specified.

The IBM SPSS Statistics Web Report Application Server can be downloaded from <http://www.ibm.com/developerworks/spssdevcentral>.

Word/RTF options

The following options are available for exporting output in Word format:

Layers in pivot tables. By default, inclusion or exclusion of pivot table layers is controlled by the table properties for each pivot table. You can override this setting and include all layers or exclude all but the currently visible layer. See the topic “Table properties: printing” on page 144 for more information.

Wide Pivot Tables. Controls the treatment of tables that are too wide for the defined document width. By default, the table is wrapped to fit. The table is divided into sections, and row labels are repeated for each section of the table. Alternatively, you can shrink wide tables or make no changes to wide tables and allow them to extend beyond the defined document width.

Preserve break points. If you have defined break points, these settings will be preserved in the Word tables.

Include footnotes and captions. Controls the inclusion or exclusion of all pivot table footnotes and captions.

Views of Models. By default, inclusion or exclusion of model views is controlled by the model properties for each model. You can override this setting and include all views or exclude all but the currently visible view. See the topic “Model properties” on page 152 for more information. (Note: all model views, including tables, are exported as graphics.)

Page Setup for Export. This opens a dialog where you can define the page size and margins for the exported document. The document width used to determine wrapping and shrinking behavior is the page width minus the left and right margins.

To set Word export options

1. Select **Word/RTF** as the export format.
2. Click **Change Options**.

Excel options

The following options are available for exporting output in Excel format:

Create a worksheet or workbook or modify an existing worksheet. By default, a new workbook is created. If a file with the specified name already exists, it will be overwritten. If you select the option to create a worksheet, if a worksheet with the specified name already exists in the specified file, it will be overwritten. If you select the option to modify an existing worksheet, you must also specify the worksheet name. (This is optional for creating a worksheet.) Worksheet names cannot exceed 31 characters and cannot contain forward or back slashes, square brackets, question marks, or asterisks.

When exporting to Excel 97-2004, if you modify an existing worksheet, charts, model views, and tree diagrams are not included in the exported output.

Location in worksheet. Controls the location within the worksheet for the exported output. By default, exported output will be added after the last column that has any content, starting in the first row, without modifying any existing contents. This is a good choice for adding new columns to an existing worksheet. Adding exported output after the last row is a good choice for adding new rows to an existing worksheet. Adding exported output starting at a specific cell location will overwrite any existing content in the area where the exported output is added.

Layers in pivot tables. By default, inclusion or exclusion of pivot table layers is controlled by the table properties for each pivot table. You can override this setting and include all layers or exclude all but the currently visible layer. See the topic “Table properties: printing” on page 144 for more information.

Include footnotes and captions. Controls the inclusion or exclusion of all pivot table footnotes and captions.

Views of Models. By default, inclusion or exclusion of model views is controlled by the model properties for each model. You can override this setting and include all views or exclude all but the currently visible view. See the topic “Model properties” on page 152 for more information. (Note: all model views, including tables, are exported as graphics.)

To set Excel export options

1. Select **Excel** as the export format.
2. Click **Change Options**.

PowerPoint options

The following options are available for PowerPoint:

Layers in pivot tables. By default, inclusion or exclusion of pivot table layers is controlled by the table properties for each pivot table. You can override this setting and include all layers or exclude all but the currently visible layer. See the topic “Table properties: printing” on page 144 for more information.

Wide Pivot Tables. Controls the treatment of tables that are too wide for the defined document width. By default, the table is wrapped to fit. The table is divided into sections, and row labels are repeated for each section of the table. Alternatively, you can shrink wide tables or make no changes to wide tables and allow them to extend beyond the defined document width.

Include footnotes and captions. Controls the inclusion or exclusion of all pivot table footnotes and captions.

Use Viewer outline entries as slide titles. Includes a title on each slide that is created by the export. Each slide contains a single item that is exported from the Viewer. The title is formed from the outline entry for the item in the outline pane of the Viewer.

Views of Models. By default, inclusion or exclusion of model views is controlled by the model properties for each model. You can override this setting and include all views or exclude all but the currently visible view. See the topic “Model properties” on page 152 for more information. (Note: all model views, including tables, are exported as graphics.)

Page Setup for Export. This opens a dialog where you can define the page size and margins for the exported document. The document width used to determine wrapping and shrinking behavior is the page width minus the left and right margins.

To set PowerPoint export options

1. Select **PowerPoint** as the export format.
2. Click **Change Options**.

Note: Export to PowerPoint is available only on Windows operating systems.

PDF options

The following options are available for PDF:

Embed bookmarks. This option includes bookmarks in the PDF document that correspond to the Viewer outline entries. Like the Viewer outline pane, bookmarks can make it much easier to navigate documents with a large number of output objects.

Embed fonts. Embedding fonts ensures that the PDF document will look the same on all computers. Otherwise, if some fonts used in the document are not available on the computer being used to view (or print) the PDF document, font substitution may yield suboptimal results.

Layers in pivot tables. By default, inclusion or exclusion of pivot table layers is controlled by the table properties for each pivot table. You can override this setting and include all layers or exclude all but the currently visible layer. See the topic “Table properties: printing” on page 144 for more information.

Views of Models. By default, inclusion or exclusion of model views is controlled by the model properties for each model. You can override this setting and include all views or exclude all but the currently visible view. See the topic “Model properties” on page 152 for more information. (Note: all model views, including tables, are exported as graphics.)

To set PDF export options

1. Select **Portable Document Format** as the export format.
2. Click **Change Options**.

Other Settings That Affect PDF Output

Page Setup/Page Attributes. Page size, orientation, margins, content and display of page headers and footers, and printed chart size in PDF documents are controlled by page setup and page attribute options.

Table Properties/TableLooks. Scaling of wide and/or long tables and printing of table layers are controlled by table properties for each table. These properties can also be saved in TableLooks.

Default/Current Printer. The resolution (DPI) of the PDF document is the current resolution setting for the default or currently selected printer (which can be changed using Page Setup). The maximum resolution is 1200 DPI. If the printer setting is higher, the PDF document resolution will be 1200 DPI.

Note: High-resolution documents may yield poor results when printed on lower-resolution printers.

Text options

The following options are available for text export:

Pivot Table Format. Pivot tables can be exported in tab-separated or space-separated format. For space-separated format, you can also control:

- **Column Width.** **Autofit** does not wrap any column contents, and each column is as wide as the widest label or value in that column. **Custom** sets a maximum column width that is applied to all columns in the table, and values that exceed that width wrap onto the next line in that column.
- **Row/Column Border Character.** Controls the characters used to create row and column borders. To suppress display of row and column borders, enter blank spaces for the values.

Layers in pivot tables. By default, inclusion or exclusion of pivot table layers is controlled by the table properties for each pivot table. You can override this setting and include all layers or exclude all but the currently visible layer. See the topic “Table properties: printing” on page 144 for more information.

Include footnotes and captions. Controls the inclusion or exclusion of all pivot table footnotes and captions.

Views of Models. By default, inclusion or exclusion of model views is controlled by the model properties for each model. You can override this setting and include all views or exclude all but the currently visible view. See the topic “Model properties” on page 152 for more information. (Note: all model views, including tables, are exported as graphics.)

To set text export options

1. Select **Text** as the export format.
2. Click **Change Options**.

Graphics only options

The following options are available for exporting graphics only:

Views of Models. By default, inclusion or exclusion of model views is controlled by the model properties for each model. You can override this setting and include all views or exclude all but the currently visible view. See the topic “Model properties” on page 152 for more information. (Note: all model views, including tables, are exported as graphics.)

Graphics format options

For HTML and text documents and for exporting charts only, you can select the graphic format, and for each graphic format you can control various optional settings.

To select the graphic format and options for exported charts:

1. Select **HTML**, **Text**, or **None (Graphics only)** as the document type.
2. Select the graphic file format from the drop-down list.
3. Click **Change Options** to change the options for the selected graphic file format.

JPEG Chart Export Options

- **Image size.** Percentage of original chart size, up to 200 percent.
- **Convert to grayscale.** Converts colors to shades of gray.

BMP chart export options

- **Image size.** Percentage of original chart size, up to 200 percent.
- **Compress image to reduce file size.** A lossless compression technique that creates smaller files without affecting image quality.

PNG chart export options

Image size. Percentage of original chart size, up to 200 percent.

Color Depth. Determines the number of colors in the exported chart. A chart that is saved under any depth will have a minimum of the number of colors that are actually used and a maximum of the number of colors that are allowed by the depth. For example, if the chart contains three colors--red, white, and black--and you save it as 16 colors, the chart will remain as three colors.

- If the number of colors in the chart exceeds the number of colors for that depth, the colors will be dithered to replicate the colors in the chart.
- **Current screen depth** is the number of colors currently displayed on your computer monitor.

EMF and TIFF chart export options

Image size. Percentage of original chart size, up to 200 percent.

Note: EMF (enhanced metafile) format is available only on Windows operating systems.

EPS chart export options

Image size. You can specify the size as a percentage of the original image size (up to 200 percent), or you can specify an image width in pixels (with height determined by the width value and the aspect ratio). The exported image is always proportional to the original.

Include TIFF preview image. Saves a preview with the EPS image in TIFF format for display in applications that cannot display EPS images on screen.

Fonts. Controls the treatment of fonts in EPS images.

- **Use font references.** If the fonts that are used in the chart are available on the output device, the fonts are used. Otherwise, the output device uses alternate fonts.
- **Replace fonts with curves.** Turns fonts into PostScript curve data. The text itself is no longer editable as text in applications that can edit EPS graphics. This option is useful if the fonts that are used in the chart are not available on the output device.

Viewer printing

There are two options for printing the contents of the Viewer window:

All visible output. Prints only items that are currently displayed in the contents pane. Hidden items (items with a closed book icon in the outline pane or hidden in collapsed outline layers) are not printed.

Selection. Prints only items that are currently selected in the outline and/or contents panes.

To print output and charts

1. Make the Viewer the active window (click anywhere in the window).
2. From the menus choose:
 File > Print...
3. Select the print settings that you want.
4. Click **OK** to print.

Print Preview

Print Preview shows you what will print on each page for Viewer documents. It is a good idea to check Print Preview before actually printing a Viewer document, because Print Preview shows you items that may not be visible by looking at the contents pane of the Viewer, including:

- Page breaks
- Hidden layers of pivot tables
- Breaks in wide tables
- Headers and footers that are printed on each page

If any output is currently selected in the Viewer, the preview displays only the selected output. To view a preview for all output, make sure nothing is selected in the Viewer.

Page Attributes: Headers and Footers

Headers and footers are the information that is printed at the top and bottom of each page. You can enter any text that you want to use as headers and footers. You can also use the toolbar in the middle of the dialog box to insert:

- Date and time
- Page numbers
- Viewer filename
- Outline heading labels
- Page titles and subtitles
- **Make Default** uses the settings specified here as the default settings for new Viewer documents. (Note: this makes the current settings on both the Header/Footer tab and the Options tab the default settings.)

- Outline heading labels indicate the first-, second-, third-, and/or fourth-level outline heading for the first item on each page.
- Page titles and subtitles print the current page titles and subtitles. These can be created with New Page Title on the Viewer Insert menu or with the TITLE and SUBTITLE commands. If you have not specified any page titles or subtitles, this setting is ignored.

Note: Font characteristics for new page titles and subtitles are controlled on the Viewer tab of the Options dialog box (accessed by choosing Options on the Edit menu). Font characteristics for existing page titles and subtitles can be changed by editing the titles in the Viewer.

To see how your headers and footers will look on the printed page, choose Print Preview from the File menu.

To insert page headers and footers

1. Make the Viewer the active window (click anywhere in the window).
2. From the menus, choose:
File > Page Attributes...
3. Click the **Header/Footer** tab.
4. Enter the header and/or footer that you want to appear on each page.

Page Attributes: Options

This dialog box controls the printed chart size, the space between printed output items, and page numbering.

- **Printed Chart Size.** Controls the size of the printed chart relative to the defined page size. The chart's aspect ratio (width-to-height ratio) is not affected by the printed chart size. The overall printed size of a chart is limited by both its height and width. When the outer borders of a chart reach the left and right borders of the page, the chart size cannot increase further to fill more page height.
- **Space between items.** Controls the space between printed items. Each pivot table, chart, and text object is a separate item. This setting does not affect the display of items in the Viewer.
- **Number pages starting with.** Numbers pages sequentially, starting with the specified number.
- **Make Default.** This option uses the settings that are specified here as the default settings for new Viewer documents. (Note that this option makes the current Header/Footer settings and Options settings the default.)

To change printed chart size, page numbering, and space between printed Items

1. Make the Viewer the active window (click anywhere in the window).
2. From the menus choose:
File > Page Attributes...
3. Click the **Options** tab.
4. Change the settings and click **OK**.

Saving output

The contents of the Viewer can be saved to several formats:

- **Viewer files (*.spv).** The format that is used to display files in the Viewer window.
- **SPSS Web Report (*.htm).** A web report is an interactive document that is compatible with most browsers. Many of the interactive features of pivot tables available in the Viewer are also available in web reports. This format requires a browser that is compatible with HTML 5.
- **Cognos Active Report (*.mht).** This format requires a browser that supports MHT format files or the Cognos Active Report application.

To control options for saving web reports or save results in other formats (for example, text, Word, Excel), use **Export** on the **File** menu.

To save a Viewer document

1. From the Viewer window menus choose:

File > Save

2. Enter the name of the document, and then click **Save**.

Optionally, you can do the following:

Lock files to prevent editing in IBM SPSS Smartreader

If a Viewer document is locked, you can manipulate pivot tables (swap rows and columns, change the displayed layer, etc.) but you cannot edit any output or save any changes to the Viewer document in IBM SPSS Smartreader (a separate product for working with Viewer documents). This setting has no effect on Viewer documents opened in IBM SPSS Statistics or IBM SPSS Modeler.

Encrypt files with a password

You can protect confidential information stored in a Viewer document by encrypting the document with a password. Once encrypted, the document can only be opened by providing the password. IBM SPSS Smartreader users will also be required to provide the password in order to open the file.

To encrypt a Viewer document:

- a. Select **Encrypt file with password** in the Save Output As dialog box.
- b. Click **Save**.
- c. In the Encrypt File dialog box, provide a password and re-enter it in the Confirm password text box. Passwords are limited to 10 characters and are case-sensitive.

Warning: Passwords cannot be recovered if they are lost. If the password is lost the file cannot be opened.

Creating strong passwords

- Use eight or more characters.
- Include numbers, symbols and even punctuation in your password.
- Avoid sequences of numbers or characters, such as "123" and "abc", and avoid repetition, such as "111aaa".
- Do not create passwords that use personal information such as birthdays or nicknames.
- Periodically change the password.

Note: Storing encrypted files to an IBM SPSS Collaboration and Deployment Services Repository is not supported.

Modifying encrypted files

- If you open an encrypted file, make modifications to it and choose **File > Save**, the modified file will be saved with the same password.
- You can change the password on an encrypted file by opening the file, repeating the steps for encrypting it, and specifying a different password in the Encrypt File dialog box.
- You can save an unencrypted version of an encrypted file by opening the file, choosing **File > Save As** and deselecting **Encrypt file with password** in the Save Output As dialog box.

Note: Encrypted data files and output documents cannot be opened in versions of IBM SPSS Statistics prior to version 21. Encrypted syntax files cannot be opened in versions prior to version 22.

Store required model information with the output document

This option applies only when there are model viewer items in the output document that

require auxiliary information to enable some of the interactive features. Click **More Info** to display a list of these model viewer items and the interactive features that require auxiliary information. Storing this information with the output document might substantially increase the document size. If you choose not to store this information, you can still open these output items but the specified interactive features will not be available.

Chapter 11. Pivot tables

Pivot tables

Many results are presented in tables that can be pivoted interactively. That is, you can rearrange the rows, columns, and layers.

Note: If you require tables that are compatible with IBM SPSS Statistics releases prior to 20, then it is recommended to render them as legacy tables. See the topic “Legacy tables” on page 149 for more information.

Manipulating a pivot table

Options for manipulating a pivot table include:

- Transposing rows and columns
- Moving rows and columns
- Creating multidimensional layers
- Grouping and ungrouping rows and columns
- Showing and hiding rows, columns, and other information
- Rotating row and column labels
- Finding definitions of terms

Activating a pivot table

Before you can manipulate or modify a pivot table, you need to **activate** the table. To activate a table:

1. Double-click the table.
or
2. Right-click the table and from the pop-up menu choose **Edit Content**.
3. From the sub-menu choose either **In Viewer** or **In Separate Window**.

Pivoting a table

1. Activate the pivot table.
2. From the menus choose:
Pivot > Pivoting Trays

A table has three dimensions: rows, columns, and layers. A dimension can contain multiple elements (or none at all). You can change the organization of the table by moving elements between or within dimensions. To move an element, just drag and drop it where you want it.

Changing display order of elements within a dimension

To change the display order of elements within a table dimension (row, column, or layer):

1. If pivoting trays are not already on, from the Pivot Table menu choose:
Pivot > Pivoting Trays
2. Drag and drop the elements within the dimension in the pivoting tray.

Moving rows and columns within a dimension element

1. In the table itself (not the pivoting trays), click the label for the row or column you want to move.
2. Drag the label to the new position.

Transposing rows and columns

If you just want to flip the rows and columns, there's a simple alternative to using the pivoting trays:

1. From the menus choose:

Pivot > Transpose Rows and Columns

This has the same effect as dragging all of the row elements into the column dimension and dragging all of the column elements into the row dimension.

Grouping rows or columns

1. Select the labels for the rows or columns that you want to group together (click and drag or Shift+click to select multiple labels).

2. From the menus choose:

Edit > Group

A group label is automatically inserted. Double-click the group label to edit the label text.

Note: To add rows or columns to an existing group, you must first ungroup the items that are currently in the group. Then you can create a new group that includes the additional items.

Ungrouping rows or columns

1. Click anywhere in the group label for the rows or columns that you want to ungroup.

2. From the menus choose:

Edit > Ungroup

Ungrouping automatically deletes the group label.

Rotating row or column labels

You can rotate labels between horizontal and vertical display for the innermost column labels and the outermost row labels in a table.

1. From the menus choose:

Format > Rotate Inner Column Labels

or

Format > Rotate Outer Row Labels

Only the innermost column labels and the outermost row labels can be rotated.

Sorting rows

To sort the rows of a pivot table:

1. Activate the table.
2. Select any cell in the column you want to use to sort on. To sort just a selected group of rows, select two or more contiguous cells in the column you want to use to sort on.

3. From the menus, choose:

Edit > Sort rows

4. Select **Ascending** or **Descending** from the submenu.

- If the row dimension contains groups, sorting affects only the group that contains the selection.
- You cannot sort across group boundaries.
- You cannot sort tables with more than one item in the row dimension.

Note: This feature is not available in legacy tables.

Related information:

"Legacy tables" on page 149

Inserting rows and columns

To insert a row or column in a pivot table:

1. Activate the table.
2. Select any cell in the table.
3. From the menus, choose:

Insert Before

or

Insert After

From the submenu, choose:

Row

or

Column

- A plus sign (+) is inserted in each cell of the new row or column to prevent the new row or column from being automatically hidden because it is empty.
- In a table with nested or layered dimensions, a column or row is inserted at every corresponding dimension level.

Note: This feature is not available in legacy tables.

Related information:

"Legacy tables" on page 149

Controlling display of variable and value labels

If variables contain descriptive variable or value labels, you can control the display of variable names and labels and data values and value labels in pivot tables.

1. Activate the pivot table.
2. From the menus, choose:

View > Variable labels

or

View > Value labels

3. Select one of the follow options from the submenu:

- **Name or Value.** Only variable names (or values) are displayed. Descriptive labels are not displayed.
- **Label.** Only descriptive labels are displayed. Variable names (or values) are not displayed.
- **Both.** Both names (or values) and descriptive labels are displayed.

Note: This feature is not available for legacy tables.

To control the default label display for pivot tables and other output objects, use Edit>Options>Output.

Related information:

"Legacy tables" on page 149

"Output options" on page 200

Changing the output language

To change the output language in a pivot table:

1. Activate the table

2. From the menus, choose:
View > Language
3. Select one of the available languages.

Changing the language affects only text that is generated by the application, such as table titles, row and column labels, and footnote text. Variable names and descriptive variable and value labels are not affected.

Note: This feature is not available for legacy tables.

To control the default language for pivot tables and other output objects, use **Edit>Options>Language**.

Related information:

"Legacy tables" on page 149

"Language options" on page 199

Navigating large tables

To use the navigation window to navigate large tables:

1. Activate the table.
2. From the menus choose:
View > Navigation

Undoing changes

You can undo the most recent change or all changes to an activated pivot table. Both actions apply only to changes made since the most recent activation of the table.

To undo the most recent change:

1. From the menus, choose:
Edit > Undo
To undo all changes:
2. From the menus, choose:
Edit > Restore
- Note: **Edit > Restore** is not available for legacy tables.

Working with layers

You can display a separate two-dimensional table for each category or combination of categories. The table can be thought of as stacked in layers, with only the top layer visible.

Creating and displaying layers

To create layers:

1. Activate the pivot table.
2. If pivoting trays are not already on, from the **Pivot Table** menu, choose:
Pivot > Pivoting Trays
3. Drag an element from the row or column dimension into the layer dimension.

Moving elements to the layer dimension creates a multidimensional table, but only a single two-dimensional "slice" is displayed. The visible table is the table for the top layer. For example, if a yes/no categorical variable is in the layer dimension, then the multidimensional table has two layers: one for the "yes" category and one for the "no" category.

Changing the displayed layer

1. Choose a category from the drop-down list of layers (in the pivot table itself, not the pivoting tray).

Go to layer category

Go to Layer Category allows you to change layers in a pivot table. This dialog box is particularly useful when there are many layers or the selected layer has many categories.

Showing and hiding items

Many types of cells can be hidden, including:

- Dimension labels
- Categories, including the label cell and data cells in a row or column
- Category labels (without hiding the data cells)
- Footnotes, titles, and captions

Hiding rows and columns in a table

Showing hidden rows and columns in a table

1. From the menus choose:

View > Show All Categories

This displays all hidden rows and columns in the table. (If **Hide empty rows and columns** is selected in Table Properties for this table, a completely empty row or column remains hidden.)

Hiding and showing dimension labels

1. Select the dimension label or any category label within the dimension.
2. From the View menu or the pop-up menu choose **Hide Dimension Label** or **Show Dimension Label**.

Hiding and showing table titles

To hide a title:

1. Activate the pivot table.
2. Select the title.
3. From the View menu choose **Hide**.

To show hidden titles:

4. From the View menu choose **Show All**.

TableLooks

A TableLook is a set of properties that define the appearance of a table. You can select a previously defined TableLook or create your own TableLook.

- Before or after a TableLook is applied, you can change cell formats for individual cells or groups of cells by using cell properties. The edited cell formats remain intact, even when you apply a new TableLook.
- Optionally, you can reset all cells to the cell formats that are defined by the current TableLook. This option resets any cells that were edited. If **As Displayed** is selected in the TableLook Files list, any edited cells are reset to the current table properties.
- Only table properties that are defined in the Table Properties dialog are saved in TableLooks. TableLooks do not include individual cell modifications.

Note: TableLooks created in earlier versions of IBM SPSS Statistics cannot be used in version 16.0 or later.

To apply a TableLook

1. Activate a pivot table.
2. From the menus, choose:
Format > TableLooks...
3. Select a TableLook from the list of files. To select a file from another directory, click **Browse**.
4. Click **OK** to apply the TableLook to the selected pivot table.

To edit or create a TableLook

1. In the TableLooks dialog box, select a TableLook from the list of files.
2. Click **Edit Look**.
3. Adjust the table properties for the attributes that you want, and then click **OK**.
4. Click **Save Look** to save the edited TableLook, or click **Save As** to save it as a new TableLook.
 - Editing a TableLook affects only the selected pivot table. An edited TableLook is not applied to any other tables that use that TableLook unless you select those tables and reapply the TableLook.
 - Only table properties that are defined in the Table Properties dialog are saved in TableLooks. TableLooks do not include individual cell modifications.

Related information:

Detailed examples

Table properties

Table Properties allows you to set general properties of a table, set cell styles for various parts of a table, and save a set of those properties as a TableLook. You can:

- Control general properties, such as hiding empty rows or columns and adjusting printing properties.
- Control the format and position of footnote markers.
- Determine specific formats for cells in the data area, for row and column labels, and for other areas of the table.
- Control the width and color of the lines that form the borders of each area of the table.

To change pivot table properties

1. Activate the pivot table.
2. From the menus choose:
Format > Table Properties...
3. Select a tab (**General**, **Footnotes**, **Cell Formats**, **Borders**, or **Printing**).
4. Select the options that you want.
5. Click **OK** or **Apply**.

The new properties are applied to the selected pivot table. To apply new table properties to a TableLook instead of just the selected table, edit the TableLook (Format menu, TableLooks).

Table properties: general

Several properties apply to the table as a whole. You can:

- Show or hide empty rows and columns. (An empty row or column has nothing in any of the data cells.)

- Control the default number of rows to display in long tables. To display all the rows in a table, regardless of how long it is, clear **Display table by rows**. Note: This feature applies only to legacy tables.
- Control the placement of row labels, which can be in the upper left corner or nested.
- Control maximum and minimum column width (expressed in points).

To change general table properties:

1. Click the **General** tab.
2. Select the options that you want.
3. Click **OK** or **Apply**.

Set rows to display

Note: This feature only applies to legacy tables.

By default, tables with many rows are displayed in sections of 100 rows. To control the number of rows displayed in a table:

1. Select **Display table by rows**.
2. Click **Set Rows to Display**.
- or
3. From the View menu of an activated pivot table, choose **Display table by rows** and **Set Rows to Display**.

Rows to display. Controls the maximum number of rows to display at one time. Navigation controls allow you move to different sections of the table. The minimum value is 10. The default is 100.

Widow/orphan tolerance. Controls the maximum number of rows of the inner most row dimension of the table to split across displayed views of the table. For example, if there are six categories in each group of the inner most row dimension, specifying a value of six would prevent any group from splitting across displayed views. This setting can cause the total number of rows in a displayed view to exceed the specified maximum number of rows to display.

Table properties: notes

The Notes tab of the Table Properties dialog controls footnote formatting and table comment text.

Footnotes. The properties of footnote markers include style and position in relation to text.

- The style of footnote markers is either numbers (1, 2, 3, ...) or letters (a, b, c, ...).
- The footnote markers can be attached to text as superscripts or subscripts.

Comment Text. You can add comment text to each table.

- Comment text is displayed in a tooltip when you hover over a table in the Viewer.
- Screen readers read the comment text when the table has focus.
- The tooltip in the Viewer displays only the first 200 characters of the comment, but screen readers read the entire text.
- When you export output to HTML or a web report, the comment text is used as alt text.

You can automatically add comments to all tables when they are created (Edit menu>Options>Pivot Tables).

Table properties: cell formats

For formatting, a table is divided into areas: title, layers, corner labels, row labels, column labels, data, caption, and footnotes. For each area of a table, you can modify the associated cell formats. Cell formats include text characteristics (such as font, size, color, and style), horizontal and vertical alignment, background colors, and inner cell margins.

Cell formats are applied to areas (categories of information). They are not characteristics of individual cells. This distinction is an important consideration when pivoting a table.

For example,

- If you specify a bold font as a cell format of column labels, the column labels will appear bold no matter what information is currently displayed in the column dimension. If you move an item from the column dimension to another dimension, it does not retain the bold characteristic of the column labels.
- If you make column labels bold simply by highlighting the cells in an activated pivot table and clicking the Bold button on the toolbar, the contents of those cells will remain bold no matter what dimension you move them to, and the column labels will not retain the bold characteristic for other items moved into the column dimension.

To change cell formats:

1. Select the **Cell Formats** tab.
2. Select an Area from the drop-down list or click an area of the sample.
3. Select characteristics for the area. Your selections are reflected in the sample.
4. Click **OK** or **Apply**.

Alternating row colors

To apply a different background and/or text color to alternate rows in the Data area of the table:

1. Select **Data** from the Area drop-down list.
2. Select (check) **Alternate row color** in the Background Color group.
3. Select the colors to use for the alternate row background and text.

Alternate row colors affect only the Data area of the table. They do not affect row or column label areas.

Table properties: borders

For each border location in a table, you can select a line style and a color. If you select **None** as the style, there will be no line at the selected location.

To change table borders:

1. Click the **Borders** tab.
2. Select a border location, either by clicking its name in the list or by clicking a line in the Sample area.
3. Select a line style or select **None**.
4. Select a color.
5. Click **OK** or **Apply**.

Table properties: printing

You can control the following properties for printed pivot tables:

- Print all layers or only the top layer of the table, and print each layer on a separate page.
- Shrink a table horizontally or vertically to fit the page for printing.

- Control widow/orphan lines by controlling the minimum number of rows and columns that will be contained in any printed section of a table if the table is too wide and/or too long for the defined page size.
Note: If a table is too long to fit on the current page because there is other output above it, but it will fit within the defined page length, the table is automatically printed on a new page, regardless of the widow/orphan setting.
- Include continuation text for tables that don't fit on a single page. You can display continuation text at the bottom of each page and at the top of each page. If neither option is selected, the continuation text will not be displayed.

To control pivot table printing properties:

1. Click the **Printing** tab.
2. Select the printing options that you want.
3. Click **OK** or **Apply**.

Cell properties

Cell properties are applied to a selected cell. You can change the font, value format, alignment, margins, and colors. Cell properties override table properties; therefore, if you change table properties, you do not change any individually applied cell properties.

To change cell properties:

1. Activate a table and select the cell(s) in the table.
2. From the Format menu or the pop-up menu choose **Cell Properties**.

Font and background

The Font and Background tab controls the font style and color and background color for the selected cells in the table.

Format value

The Format Value tab controls value formats for the selected cells. You can select formats for numbers, dates, time, or currencies, and you can adjust the number of decimal digits that are displayed.

Note: The list of Currency formats contains Dollar format (numbers with a leading dollar sign) and five custom currency formats. By default, all custom currency formats are set to the default number format, which contains no currency or other custom symbols. Use **Edit>Options>Currency** to define custom currency formats.

Alignment and margins

The Alignment and Margins tab controls horizontal and vertical alignment of values and top, bottom, left, and right margins for the selected cells. **Mixed** horizontal alignment aligns the content of each cell according to its type. For example, dates are right-aligned and text values are left-aligned.

Footnotes and captions

You can add footnotes and captions to a table. You can also hide footnotes or captions, change footnote markers, and renumber footnotes.

Adding footnotes and captions

To add a caption to a table:

1. From the Insert menu choose **Caption**.

A footnote can be attached to any item in a table. To add a footnote:

1. Click a title, cell, or caption within an activated pivot table.
2. From the Insert menu choose **Footnote**.
3. Insert the footnote text in the provided area.

To hide or show a caption

To hide a caption:

1. Select the caption.
2. From the View menu choose **Hide**.

To show hidden captions:

1. From the View menu choose **Show All**.

To hide or show a footnote in a table

To hide a footnote:

1. Right-click the cell that contains the footnote reference and select **Hide Footnotes** from the pop-up menu
or
2. Select the footnote in the footnote area of the table and select **Hide** from the pop-up menu.

Note: For legacy tables, select the footnote area of the table, select **Edit Footnote** from the pop-up menu, and then deselect (clear) the **Visible** property for any footnotes you want to hide.

If a cell contains multiple footnotes, use the latter method to selectively hide footnotes.

To hide all footnotes in the table:

1. Select all of the footnotes in the footnote area of the table (use click and drag or Shift+click to select the footnotes) and select **Hide** from the View menu.

Note: For legacy tables, select the footnote area of the table and select **Hide** from the View menu.

To show hidden footnotes:

1. Select **Show All Footnotes** from the View menu.

Footnote marker

Footnote Marker changes the characters that can be used to mark a footnote. By default, standard footnote markers are sequential letters or numbers, depending on the table properties settings. You can also assign a special marker. Special markers are not affected when you renumber footnotes or switch between numbers and letters for standard markers. The display of numbers or letters for standard markers and the subscript or superscript position of footnote markers are controlled by the Footnotes tab of the Table Properties dialog.

Note: For changing footnote markers in legacy tables, see “Editing footnotes in legacy tables” on page 147.

To change footnote markers:

1. Select a footnote.
2. From the **Format** menu, choose **Footnote Marker**.

Special markers are limited to 2 characters. Footnotes with special markers precede footnotes with sequential letters or numbers in the footnote area of the table; so changing to a special marker can reorder the footnote list.

Renumbering footnotes

When you have pivoted a table by switching rows, columns, and layers, the footnotes may be out of order. To renumber the footnotes:

1. From the Format menu choose **Renumber Footnotes**.

Editing footnotes in legacy tables

For legacy tables, you can use the Edit Footnotes dialog to enter and modify footnote text and font settings, change footnote markers, and selectively hide or delete footnotes.

When you insert a new footnote in a legacy table, the Edit Footnotes dialog automatically opens. To use the Edit Footnotes dialog to edit existing footnotes (without creating a new footnote):

1. Double-click the footnote area of the table or from the menus choose: Format > Edit Footnote.

Marker. By default, standard footnote markers are sequential letters or numbers, depending on the table properties settings. To assign a special marker, simply enter the new marker value in the Marker column. Special markers are not affected when you renumber footnotes or switch between numbers and letters for standard markers. The display of numbers or letters for standard markers and the subscript or superscript position of footnote markers are controlled by the Footnotes tab of the Table Properties dialog. See the topic “Table properties: notes” on page 143 for more information.

To change a special marker back to a standard marker, right-click on the marker in the Edit Footnotes dialog, select **Footnote Marker** from the pop-up menu, and select Standard marker in the Footnote Marker dialog box.

Footnote. The content of the footnote. The display reflects the current font and background settings. The font settings can be changed for individual footnotes using the Format subdialog. See the topic “Footnote font and color settings” for more information. A single background color is applied to all footnotes and can be changed in the Font and Background tab of the Cell Properties dialog. See the topic “Font and background” on page 145 for more information.

Visible. All footnotes are visible by default. Deselect (clear) the Visible checkbox to hide a footnote.

Footnote font and color settings

For legacy tables, you can use the Format dialog to change the font family, style, size and color for one or more selected footnotes:

1. In the Edit Footnotes dialog, select (click) one or more footnotes in the Footnotes grid.
2. Click the **Format** button.

The selected font family, style, size, and colors are applied to all the selected footnotes.

Background color, alignment, and margins can be set in the Cell Properties dialog and apply to all footnotes. You cannot change these settings for individual footnotes. See the topic “Font and background” on page 145 for more information.

Data cell widths

Set Data Cell Width is used to set all data cells to the same width.

To set the width for all data cells:

1. From the menus choose:
Format > Set Data Cell Widths...
2. Enter a value for the cell width.

Changing column width

1. Click and drag the column border.
-

Displaying hidden borders in a pivot table

For tables without many visible borders, you can display the hidden borders. This can simplify tasks like changing column widths.

1. From the View menu choose **Gridlines**.
-

Selecting rows, columns, and cells in a pivot table

You can select an entire row or column or a specified set of data and label cells.

To select multiple cells:

Select > Data and Label Cells

Printing pivot tables

Several factors can affect the way that printed pivot tables look, and these factors can be controlled by changing pivot table attributes.

- For multidimensional pivot tables (tables with layers), you can either print all layers or print only the top (visible) layer. See the topic “Table properties: printing” on page 144 for more information.
- For long or wide pivot tables, you can automatically resize the table to fit the page or control the location of table breaks and page breaks. See the topic “Table properties: printing” on page 144 for more information.
- For tables that are too wide or too long for a single page, you can control the location of table breaks between pages.

Use Print Preview on the File menu to see how printed pivot tables will look.

Controlling table breaks for wide and long tables

Pivot tables that are either too wide or too long to print within the defined page size are automatically split and printed in multiple sections. You can:

- Control the row and column locations where large tables are split.
- Specify rows and columns that should be kept together when tables are split.
- Rescale large tables to fit the defined page size.

To specify row and column breaks for printed pivot tables:

1. Activate the pivot table.
2. Click any cell in the column to the left of where you want to insert the break, or click any cell in the row before the row where you want to insert the break.

Note: For legacy tables, you must click a column label or row label cell.

3. From the menus, choose:

Format > Breakpoints > Vertical Breakpoint

or

Format > Breakpoints > Horizontal Breakpoint

Note: For legacy tables, choose **Format > Break Here** for both vertical and horizontal breakpoints.

To specify rows or columns to keep together:

1. Select the labels of the rows or columns that you want to keep together. Click and drag or Shift+click to select multiple row or column labels.
2. From the menus, choose:

Format > Breakpoints > Keep Together

Note: For legacy tables, choose **Format > Keep Together**.

To view breakpoints and keep together groups:

1. From the menus, choose:

Format > Breakpoints > Display Breakpoints

Breakpoints are shown as vertical or horizontal lines. Keep together groups appear as grayed out rectangular regions that are enclosed by a darker border.

Note: Displaying breakpoints and keep together groups is not supported for legacy tables.

To clear breakpoints and keep together groups

To clear a breakpoint:

1. Click any cell in the column to the left of a vertical breakpoint, or click any cell in the row above a horizontal breakpoint.

Note: For legacy tables, you must click a column label or row label cell.

2. From the menus, choose:

Format > Breakpoints > Clear Breakpoint or Group

Note: For legacy tables, choose **Format > Remove Break Here**.

To clear a keep together group:

3. Select the column or row labels that specify the group.

4. From the menus, choose:

Format > Breakpoints > Clear Breakpoint or Group

Note: For legacy tables, choose **Format > Remove Keep Together**.

All breakpoints and keep together groups are automatically cleared when you pivot or reorder any row or column. This behavior does not apply to legacy tables.

Creating a chart from a pivot table

1. Double-click the pivot table to activate it.
2. Select the rows, columns, or cells you want to display in the chart.
3. Right-click anywhere in the selected area.
4. Choose **Create Graph** from the pop-up menu and select a chart type.

Legacy tables

You can choose to render tables as legacy tables (referred to as full-featured tables in release 19) which are then fully compatible with IBM SPSS Statistics releases prior to 20. Legacy tables may render slowly and are only recommended if compatibility with releases prior to 20 is required. For information on how to create legacy tables, see “Pivot table options” on page 203.

Chapter 12. Models

Some results are presented as models, which appear in the output Viewer as a special type of visualization. The visualization displayed in the output Viewer is not the only view of the model that is available. A single model contains many different views. You can activate the model in the Model Viewer and interact with the model directly to display the available model views. You can also choose to print and export all the views in the model.

Interacting with a model

To interact with a model, you first activate it:

1. Double-click the model.
- or
2. Right-click the model and from the pop-up menu choose **Edit Content**.
3. From the submenu choose **In Separate Window**.

Activating the model displays the model in the Model Viewer. See the topic “Working with the Model Viewer” for more information.

Working with the Model Viewer

The Model Viewer is an interactive tool for displaying the available model views and editing the look of the model views. (For information about displaying the Model Viewer, see “Interacting with a model”.) There are two different styles of Model Viewer:

- **Split into main/auxiliary views.** In this style the **main view** appears in the left part of the Model Viewer. The main view displays some general visualization (for example, a network graph) for the model. The main view itself may have more than one model view. The drop-down list below the main view allows you to choose from the available main views.
The **auxiliary view** appears in the right part of the Model Viewer. The auxiliary view typically displays a more detailed visualization (including tables) of the model compared to the general visualization in the main view. Like the main view, the auxiliary view may have more than one model view. The drop-down list below the auxiliary view allows you to choose from the available main views. The auxiliary can also display specific visualizations for elements that are selected in the main view. For example, depending on the type of model, you may be able to select a variable node in the main view to display a table for that variable in the auxiliary view.
- **One view at a time, with thumbnails.** In this style there is only one view visible, and other views are accessed via thumbnails on the left of the Model Viewer. Each view displays some visualization for the model.

The specific visualizations that are displayed depend on the procedure that created the model. For information about working with specific models, refer to the documentation for the procedure that created the model.

Model view tables

Tables displayed in the Model Viewer are not pivot tables. You cannot manipulate these tables as you can manipulate pivot tables.

Setting model properties

Within the Model Viewer, you can set specific properties for the model. See the topic “Model properties” for more information.

Copying model views

You can also copy individual model views within the Model Viewer. See the topic “Copying model views” for more information.

Model properties

Depending on your Model Viewer, choose:

File > Properties

or

File > Print View

Each model has associated properties that let you specify which views are printed from the output Viewer. By default, only the view that is visible in the output Viewer is printed. This is always a main view, and only one main view. You can also specify that all available model views are printed. These include all the main views and all the auxiliary views (except for auxiliary views based on selection in the main view; these are not printed). Note that you can also print individual model views within the Model Viewer itself. See the topic “Printing a model” for more information.

Copying model views

From the Edit menu within the Model Viewer, you can copy the currently displayed main view or the currently displayed auxiliary view. Only one model view is copied. You can paste the model view into the output Viewer, where the individual model view is subsequently rendered as a visualization that can be edited in the Graphboard Editor. Pasting into the output Viewer allows you to display multiple model views simultaneously. You can also paste into other applications, where the view may appear as an image or a table depending on the target application.

Printing a model

Printing from the Model Viewer

You can print a single model view within the Model Viewer itself.

1. Activate the model in the Model Viewer. See the topic “Interacting with a model” on page 151 for more information.
2. From the menus choose View > Edit Mode, if available.
3. On the General toolbar palette in the main or auxiliary view (depending on which one you want to print), click the print icon. (If this palette is not displayed, choose Palettes>General from the View menu.)

Note: If your Model Viewer does not support a print icon, then choose File > Print.

Printing from the output Viewer

When you print from the output Viewer, the number of views that are printed for a specific model depend on the model's properties. The model can be set to print only the displayed view or all of the available model views. See the topic “Model properties” for more information.

Exporting a model

By default, when you export models from the output Viewer, inclusion or exclusion of model views is controlled by the model properties for each model. For more information about model properties, see “Model properties” on page 152. On export, you can override this setting and include all model views or only the currently visible model view. In the Export Output dialog box, click **Change Options...** in the Document group. For more information about exporting and this dialog box, see “Export output” on page 126. Note that all model views, including tables, are exported as graphics. Also note that auxiliary views based on selections in the main view are never exported.

Saving fields used in the model to a new dataset

You can save fields used in the model to a new dataset.

1. Activate the model in the Model Viewer. See the topic “Interacting with a model” on page 151 for more information.
2. From the menus choose:

Generate > Field Selection (Model input and target)

Dataset name. Specify a valid dataset name. Datasets are available for subsequent use in the same session but are not saved as files unless explicitly saved prior to the end of the session. Dataset names must conform to variable naming rules. See the topic “Variable names” on page 50 for more information.

Saving predictors to a new dataset based on importance

You can save predictors to a new dataset based on the information in the predictor importance chart.

1. Activate the model in the Model Viewer. See the topic “Interacting with a model” on page 151 for more information.
2. From the menus choose:

Generate > Field Selection (Predictor Importance)

Top number of variables. Includes or excludes the most important predictors up to the specified number.

Importance greater than. Includes or excludes all predictors with relative importance greater than the specified value.

3. After you click **OK**, the New Dataset dialog appears.

Dataset name. Specify a valid dataset name. Datasets are available for subsequent use in the same session but are not saved as files unless explicitly saved prior to the end of the session. Dataset names must conform to variable naming rules. See the topic “Variable names” on page 50 for more information.

Ensemble Viewer

Models for Ensembles

The model for an ensemble provides information about the component models in the ensemble and the performance of the ensemble as a whole.

The main (view-independent) toolbar allows you to choose whether to use the ensemble or a reference model for scoring. If the ensemble is used for scoring you can also select the combining rule. These changes do not require model re-execution; however, these choices are saved to the model for scoring and/or downstream model evaluation. They also affect PMML exported from the ensemble viewer.

Combining Rule. When scoring an ensemble, this is the rule used to combine the predicted values from the base models to compute the ensemble score value.

- Ensemble predicted values for **categorical** targets can be combined using voting, highest probability, or highest mean probability. **Voting** selects the category that has the highest probability most often across the base models. **Highest probability** selects the category that achieves the single highest probability across all base models. **Highest mean probability** selects the category with the highest value when the category probabilities are averaged across base models.
- Ensemble predicted values for **continuous** targets can be combined using the mean or median of the predicted values from the base models.

The default is taken from the specifications made during model building. Changing the combining rule recomputes the model accuracy and updates all views of model accuracy. The Predictor Importance chart also updates. This control is disabled if the reference model is selected for scoring.

Show All Combining rules. When selected, results for all available combining rules are shown in the model quality chart. The Component Model Accuracy chart is also updated to show reference lines for each voting method.

Model Summary

The Model Summary view is a snapshot, at-a-glance summary of the ensemble quality and diversity.

Quality. The chart displays the accuracy of the final model, compared to a reference model and a naive model. Accuracy is presented in larger is better format; the "best" model will have the highest accuracy. For a categorical target, accuracy is simply the percentage of records for which the predicted value matches the observed value. For a continuous target, accuracy is 1 minus the ratio of the mean absolute error in prediction (the average of the absolute values of the predicted values minus the observed values) to the range of predicted values (the maximum predicted value minus the minimum predicted value).

For bagging ensembles, the reference model is a standard model built on the whole training partition. For boosted ensembles, the reference model is the first component model.

The naive model represents the accuracy if no model were built, and assigns all records to the modal category. The naive model is not computed for continuous targets.

Diversity. The chart displays the "diversity of opinion" among the component models used to build the ensemble, presented in larger is more diverse format. It is a measure of how much predictions vary across the base models. Diversity is not available for boosted ensemble models, nor is it shown for continuous targets.

Predictor Importance

Typically, you will want to focus your modeling efforts on the predictor fields that matter most and consider dropping or ignoring those that matter least. The predictor importance chart helps you do this by indicating the relative importance of each predictor in estimating the model. Since the values are relative, the sum of the values for all predictors on the display is 1.0. Predictor importance does not relate to model accuracy. It just relates to the importance of each predictor in making a prediction, not whether or not the prediction is accurate.

Predictor importance is not available for all ensemble models. The predictor set may vary across component models, but importance can be computed for predictors used in at least one component model.

Predictor Frequency

The predictor set can vary across component models due to the choice of modeling method or predictor selection. The Predictor Frequency plot is a dot plot that shows the distribution of predictors across component models in the ensemble. Each dot represents one or more component models containing the predictor. Predictors are plotted on the y-axis, and are sorted in descending order of frequency; thus the

topmost predictor is the one that is used in the greatest number of component models and the bottommost one is the one that was used in the fewest. The top 10 predictors are shown.

Predictors that appear most frequently are typically the most important. This plot is not useful for methods in which the predictor set cannot vary across component models.

Component Model Accuracy

The chart is a dot plot of predictive accuracy for component models. Each dot represents one or more component models with the level of accuracy plotted on the y-axis. Hover over any dot to obtain information on the corresponding individual component model.

Reference lines. The plot displays color coded lines for the ensemble as well as the reference model and naïve models. A checkmark appears next to the line corresponding to the model that will be used for scoring.

Interactivity. The chart updates if you change the combining rule.

Boosted ensembles. A line chart is displayed for boosted ensembles.

Component Model Details

The table displays information on component models, listed by row. By default, component models are sorted in ascending model number order. You can sort the rows in ascending or descending order by the values of any column.

Model. A number representing the sequential order in which the component model was created.

Accuracy. Overall accuracy formatted as a percentage.

Method. The modeling method.

Predictors. The number of predictors used in the component model.

Model Size. Model size depends on the modeling method: for trees, it is the number of nodes in the tree; for linear models, it is the number of coefficients; for neural networks, it is the number of synapses.

Records. The weighted number of input records in the training sample.

Automatic Data Preparation

This view shows information about which fields were excluded and how transformed fields were derived in the automatic data preparation (ADP) step. For each field that was transformed or excluded, the table lists the field name, its role in the analysis, and the action taken by the ADP step. Fields are sorted by ascending alphabetical order of field names.

The action **Trim outliers**, if shown, indicates that values of continuous predictors that lie beyond a cutoff value (3 standard deviations from the mean) have been set to the cutoff value.

Split Model Viewer

The Split Model Viewer lists the models for each split, and provides summaries about the split models.

Split. The column heading shows the field(s) used to create splits, and the cells are the split values. Double-click any split to open a Model Viewer for the model built for that split.

Accuracy. Overall accuracy formatted as a percentage.

Model Size. Model size depends on the modeling method: for trees, it is the number of nodes in the tree; for linear models, it is the number of coefficients; for neural networks, it is the number of synapses.

Records. The weighted number of input records in the training sample.

Chapter 13. Automated Output Modification

Automated output modification applies formatting and other changes to the contents of the active Viewer window. Changes that can be applied include:

- All or selected viewer objects
- Selected types of output objects (for example, charts, logs, pivot tables)
- Pivot table content based on conditional expressions
- Outline (navigation) pane content

The types of changes you can make include:

- Delete objects
- Index objects (add a sequential numbering scheme)
- Change the visible property of objects
- Change the outline label text
- Transpose rows and columns in pivot tables
- Change the selected layer of pivot tables
- Change the formatting of selected areas or specific cells in a pivot table based on conditional expressions (for example, make all significance values less than 0.05 bold)

To specify automated output modification:

1. From the menus, choose:**Utilities > Style Output**
2. Select one or more objects in the Viewer.
3. Select the options you want in the Select dialog. (You can also select objects before you open the dialog.)
4. Select the output changes you want in the Style Output dialog.

Style Output: Select

The Style Output: Select dialog specifies basic selection criteria for changes that you specify on the Style Output dialog.

You can also select objects in the Viewer after opening the Style Output: Select dialog.

Selected only. Changes are applied only to the selected objects that meet the specified criteria.

- **Select as last command.** If selected, changes are only applied to the output from the last procedure. If not selected, changes are applied to the specific instance of the procedure. For example, if there are three instances of the Frequencies procedure and you select the second instance, changes are applied only to that instance. If you paste the syntax based on your selections, this option selects the second instance of that procedure. If output from multiple procedures is selected, this option applies only to the procedure for which the output is the last block of output in the Viewer.
- **Select as a group.** If selected, all of the objects in the selection are treated as a single group on the main Style Output dialog. If not selected, the selected objects are treated as individual selections, and you can set the properties for each object individually.

All objects of this type. Changes are applied to all objects of the selected type that meet the specified criteria. This option is only available if there is a single object type selected in the Viewer. Object types include tables, warnings, logs, charts, tree diagrams, text, models, and outline headers.

All objects of this sub-type. Changes are applied to all tables of the same subtype as the selected tables that meet the specified criteria. This option is only available if there is a single table subtype selected in the Viewer. For example, the selection can include two separate Frequencies tables, but not a Frequencies table and a Descriptives table.

Objects with a similar name. Changes are applied to all objects with a similar name that meet the specified criteria.

- **Criteria.** The options are Contains, Exactly, Starts with, and Ends with.
- **Value.** The name as it is displayed in the outline pane of the Viewer.
- **Update.** Selects all the objects in the Viewer that meet the specified criteria for the specified value.

Style Output

The Style Output dialog specifies the changes you want to make to the selected output objects in the Viewer.

Create a Backup of the Output. Changes made by the automated output modification process cannot be undone. To preserve the original Viewer document, create a backup copy.

Selections and Properties

The list of objects or groups of objects that you can modify is determined by the objects you select in the Viewer and the selections you make in the Style Output: Select dialog.

Selection. The name of the selected procedure or group of object types. When there is an integer in parentheses after the selection text, changes are applied only to that instance of that procedure in the sequence of objects in the Viewer. For example, "Frequencies(2)" applies change only to the second instance of the Frequencies procedure in the Viewer output.

Type. The type of object. For example, log, title, table, chart. For individual table types, the table subtype is also displayed.

Delete. Specifies whether the selection should be deleted.

Visible. Specifies whether the selection should be visible or hidden. The default option is "As Is," which means that current visibility property of the selection is preserved.

Properties. A summary of changes to apply to the selection.

Add. Adds a row to the list and opens the Style Output: Select dialog. You can select other objects in the Viewer and specify the selection conditions.

Duplicate.Duplicates the selected row.

Move Up and Move Down. Moves the selected row up or down in the list. The order can be important since changes specified in subsequent rows can overwrite changes that are specified in previous rows.

Create a report of the property changes. Displays a table that summarizes the changes in the Viewer.

Object Properties

You specify the changes that you want to make to each selection from the Selections and Properties section in the Object Properties section. The available properties are determined by the selected row in the Selections and Properties section.

Command. The name of the procedure if the selection refers to a single procedure. The selection can include multiple instances of the same procedure.

Type. The type of object.

Subtype. If the selection refers to a single table type, the table subtype name is displayed.

Outline Label. The label in the outline pane that is associated with the selection. You can replace the label text or add information to the label. For more information, see the topic Style Output: Labels and Text.

Indexing Format. Adds a sequential number, letter, or roman numeral to the objects in the selection. For more information, see the topic Style Output: Indexing.

Table Title. The title of table or tables. You can replace the title or add information to the title. For more information, see the topic Style Output: Labels and Text.

TableLook. The TableLook used for tables. For more information, see the topic Style Output: TableLooks.

Transpose. Transposes rows and columns in tables.

Top Layer. For tables with layers, the category that is displayed for each layer.

Conditional styling. Conditional style changes for tables. For more information, see the topic Table Style.

Sort. Sorts table contents by the values of the selected column label. The available column labels are displayed in a dropdown list. This option is only available if the selection contains a single table subtype.

Sort Direction. Specifies the sort direction for tables.

Comment Text. You can add comment text to each table.

- Comment text is displayed in a tooltip when you hover over a table in the Viewer.
- Screen readers read the comment text when the table has focus.
- The tooltip in the Viewer displays only the first 200 characters of the comment, but screen readers read the entire text.
- When you export output to HTML or a web report, the comment text is used as alt text.

Contents. The text of logs, titles, and text objects. You can replace the text or add information to the text. For more information, see the topic Style Output: Labels and Text.

Font. The font for logs, titles, and text objects.

Font Size. The font size for logs, titles, and text objects.

Text Color. Text color for logs, titles, and text objects.

Chart Template. The chart template that is used for charts, except charts that are created with the Graphboard Template Chooser.

Graphboard Stylesheet. The style sheet used for charts that are created with the Graphboard Template Chooser.

Size. The size of charts and tree diagrams.

Special Variables in Comment Text

You can include special variables to insert date, time, and other values in the **Comment Text** field.

)DATE

Current date in the form dd-mmm-yyyy.

)ADATE

Current date in the form mm/dd/yyyy.

)SDATE

Current date in the form yyyy/mm/dd.

)EDATE

Current date in the form dd.mm.yyyy.

)TIME

Current 12-hour clock time in the form hh:mm:ss.

)ETIME

Current 24-hour clock time in the form hh:mm:ss.

)INDEX

The defined index value. For more information, see the topic Style Output: Indexing.

)TITLE

The text of the outline label for the table.

)PROCEDURE

The name of the procedure that created the table.

)DATASET

The name of the dataset used to create the table.

\n

Inserts a line break.

Style Output: Labels and Text

The Style Output: Labels and Text dialog replaces or adds text to outline labels, text objects, and table titles. It also specifies the inclusion and placement of index values for outline labels, text objects, and table titles.

Add text to the label or text object. You can add the text before or after the existing text or replace the existing text.

Add indexing. Adds a sequential letter, number, or roman numeral. You can place the index before or after the text. You can also specify one or more characters that are used as a separator between the text and the index. For information about index formatting, see the topic Style Output: Indexing.

Style Output: Indexing

The Style Output: Indexing dialog specifies the index format and the starting value.

Type: The sequential index values can be numbers, lower or uppercase letters, lower or uppercase roman numerals.

Starting value. The starting value can be any value that is valid for the type that is selected.

To display index values in the output, you must select **Add indexing** in the Style Output: Labels and Text dialog for the selected object type.

- For outline labels, select **Outline label** in the Properties column in the Style Output dialog.
- For table titles, select **Table Title** in the Properties column in the Style Output dialog.
- For text objects, select **Contents** in the Properties column in the Style Output dialog.

Style Output: TableLooks

A TableLook is a set of properties that define the appearance of a table. You can select a previously defined TableLook or create your own TableLook.

- Before or after a TableLook is applied, you can change cell formats for individual cells or groups of cells by using cell properties. The edited cell formats remain intact, even when you apply a new TableLook.
- Optionally, you can reset all cells to the cell formats that are defined by the current TableLook. This option resets any cells that were edited. If **As Displayed** is selected in the TableLook Files list, any edited cells are reset to the current table properties.
- Only table properties that are defined in the Table Properties dialog are saved in TableLooks. TableLooks do not include individual cell modifications.

Style Output: Size

The Style Output: Size dialog controls the size of charts and tree diagrams. You can specify the height and width in centimeters, inches, or points.

Table Style

The Table Style dialog specifies conditions for automatically changing properties of pivot tables based on specific conditions. For example, you can make all significance values less than 0.05 bold and red. The Table Style dialog can be accessed from the Style Output dialog or from the dialogs for specific statistical procedures.

- The Table Style dialog can be accessed from the Style Output dialog or from the dialogs for specific statistical procedures.
- The statistical procedure dialogs that support the Table Style dialog are Bivariate Correlations, Crosstabs, Custom Tables, Descriptives, Frequencies, Logistic Regression, Linear Regression, and Means.

Table. The table or tables to which the conditions apply. When you access this dialog from the Style Output dialog, the only choice is "All applicable tables." When you access this dialog from a statistical procedure dialog, you can select the table type from a list of procedure-specific tables.

Value. The row or column label value that defines the area of the table to search for values that meet the conditions. You can select a value from the list or enter a value. The values in the list are not affected by the output language and apply to numerous variations of the value. The values available in the list depend on the table type.

- **Count.** Rows or columns with any of these labels or the equivalent in the current output language: "Frequency", "Count", "N".
- **Mean.** Rows or columns with the label "Mean" or the equivalent in the current output language.
- **Median** Rows or columns with the label "Median" or the equivalent in the current output language.
- **Percent.** Rows or columns with the label "Percent" or the equivalent in the current output language.
- **Residual.** Rows or columns with any of these labels or the equivalent in the current output language: "Resid", "Residual", "Std. Residual".
- **Correlation.** Rows or columns with any of these labels or the equivalent in the current output language: "Adjusted R Square", "Correlation Coefficient", "Correlations", "Pearson Correlation", "R Square".
- **Significance.** Rows or columns with any of these labels or the equivalents in the current output language: "Approx Sig.", "Asymp. Sig (2-sided)", "Exact Sig.", "Exact Sig. (1-sided)", "Exact Sig. (2-sided)", "Sig.", "Sig. (1-tailed)", "Sig. (2-tailed)"
- **All data cells.** All data cells are included.

Dimension. Specifies whether to search rows, columns, or both for a label with the specified value.

Condition. Specifies the condition to find. For more information, see the topic Table Style: Condition.

Formatting. Specifies the formatting to apply to the table cells or areas that meet the condition. For more information, see the topic Table Style: Format.

Add. Adds a row to the list.

Duplicate. Duplicates the selected row.

Move Up and **Move Down**. Moves the selected row up or down in the list. The order can be important since changes specified in subsequent rows can overwrite changes that are specified in previous rows.

Create a report of the conditional styling. Displays a table that summarizes the changes in the Viewer. This option is available when the Table Style dialog is accessed from a statistical procedure dialog. The Style Output dialog has a separate option to create a report.

Table Style: Condition

The Table Style: Condition dialog specifies the conditions under which the changes are applied. There are two options.

- **To all values of this type.** The only condition is the value specified in the Value column in the Table Style dialog. This option is the default.
- **Based on the following conditions.** Within the table area specified by the Value and Dimension columns in the Table Style dialog, find values that meet the specified conditions.

Values. The list contains comparison expressions, such as **Exactly**, **Less than**, **Greater Than**, and **Between**.

- **Absolute value** is available for comparison expressions that only require one value. For example, you could find correlations with an absolute value greater than 0.5.
- **Top** and **Bottom** are the highest and lowest *n* values in the specified table area. The value for **Number** must be an integer.
- **System-missing.** Finds system-missing values in the specified table area.

Table Style: Format

The Table Style: Format dialog specifies changes to apply, based on the conditions specified in the Table Style: Conditions dialog.

Use TableLook defaults. If no previous format changes have been made, either manually or by automated output modification, this is equivalent to making no format changes. If previous changes have been made, this removes those changes and restores the affected areas of the table to their default format.

Apply new formatting. Applies the specified format changes. Format changes include font style and color, background color, format for numeric values (including dates and times), and number of decimals displayed.

Apply to. Specifies the area of the table to which to apply the changes.

- **Cells only.** Applies changes only to table cells that meet the condition.
- **Entire column.** Applies changes to the entire column that contains a cell that meet the condition. This option includes the column label.
- **Entire row.** Applies changes to the entire row that contains a cell that meets the condition. This option includes the row label.

Replace value. Replaces values with the specified new value. For **Cells only**, this option replaces individual cell values that meet the condition. For **Entire row** and **Entire column**, this option replaces all values in the row or column.

Chapter 14. Working with Command Syntax

The powerful command language allows you to save and automate many common tasks. It also provides some functionality not found in the menus and dialog boxes.

Most commands are accessible from the menus and dialog boxes. However, some commands and options are available only by using the command language. The command language also allows you to save your jobs in a syntax file so that you can repeat your analysis at a later date or run it in an automated job with the a production job.

A syntax file is simply a text file that contains commands. While it is possible to open a syntax window and type in commands, it is often easier if you let the software help you build your syntax file using one of the following methods:

- Pasting command syntax from dialog boxes
- Copying syntax from the output log
- Copying syntax from the journal file

Detailed command syntax reference information is available in two forms: integrated into the overall Help system and as a separate PDF file, called the *Command Syntax Reference*, also available from the Help menu. Context-sensitive Help for the current command in a syntax window is available by pressing the F1 key.

Syntax Rules

When you run commands from a command syntax window during a session, you are running commands in **interactive** mode.

The following rules apply to command specifications in interactive mode:

- Each command must start on a new line. Commands can begin in any column of a command line and continue for as many lines as needed. The exception is the END DATA command, which must begin in the first column of the first line after the end of data.
- Each command should end with a period as a command terminator. It is best to omit the terminator on BEGIN DATA, however, so that inline data are treated as one continuous specification.
- The command terminator must be the last nonblank character in a command.
- In the absence of a period as the command terminator, a blank line is interpreted as a command terminator.

Note: For compatibility with other modes of command execution (including command files run with INSERT or INCLUDE commands in an interactive session), each line of command syntax should not exceed 256 characters.

- Most subcommands are separated by slashes (/). The slash before the first subcommand on a command is usually optional.
- Variable names must be spelled out fully.
- Text included within apostrophes or quotation marks must be contained on a single line.
- A period (.) must be used to indicate decimals, regardless of your regional or locale settings.
- Variable names ending in a period can cause errors in commands created by the dialog boxes. You cannot create such variable names in the dialog boxes, and you should generally avoid them.

Command syntax is case insensitive, and three- or four-letter abbreviations can be used for many command specifications. You can use as many lines as you want to specify a single command. You can

add space or break lines at almost any point where a single blank is allowed, such as around slashes, parentheses, arithmetic operators, or between variable names. For example,

```
FREQUENCIES  
  VARIABLES=JOBCAT GENDER  
  /PERCENTILES=25 50 75  
  /BARCHART.
```

and

```
freq var=jobcat gender /percent=25 50 75 /bar.
```

are both acceptable alternatives that generate the same results.

INCLUDE Files

For command files run via the INCLUDE command, batch mode syntax rules apply.

The following rules apply to command specifications in batch mode:

- All commands in the command file must begin in column 1. You can use plus (+) or minus (-) signs in the first column if you want to indent the command specification to make the command file more readable.
- If multiple lines are used for a command, column 1 of each continuation line must be blank.
- Command terminators are optional.
- A line cannot exceed 256 characters; any additional characters are truncated.

Unless you have existing command files that already use the INCLUDE command, you should probably use the INSERT command instead, since it can accommodate command files that conform to either set of rules. If you generate command syntax by pasting dialog box choices into a syntax window, the format of the commands is suitable for any mode of operation. See the *Command Syntax Reference* (available in PDF format from the Help menu) for more information.

Pasting Syntax from Dialog Boxes

The easiest way to build a command syntax file is to make selections in dialog boxes and paste the syntax for the selections into a syntax window. By pasting the syntax at each step of a lengthy analysis, you can build a job file that allows you to repeat the analysis at a later date or run an automated job with the Production Facility.

In the syntax window, you can run the pasted syntax, edit it, and save it in a syntax file.

To Paste Syntax from Dialog Boxes

1. Open the dialog box and make the selections that you want.
2. Click **Paste**.

The command syntax is pasted to the designated syntax window. If you do not have an open syntax window, a new syntax window opens automatically, and the syntax is pasted there. By default, the syntax is pasted after the last command. You can choose to have syntax pasted at the position of the cursor or to overwrite selected syntax. The setting is specified from the Syntax Editor tab in the Options dialog box.

Copying Syntax from the Output Log

You can build a syntax file by copying command syntax from the log that appears in the Viewer. To use this method, you must select **Display commands in the log** in the Viewer settings (Edit menu, Options, Viewer tab) before running the analysis. Each command will then appear in the Viewer along with the output from the analysis.

In the syntax window, you can run the pasted syntax, edit it, and save it in a syntax file.

To Copy Syntax from the Output Log

1. Before running the analysis, from the menus choose:

Edit > Options...

2. On the Viewer tab, select **Display commands in the log**.

As you run analyses, the commands for your dialog box selections are recorded in the log.

3. Open a previously saved syntax file or create a new one. To create a new syntax file, from the menus choose:

File > New > Syntax

4. In the Viewer, double-click a log item to activate it.

5. Select the text that you want to copy.

6. From the Viewer menus choose:

Edit > Copy

7. In a syntax window, from the menus choose:

Edit > Paste

Using the Syntax Editor

The Syntax Editor provides an environment specifically designed for creating, editing, and running command syntax. The Syntax Editor features:

- **Auto-Completion.** As you type, you can select commands, subcommands, keywords, and keyword values from a context-sensitive list. You can choose to be prompted automatically with the list or display the list on demand.
- **Color Coding.** Recognized elements of command syntax (commands, subcommands, keywords, and keyword values) are color coded so, at a glance, you can spot unrecognized terms. Also, a number of common syntactical errors--such as unmatched quotes--are color coded for quick identification.
- **Breakpoints.** You can stop execution of command syntax at specified points, allowing you to inspect the data or output before proceeding.
- **Bookmarks.** You can set bookmarks that allow you to quickly navigate large command syntax files.
- **Auto-Indentation.** You can automatically format your syntax with an indentation style similar to syntax pasted from a dialog box.
- **Step Through.** You can step through command syntax one command at a time, advancing to the next command with a single click.

Note: When working with right to left languages, it is recommended to check the Optimize for right to left languages box on the Syntax Editor tab in the Options dialog box.

Syntax Editor Window

The Syntax Editor window is divided into four areas:

- The editor pane is the main part of the Syntax Editor window and is where you enter and edit command syntax.
- The gutter is adjacent to the editor pane and displays information such as line numbers and breakpoint positions.
- The navigation pane is to the left of the gutter and editor pane and displays a list of all commands in the Syntax Editor window and provides single click navigation to any command.
- The error pane is below the editor pane and displays runtime errors.

Gutter Contents

Line numbers, breakpoints, bookmarks, command spans, and a progress indicator are displayed in the gutter to the left of the editor pane in the syntax window.

- Line numbers do not account for any external files referenced in INSERT and INCLUDE commands. You can show or hide line numbers by choosing View > Show Line Numbers from the menus.
- Breakpoints stop execution at specified points and are represented as a red circle adjacent to the command on which the breakpoint is set.
- Bookmarks mark specific lines in a command syntax file and are represented as a square enclosing the number (1-9) assigned to the bookmark. Hovering over the icon for a bookmark displays the number of the bookmark and the name, if any, assigned to the bookmark.
- Command spans are icons that provide visual indicators of the start and end of a command. You can show or hide command spans by choosing View > Show Command Spans from the menus.
- The progress of a given syntax run is indicated with a downward pointing arrow in the gutter, stretching from the first command run to the last command run. This is most useful when running command syntax containing breakpoints and when stepping through command syntax. See the topic “Running Command Syntax” on page 176 for more information.

Navigation Pane

The navigation pane contains a list of all recognized commands in the syntax window, displayed in the order in which they occur in the window. Clicking on a command in the navigation pane positions the cursor at the start of the command.

- You can use the Up and Down arrow keys to move through the list of commands or click a command to navigate to it. A double click will select the command.
- Command names for commands containing certain types of syntactical errors--such as unmatched quotes--are colored red and in bold text by default. See the topic “Color Coding” on page 169 for more information.
- The first word of each line of unrecognized text is shown in gray.
- You can show or hide the navigation pane by choosing View > Show Navigation Pane from the menus.

Error Pane

The error pane displays runtime errors from the most previous run.

- The information for each error contains the starting line number of the command containing the error.
- You can use the Up and Down arrow keys to move through the list of errors.
- Clicking on an entry in the list will position the cursor on the first line of the command that generated the error.
- You can show or hide the error pane by choosing View > Show Error Pane from the menus.

Using Multiple Views

You can split the editor pane into two panes arranged with one above the other.

1. From the menus choose:

Window > Split

Actions in the navigation and error panes--such as clicking on an error--act on the pane where the cursor is positioned.

You can remove the splitter by double-clicking it or choosing Window > Remove Split.

Terminology

Commands. The basic unit of syntax is the command. Each command begins with the command name, which consists of one, two, or three words--for example, DESCRIPTIVES, SORT CASES, or ADD VALUE LABELS.

Subcommands. Most commands contain subcommands. Subcommands provide for additional specifications and begin with a forward slash followed by the name of the subcommand.

Keywords. Keywords are fixed terms that are typically used within a subcommand to specify options available for the subcommand.

Keyword Values. Keywords can have values such as a fixed term that specifies an option or a numeric value.

Example

```
CODEBOOK gender jobcat salary  
/VARINFO VALUELABELS MISSING  
/OPTIONS VARORDER=MEASURE.
```

- The name of the command is CODEBOOK.
- VARINFO and OPTIONS are subcommands.
- VALUELABELS, MISSING, and VARORDER are keywords.
- MEASURE is a keyword value associated with VARORDER.

Auto-Completion

The Syntax Editor provides assistance in the form of auto-completion of commands, subcommands, keywords, and keyword values. By default, you are prompted with a context-sensitive list of available terms as you type. Pressing Enter or Tab will insert the currently highlighted item in the list at the position of the cursor. You can display the list on demand by pressing Ctrl+Spacebar and you can close the list by pressing the Esc key.

The Auto Complete menu item on the Tools menu toggles the automatic display of the auto-complete list on or off. You can also enable or disable automatic display of the list from the Syntax Editor tab in the Options dialog box. Toggling the Auto Complete menu item overrides the setting on the Options dialog but does not persist across sessions.

Note: The auto-completion list will close if a space is entered. For commands consisting of multiple words--such as ADD FILES--select the command before entering any spaces.

Color Coding

The Syntax Editor color codes recognized elements of command syntax, such as commands and subcommands, as well as various syntactical errors like unmatched quotes or parentheses. Unrecognized text is not color coded.

Commands. By default, recognized commands are colored blue and in bold text. If, however, there is a recognized syntactical error within the command--such as a missing parenthesis--the command name is colored red and in bold text by default.

Note: Abbreviations of command names--such as FREQ for FREQUENCIES--are not colored, but such abbreviations are valid.

Subcommands. Recognized subcommands are colored green by default. If, however, the subcommand is missing a required equals sign or an invalid equals sign follows it, the subcommand name is colored red by default.

Keywords. Recognized keywords are colored maroon by default. If, however, the keyword is missing a required equals sign or an invalid equals sign follows it, the keyword is colored red by default.

Keyword values. Recognized keyword values are colored orange by default. User-specified values of keywords such as integers, real numbers, and quoted strings are not color coded.

Comments. Text within a comment is colored gray by default.

Quotes. Quotes and text within quotes are colored black by default.

Syntactical Errors. Text associated with the following syntactical errors is colored red by default.

- **Unmatched Parentheses, Brackets, and Quotes.** Unmatched parentheses and brackets within comments and quoted strings are not detected. Unmatched single or double quotes within quoted strings are syntactically valid.

Certain commands contain blocks of text that are not command syntax--such as BEGIN DATA-END DATA, BEGIN GPL-END GPL, and BEGIN PROGRAM-END PROGRAM. Unmatched values are not detected within such blocks.

- **Long lines.** Long lines are lines containing more than 251 characters.
- **End statements.** Several commands require either an END statement prior to the command terminator (for example, BEGIN DATA-END DATA) or require a matching END command at some point later in the command stream (for example, LOOP-END LOOP). In both cases, the command will be colored red, by default, until the required END statement is added.

Note: You can navigate to the next or previous syntactical error by choosing Next Error or Previous error from the Validation Errors submenu of the Tools menu.

From the Syntax Editor tab in the Options dialog box, you can change default colors and text styles and you can turn color coding off or on. You can also turn color coding of commands, subcommands, keywords, and keyword values off or on by choosing Tools > Color Coding from the menus. You can turn color coding of syntactical errors off or on by choosing Tools > Validation. Choices made on the Tools menu override settings in the Options dialog box but do not persist across sessions.

Note: Color coding of command syntax within macros is not supported.

Breakpoints

Breakpoints allow you to stop execution of command syntax at specified points within the syntax window and continue execution when ready.

- Breakpoints are set at the level of a command and stop execution prior to running the command.
- Breakpoints cannot occur within LOOP-END LOOP, DO IF-END IF, DO REPEAT-END REPEAT, INPUT PROGRAM-END INPUT PROGRAM, and MATRIX-END MATRIX blocks. They can, however, be set at the beginning of such blocks and will stop execution prior to running the block.
- Breakpoints cannot be set on lines containing non-IBM SPSS Statistics command syntax, such as occur within BEGIN PROGRAM-END PROGRAM, BEGIN DATA-END DATA, and BEGIN GPL-END GPL blocks.
- Breakpoints are not saved with the command syntax file and are not included in copied text.
- By default, breakpoints are honored during execution. You can toggle whether breakpoints are honored or not from Tools > Honor Breakpoints.

To Insert a Breakpoint

1. Click anywhere in the gutter to the left of the command text.
or
2. Position the cursor within the command.
3. From the menus choose:

Tools > Toggle Breakpoint

The breakpoint is represented as a red circle in the gutter to the left of the command text and on the same line as the command name.

Clearing Breakpoints

To clear a single breakpoint:

1. Click the icon representing the breakpoint in the gutter to the left of the command text.
or
2. Position the cursor within the command.
3. From the menus choose:
Tools > Toggle Breakpoint
To clear all breakpoints:
4. From the menus choose:
Tools > Clear All Breakpoints

See “Running Command Syntax” on page 176 for information about the run-time behavior in the presence of breakpoints.

Bookmarks

Bookmarks allow you to quickly navigate to specified positions in a command syntax file. You can have up to 9 bookmarks in a given file. Bookmarks are saved with the file, but are not included when copying text.

To Insert a Bookmark

1. Position the cursor on the line where you want to insert the bookmark.
2. From the menus choose:
Tools > Toggle Bookmark

The new bookmark is assigned the next available number, from 1 to 9. It is represented as a square enclosing the assigned number and displayed in the gutter to the left of the command text.

Clearing Bookmarks

To clear a single bookmark:

1. Position the cursor on the line containing the bookmark.
2. From the menus choose:
Tools > Toggle Bookmark

To clear all bookmarks:

1. From the menus choose:
Tools > Clear All Bookmarks

Renaming a Bookmark

You can associate a name with a bookmark. This is in addition to the number (1-9) assigned to the bookmark when it was created.

1. From the menus choose:
Tools > Rename Bookmark
2. Enter a name for the bookmark and click **OK**.

The specified name replaces any existing name for the bookmark.

Navigating with Bookmarks

To navigate to the next or previous bookmark:

1. From the menus choose:

Tools > Next Bookmark

or

Tools > Previous Bookmark

To navigate to a specific bookmark:

1. From the menus choose:

Tools > Go To Bookmark

2. Select the bookmark.

Commenting or Uncommenting Text

You can comment out entire commands as well as text that is not recognized as command syntax and you can uncomment text that has previously been commented out.

To comment out text

1. Select the text. Note that a command will be commented out if any part of it is selected.

2. From the menus choose:

Tools > Toggle Comment Selection

You can comment out a single command by positioning the cursor anywhere within the command and choosing Tools > Toggle Comment Selection.

To uncomment text

1. Select the text to uncomment. Note that a command will be uncommented if any part of it is selected.

2. From the menus choose:

Tools > Toggle Comment Selection

You can uncomment a single command by positioning the cursor anywhere within the command and choosing Tools > Toggle Comment Selection. Note that this feature will not remove comments within a command (text set off by /* and */) or comments created with the COMMENT keyword.

Formatting Syntax

There are a number of formatting options available when working directly with syntax. The following formatting options are available:

Indenting syntax

You can indent or outdent selected lines of syntax and you can automatically indent selections so that the syntax is formatted in a manner similar to syntax pasted from a dialog box.

- The default indent is four spaces and applies to indenting selected lines of syntax as well as to automatic indentation. You can change the indent size from the **Syntax Editor** tab in the Options dialog box.
- Note that using the Tab key in the Syntax Editor does not insert a tab character. It inserts a space.

Indenting text

1. Select the text or position the cursor on a single line that you want to indent.
2. From the menus choose:

Tools > Indent Syntax > Indent

You can also indent a selection or line by pressing the Tab key.

Outdenting text

1. Select the text or position the cursor on a single line that you want to outdent.
2. From the menus choose:

Tools > Indent Syntax > Outdent

Automatically indenting text

1. Select the text.
2. From the menus choose:
Tools > Indent Syntax > Auto Indent

When you automatically indent text, any existing indentation is removed and replaced with the automatically generated indents. Note that automatically indenting code within a BEGIN PROGRAM block may break the code if it depends on specific indentation to function, such as Python code containing loops and conditional blocks.

Syntax formatted with the auto-indent feature may not run in batch mode. For example, auto-indenting an INPUT PROGRAM-END INPUT PROGRAM, LOOP-END LOOP, DO IF-END IF or DO REPEAT-END REPEAT block will cause the syntax to fail in batch mode because commands in the block will be indented and will not start in column 1 as required for batch mode. You can, however, use the -i switch in batch mode to force the Batch Facility to use interactive syntax rules. See the topic “Syntax Rules” on page 165 for more information.

Duplicating syntax

You can duplicate one or more syntax lines with a keyboard shortcut. Duplicated syntax is inserted immediately after the last line of the selected syntax. The **Duplicate lines** feature is available through the **Edit** menu, or by using the Ctrl+U keyboard shortcut.

1. Select one or more syntax lines that you want to duplicate.
2. Select **Duplicate lines** from the **Edit** menu, or use the Ctrl+U keyboard shortcut.

The selected syntax lines are duplicated immediately underneath the selection.

Note: If you do not select any syntax lines, and then select **Edit > Duplicate lines** or press Ctrl+U, the line where the cursor is placed is duplicated.

Joining syntax

You can join one or more syntax lines with a keyboard shortcut. Joining syntax lines effectively removes line breaks between the selected lines and places all selected syntax on a single line. The **Join lines** feature is available through the **Edit** menu, or by using the Ctrl+J keyboard shortcut.

1. Select one or more syntax lines that you want to join.
2. Select **Join lines** from the **Edit** menu, or use the Ctrl+J keyboard shortcut.

The selected syntax lines are joined and all selected syntax is presented on a single line.

Removing empty syntax

You can remove empty syntax lines with a keyboard shortcut. The **Remove empty lines** feature is available through the **Edit** menu, or by using the Ctrl+M keyboard shortcut.

1. Select an empty line, or select a section of syntax that contains empty lines.
2. Select **Remove empty lines** from the **Edit** menu, or use the Ctrl+M keyboard shortcut.

The empty lines are removed from the selected syntax.

Note: If you do not select any syntax lines, and then select **Edit > Remove empty lines** or press Ctrl+M, the line where the cursor is placed is removed (if the line is empty).

Moving syntax

You can move one or more syntax lines up or down with keyboard shortcuts. Syntax lines are moved up or down either before the first line or after the last line of the selected syntax. The **Move lines up** and **Move lines down** features are available through the **Edit** menu, or by using the Alt+Up and Alt+Down keyboard shortcuts.

1. Select one or more syntax lines that you want to move.
2. Select **Move lines up** or **Move lines down** from the **Edit** menu, or use the Alt+Up or Alt+Down keyboard shortcuts.

The selected syntax lines are moved either up or down before the first line or after the last line of the selected syntax.

Note: If you do not select any syntax lines, and then select **Edit > Move lines up/Move lines down** or press Alt+Up/Alt+Down, the line where the cursor is placed is moved.

Column edit mode

You can place selected syntax lines into column edit mode with a keyboard shortcut. In column edit mode, you can apply similar changes across multiple syntax lines. The **Toggle column edit mode** and **Exit column edit mode** features are available through the **Edit** menu, or by using the Ctrl+G and ESC keyboard shortcuts.

The gutter pane shows which lines are in column edit mode. A dark blue background in the gutter pane indicates that the selected lines are in column edit mode. When column edit mode is activated, any edit that is made on one line is mirrored to all of the selected syntax lines.

Note: When column edit mode is activated, edits apply to only the selected syntax lines.

Toggling column edit mode

1. Select the syntax lines that you want edit.
2. Select **Toggle column edit mode** from the **Edit** menu, or use the Ctrl+G keyboard shortcut.
3. Make an edit on a single line. The change is propagated to the other lines that are actively in column edit mode.
4. Press the Esc key to exit column edit mode.

Note: Column edits are based on character position, which means that if you delete the first character in a line, the first character in the other selected lines are also deleted (regardless of whether the characters match or not). The same applies to additions. If you add word to a line, the same word is added to the other selected lines at the same character position.

Pasting syntax in column edit mode

Column edit mode functions differently depending on whether you paste single or multiple syntax lines:

- When a single syntax line is pasted, the line is copied to all selected lines.
- When multiple syntax lines are pasted, the following rules apply.
 - If the number of pasted syntax lines is less than the number of syntax lines in column edit mode, all lines are pasted.
 - If the number of pasted syntax lines is equal to or greater than the number of syntax lines in column edit mode, the number of syntax lines that are pasted cannot exceed the number of selected syntax lines. For example, when the number of syntax lines in column edit mode is 5 and you attempt to paste 7 syntax lines, only 5 of out 7 lines are pasted.

Note: When you delete or replace multiple syntax lines while in column edit mode, the interface first exits column edit mode and then processes the action.

Trimming leading and trailing spaces

You can trim leading or trailing spaces from syntax lines with keyboard shortcuts. The **Trim leading spaces**, **Trim trailing spaces**, and **Trim leading and trailing spaces** features are available through the **Edit** menu, or by using the Ctrl+L, Ctrl+T, and Ctrl+B keyboard shortcuts.

1. Select one or more syntax lines that contain trailing and/or leading space characters.
2. Select **Trim leading spaces**, **Trim trailing spaces**, or **Trim leading and trailing spaces** from the **Edit** menu, or use the Ctrl+L, Ctrl+T, and Ctrl+B keyboard shortcuts.

The leading and/or trailing space characters are removed from the selected syntax lines.

Note: If you do not select any syntax lines, and then select **Trim leading spaces**, **Trim trailing spaces**, or **Trim leading and trailing spaces** or press Ctrl+L, Ctrl+T, or Ctrl+B, the leading and/or trailing spaces are removed from the syntax line where the cursor is placed.

Syntax Editor keyboard shortcuts

The **Edit** menu provides an option to view all available Syntax Editor keyboard shortcuts (**Edit > Syntax keyboard shortcuts**).

Table 22. Syntax Editor keyboard shortcuts

Keystroke	Action
Ctrl+J	Join lines (remove line breaks, creating single line)
Ctrl+U	Duplicate current line, or selection
Ctrl+D	Delete current line, or selection
Ctrl+M	Remove any empty lines within selection
Ctrl+G	Activates column editing mode on selected lines
Esc	Deactivates column editing mode
Alt+Up	Move selected text up
Alt+Down	Move selected text down
Ctrl+L	Trim leading spaces
Ctrl+T	Trim trailing spaces
Ctrl+B	Trim leading and trailing spaces
Ctrl+R	Run selection
Ctrl+I	Auto indent selected text
Tab	Indent selected text
Shift+Tab	Outdent selected text
Ctrl+E	Go to next error
Ctrl+Space	Open auto complete list
Ctrl+Shift+Number	Adds bookmark shortcut number
Ctrl+Number	Go to bookmark shortcut number
Ctrl+Equals	Go to next bookmark
Ctrl+Minus	Go to previous bookmark
Ctrl+Home	Set focus in navigation pane
Insert	Toggle overwrite mode
Ctrl+C	Copy to clipboard
Ctrl+X	Cut to clipboard
Ctrl+V	Paste from clipboard

Running Command Syntax

1. Highlight the commands that you want to run in the syntax window.
2. Click the Run button (the right-pointing triangle) on the Syntax Editor toolbar. It runs the selected commands or the command where the cursor is located if there is no selection.

or
3. Choose one of the items from the Run menu.
 - **All.** Runs all commands in the syntax window, honoring any breakpoints.
 - **Selection.** Runs the currently selected commands, honoring any breakpoints. This includes any partially highlighted commands. If there is no selection, the command where the cursor is positioned is run.
 - **To End.** Runs all commands starting from the first command in the current selection to the last command in the syntax window, honoring any breakpoints. If nothing is selected, the run starts from the command where the cursor is positioned.
 - **Step Through.** Runs the command syntax one command at a time starting from the first command in the syntax window (Step Through From Start) or from the command where the cursor is positioned (Step Through From Current). If there is selected text, the run starts from the first command in the selection. After a given command has run, the cursor advances to the next command and you continue the step through sequence by choosing Continue.
- LOOP-END LOOP, DO IF-END IF, DO REPEAT-END REPEAT, INPUT PROGRAM-END INPUT PROGRAM, and MATRIX-END MATRIX blocks are treated as single commands when using Step Through. You can not step into one of these blocks.
- **Continue.** Continues a run stopped by a breakpoint or Step Through.

Progress Indicator

The progress of a given syntax run is indicated with a downward pointing arrow in the gutter, spanning the last set of commands run. For instance, you choose to run all commands in a syntax window that contains breakpoints. At the first breakpoint, the arrow will span the region from the first command in the window to the command prior to the one containing the breakpoint. At the second breakpoint, the arrow will stretch from the command containing the first breakpoint to the command prior to the one containing the second breakpoint.

Run-time Behavior with Breakpoints

- When running command syntax containing breakpoints, execution stops at each breakpoint. Specifically, the block of command syntax from a given breakpoint (or beginning of the run) to the next breakpoint (or end of the run) is submitted for execution exactly as if you had selected that syntax and chosen Run > Selection.
- You can work with multiple syntax windows, each with its own set of breakpoints, but there is only one queue for executing command syntax. Once a block of command syntax has been submitted--such as the block of command syntax up to the first breakpoint--no other block of command syntax will be executed until the previous block has completed, regardless of whether the blocks are in the same or different syntax windows.
- With execution stopped at a breakpoint, you can run command syntax in other syntax windows, and inspect Data Editor or Viewer windows. However, modifying the contents of the syntax window containing the breakpoint or changing the cursor position in that window will cancel the run.

Character Set Encoding in Syntax Files

The character set encoding of a syntax file can be either Unicode or code page encoding. A Unicode file can contain characters from many different character sets. Code page files are restricted to characters supported in a specific language or locale. For example, a code page file in a western European encoding cannot contain Japanese or Chinese characters.

Reading syntax files

To read syntax files correctly, the syntax editor needs to know the character encoding of the file.

- Files with a Unicode UTF-8 byte order mark are read as Unicode UTF-8 encoding, regardless of any encoding selection you make. This byte order mark is at the beginning of the file, but it is not displayed.
- By default, files without any encoding information are read as Unicode UTF-8 in Unicode mode or the current locale character encoding in code page mode. To override the default behavior, select **Unicode (UTF-8)** or **Local Encoding**.
- **As Declared** is enabled if the syntax file contains a code page encoding identifier at the top of the file. Starting with release 23, a comment is automatically inserted in syntax files that are saved in code page encoding. For example, the first line in the file could be:
* Encoding: en_US.windows-1252.

If you select **As Declared**, that encoding is used to read the file.

Saving syntax files

By default, syntax files are saved as Unicode UTF-8 in Unicode mode or the current locale character encoding in code page mode. To override the default behavior, select **Unicode (UTF-8)** or **Local Encoding** in the Save Syntax As dialog.

- If you save a new syntax file or save the file in a different encoding, a comment is inserted at the top of the file that identifies the encoding. If an encoding comment is already present, it is replaced.
- If you save a syntax file and then save it again without closing it, it is saved in the same encoding.

Multiple Execute Commands

Syntax pasted from dialog boxes or copied from the log or the journal may contain EXECUTE commands. When you run commands from a syntax window, EXECUTE commands are generally unnecessary and may slow performance, particularly with larger data files, because each EXECUTE command reads the entire data file. For more information, see the EXECUTE command in the *Command Syntax Reference* (available from the Help menu in any IBM SPSS Statistics window).

Lag Functions

One notable exception is transformation commands that contain lag functions. In a series of transformation commands without any intervening EXECUTE commands or other commands that read the data, lag functions are calculated after all other transformations, regardless of command order. For example,

```
COMPUTE lagvar=LAG(var1).  
COMPUTE var1=var1*2.
```

and

```
COMPUTE lagvar=LAG(var1).  
EXECUTE.  
COMPUTE var1=var1*2.
```

yield very different results for the value of *lagvar*, since the former uses the transformed value of *var1* while the latter uses the original value.

Character Set Encoding in Syntax Files

The character set encoding of a syntax file can be either Unicode or code page encoding. A Unicode file can contain characters from many different character sets. Code page files are restricted to characters supported in a specific language or locale. For example, a code page file in a western European encoding cannot contain Japanese or Chinese characters.

Reading syntax files

To read syntax files correctly, the syntax editor needs to know the character encoding of the file.

- Files with a Unicode UTF-8 byte order mark are read as Unicode UTF-8 encoding, regardless of any encoding selection you make. This byte order mark is at the beginning of the file, but it is not displayed.
- By default, files without any encoding information are read as Unicode UTF-8 in Unicode mode or the current locale character encoding in code page mode. To override the default behavior, select **Unicode (UTF-8) or Local Encoding**.
- **As Declared** is enabled if the syntax file contains a code page encoding identifier at the top of the file. Starting with release 23, a comment is automatically inserted in syntax files that are saved in code page encoding. For example, the first line in the file could be:
* Encoding: en_US.windows-1252.

If you select **As Declared**, that encoding is used to read the file.

Saving syntax files

By default, syntax files are saved as Unicode UTF-8 in Unicode mode or the current locale character encoding in code page mode. To override the default behavior, select **Unicode (UTF-8) or Local Encoding** in the Save Syntax As dialog.

- If you save a new syntax file or save the file in a different encoding, a comment is inserted at the top of the file that identifies the encoding. If an encoding comment is already present, it is replaced.
- If you save a syntax file and then save it again without closing it, it is saved in the same encoding.

Multiple Execute Commands

Syntax pasted from dialog boxes or copied from the log or the journal may contain EXECUTE commands. When you run commands from a syntax window, EXECUTE commands are generally unnecessary and may slow performance, particularly with larger data files, because each EXECUTE command reads the entire data file. For more information, see the EXECUTE command in the *Command Syntax Reference* (available from the Help menu in any IBM SPSS Statistics window).

Lag Functions

One notable exception is transformation commands that contain lag functions. In a series of transformation commands without any intervening EXECUTE commands or other commands that read the data, lag functions are calculated after all other transformations, regardless of command order. For example,

```
COMPUTE lagvar=LAG(var1).  
COMPUTE var1=var1*2.
```

and

```
COMPUTE lagvar=LAG(var1).  
EXECUTE.  
COMPUTE var1=var1*2.
```

yield very different results for the value of *lagvar*, since the former uses the transformed value of *var1* while the latter uses the original value.

Encrypting syntax files

You can protect your syntax files by encrypting them with a password. Encrypted files can be opened only by providing the password.

Note: Encrypted syntax files cannot be used in production jobs or with the IBM SPSS Statistics Batch Facility (available with IBM SPSS Statistics Server).

To save the contents of the syntax editor as an encrypted syntax file:

1. Make the Syntax Editor the active window (click anywhere in the window to make it active).
2. From the menus, choose:
File > Save As...
3. Select **Encrypted Syntax** from the Save as type drop-down list.
4. Click **Save**.
5. In the Encrypt File dialog box, provide a password and reenter it in the Confirm password text box. Passwords are limited to 10 characters and are case-sensitive.

Warning: Passwords cannot be recovered if they are lost. If the password is lost the file cannot be opened.

Creating strong passwords

- Use eight or more characters.
- Include numbers, symbols and even punctuation in your password.
- Avoid sequences of numbers or characters, such as "123" and "abc", and avoid repetition, such as "111aaa".
- Do not create passwords that use personal information such as birthdays or nicknames.
- Periodically change the password.

Note: Storing encrypted files to an IBM SPSS Collaboration and Deployment Services Repository is not supported.

Modifying encrypted files

- If you open an encrypted file, modify the file and choose File > Save, the modified file is saved with the same password.
- You can change the password on an encrypted file by opening the file, repeating the steps for encrypting it, and specifying a different password in the Encrypt File dialog box.
- You can save an unencrypted version of an encrypted file by opening the file, choosing File > Save As and selecting **Syntax** from the Save as type drop-down list.

Note: Encrypted syntax files cannot be opened in versions of IBM SPSS Statistics prior to version 22.

Chapter 15. Overview of the chart facility

High-resolution charts and plots are created by the procedures on the Graphs menu and by many of the procedures on the Analyze menu. This chapter provides an overview of the chart facility.

Building and editing a chart

Before you can create a chart, you need to have your data in the Data Editor. You can enter the data directly into the Data Editor; open a previously saved data file; or read a spreadsheet, tab-delimited data file, or database file. The Tutorial selection on the Help menu has online examples of creating and modifying a chart, and the online Help system provides information about creating and modifying all chart types.

Building Charts

The Chart Builder allows you to build charts from predefined gallery charts or from the individual parts (for example, axes and bars). You build a chart by dragging and dropping the gallery charts or basic elements onto the **canvas**, which is the large area to the right of the Variables list in the Chart Builder dialog box.

As you are building the chart, the canvas displays a preview of the chart. Although the preview uses defined variable labels and measurement levels, it does not display your actual data. Instead, it uses randomly generated data to provide a rough sketch of how the chart will look.

Using the gallery is the preferred method for new users. For information about using the gallery, see “Building a Chart from the Gallery.”

How to Start the Chart Builder

1. From the menus choose:

Graphs > Chart Builder

Building a Chart from the Gallery

The easiest method for building charts is to use the gallery. Following are general steps for building a chart from the gallery.

1. Click the **Gallery** tab if it is not already displayed.
2. In the Choose From list, select a category of charts. Each category offers several types.
3. Drag the picture of the chart you want onto the canvas. You can also double-click the picture. If the canvas already displays a chart, the gallery chart replaces the axis set and graphic elements on the chart.
 - a. Drag variables from the Variables list and drop them into the axis drop zones and, if available, the grouping drop zone. If an axis drop zone already displays a statistic and you want to use that statistic, you do not have to drag a variable into the drop zone. You need to add a variable to a zone only when the text in the zone is blue. If the text is black, the zone already contains a variable or statistic.

Note: The measurement level of your variables is important. The Chart Builder sets defaults based on the measurement level while you are building the chart. Furthermore, the resulting chart may also look different for different measurement levels. You can temporarily change a variable's measurement level by right-clicking the variable and choosing an option.

- b. If you need to filter chart data, drag-and-drop a categorical variable into the filter drop zone on the chart canvas, and then exclude categories for the selected variable. For more information on chart filtering, see Editing variable filters.

- c. If you need to change statistics or modify attributes of the axes or legends (such as the scale range), click the **Element Properties** tab in the side bar of the Chart Builder. (If the side bar is not displayed, click the button in the upper right corner of the Chart Builder to display the side bar.)
 - d. In the Edit Properties Of list, select the item you want to change. (For information about the specific properties, click **Help**.)
 - e. If you need to add more variables to the chart (for example, for clustering or paneling), click the **Groups/Point ID** tab in the Chart Builder dialog box and select one or more options. Then drag categorical variables to the new drop zones that appear on the canvas.
4. If you want to transpose the chart (for example, to make the bars horizontal), click the **Basic Elements** tab and then click **Transpose**.
 5. Click **OK** to create the chart. The chart is displayed in the Viewer.

Editing Charts

The Chart Editor provides a powerful, easy-to-use environment where you can customize your charts and explore your data. The Chart Editor features:

- **Simple, intuitive user interface.** You can quickly select and edit parts of the chart using menus and toolbars. You can also enter text directly on a chart.
- **Wide range of formatting and statistical options.** You can choose from a full range of styles and statistical options.
- **Powerful exploratory tools.** You can explore your data in various ways, such as by labeling, reordering, and rotating it. You can change chart types and the roles of variables in the chart. You can also add distribution curves and fit, interpolation, and reference lines.
- **Flexible templates for consistent look and behavior.** You can create customized templates and use them to easily create charts with the look and options that you want. For example, if you always want a specific orientation for axis labels, you can specify the orientation in a template and apply the template to other charts.

How to View the Chart Editor

1. Double-click a chart in the Viewer.

Chart Editor Fundamentals

The Chart Editor provides various methods for manipulating charts.

Menus

Many actions that you can perform in the Chart Editor are done with the menus, especially when you are adding an item to the chart. For example, you use the menus to add a fit line to a scatterplot. After adding an item to the chart, you often use the Properties dialog box to specify options for the added item.

Properties Dialog Box

Options for the chart and its chart elements can be found in the Properties dialog box.

To view the Properties dialog box, you can:

1. Double-click a chart element.
or
2. Select a chart element, and then from the menus choose:
Edit > Properties

Additionally, the Properties dialog box automatically appears when you add an item to the chart.

The Properties dialog box has tabs that allow you to set the options and make other changes to a chart. The tabs that you see in the Properties dialog box are based on your current selection.

Some tabs include a preview to give you an idea of how the changes will affect the selection when you apply them. However, the chart itself does not reflect your changes until you click **Apply**. You can make changes on more than one tab before you click **Apply**. If you have to change the selection to modify a different element on the chart, click **Apply** before changing the selection. If you do not click **Apply** before changing the selection, clicking **Apply** at a later point will apply changes only to the element or elements currently selected.

Depending on your selection, only certain settings will be available. The help for the individual tabs specifies what you need to select to view the tabs. If multiple elements are selected, you can change only those settings that are common to all the elements.

Toolbars

The toolbars provide a shortcut for some of the functionality in the Properties dialog box. For example, instead of using the Text tab in the Properties dialog box, you can use the Edit toolbar to change the font and style of the text.

Saving the Changes

Chart modifications are saved when you close the Chart Editor. The modified chart is subsequently displayed in the Viewer.

Chart definition options

When you are defining a chart in the Chart Builder, you can add titles and change options for the chart creation.

Adding and Editing Titles and Footnotes

You can add titles and footnotes to the chart to help a viewer interpret it. The Chart Builder also automatically displays error bar information in the footnotes.

Add or modify titles and footnotes

1. Click the **Titles/Footnotes** tab or click a title or footnote displayed in the preview.
2. Use the **Element Properties** tab to edit the title/footnote text.

For **Title 1**, an **Automatic** title is generated, based on the chart type and the variables used in the chart. This automatic title is used by default unless you select **Custom** or **None**.

How to Remove a Title or Footnote

1. Click the **Titles/Footnotes** tab.
2. Deselect the title or footnote that you want to remove.

How to Edit the Title or Footnote Text

When you add titles and footnotes, you cannot edit their associated text directly on the chart. As with other items in the Chart Builder, you edit them using the **Element Properties** tab.

1. In the **Edit Properties Of** list, select a title, subtitle, or footnote (for example, **Title 1**).
2. In the content box, type the text associated with the title, subtitle, or footnote.

Setting General Options

The Chart Builder offers general options for the chart. These are options that apply to the overall chart, rather than a specific item on the chart. General options include missing value handling, chart size, and panel wrapping.

1. Click the **Options** tab in the side bar of the Chart Builder. (If the side bar is not displayed, click the  button in the upper right corner of the Chart Builder to display the side bar.)
2. Modify the general options. Details about these follow.

User-Missing Values

Break Variables. If there are missing values for the variables used to define categories or subgroups, select **Include** so that the category or categories of user-missing values (values identified as missing by the user) are included in the chart. These "missing" categories also act as break variables in calculating the statistic. The "missing" category or categories are displayed on the category axis or in the legend, adding, for example, an extra bar or a slice to a pie chart. If there are no missing values, the "missing" categories are not displayed.

If you select this option and want to suppress display after the chart is drawn, open the chart in the Chart Editor and choose **Properties** from the Edit menu. Use the Categories tab to move the categories that you want to suppress to the Excluded list. Note, however, that the statistics are not recalculated if you hide the "missing" categories. Therefore, something like a percent statistic will still take the "missing" categories into account.

Note: This control does not affect system-missing values. These are always excluded from the chart.

Summary Statistics and Case Values. You can choose one of the following alternatives for exclusion of cases having missing values:

- **Exclude listwise to obtain a consistent base for the chart.** If any of the variables in the chart has a missing value for a given case, the whole case is excluded from the chart.
- **Exclude variable-by-variable to maximize the use of the data.** If a selected variable has any missing values, the cases having those missing values are excluded when the variable is analyzed.

Chart Size and Panels

Chart Size. Specify a percentage greater than 100 to enlarge the chart or less than 100 to shrink it. The percentage is relative to the default chart size.

Panels. When there are many panel columns, select **Wrap Panels** to allow panels to wrap across rows rather than being forced to fit in a specific row. Unless this option is selected, the panels are shrunk to force them to fit in a row.

Chapter 16. Scoring data with predictive models

The process of applying a predictive model to a set of data is referred to as **scoring** the data. IBM SPSS Statistics has procedures for building predictive models such as regression, clustering, tree, and neural network models. Once a model has been built, the model specifications can be saved in a file that contains all of the information necessary to reconstruct the model. You can then use that model file to generate predictive scores in other datasets. *Note:* Some procedures produce a model XML file, and some procedures produce a compressed file archive (.zip file).

Example. The direct marketing division of a company uses results from a test mailing to assign propensity scores to the rest of their contact database, using various demographic characteristics to identify contacts most likely to respond and make a purchase.

Scoring is treated as a transformation of the data. The model is expressed internally as a set of numeric transformations to be applied to a given set of fields (variables)--the predictors specified in the model--in order to obtain a predicted result. In this sense, the process of scoring data with a given model is inherently the same as applying any function, such as a square root function, to a set of data.

The scoring process consists of two basic steps:

1. Build the model and save the model file. You build the model using a dataset for which the outcome of interest (often referred to as the **target**) is known. For example, if you want to build a model that will predict who is likely to respond to a direct mail campaign, you need to start with a dataset that already contains information on who responded and who did not respond. For example, this might be the results of a test mailing to a small group of customers or information on responses to a similar campaign in the past.

Note: For some model types there is no target outcome of interest. Clustering models, for example, do not have a target, and some nearest neighbor models do not have a target.

2. Apply that model to a different dataset (for which the outcome of interest is not known) to obtain predicted outcomes.

Scoring Wizard

You can use the Scoring Wizard to apply a model created with one dataset to another dataset and generate **scores**, such as the predicted value and/or predicted probability of an outcome of interest.

To score a dataset with a predictive model

1. Open the dataset that you want to score.
2. Open the Scoring Wizard. From the menus choose:
Utilities > Scoring Wizard.
3. Select a model XML file or compressed file archive (.zip file). Use the **Browse** button to navigate to a different location to select a model file.
4. Match fields in the active dataset to fields used in the model. See the topic "Matching model fields to dataset fields" on page 186 for more information.
5. Select the scoring functions you want to use. See the topic "Selecting scoring functions" on page 187 for more information.

Select a Scoring Model. The model file can be an XML file or a compressed file archive (.zip file) that contains model PMML. The list only displays files with a .zip or .xml extension; the file extensions are not displayed in the list. You can use any model file created by IBM SPSS Statistics. You can also use

some model files created by other applications, such as IBM SPSS Modeler, but some model files created by other applications cannot be read by IBM SPSS Statistics, including any models that have multiple target fields (variables).

Model Details. This area displays basic information about the selected model, such as model type, target (if any), and predictors used to build the model. Since the model file has to be read to obtain this information, there may be a delay before this information is displayed for the selected model. If the XML file or .zip file is not recognized as a model that IBM SPSS Statistics can read, a message indicating that the file cannot be read is displayed.

Matching model fields to dataset fields

In order to score the active dataset, the dataset must contain fields (variables) that correspond to all the predictors in the model. If the model also contains split fields, then the dataset must also contain fields that correspond to all the split fields in the model.

- By default, any fields in the active dataset that have the same name and type as fields in the model are automatically matched.
- Use the drop-down list to match dataset fields to model fields. The data type for each field must be the same in both the model and the dataset in order to match fields.
- You cannot continue with the wizard or score the active dataset unless all predictors (and split fields if present) in the model are matched with fields in the active dataset.

Dataset Fields. The drop-down list contains the names of all the fields in the active dataset. Fields that do not match the data type of the corresponding model field cannot be selected.

Model Fields. The fields used in the model.

Role. The displayed role can be one of the following:

- **Predictor.** The field is used as a predictor in the model. That is, values of the predictors are used to "predict" values of the target outcome of interest.
- **Split.** The values of the split fields are used to define subgroups, which are each scored separately. There is a separate subgroup for each unique combination of split field values. (Note: splits are only available for some models.)
- **Record ID.** Record (case) identifier.

Measure. Measurement level for the field as defined in the model. For models in which measurement level can affect the scores, the measurement level as defined in the model is used, not the measurement level as defined in the active dataset. For more information on measurement level, see "Variable measurement level" on page 51.

Type. Data type as defined in the model. The data type in the active dataset must match the data type in the model. Data type can be one of the following:

- **String.** Fields with a data type of string in the active dataset match the data type of string in the model.
- **Numeric.** Numeric fields with display formats other than date or time formats in the active dataset match the numeric data type in the model. This includes F (numeric), Dollar, Dot, Comma, E (scientific notation), and custom currency formats. Fields with Wkday (day of week) and Month (month of year) formats are also considered numeric, not dates. For some model types, date and time fields in the active dataset are also considered a match for the numeric data type in the model.
- **Date.** Numeric fields with display formats that include the date but not the time in the active dataset match the date type in the model. This includes Date (dd-mm-yyyy), Adate (mm/dd/yyyy), Edate (dd.mm.yyyy), Sdate (yyyy/mm/dd), and Jdate (dddyyyy).
- **Time.** Numeric fields with display formats that include the time but not the date in the active dataset match the time data type in the model. This includes Time (hh:mm:ss) and Dtime (dd hh:mm:ss)

- **Timestamp.** Numeric fields with a display format that includes both the date and the time in the active dataset match the timestamp data type in the model. This corresponds to the Datetime format (dd-mm-yyyy hh:mm:ss) in the active dataset.

Note: In addition to field name and type, you should make sure that the actual data values in the dataset being scored are recorded in the same fashion as the data values in the dataset used to build the model. For example, if the model was built with an *Income* field that has income divided into four categories, and *IncomeCategory* in the active dataset has income divided into six categories or four different categories, those fields don't really match each other and the resulting scores will not be reliable.

Missing Values

This group of options controls the treatment of missing values, encountered during the scoring process, for the predictor variables defined in the model. A missing value in the context of scoring refers to one of the following:

- A predictor contains no value. For numeric fields (variables), this means the system-missing value. For string fields, this means a null string.
- The value has been defined as user-missing, in the model, for the given predictor. Values defined as user-missing in the active dataset, but not in the model, are not treated as missing values in the scoring process.
- The predictor is categorical and the value is not one of the categories defined in the model.

Use Value Substitution. Attempt to use value substitution when scoring cases with missing values. The method for determining a value to substitute for a missing value depends on the type of predictive model.

- **Linear Regression and Discriminant models.** For independent variables in linear regression and discriminant models, if mean value substitution for missing values was specified when building and saving the model, then this mean value is used in place of the missing value in the scoring computation, and scoring proceeds. If the mean value is not available, then the system-missing value is returned.
- **Decision Tree models.** For the CHAID and Exhaustive CHAID models, the biggest child node is selected for a missing split variable. The biggest child node is the one with the largest population among the child nodes using learning sample cases. For C&RT and QUEST models, surrogate split variables (if any) are used first. (Surrogate splits are splits that attempt to match the original split as closely as possible using alternate predictors.) If no surrogate splits are specified or all surrogate split variables are missing, the biggest child node is used.
- **Logistic Regression models.** For covariates in logistic regression models, if a mean value of the predictor was included as part of the saved model, then this mean value is used in place of the missing value in the scoring computation, and scoring proceeds. If the predictor is categorical (for example, a factor in a logistic regression model), or if the mean value is not available, then the system-missing value is returned.

Use System-Missing. Return the system-missing value when scoring a case with a missing value.

Selecting scoring functions

The scoring functions are the types of "scores" available for the selected model. For example, predicted value of the target, probability of the predicted value, or probability of a selected target value.

Scoring function. The scoring functions available are dependent on the model. One or more of the following will be available in the list:

- **Predicted value.** The predicted value of the target outcome of interest. This is available for all models, except those that do not have a target.

- **Probability of predicted value.** The probability of the predicted value being the correct value, expressed as a proportion. This is available for most models with a categorical target.
- **Probability of selected value.** The probability of the selected value being the correct value, expressed as a proportion. Select a value from the drop-down list in the Value column. The available values are defined by the model. This is available for most models with a categorical target.
- **Confidence.** A probability measure associated with the predicted value of a categorical target. For Binary Logistic Regression, Multinomial Logistic Regression, and Naive Bayes models, the result is identical to the probability of the predicted value. For Tree and Ruleset models, the confidence can be interpreted as an adjusted probability of the predicted category and is always less than the probability of the predicted value. For these models, the confidence value is more reliable than the probability of the predicted value.
- **Node number.** The predicted terminal node number for Tree models.
- **Standard error.** The standard error of the predicted value. Available for Linear Regression models, General Linear models, and Generalized Linear models with a scale target. This is available only if the covariance matrix is saved in the model file.
- **Cumulative Hazard.** The estimated cumulative hazard function. The value indicates the probability of observing the event at or before the specified time, given the values of the predictors.
- **Nearest neighbor.** The ID of the nearest neighbor. The ID is the value of the case labels variable, if supplied, and otherwise the case number. Applies only to nearest neighbor models.
- **Kth nearest neighbor.** The ID of the k th nearest neighbor. Enter an integer for the value of k in the Value column. The ID is the value of the case labels variable, if supplied, and otherwise the case number. Applies only to nearest neighbor models.
- **Distance to nearest neighbor.** The distance to the nearest neighbor. Depending on the model, either Euclidean or City Block distance will be used. Applies only to nearest neighbor models.
- **Distance to k th nearest neighbor.** The distance to the k th nearest neighbor. Enter an integer for the value of k in the Value column. Depending on the model, either Euclidean or City Block distance will be used. Applies only to nearest neighbor models.

Field Name. Each selected scoring function saves a new field (variable) in the active dataset. You can use the default names or enter new names. If fields with those names already exist in the active dataset, they will be replaced. For information on field naming rules, see “Variable names” on page 50.

Value. See the descriptions of the scoring functions for descriptions of functions that use a Value setting.

Scoring the active dataset

On the final step of the wizard, you can score the active dataset or paste the generated command syntax to a syntax window. You can then modify and/or save the generated command syntax.

Merging model and transformation XML files

Some predictive models are built with data that have been modified or transformed in various ways. In order to apply those models to other datasets in a meaningful way, the same transformations must also be performed on the dataset being scored, or the transformations must also be reflected in the model file. Including transformations in the model file is a two-step process:

1. Save the transformations in a transformation XML file. This can only be done using TMS BEGIN and TMS END in command syntax.
2. Combine the model file (XML file or .zip file) and the transformation XML file in a new, merged model XML file.

To combine a model file and a transformation XML file in a new, merged model file:

3. From the menus choose:
4. Utilities > Merge Model XML
5. Select the model XML file.

5. Select the transformation XML file.
6. Enter a path and name for the new merged model XML file, or use **Browse** to select the location and name.

Note: You cannot merge model .zip files for models that contain splits (separate model information for each split group) or ensemble models with transformation XML files.

Chapter 17. Utilities

Utilities

This chapter describes the functions found on the Utilities menu and the ability to reorder target variable lists.

- For information on the Scoring Wizard, see Chapter 16, “Scoring data with predictive models,” on page 185.
 - For information on merging model and transformation XML files, see “Merging model and transformation XML files” on page 188.
-

Variable information

The Variables dialog box displays variable definition information for the currently selected variable, including:

- Variable label
- Data format
- User-missing values
- Value labels
- Measurement level

Visible. The Visible column in the variable list indicates if the variable is currently visible in the Data Editor and in dialog box variable lists.

Go To. Goes to the selected variable in the Data Editor window.

Paste. Pastes the selected variables into the designated syntax window at the cursor location.

To modify variable definitions, use the Variable view in the Data Editor.

There are two ways to open the Variables dialog box.

- From the menus, choose **Utilities > Variables....**
- Right-click a variable name in the Data Editor and select **Variable Information...** from the context menu.

Data file comments

You can include descriptive comments with a data file. For IBM SPSS Statistics data files, these comments are saved with the data file.

To add, modify, delete, or display data file comments

1. From the menus choose:
Utilities > Data File Comments...
2. To display the comments in the Viewer, select **Display comments in output.**

Comments can be any length but are limited to 80 bytes (typically 80 characters in single-byte languages) per line; lines will automatically wrap at 80 characters. Comments are displayed in the same font as text output to accurately reflect how they will appear when displayed in the Viewer.

A date stamp (the current date in parentheses) is automatically appended to the end of the list of comments whenever you add or modify comments. This may lead to some ambiguity concerning the dates associated with comments if you modify an existing comment or insert a new comment between existing comments.

Variable sets

You can restrict the variables that are displayed in the Data Editor and in dialog box variable lists by defining and using variable sets. This is particularly useful for data files with a large number of variables. Small variable sets make it easier to find and select the variables for your analysis.

Defining variable sets

Define Variable Sets creates subsets of variables to display in the Data Editor and in dialog box variable lists. Defined variable sets are saved with IBM SPSS Statistics data files.

Set Name. Set names can be up to 64 bytes. Any characters, including blanks, can be used.

Variables in Set. Any combination of numeric and string variables can be included in a set. The order of variables in the set has no effect on the display order of the variables in the Data Editor or in dialog box variable lists. A variable can belong to multiple sets.

To define variable sets

1. From the menus choose:
 Utilities > Define Variable Sets...
2. Select the variables that you want to include in the set.
3. Enter a name for the set (up to 64 bytes).
4. Click **Add Set**.

Using variable sets to show and hide variables

Use Variable Sets restricts the variables displayed in the Data Editor and in dialog box variable lists to the variables in the selected (checked) sets.

- The set of variables displayed in the Data Editor and in dialog box variable lists is the union of all selected sets.
- A variable can be included in multiple selected sets.
- The order of variables in the selected sets and the order of selected sets have no effect on the display order of variables in the Data Editor or in dialog box variable lists.
- Although the defined variable sets are saved with IBM SPSS Statistics data files, the list of currently selected sets is reset to the default, built-in sets each time you open the data file.

The list of available variable sets includes any variable sets defined for the active dataset, plus two built-in sets:

- **ALLVARIABLES.** This set contains all variables in the data file, including new variables created during a session.
- **NEWVARIABLES.** This set contains only new variables created during the session.

Note: Even if you save the data file after creating new variables, the new variables are still included in the **NEWVARIABLES** set until you close and reopen the data file.

At least one variable set must be selected. If **ALLVARIABLES** is selected, any other selected sets will not have any visible effect, since this set contains all variables.

1. From the menus choose:

Utilities > Use Variable Sets...

2. Select the defined variable sets that contain the variables that you want to appear in the Data Editor and in dialog box variable lists.

To display all variables

1. From the menus choose:

Utilities > Show All Variables

Reordering target variable lists

Variables appear on dialog box target lists in the order in which they are selected from the source list. If you want to change the order of variables on a target list—but you don't want to deselect all of the variables and reselect them in the new order—you can move variables up and down on the target list using the Ctrl key (Macintosh: Command key) with the up and down arrow keys. You can move multiple variables simultaneously if they are contiguous (grouped together). You cannot move noncontiguous groups of variables.

Chapter 18. Options

Options

Options control a wide variety of settings:

- Session journal, which keeps a record of all commands run in every session
- Display order for variables in dialog box source lists
- Items that are displayed and hidden in new output results
- TableLook for new pivot tables
- Custom currency formats

Options control various settings.

To change options settings

1. From the menus, choose:
Edit > Options...
2. Click the tabs for the settings that you want to change.
3. Change the settings.
4. Click **OK** or **Apply**.

General options

Variable Lists

These settings control the display of variables in dialog box lists. You can display variable names or variable labels. Names or labels can be displayed in alphabetical or file order or grouped by measurement level. Display order affects only source variable lists. Target variable lists always reflect the order in which variables were selected.

Roles

Some dialogs support the ability to pre-select variables for analysis based on defined roles. See the topic “Roles” on page 54 for more information.

- **Use predefined roles.** By default, pre-select variables based on defined roles.
- **Use custom assignments.** By default, do *not* use roles to pre-select variables.

You can also switch between predefined roles and custom assignment within the dialogs that support this functionality. The setting here controls only the initial default behavior in effect for each dataset.

Maximum Number of Threads

The number of threads that multithreaded procedures use when calculating results. The **Automatic** setting is based on the number of available processing cores. Specify a lower value if you want to make more processing resources available to other applications while multithreaded procedures are running. This option is disabled in distributed analysis mode.

Windows

Look and feel. Controls the basic appearance of windows and dialog boxes. If you notice any display issues after changing the look and feel, try shutting down and restarting the application.

Open syntax window at startup. Syntax windows are text file windows used to enter, edit, and run commands. If you frequently work with command syntax, select this option to automatically open a syntax window at the beginning of each session. This is useful primarily for experienced users who prefer to work with command syntax instead of dialog boxes.

Open only one dataset at a time. Closes the currently open data source each time you open a different data source using the menus and dialog boxes. By default, every time you use the menus and dialog boxes to open a new data source, that data source opens in a new Data Editor window, and any other data sources open in other Data Editor windows remain open and available during the session until explicitly closed.

When you select this option, it takes effect immediately but does not close any datasets that were open at the time the setting was changed. This setting has no effect on data sources opened using command syntax, which relies on DATASET commands to control multiple datasets. See the topic Chapter 6, "Working with Multiple Data Sources," on page 65 for more information.

Output

No scientific notation for small numbers in tables. Suppresses the display of scientific notation for small decimal values in output. Very small decimal values will be displayed as 0 (or 0.000). This setting only affects output with the "general" format, which is determined by the application. This setting does not affect the display of significance levels or other statistics with a standard range of values. The format of many numeric values in pivot tables is based on the format of the variable associated with the numeric value.

Apply locale's digit grouping format to numeric values. Applies the current locale's digit grouping format to numeric values in pivot tables and charts as well as in the Data Editor. For example, in a French locale with this setting enabled the value 34419,57 will be displayed as 34 419,57.

The grouping format does not apply to trees, Model Viewer items, numeric values with the DOT or COMMA format, or numeric values with a DOLLAR or custom currency format. It does, however, apply to the display of the days value for numeric values with a DTIME format—for example, to the value of ddd in the format ddd hh:mm.

Display a leading zero for decimal values. Displays leading zeros for numeric values that consist only of a decimal part. For example, when leading zeros are displayed, the value .123 is displayed as 0.123. This setting does not apply to numeric values that have a currency or percent format. Except for fixed ASCII files (*.dat), leading zeros are not included when the data are saved to an external file.

Measurement System. The measurement system used (points, inches, or centimeters) for specifying attributes such as pivot table cell margins, cell widths, and space between tables for printing.

Notification. Controls the manner in which the program notifies you that it has finished running a procedure and that the results are available in the Viewer.

Viewer options

Viewer output display options affect only new output that is produced after you change the settings. Output that is already displayed in the Viewer is not affected by changes in these settings.

Initial Output State. Controls which items are automatically displayed or hidden each time that you run a procedure and how items are initially aligned. You can control the display of the following items: log, warnings, notes, titles, pivot tables, charts, tree diagrams, and text output. You can also turn the display of commands in the log on or off. You can copy command syntax from the log and save it in a syntax file.

Note: All output items are displayed left-aligned in the Viewer. Only the alignment of printed output is affected by the justification settings. Centered and right-aligned items are identified by a small symbol.

Title. Controls the font style, size, and color for new output titles.

Page Title. Controls the font style, size, and color for new page titles and page titles that are generated by TITLE and SUBTITLE command syntax or created by **New Page Title** on the **Insert** menu.

Text Output Font that is used for text output. Text output is designed for use with a monospaced (fixed-pitch) font. If you select a proportional font, tabular output does not align properly.

Default Page Setup. Controls the default options for orientation and margins for printing.

Data Options

Transformation and Merge Options. Each time the program executes a command, it reads the data file. Some data transformations (such as Compute and Recode) and file transformations (such as Add Variables and Add Cases) do not require a separate pass of the data, and execution of these commands can be delayed until the program reads the data to execute another command, such as a statistical or charting procedure.

- For large data files, where reading the data can take some time, you may want to select **Calculate values before used** to delay execution and save processing time. When this option is selected, the results of transformations you make using dialog boxes such as Compute Variable will not appear immediately in the Data Editor; new variables created by transformations will be displayed without any data values; and data values in the Data Editor cannot be changed while there are pending transformations. Any command that reads the data, such as a statistical or charting procedure, will execute the pending transformations and update the data displayed in the Data Editor. Alternatively, you can use **Run Pending Transforms** on the Transform menu.
- With the default setting of **Calculate values immediately**, when you paste command syntax from dialogs, an EXECUTE command is pasted after each transformation command. See the topic “Multiple Execute Commands” on page 177 for more information.

Display Format for New Numeric Variables. Controls the default display width and number of decimal places for new numeric variables. There is no default display format for new string variables. If a value is too large for the specified display format, first decimal places are rounded and then values are converted to scientific notation. Display formats do not affect internal data values. For example, the value 123456.78 may be rounded to 123457 for display, but the original unrounded value is used in any calculations.

Set Century Range for 2-Digit Years. Defines the range of years for date-format variables entered and/or displayed with a two-digit year (for example, 10/28/86, 29-OCT-87). The automatic range setting is based on the current year, beginning 69 years prior to and ending 30 years after the current year (adding the current year makes a total range of 100 years). For a custom range, the ending year is automatically determined based on the value that you enter for the beginning year.

Random Number Generator. Two different random number generators are available:

- *Version 12 Compatible.* The random number generator used in version 12 and previous releases. If you need to reproduce randomized results generated in previous releases based on a specified seed value, use this random number generator.

- *Mersenne Twister*. A newer random number generator that is more reliable for simulation purposes. If reproducing randomized results from version 12 or earlier is not an issue, use this random number generator.

Assigning Measurement Level. For data read from external file formats, older IBM SPSS Statistics data files (prior to release 8.0) and new fields created in a session, measurement level for numeric fields is determined by a set of rules, including the number of unique values. You can specify the minimum number of data values for a numeric variable used to classify the variable as continuous (scale) or nominal. Variables with fewer than the specified number of unique values are classified as nominal.

There are numerous other conditions that are evaluated prior to applying the minimum number of data values rule when determining to apply the continuous (scale) or nominal measurement level. Conditions are evaluated in the order listed in the table below. The measurement level for the first condition that matches the data is applied.

Table 23. Rules for determining default measurement level

Condition	Measurement Level
Format is dollar or custom-currency	Continuous
Format is date or time (excluding Month and Wkday)	Continuous
All values of a variable are missing	Nominal
Variable contains at least one non-integer value	Continuous
Variable contains at least one negative value	Continuous
Variable contains no valid values less than 10,000	Continuous
Variable has N or more valid, unique values*	Continuous
Variable has no valid values less than 10	Continuous
Variable has less than N valid, unique values*	Nominal

* N is the user-specified cut-off value. The default is 24.

Rounding and Truncation of Numeric Values. For the RND and TRUNC functions, this setting controls the default threshold for rounding up values that are very close to a rounding boundary. The setting is specified as a number of bits and is set to 6 at install time, which should be sufficient for most applications. Setting the number of bits to 0 produces the same results as in release 10. Setting the number of bits to 10 produces the same results as in releases 11 and 12.

- For the RND function, this setting specifies the number of least-significant bits by which the value to be rounded may fall short of the threshold for rounding up but still be rounded up. For example, when rounding a value between 1.0 and 2.0 to the nearest integer this setting specifies how much the value can fall short of 1.5 (the threshold for rounding up to 2.0) and still be rounded up to 2.0.
- For the TRUNC function, this setting specifies the number of least-significant bits by which the value to be truncated may fall short of the nearest rounding boundary and be rounded up before truncating. For example, when truncating a value between 1.0 and 2.0 to the nearest integer this setting specifies how much the value can fall short of 2.0 and be rounded up to 2.0.

Customize Variable View. Controls the default display and order of attributes in Variable View. See the topic “Changing the default variable view” on page 199 for more information.

Change Dictionary. Controls the language version of the dictionary used for checking the spelling of items in Variable View. See the topic “Spell checking” on page 57 for more information.

Changing the default variable view

You can use Customize Variable View to control which attributes are displayed by default in Variable View (for example, name, type, label) and the order in which they are displayed.

Click **Customize Variable View**.

1. Select (check) the variable attributes you want to display.
2. Use the up and down arrow buttons to change the display order of the attributes.

Language options

Language

Output language. Controls the language that is used in the output. Does not apply to simple text output. The list of available languages depends on the currently installed language files. (Note: This setting does not affect the user interface language.) Depending on the language, you might also need to use Unicode character encoding for characters to render properly.

Note: Custom scripts that rely on language-specific text strings in the output might not run correctly when you change the output language. For more information, see the topic “Script options” on page 205.

User Interface. This setting controls the language that is used in menus, dialogs, and other user interface features. (Note: This setting does not affect the output language.)

Character Encoding and Locale

This controls the default behavior for determining the encoding for reading and writing data files and syntax files. You can change these settings only when there are no open data sources, and the settings remain in effect for subsequent sessions until explicitly changed.

Unicode (universal character set). Use Unicode encoding (UTF-8) for reading and writing files. This mode is referred to as Unicode mode.

Locale's writing system. Use the current locale selection to determine the encoding for reading and writing files. This mode is referred to as code page mode.

Locale. You can select a locale from the list or enter any valid locale value. OS writing system sets the locale to the writing system of the operating system. The application gets this information from the operating system each time you start the application. The locale setting is primarily relevant in code page mode, but it can also affect how some characters are rendered in Unicode mode.

There are a number of important implications for Unicode mode and Unicode files:

- IBM SPSS Statistics data files and syntax files that are saved in Unicode encoding should not be used in releases of IBM SPSS Statistics before 16.0. For syntax files, you can specify the encoding when you save the file. For data files, you should open the data file in code page mode and then resave it if you want to read the file with earlier versions.
- When code page data files are read in Unicode mode, the defined width of all string variables is tripled. To automatically set the width of each string variable to the longest observed value for that variable, select **Minimize string widths based on observed values** in the Open Data dialog box.

Bidirectional Text

If you use a combination of right-to-left (for example, Arabic or Hebrew) languages and left-to-right languages (for example, English), select the direction for text flow. Individual words will still flow in the

correct direction, based on the language. This option controls only the text flow for complete blocks of text (for example, all the text that is entered in an edit field).

- **Automatic.** Text flow is determined by characters that are used in each word. This option is the default.
- **Right-to-left.** Text flows right to left.
- **Left-to-right.** Text flows left to right.

Currency options

You can create up to five custom currency display formats that can include special prefix and suffix characters and special treatment for negative values.

The five custom currency format names are CCA, CCB, CCC, CCD, and CCE. You cannot change the format names or add new ones. To modify a custom currency format, select the format name from the source list and make the changes that you want.

Prefixes, suffixes, and decimal indicators defined for custom currency formats are for display purposes only. You cannot enter values in the Data Editor using custom currency characters.

To create custom currency formats

1. Click the **Currency** tab.
2. Select one of the currency formats from the list (CCA, CCB, CCC, CCD, and CCE).
3. Enter the prefix, suffix, and decimal indicator values.
4. Click **OK** or **Apply**.

Output options

Output options control the default setting for a number of output options.

Outline labeling. Controls display of variable names, variable labels, data values, and value labels in the outline pane of the Viewer.

Pivot table labeling. Controls display of variable names, variable labels, data values, and value labels in pivot tables.

Descriptive variable and value labels (Variable view in the Data Editor, *Label* and *Values* columns) often make it easier to interpret your results. However, long labels can be awkward in some tables. Changing labeling options here only affects the default display for new output.

One click descriptives. Controls options for descriptive statistics generated for selected variables in the Data Editor. See the topic “[Obtaining Descriptive Statistics for Selected Variables](#)” on page 62 for more information.

- **Suppress tables with many categories.** For variables with more than the specified number of unique values, frequency tables will not be displayed.
- **Include a chart in the output.** For nominal and ordinal variables and variables with an unknown measurement level, a bar chart is displayed. For continuous (scale) variables, a histogram is displayed.

Output display. Controls which type of output is generated for procedures that can generate either Model Viewer output or pivot table and chart output. This setting only applies to the Generalized Linear Mixed Models and Nonparametric Tests procedures.

Screen Reader Accessibility. Controls how pivot table row and column labels are read by screen readers. You can read full row and column labels for each data cell or read only the labels that change as you move between data cells in the table.

Chart options

Chart Template. New charts can use either the settings selected here or the settings from a chart template file. Click **Browse** to select a chart template file. To create a chart template file, create a chart with the attributes that you want and save it as a template (choose **Save Chart Template** from the File menu).

Chart Aspect Ratio. The width-to-height ratio of the outer frame of new charts. You can specify a width-to-height ratio from 0.1 to 10.0. Values less than 1 make charts that are taller than they are wide. Values greater than 1 make charts that are wider than they are tall. A value of 1 produces a square chart. Once a chart is created, its aspect ratio cannot be changed.

Current Settings. Available settings include:

- **Font.** Font used for all text in new charts.
- **Style Cycle Preference.** The initial assignment of colors and patterns for new charts. **Cycle through colors only** uses only colors to differentiate chart elements and does not use patterns. **Cycle through patterns only** uses only line styles, marker symbols, or fill patterns to differentiate chart elements and does not use color.
- **Frame.** Controls the display of inner and outer frames on new charts.
- **Grid Lines.** Controls the display of scale and category axis grid lines on new charts.

Style Cycles. Customizes the colors, line styles, marker symbols, and fill patterns for new charts. You can change the order of the colors and patterns that are used when a new chart is created.

Data Element Colors

Specify the order in which colors should be used for the data elements (such as bars and markers) in your new chart. Colors are used whenever you select a choice that includes *color* in the Style Cycle Preference group in the main Chart Options dialog box.

For example, if you create a clustered bar chart with two groups and you select **Cycle through colors, then patterns** in the main Chart Options dialog box, the first two colors in the Grouped Charts list are used as the bar colors on the new chart.

To Change the Order in Which Colors Are Used

1. Select **Simple Charts** and then select a color that is used for charts without categories.
2. Select **Grouped Charts** to change the color cycle for charts with categories. To change a category's color, select a category and then select a color for that category from the palette.

Optionally, you can:

- Insert a new category above the selected category.
- Move a selected category.
- Remove a selected category.
- Reset the sequence to the default sequence.
- Edit a color by selecting its well and then clicking **Edit**.

Data Element Lines

Specify the order in which styles should be used for the line data elements in your new chart. Line styles are used whenever your chart includes line data elements and you select a choice that includes *patterns* in the Style Cycle Preference group in the main Chart Options dialog box.

For example, if you create a line chart with two groups and you select **Cycle through patterns only** in the main Chart Options dialog box, the first two styles in the Grouped Charts list are used as the line patterns on the new chart.

To Change the Order in Which Line Styles Are Used

1. Select **Simple Charts** and then select a line style that is used for line charts without categories.
2. Select **Grouped Charts** to change the pattern cycle for line charts with categories. To change a category's line style, select a category and then select a line style for that category from the palette.

Optionally, you can:

- Insert a new category above the selected category.
- Move a selected category.
- Remove a selected category.
- Reset the sequence to the default sequence.

Data Element Markers

Specify the order in which symbols should be used for the marker data elements in your new chart. Marker styles are used whenever your chart includes marker data elements and you select a choice that includes *patterns* in the Style Cycle Preference group in the main Chart Options dialog box.

For example, if you create a scatterplot chart with two groups and you select **Cycle through patterns only** in the main Chart Options dialog box, the first two symbols in the Grouped Charts list are used as the markers on the new chart.

To Change the Order in Which Marker Styles Are Used

1. Select **Simple Charts** and then select a marker symbol that is used for charts without categories.
2. Select **Grouped Charts** to change the pattern cycle for charts with categories. To change a category's marker symbol, select a category and then select a symbol for that category from the palette.

Optionally, you can:

- Insert a new category above the selected category.
- Move a selected category.
- Remove a selected category.
- Reset the sequence to the default sequence.

Data Element Fills

Specify the order in which fill styles should be used for the bar and area data elements in your new chart. Fill styles are used whenever your chart includes bar or area data elements, and you select a choice that includes *patterns* in the Style Cycle Preference group in the main Chart Options dialog box.

For example, if you create a clustered bar chart with two groups and you select **Cycle through patterns only** in the main Chart Options dialog box, the first two styles in the Grouped Charts list are used as the bar fill patterns on the new chart.

To Change the Order in Which Fill Styles Are Used

1. Select **Simple Charts** and then select a fill pattern that is used for charts without categories.
2. Select **Grouped Charts** to change the pattern cycle for charts with categories. To change a category's fill pattern, select a category and then select a fill pattern for that category from the palette.

Optionally, you can:

- Insert a new category above the selected category.

- Move a selected category.
- Remove a selected category.
- Reset the sequence to the default sequence.

Pivot table options

Pivot Table options set various options for the display of pivot tables.

TableLook

Select a TableLook from the list of files and click **OK** or **Apply**. You can use one of the TableLooks provided with IBM SPSS Statistics, or you can create your own in the Pivot Table Editor (choose **TableLooks** from the Format menu).

- **Browse.** Allows you to select a TableLook from another directory.
- **Set TableLook Directory.** Allows you to change the default TableLook directory. Use **Browse** to navigate to the directory you want to use, select a TableLook in that directory, and then select **Set TableLook Directory**.

Note: TableLooks created in earlier versions of IBM SPSS Statistics cannot be used in version 16.0 or later.

Column Widths

These options control the automatic adjustment of column widths in pivot tables.

- **Adjust for labels only.** Adjusts column width to the width of the column label. This produces more compact tables, but data values wider than the label may be truncated.
- **Adjust for labels and data for all tables.** Adjusts column width to whichever is larger: the column label or the largest data value. This produces wider tables, but it ensures that all values will be displayed.

Table Comments

You can automatically include comments for each table.

- Comment text is displayed in a tooltip when you hover over a table in the Viewer.
- Screen readers read the comment text when the table has focus.
- The tooltip in the Viewer displays only the first 200 characters of the comment, but screen readers read the entire text.
- When you export output to HTML or a web report, the comment text is used as alt text.

Title. Include the table title in the comment.

Procedure. Include the name of the procedure that created the table.

Date. Includes the date the table was created.

Dataset. Includes the name of the dataset that was used to create the table.

Default Editing Mode

This option controls activation of pivot tables in the Viewer window or in a separate window. By default, double-clicking a pivot table activates all but very large tables in the Viewer window. You can choose to activate pivot tables in a separate window or select a size setting that will open smaller pivot tables in the Viewer window and larger pivot tables in a separate window.

Copying wide tables to the clipboard in rich text format

When pivot tables are pasted in Word/RTF format, tables that are too wide for the document width will either be wrapped, scaled down to fit the document width, or left unchanged.

File locations options

Options on the File Locations tab control the default location that the application will use for opening and saving files at the start of each session, the location of the journal file, the location of the temporary folder, the number of files that appear in the recently used file list, and the installation of Python 2.7 that is used by the IBM SPSS Statistics - Integration Plug-in for Python.

Startup Folders for Open and Save Dialog Boxes

- **Specified folder.** The specified folder is used as the default location at the start of each session. You can specify different default locations for data and other (non-data) files.
- **Last folder used.** The last folder used to either open or save files in the previous session is used as the default at the start of the next session. This applies to both data and other (non-data) files.

These settings apply only to dialog boxes for opening and saving files—and the "last folder used" is determined by the last dialog box used to open or save a file. Files opened or saved via command syntax have no effect on—and are not affected by—these settings. These settings are only available in local analysis mode. In distributed analysis mode connected to a remote server (requires IBM SPSS Statistics Server), you cannot control the startup folder locations.

Session Journal

You can use the session journal to automatically record commands run in a session. This includes commands entered and run in syntax windows and commands generated by dialog box choices. You can edit the journal file and use the commands again in other sessions. You can turn journaling off and on, append or overwrite the journal file, and select the journal filename and location. You can copy command syntax from the journal file and save it in a syntax file.

Temporary Folder

This specifies the location of temporary files created during a session. In distributed mode (available with the server version), this does not affect the location of temporary data files. In distributed mode, the location of temporary data files is controlled by the environment variable *SPSSTMPDIR*, which can be set only on the computer running the server version of the software. If you need to change the location of the temporary directory, contact your system administrator.

Recently Used File List

This controls the number of recently used files that appear on the File menu.

Python 2.7 and Python 3 Locations

These settings specify the installations of Python 2.7 and 3.4 that are used by the IBM SPSS Statistics - Integration Plug-in for Python when you are running Python from SPSS Statistics. By default, the distributions of Python 2.7 and 3.4 that are installed with SPSS Statistics (as part of IBM SPSS Statistics - Essentials for Python) are used. They are in the Python (contains Python 2.7) and Python3 directories under the directory where SPSS Statistics is installed. To use a different installation of Python 2.7 or Python 3.4 on your computer, specify the path to the root directory of that installation of Python. The setting takes effect in the current session unless Python was already started by running a Python program, a Python script, or an extension command that is implemented in Python. In that case, the setting will be in effect in the next session. This setting is only available in local analysis mode. In

distributed analysis mode (requires IBM SPSS Statistics Server), the Python location on the remote server is set from the IBM SPSS Statistics Administration Console. Contact your system administrator for assistance.

Script options

Use the Scripts tab to specify the default script language and any autoscripts you want to use. You can use scripts to automate many functions, including customizing pivot tables.

Note: Legacy Sax Basic users must manually convert any custom autoscripts. The autoscripts installed with pre-16.0 versions are available as a set of separate script files located in the *Samples* subdirectory of the directory where IBM SPSS Statistics is installed. By default, no output items are associated with autoscripts. You must manually associate all autoscripts with the output items, as described below. For information on converting legacy autoscripts, see “Compatibility with Versions Prior to 16.0” on page 279.

Default script language. The default script language determines the script editor that is launched when new scripts are created. It also specifies the default language whose executable will be used to run autoscripts. The available scripting languages depend on your platform. For Windows, the available scripting languages are Basic, which is installed with the Core system, and the Python programming language. For all other platforms, scripting is available with the Python programming language. For the Python programming language, you can choose Python 2 or Python 3 as the default script language.

To use scripting with the Python programming language, you need IBM SPSS Statistics - Essentials for Python, which is installed by default with your IBM SPSS Statistics product.

Enable Autoscripting. This check box allows you to enable or disable autoscripting. By default, autoscripting is enabled.

Base Autoscript. An optional script that is applied to all new Viewer objects before any other autoscripts are applied. Specify the script file to be used as the base autoscript as well as the language whose executable will be used to run the script.

To Apply Autoscripts to Output Items

1. In the Command Identifiers grid, select a command that generates output items to which autoscripts will be applied.
The *Objects* column, in the Objects and Scripts grid, displays a list of the objects associated with the selected command. The *Script* column displays any existing script for the selected command.
2. Specify a script for any of the items shown in the *Objects* column. Click the corresponding *Script* cell. Enter the path to the script or click the ellipsis (...) button to browse for the script.
3. Specify the language whose executable will be used to run the script. *Note:* The selected language is not affected by changing the default script language.
4. Click **Apply** or **OK**.

To remove autoscript associations

1. In the Objects and Scripts grid, click the cell in the *Script* column corresponding to the script to dissociate.
2. Delete the path to the script and then click any other cell in the Objects and Scripts grid.
3. Click **Apply** or **OK**.

Syntax editor options

Syntax Color Coding

You can turn color coding of commands, subcommands, keywords, keyword values, and comments off or on and you can specify the font style and color for each.

Error Color Coding

You can turn color coding of certain syntactical errors off or on and you can specify the font style and color used. Both the command name and the text (within the command) containing the error are color coded, and you can choose different styles for each. See the topic “Color Coding” on page 169 for more information.

Auto-Complete Settings

You can turn automatic display of the auto-complete control off or on. The auto-complete control can always be displayed by pressing **Ctrl+Spacebar**. See the topic “Auto-Completion” on page 169 for more information.

Indent size

Specifies the number of spaces in an indent. The setting applies to indenting selected lines of syntax as well as to automatic indentation.

Gutter

These options specify the default behavior for showing or hiding line numbers and command spans within the Syntax Editor’s gutter—the region to the left of the text pane that is reserved for line numbers, bookmarks, breakpoints, and command spans. Command spans are icons that provide visual indicators of the start and end of a command.

Panes

Display the navigation pane. This specifies the default for showing or hiding the navigation pane. The navigation pane contains a list of all recognized commands in the syntax window, displayed in the order in which they occur. Clicking on a command in the navigation pane positions the cursor at the start of the command.

Automatically open Error Tracking pane when errors are found. This specifies the default for showing or hiding the error tracking pane when run-time errors are found.

Optimize for right to left languages. Check this box for the best user experience when working with right to left languages.

Paste syntax from dialogs. Specifies the position at which syntax is inserted into the designated syntax window when pasting syntax from a dialog. **After last command** inserts pasted syntax after the last command. **At cursor or selection** inserts pasted syntax at the position of the cursor; or, if a block of syntax is selected, then the selection will be replaced by the pasted syntax.

Multiple imputations options

The Multiple Imputations tab controls two kinds of preferences related to Multiple Imputations:

Appearance of Imputed Data. By default, cells containing imputed data will have a different background color than cells containing nonimputed data. The distinctive appearance of the imputed data should make it easy for you to scroll through a dataset and locate those cells. You can change the default cell background color, the font, and make the imputed data display in bold type.

Analysis Output. This group controls the type of Viewer output produced whenever a multiply imputed dataset is analyzed. By default, output will be produced for the original (pre-imputation) dataset and for each of the imputed datasets. In addition, for those procedures that support pooling of imputed data, final pooled results will be generated. When univariate pooling is performed, pooling diagnostics will also display. However, you can suppress any output you do not want to see.

To Set Multiple Imputation Options

From the menus, choose:

Edit > Options

Click the Multiple Imputation tab.

Chapter 19. Customizing Menus and Toolbars

Customizing Menus and Toolbars

Menu Editor

You can use the Menu Editor to customize your menus. With the Menu Editor you can:

- Add menu items that run customized scripts.
- Add menu items that run command syntax files.
- Add menu items that launch other applications and automatically send data to other applications.

You can send data to other applications in the following formats: IBM SPSS Statistics, Excel, Lotus 1-2-3, tab-delimited, and dBASE IV.

To Add Items to Menus

1. From the menus choose:
View > Menu Editor...
2. In the Menu Editor dialog box, double-click the menu (or click the plus sign icon) to which you want to add a new item.
3. Select the menu item above which you want the new item to appear.
4. Click **Insert Item** to insert a new menu item.
5. Enter the text for the new item. On Windows operating systems, an ampersand (&) before a letter specifies that letter should be used as the underlined mnemonic key.
6. Select the file type for the new item (script file, command syntax file, or external application).
7. Click **Browse** to select a file to attach to the menu item.

You can also add entirely new menus and separators between menu items. Optionally, you can automatically send the contents of the Data Editor to another application when you select that application on the menu.

Customizing Toolbars

You can customize toolbars and create new toolbars. Toolbars can contain any of the available tools, including tools for all menu actions. They can also contain custom tools that launch other applications, run command syntax files, or run script files.

Show Toolbars

Use Show Toolbars to show or hide toolbars, customize toolbars, and create new toolbars. Toolbars can contain any of the available tools, including tools for all menu actions. They can also contain custom tools that launch other applications, run command syntax files, or run script files.

To Customize Toolbars

1. From the menus choose:
View > Toolbars > Customize
2. Select the toolbar you want to customize and click **Edit**, or click **New** to create a new toolbar.
3. For new toolbars, enter a name for the toolbar, select the windows in which you want the toolbar to appear, and click **Edit**.

4. Select an item in the Categories list to display available tools in that category.
5. Drag and drop the tools you want onto the toolbar displayed in the dialog box.
6. To remove a tool from the toolbar, drag it anywhere off the toolbar displayed in the dialog box.
To create a custom tool to open a file, to run a command syntax file, or to run a script:
7. Click **New Tool** in the Edit Toolbar dialog box.
8. Enter a descriptive label for the tool.
9. Select the action you want for the tool (open a file, run a command syntax file, or run a script).
10. Click **Browse** to select a file or application to associate with the tool.

New tools are displayed in the User-Defined category, which also contains user-defined menu items.

Toolbar Properties

Use Toolbar Properties to select the window types in which you want the selected toolbar to appear. This dialog box is also used for creating names for new toolbars.

To Set Toolbar Properties

1. From the menus choose:
View > Toolbars > Customize
2. For existing toolbars, click **Edit**, and then click **Properties** in the Edit Toolbar dialog box.
3. For new toolbars, click **New Tool**.
4. Select the window types in which you want the toolbar to appear. For new toolbars, also enter a toolbar name.

Edit Toolbar

Use the Edit Toolbar dialog box to customize existing toolbars and create new toolbars. Toolbars can contain any of the available tools, including tools for all menu actions. They can also contain custom tools that launch other applications, run command syntax files, or run script files.

To Change Toolbar Images

1. Select the tool for which you want to change the image on the toolbar display.
2. Click **Change Image**.
3. Select the image file that you want to use for the tool. The following image formats are supported:
BMP, PNG, JPG.
 - Images should be square. Images that are not square are clipped to a square area.
 - Images are automatically scaled to fit. For optimal display, use 16x16 pixel images for small toolbar images or 32x32 pixel images for large toolbar images.

Create New Tool

Use the Create New Tool dialog box to create custom tools to launch other applications, run command syntax files, and run script files.

Chapter 20. Extensions

Extensions are custom components that extend the capabilities of IBM SPSS Statistics. Extensions are packaged in extension bundles (.spe files) and are installed to IBM SPSS Statistics. Extensions can be created by any user and shared with other users by sharing the associated extension bundle.

The following utilities are provided for working with extensions:

- The “Extension Hub,” which is accessed from **Extensions > Extension Hub**, is an interface for searching for, downloading, and installing extensions from the IBM SPSS Predictive Analytics collection on GitHub. From the Extension Hub dialog, you can also view details of the extensions that are installed on your computer, get updates for installed extensions, and remove extensions.
- You can install an extension bundle that is stored on your local computer from **Extensions > Install Local Extension Bundle**.
- You can use the Custom Dialog Builder for Extensions to create an extension that includes a user interface, which is referred to as a custom dialog. Custom dialogs generate SPSS Statistics command syntax that carries out the tasks that are associated with the extension. You design the generated syntax as part of designing the custom dialog.
- You can use the Create Extension Bundle and Edit Extension Bundle dialogs for cases where you can't use the Custom Dialog Builder for Extensions. For more information, see the topic “Creating and editing extension bundles” on page 252.

Extension Hub

From the Extension Hub dialog, you can do the following tasks:

- Explore extensions that are available from the IBM SPSS Predictive Analytics collection on GitHub. You can select extensions to install now or you can download selected extensions and install them later.
- Get updated versions of extensions that are already installed on your computer.
- View details about the extensions that are already installed on your computer.
- Remove extensions that are installed on your computer.

To download or remove extensions:

1. From the menus, choose: **Extensions > Extension Hub**
2. Select the extensions that you want to download or remove and click **OK**. All selections that are made on the Explore and Installed tabs are processed when you click **OK**.

By default, the extensions that are selected for download are downloaded and installed on your computer. From the Settings tab, you can choose to download the selected extensions to a specified location without installing them. You can then install them later by choosing **Extensions > Install Local Extension Bundle**. For Windows and Mac, you can install an extension by double-clicking the extension bundle file.

Important:

- Extensions are always installed, or downloaded, to your local computer. If you work in distributed analysis mode, then see the topic “Installing local extension bundles” on page 214 for further details.
- For Windows 7 and later, installing an updated version of an existing extension bundle might require running IBM SPSS Statistics with administrator privileges. You can start IBM SPSS Statistics with administrator privileges by right-clicking the icon for IBM SPSS Statistics and choosing **Run as administrator**. In particular, if you receive an error message that states that one or more extension bundles could not be installed, then try running with administrator privileges.

Note: The license that you agree to when you install an extension can be viewed at any later time by clicking **More info...** for the extension on the Installed tab.

Explore tab

The Explore tab displays all of the extensions that are available from the IBM SPSS Predictive Analytics collection on GitHub (<https://ibmpredictiveanalytics.github.io/>). From the Explore tab, you can select new extensions to download and install, and you can select updates for extensions that are already installed on your computer. The Explore tab requires an internet connection.

- For each extension, the number of the latest version and the associated date of that version are displayed. A brief summary of the extension is also provided. For extensions that are already installed on your computer, the installed version number is also displayed.
- You can view detailed information about an extension by clicking **More info**. When an update is available, **More info** displays information about the update.
- You can view the prerequisites for running an extension, such as whether the IBM SPSS Statistics - Integration Plug-in for R is required, by clicking **Prerequisites**. When an update is available, **Prerequisites** displays information about the update.

Refine by

You can refine the set of extensions that are displayed. You can refine by general categories of extensions, the language in which the extension is implemented, the type of organization that provided the extension, or the state of the extension. For each group, such as Category, you can select multiple items by which to refine the displayed list of extensions. You can also refine by search terms. Searches are not case-sensitive, and the asterisk (*) is treated as any other character and does not indicate a wildcard search.

- To refine the displayed list of extensions, click **Apply**. Pressing the enter key when the cursor is in the **Search** box has the same effect as clicking **Apply**.
- To reset the list to display all available extensions, delete any text in the **Search** box, deselect all items, and click **Apply**.

Installed tab

The Installed tab displays all of the extensions that are installed on your computer. From the Installed tab, you can select updates for installed extensions that are available from the IBM SPSS Predictive Analytics collection on GitHub and you can remove extensions. To get updates for installed extensions, you must have an internet connection.

- For each extension, the installed version number is displayed. When an internet connection is available, the number of the latest version and the associated date of that version are displayed. A brief summary of the extension is also provided.
- You can view detailed information about an extension by clicking **More info**. When an update is available, **More info** displays information about the update.
- You can view the prerequisites for running an extension, such as whether the IBM SPSS Statistics - Integration Plug-in for R is required, by clicking **Prerequisites**. When an update is available, **Prerequisites** displays information about the update.

Refine by

You can refine the set of extensions that are displayed. You can refine by general categories of extensions, the language in which the extension is implemented, the type of organization that provided the extension, or the state of the extension. For each group, such as Category, you can select multiple items by which to refine the displayed list of extensions. You can also refine by search terms. Searches are not case-sensitive, and the asterisk (*) is treated as any other character and does not indicate a wildcard search.

- To refine the displayed list of extensions, click **Apply**. Pressing the enter key when the cursor is in the **Search** box has the same effect as clicking **Apply**.
- To reset the list to display all available extensions, delete any text in the **Search** box, deselect all items, and click **Apply**.

Private extensions

Private extensions are extensions that are installed on your computer but are not available from the IBM SPSS Predictive Analytics collection on GitHub. The features for refining the set of displayed extensions and for viewing the prerequisites to run an extension are not available for private extensions.

Note: When using the Extension Hub without an internet connection, some of the features of the Installed tab might not be available.

Settings

The Settings tab specifies whether extensions that are selected for download are downloaded and then installed or downloaded but not installed. This setting applies to new extensions and updates to existing extensions. You might choose to download extensions without installing them if you are downloading extensions to distribute to other users within your organization. You might also choose to download, but not install, extensions if you don't have the prerequisites for running the extensions but plan to get the prerequisites.

If you choose to download extensions without installing them, you can install them later by choosing **Extensions > Install Local Extension Bundle**. For Windows and Mac, you can install an extension by double-clicking the extension bundle file.

Extension Details

The Extension Details dialog box displays the information that was provided by the author of the extension. In addition to required information, such as Summary, and Version, the author might have included URLs to locations of relevance, such as the author's home page. If the extension was downloaded from the Extension Hub, then it includes a license that can be viewed by clicking **View license**.

Components. The Components group lists the custom dialogs, if any, and the names of any extension commands that are included in the extension. Extension commands can be run from the syntax editor in the same manner as built-in IBM SPSS Statistics commands. Help for an extension command might be available by positioning the cursor within the command (in a syntax window) and pressing the F1 key. Help might also be available by running `CommandName /HELP` in the syntax editor.

Note: Installing an extension that contains a custom dialog might require a restart of IBM SPSS Statistics to see the entry for the dialog in the Components table.

Dependencies. The Dependencies group lists add-ons that are required to run the components included in the extension.

- **Integration Plug-Ins for Python and R.** The components for an extension may require the Integration Plug-in for Python, or the Integration Plug-in for R, or both. The Integration Plug-in for Java™ is installed with the Core system and does not require separate installation.
- **R packages.** Lists any R packages that are required by the extension. During installation of the extension, the installer attempts to download and install the necessary packages on your machine. If this process fails, you will be alerted and you will then need to manually install the packages. See the topic "Required R packages" on page 215 for more information.
- **Python modules.** Lists any Python modules that are required by the extension. Any such modules may be available from the IBM SPSS Predictive Analytics community at <https://developer.ibm.com/>

`predictiveanalytics/`. Copy the modules to a location specified for extension commands as shown in the output from the SHOW EXTPATHS command. Alternatively, you can copy the modules to a location on the Python search path, such as the Python site-packages directory.

- **Extensions.** Lists the names of any extensions that are required by the current extension. Any such extensions might be available for download from the Extension Hub.

Installing local extension bundles

To install an extension bundle that is stored on your local computer:

1. From the menus choose:

Extensions > Install Local Extension Bundle...

2. Select the extension bundle. Extension bundles have a file type of spe.

You can also install extension bundles with a command line utility that allows you to install multiple bundles at once. See the topic “Batch installation of extension bundles” on page 216 for more information.

Installing extension bundles in distributed mode

If you work in distributed mode, then you must install the extension bundle on the associated IBM SPSS Statistics Server, using the command line utility. If the extension bundle contains a custom dialog, then you must also install the extension bundle on your local machine (from the menus, as described previously). If you don't know whether the extension bundle contains a custom dialog, then it is best to install the extension bundle on your local machine and on the IBM SPSS Statistics Server machine. For information on using the command line utility, see “Batch installation of extension bundles” on page 216.

Important: For users of Windows 7 and later versions of Windows, installing an updated version of an existing extension bundle might require running IBM SPSS Statistics with administrator privileges. You can start IBM SPSS Statistics with administrator privileges by right-clicking the icon for IBM SPSS Statistics and choosing **Run as administrator**. In particular, if you receive an error message that states that one or more extension bundles could not be installed, then try running with administrator privileges.

Installation locations for extensions

By default, extensions are installed to a general user-writable location for your operating system. To view the location, run the SHOW EXTPATHS syntax command. The output displays a list of locations, under the heading "Locations for extension commands," for installing extensions. Extensions are installed to the first writable location in the list.

You can override the default location by defining a path with the SPSS_EXTENSIONS_PATH environment variable. For multiple locations, separate each with a semicolon on Windows and a colon on Linux and Mac. The specified location must exist on the target computer. After you set SPSS_EXTENSIONS_PATH, you must restart IBM SPSS Statistics for the changes to take effect.

Custom dialogs, contained in extension bundles, are installed to the first writable location in the list under the heading "Locations for custom dialogs" in the output from the SHOW EXTPATHS syntax command. You can override the default location by defining a path with the SPSS_CDIALOGS_PATH environment variable, with the same specifications as for the SPSS_EXTENSIONS_PATH environment variable.

Note: For an extension that is implemented in Python 3, you can specify alternative installation locations with the SPSSEX_EXTENSIONS_PATH and SPSSEX_CDIALOGS_PATH environment variables. The SPSSEX_EXTENSIONS_PATH and SPSSEX_CDIALOGS_PATH variables have the same specifications as SPSS_EXTENSIONS_PATH and SPSS_CDIALOGS_PATH and take precedence over them and over the default installation location for extensions. The SPSSEX_EXTENSIONS_PATH and SPSSEX_CDIALOGS_PATH environment variables apply only in the case that Python 3 is specified as the Python Version for the extension. You

can view the Python Version from the Extension Details dialog, which is accessed by clicking **More info** for the associated extension in the Extension Hub dialog.

To create an environment variable on Windows, from the Control Panel:

Windows 7

1. Select User Accounts.
2. Select **Change my environment variables**.
3. Click **New**, enter the name of the environment variable (for instance, SPSS_EXTENSIONS_PATH) in the **Variable name** field and enter the path or paths in the **Variable value** field.

Windows 8 or Later

1. Select System.
2. Select the Advanced tab and click **Environment Variables**. The Advanced tab is accessed from Advanced system settings.
3. In the User variables section, click **New**, enter the name of the environment variable (for instance, SPSS_EXTENSIONS_PATH) in the **Variable name** field and enter the path or paths in the **Variable value** field.

Important: For users of Windows 7 and later versions of Windows, installing an updated version of an existing extension bundle might require running IBM SPSS Statistics with administrator privileges. You can start IBM SPSS Statistics with administrator privileges by right-clicking the icon for IBM SPSS Statistics and choosing **Run as administrator**. In particular, if you receive an error message that states that one or more extension bundles could not be installed, then try running with administrator privileges.

Required R packages

The extension installer attempts to download and install any R packages that are required by the extension and not found on your computer. If you do not have internet access, you will need to obtain the necessary packages from someone who does. If installation of the packages fails, you will be alerted with the list of required packages. You can also view the list from the Extension Details dialog box, once the extension is installed. See the topic “Extension Details” on page 213 for more information. Packages can be downloaded from and then installed from within R. For details, see the *R Installation and Administration* guide, distributed with R.

Note: For UNIX (including Linux) users, packages are downloaded in source form and then compiled. This requires that you have the appropriate tools installed on your machine. See the *R Installation and Administration* guide for details. In particular, Debian users should install the r-base-dev package from `apt-get install r-base-dev`.

Permissions

By default, required R packages are installed to the library folder under the location where R is installed (for example, C:\Program Files\R\R-3.3.3\library on Windows). If you do not have write permission to this location or would like to store R packages--installed for extensions--elsewhere, you can specify one or more alternate locations by defining the SPSS_RPACKAGES_PATH environment variable. When present, the paths specified in SPSS_RPACKAGES_PATH are added to the R library search path and take precedence over the default location. R packages will be installed to the first writable location. For multiple locations, separate each with a semicolon on Windows and a colon on Linux and Mac. The specified locations must exist on the target machine. After setting SPSS_RPACKAGES_PATH, you will need to restart IBM SPSS Statistics for the changes to take effect. For information on how to set an environment variable on Windows, see “Installation locations for extensions” on page 214.

Batch installation of extension bundles

You can install multiple extension bundles at once using the batch utility `installextbundles.bat` (`installextbundles.sh` for Mac and UNIX systems) located in the IBM SPSS Statistics installation directory. For Windows and Mac, the utility is located at the root of the installation directory. For Linux and IBM SPSS Statistics Server for UNIX, the utility is located in the `bin` subdirectory of the installation directory. The utility is designed to run from a command prompt and must be run from its installed location. The form of the command line is:

```
installextbundles [-statssrv] [-download no|yes] -source <folder> | <filename>...
```

-statssrv. Specifies that you are running the utility on a IBM SPSS Statistics Server. You should also install the same extension bundles on client machines that connect to the server.

-download no|yes. Specifies whether the utility has permission to access the internet in order to download any R packages required by the specified extension bundles. The default is no. If you keep the default or do not have internet access then you will need to manually install any required R packages. See the topic “Required R packages” on page 215 for more information.

-source <folder> | <filename>... Specifies the extension bundles to install. You can specify the path to a folder containing extension bundles or you can specify a list of filenames of extension bundles. If you specify a folder, all extension bundles (files of type `.spe`) found in the folder will be installed. Separate multiple filenames with one or more spaces. Enclose paths with double-quotes if they contain spaces.

Note: When running `installextbundles.sh` on IBM SPSS Statistics Server for UNIX, the Bash shell must be present on the server machine.

Creating and managing custom dialogs

The Custom Dialog Builder creates custom dialogs for generating command syntax.

Using the Custom Dialog Builder you can:

- Create your own version of a dialog for a built-in IBM SPSS Statistics procedure. For example, you can create a dialog for the Frequencies procedure that only allows the user to select the set of variables and then generates command syntax with pre-set options that standardize the output.
- Create a user interface that generates command syntax for an extension command. Extension commands are user-defined IBM SPSS Statistics commands that are implemented in the Python programming language, R or Java. See the topic “Custom Dialogs for Extension Commands” on page 250 for more information.
- Open a file containing the specification for a custom dialog--perhaps created by another user--and add the dialog to your installation of IBM SPSS Statistics, optionally making your own modifications.
- Save the specification for a custom dialog so that other users can add it to their installations of IBM SPSS Statistics.

The Custom Dialog Builder supports two modes of operation:

Custom Dialog Builder for Extensions

This mode creates and modifies custom dialogs within new or existing extensions. When you open the Custom Dialog Builder in this mode, a new extension that contains an empty custom dialog is created. When you save or install custom dialogs in this mode, they are saved or installed as part of an extension.

Dialogs that you create in this mode are referred to as *enhanced* custom dialogs. Enhanced custom dialogs have many new features that were not available in releases of IBM SPSS Statistics before release 24. In that regard, enhanced dialogs are incompatible with pre-24 releases.

To open the Custom Dialog Builder in this mode, from the menus choose:

Extensions > Custom Dialog Builder for Extensions

Custom Dialog Builder - Compatibility mode

This mode creates and modifies custom dialogs that are compatible with all releases of IBM SPSS Statistics and are referred to as *compatible custom dialogs*. Compatible custom dialogs do not have the added features that are available with enhanced dialogs.

- The controls and control properties that are supported in compatibility mode are the same controls and properties that are supported in the Custom Dialog Builder in pre-24 releases.
- When you save a dialog in compatibility mode, it is saved as a compatible custom dialog package (.spd) file, which you can then add to an extension.
- You can convert a dialog that is open in compatibility mode to an enhanced dialog that is then part of a new extension.

To open the Custom Dialog Builder in compatibility mode, from the menus choose:

Extensions > Utilities > Custom Dialog Builder (Compatibility mode)...

Custom Dialog Builder layout

Canvas

The canvas is the area of the Custom Dialog Builder where you design the layout of your dialog.

Properties Pane

The properties pane is the area of the Custom Dialog Builder where you specify properties of the controls that make up the dialog as well as properties of the dialog itself, such as the menu location.

Tools Palette

The tools palette provides the set of controls that can be included in a custom dialog. You can show or hide the Tools Palette by choosing Tools Palette from the **View** menu.

Syntax Template

The Syntax Template specifies the command syntax that is generated by the custom dialog. You can move the Syntax Template pane to a separate window by clicking **Move to New Window**. To move a separate Syntax Template window back into the Custom Dialog Builder, click **Restore to Main Window**.

Building a custom dialog

The basic steps involved in building a custom dialog are:

Note: If you are working in compatibility mode, then some special considerations apply as noted in what follows.

1. Specify the properties of the dialog itself, such as the title that appears when the dialog is launched and the location of the new menu item for the dialog within the IBM SPSS Statistics menus. See the topic “Dialog Properties” on page 218 for more information.
2. Specify the controls, such as source and target variable lists, that make up the dialog and any sub-dialogs. See the topic “Control types” on page 222 for more information.
3. Create the syntax template that specifies the command syntax to be generated by the dialog. See the topic “Building the Syntax Template” on page 220 for more information.
4. Specify properties of the extension that contains your dialog. See the topic “Extension Properties” on page 243 for more information.

Note: This step does not apply in Compatibility mode.

5. Install the extension that contains the dialog to IBM SPSS Statistics and/or save the extension to an extension bundle (.spe) file. See the topic “Managing custom dialogs” on page 246 for more information.

Note: In Compatibility mode, you install the dialog to IBM SPSS Statistics and/or save the specification for the dialog to a compatible custom dialog package (.spd) file. See the topic “Managing custom dialogs in compatibility mode” on page 248 for more information.

You can preview your dialog as you're building it. See the topic “Previewing a custom dialog” on page 222 for more information.

Dialog Properties

To view and set Dialog Properties:

1. Click on the canvas in an area outside of any controls. With no controls on the canvas, Dialog Properties are always visible.

Dialog Name. The Dialog Name property is required and specifies a unique name to associate with the dialog. To minimize the possibility of name conflicts, you may want to prefix the dialog name with an identifier for your organization, such as a URL.

Menu Location. Click the ellipsis (...) button to open the Menu Location dialog box, which allows you to specify the name and location of the menu item for the custom dialog.

Title. The Title property specifies the text to be displayed in the title bar of the dialog box.

Help File. The Help File property is optional and specifies the path to a help file for the dialog. This is the file that will be launched when the user clicks the **Help** button on the dialog. Help files must be in HTML format. A copy of the specified help file is included with the specifications for the dialog when the dialog is installed or saved. The Help button on the run-time dialog is hidden if there is no associated help file.

- Localized versions of the help file that exist in the same directory as the help file are automatically added to the dialog when you add the help file. Localized versions of the help file are named <Help File>_<language identifier>.htm. For more information, see the topic “Creating Localized Versions of Custom Dialogs” on page 250.
- Supporting files, such as image files and style sheets, can be added to the dialog by first saving the dialog. You then manually add the supporting files to the dialog file (.cfe or .spd). For information about accessing and manually modifying custom dialog files, see the section that is titled “To localize dialog strings” in the topic “Creating Localized Versions of Custom Dialogs” on page 250.

Web Deployment Properties. Allows you to associate a properties file with this dialog for use in building thin client applications that are deployed over the web.

Modeless. Specifies whether the dialog is modal or modeless. When a dialog is modal, it must be closed before the user can interact with the main application windows (Data, Output, and Syntax) or with other open dialogs. Modeless dialogs do not have that constraint. The default is modeless.

Data Source. Specifies the source of the fields that are displayed in Source List, Target List, and Field Chooser controls. By default, the active dataset is the data source. You can specify a Dataset Selector control as the data source. At run time, the dataset that is selected in the control is the data source. Click the ellipsis (...) button to open the dialog box and specify the data source.

Required Add-Ons. Specifies one or more add-ons--such as the Integration Plug-in for Python or the Integration Plug-in for R--that are required in order to run the command syntax generated by the dialog.

For example, if the dialog generates command syntax for an extension command implemented in R, then check the box for the Integration Plug-in for R. Users will be warned about required add-ons that are missing when they try to install or run the dialog.

Note: The Required Add-Ons property applies only in Compatibility mode. When you are working with the Custom Dialog Builder for Extensions, any required add-ons are specified as part of the extension properties.

Specifying the Data Source for a Dialog

The Data Source dialog box specifies the source of the fields that are displayed in Source List, Target List, and Field Chooser controls. By default, the active dataset is the data source. You can specify a Dataset Selector control as the data source. At run time, the dataset that is selected in the control is the data source.

Specifying the Menu Location for a Custom Dialog

The Menu Location dialog box is used to specify the name and location of the menu item for a custom dialog. Menu items for custom dialogs do not appear in the Menu Editor within IBM SPSS Statistics. If you are working in compatibility mode, then see the Compatibility Mode section for instructions.

1. Double-click the menu (or click the plus sign icon) to which you want to add the item for the new dialog. By default, the dialog is installed to all window types (Data Editor, Syntax, and Viewer) and the menus that are common to all window types are displayed. You can choose to install the dialog to a particular window type, and display the menus for that window type, by selecting the window type from the **Install to** list.
2. Select the menu item above which you want the item for the new dialog to appear and click **Add**. After the item is added, you can use **Move Up** and **Move Down** to reposition it. You can also move the item under a new submenu by clicking **Move Under New Menu**. When the dialog is installed, the submenu is added to the IBM SPSS Statistics menu system if it is not already present.
3. Enter the name of the menu item. Names within a particular menu or submenu must be unique.

Compatibility Mode

1. Double-click the menu (or click the plus sign icon) to which you want to add the item for the new dialog.
If you want to create custom menus or submenus, use the Menu Editor. For more information, see the topic “Menu Editor” on page 209, and also see the associated note in the Note section.
2. Select the menu item above which you want the item for the new dialog to appear. After the item is added, you can use the **Move Up** and **Move Down** buttons to reposition it.
3. Enter a title for the menu item. Titles within a particular menu or submenu must be unique.
4. Click **Add**.

Options

- Add a separator above or below the new menu item.
- Specify the path to an image that appears next to the menu item for the custom dialog. The supported image types are gif and png. The image cannot be larger than 16 x 16 pixels.

Note:

- The Menu Location dialog box displays menus for all add-on modules. Be sure to add the menu item for your custom dialog to a menu that is available to you or other users of your dialog.
- If you add your dialog to a menu that was created from the Menu Editor (**View>Menu Editor**), then users of your dialog must manually create the same menu from their Menu Editor. Otherwise, the dialog is added to their **Extensions** menu.

Laying out controls on the canvas

You add controls to a custom dialog by dragging them from the tools palette onto the canvas. The canvas is divided into four functional columns in which you can place controls. The following considerations apply:

- The first (leftmost) column is primarily intended for a Source List control. Source List controls must in fact be in the first column.
- Target List controls must be in the second column.
- Sub-dialog buttons must be in the rightmost column (for example, the third column if only three columns are used) and no other controls can be in the same column as Sub-dialog buttons. In that regard, the fourth column can contain only Sub-dialog buttons.

Note: In compatibility mode, only three columns are available for laying out controls.

Although not shown on the canvas, each custom dialog contains **OK**, **Paste**, **Cancel**, and **Help** buttons positioned across the bottom of the dialog. The presence and locations of these buttons is automatic, however, the **Help** button is hidden if there is no help file associated with the dialog (as specified by the Help File property in Dialog Properties).

You can change the vertical order of the controls within a column by dragging them up or down, but the exact position of the controls is determined automatically for you. At run-time, controls resize in appropriate ways when the dialog itself is resized. Controls such as source and target lists automatically expand to fill the available space below them.

Building the Syntax Template

The syntax template specifies the command syntax that will be generated by the custom dialog. A single custom dialog can generate command syntax for any number of built-in IBM SPSS Statistics commands or extension commands.

The syntax template can consist of static text that is always generated and control identifiers that are replaced at run time with the values of the associated custom dialog controls. For example, command names and subcommand specifications that don't depend on user input would be static text. The set of variables that is specified in a target list, however, would be represented with the control identifier for the target list control.

To Build the Syntax Template

1. For static command syntax that does not depend on user-specified values, enter the syntax as you would in the Syntax Editor. The Syntax Template supports the auto-completion and color coding features of the Syntax Editor. See the topic "Using the Syntax Editor" on page 167 for more information.
2. Add control identifiers of the form `%%Identifier%%` at the locations where you want to insert command syntax that is generated by controls, where `Identifier` is the value of the Identifier property for the control.
 - You can insert a control identifier by selecting a row in the table of identifiers, right-clicking and selecting **Add to syntax template**. You can also insert a control identifier by right-clicking a control on the canvas and selecting **Add to syntax template**.
 - You can also select from a list of available control identifiers by pressing **Ctrl+Spacebar**. The list contains the control identifiers followed by the items available with the syntax auto-completion feature.

If you manually enter identifiers, retain any spaces, since all spaces in identifiers are significant.

At run time, and for all controls other than check boxes and check box groups, each identifier is replaced with the current value of the Syntax property of the associated control. For check boxes and check box groups, the identifier is replaced by the current value of the Checked Syntax or Unchecked

Syntax property of the associated control, depending on the current state of the control--checked or unchecked. See the topic “Control types” on page 222 for more information.

Note: The syntax that is generated at run-time automatically includes a command terminator (period) as the very last character if one is not present.

Example: Including Run-time Values in the Syntax Template

Consider a simplified version of the Frequencies dialog that contains only a source list control and a target list control, and generates command syntax of the following form:

```
FREQUENCIES VARIABLES=var1 var2...
/FORMAT = NOTABLE
/BARCHART.
```

The syntax template to generate this syntax might look like:

```
FREQUENCIES VARIABLES=%target_list%
/FORMAT = NOTABLE
/BARCHART.
```

- `%target_list%` is the value of the Identifier property for the target list control. At run time it will be replaced by the current value of the Syntax property of the control.
- Defining the Syntax property of the target list control to be `%%ThisValue%%` specifies that at run time, the current value of the property will be the value of the control, which is the set of variables in the target list.

Example: Including Command Syntax from Container Controls

Building on the previous example, consider adding a Statistics sub-dialog that contains a single group of check boxes that allow a user to specify mean, standard deviation, minimum, and maximum. Assume that the check boxes are contained in an item group control.

An example of the generated command syntax would be:

```
FREQUENCIES VARIABLES=var1 var2...
/FORMAT = NOTABLE
/STATISTICS MEAN STDDEV
/BARCHART.
```

The syntax template to generate this syntax might look like:

```
FREQUENCIES VARIABLES=%target_list%
/FORMAT = NOTABLE
%%stats_group%%
/BARCHART.
```

- `%target_list%` is the value of the Identifier property for the target list control and `%%stats_group%%` is the value of the Identifier property for the item group control.

The following definitions illustrate one way to specify the Syntax properties of the item group and the check boxes that are contained within it to generate the correct result. The Syntax property of the target list would be set to `%%ThisValue%%`, as described in the previous example.

Syntax property of item group: `/STATISTICS %%ThisValue%%`

Checked Syntax property of mean check box: `MEAN`

Checked Syntax property of stddev check box: `STDDEV`

Checked Syntax property of min check box: `MINIMUM`

Checked Syntax property of max check box: `MAXIMUM`

At run time, `%%stats_group%%` will be replaced by the current value of the Syntax property for the item group control. Specifically, `%%ThisValue%%` will be replaced by a blank-separated list of the values of the Checked or Unchecked Syntax property of each check box, depending on its state--checked or unchecked. Since we only specified values for the Checked Syntax property, only checked boxes will contribute to `%%ThisValue%%`. For example, if the user checks the mean and standard deviation boxes, the run time value of the Syntax property for the item group will be /STATISTICS MEAN STDDEV.

If no boxes are checked, then the Syntax property for the item group control will be empty, and the generated command syntax will not contain any reference to `%%stats_group%%`. This might or might not be desirable. For example, even with no boxes checked you might still want to generate the STATISTICS subcommand. This can be accomplished by referencing the identifiers for the check boxes directly in the syntax template, as in:

```
FREQUENCIES VARIABLES=%%target_list%%
/FORMAT = NOTABLE
/STATISTICS %%stats_mean%% %%stats_stddev%% %%stats_min%% %%stats_max%%
/BARCHART.
```

Previewing a custom dialog

You can preview the dialog that is currently open in the Custom Dialog Builder. The dialog appears and functions as it would when run from the menus within IBM SPSS Statistics.

- Source variable lists are populated with dummy variables that can be moved to target lists.
- The **Paste** button pastes command syntax into the designated syntax window.
- The **OK** button closes the preview.
- If a help file is specified, the **Help** button is enabled and will open the specified file. If no help file is specified, the help button is disabled when previewing, and hidden when the actual dialog is run.

To preview a custom dialog:

1. From the menus in the Custom Dialog Builder choose:

File > Preview Dialog

Control types

The tools palette provides the controls that can be added to a custom dialog.

- **Source List:** A list of source variables from the active dataset. See the topic “Source List” on page 223 for more information.
- **Target List:** A target for variables transferred from the source list. See the topic “Target List” on page 223 for more information.
- **Field Chooser:** A list of all the fields from the active dataset. See the topic “Field Chooser” on page 224 for more information.
- **Dataset Selector:** A list of the datasets that are currently open. See the topic “Dataset Selector” on page 226 for more information.
- **Check Box:** A single check box. See the topic “Check Box” on page 226 for more information.
- **Combo Box:** A combo box for creating drop-down lists. See the topic “Combo Box” on page 227 for more information.
- **List Box:** A list box for creating single selection or multiple selection lists. See the topic “Combo Box” on page 227 for more information.
- **Text control:** A text box that accepts arbitrary text as input. See the topic “Text control” on page 230 for more information.
- **Number control:** A text box that is restricted to numeric values as input. See the topic “Number Control” on page 231 for more information.
- **Date control:** A spinner control for specifying date/time values, which include dates, times, and datetimes. See the topic “Date control” on page 232 for more information.

- **Secured Text:** A text box that masks user entry with asterisks. See the topic “Secured Text” on page 233 for more information.
- **Static Text control:** A control for displaying static text. See the topic “Static Text Control” on page 235 for more information.
- **Color Picker:** A control for specifying a color and generating the associated RGB value. See the topic “Color Picker” on page 235 for more information.
- **Table control:** A table with a fixed number of columns and a variable number of rows that are added at run time. See the topic “Table Control” on page 235 for more information.
- **Item Group:** A container for grouping a set of controls, such as a set of check boxes. See the topic “Item Group” on page 238 for more information.
- **Radio Group:** A group of radio buttons. See the topic “Radio Group” on page 238 for more information.
- **Check Box Group:** A container for a set of controls that are enabled or disabled as a group, by a single check box. See the topic “Check Box Group” on page 239 for more information.
- **File Browser:** A control for browsing the file system to open or save a file. See the topic “File Browser” on page 240 for more information.
- **Tab:** A single tab. See the topic “Tab” on page 241 for more information.
- **Sub-dialog Button:** A button for launching a sub-dialog. See the topic “Sub-dialog button” on page 242 for more information.

Source List

The Source Variable List control displays the list of variables from the active dataset that are available to the end user of the dialog. You can display all variables from the active dataset (the default) or you can filter the list based on type and measurement level—for instance, numeric variables that have a measurement level of scale. Use of a Source List control implies the use of one or more Target List controls. The Source List control has the following properties:

Identifier. The unique identifier for the control.

Title. An optional title that appears with the control. For multi-line titles or long titles, click the ellipsis (...) button and enter your title in the Title Property dialog.

ToolTip. Optional ToolTip text that appears when the user hovers over the control. The specified text only appears when hovering over the title area of the control. Hovering over one of the listed variables will display the variable name and label.

Mnemonic Key. An optional character in the title to use as a keyboard shortcut to the control. The character appears underlined in the title. The shortcut is activated by pressing Alt+[mnemonic key]. The Mnemonic Key property is not supported on Mac.

Variable Transfers. Specifies whether variables transferred from the source list to a target list remain in the source list (Copy Variables), or are removed from the source list (Move Variables).

Variable Filter. Allows you to filter the set of variables displayed in the control. You can filter on variable type and measurement level, and you can specify that multiple response sets are included in the variable list. Click the ellipsis (...) button to open the Filter dialog. You can also open the Filter dialog by double-clicking the Source List control on the canvas. See the topic “Filtering Variable Lists” on page 226 for more information.

Note: The Source List control cannot be added to a sub-dialog.

Target List

The Target List control provides a target for variables that are transferred from the source list. Use of the Target List control assumes the presence of a Source List control. You can specify that only a single

variable can be transferred to the control or that multiple variables can be transferred to it, and you can constrain which types of variables can be transferred to the control--for instance, only numeric variables with a measurement level of nominal or ordinal. The Target List control has the following properties:

Identifier. The unique identifier for the control. This is the identifier to use when referencing the control in the syntax template.

Title. An optional title that appears with the control. For multi-line titles or long titles, click the ellipsis (...) button and enter your title in the Title Property dialog.

ToolTip. Optional ToolTip text that appears when the user hovers over the control. The specified text only appears when hovering over the title area of the control. Hovering over one of the listed variables will display the variable name and label.

Target list type. Specifies whether multiple variables or only a single variable can be transferred to the control.

Mnemonic Key. An optional character in the title to use as a keyboard shortcut to the control. The character appears underlined in the title. The shortcut is activated by pressing Alt+[mnemonic key]. The Mnemonic Key property is not supported on Mac.

Separator Type. Specifies the delimiter between the selected fields in the generated syntax. The allowed separators are a blank, a comma, and a plus sign (+). You can also enter an arbitrary single character to be used as the separator.

Minimum Fields. The minimum number of fields that must be specified for the control, if any.

Maximum Fields. The maximum number of fields that can be specified for the control, if any.

Required for execution. Specifies whether a value is required in this control in order for execution to proceed. If **True** is specified, the **OK** and **Paste** buttons will be disabled until a value is specified for this control. If **False** is specified, the absence of a value in this control has no effect on the state of the **OK** and **Paste** buttons. The default is **True**.

Variable Filter. Allows you to constrain the types of variables that can be transferred to the control. You can constrain by variable type and measurement level, and you can specify whether multiple response sets can be transferred to the control. Click the ellipsis (...) button to open the Filter dialog. You can also open the Filter dialog by double-clicking the Target List control on the canvas. See the topic "Filtering Variable Lists" on page 226 for more information.

Syntax. Specifies command syntax that is generated by this control at run time and can be inserted in the syntax template.

- You can specify any valid command syntax. For multi-line syntax or long syntax, click the ellipsis (...) button and enter your syntax in the Syntax Property dialog.
- The value `%%ThisValue%%` specifies the run time value of the control, which is the list of variables transferred to the control. This is the default.

Enabling Rule. Specifies a rule that determines when the current control is enabled. Click the ellipsis (...) button to open the dialog box and specify the rule. The Enabling Rule property is visible only when other controls that can be used to specify an enabling rule exist on the canvas.

Note: The Target List control cannot be added to a sub-dialog.

Field Chooser

The Field Chooser control displays the list of fields that are available to the end user of the dialog. You can filter the list based on type and measurement level--for instance, numeric fields that have a

measurement level of scale. You can also specify any other Field Chooser or Target List control as the source of fields for the current Field Chooser. The Field Chooser control has the following properties:

Identifier. The unique identifier for the control.

Title. An optional title that appears with the control. For multi-line titles or long titles, click the ellipsis (...) button and enter your title in the Title Property dialog.

Title Position. Specifies the position of the title relative to the control. Values are Top and Left where Top is the default. This property only applies when the chooser type is set to select a single field.

ToolTip. Optional ToolTip text that appears when the user hovers over the control. The specified text only appears when hovering over the title area of the control. Hovering over one of the listed fields displays the field name and label.

Mnemonic Key. An optional character in the title to use as a keyboard shortcut to the control. The character appears underlined in the title. The shortcut is activated by pressing Alt+[mnemonic key].

Chooser Type. Specifies whether the Field Chooser in the custom dialog can be used to select a single field or multiple fields from the field list.

Separator Type. Specifies the delimiter between the selected fields in the generated syntax. The allowed separators are a blank, a comma, and a plus sign (+). You can also enter an arbitrary single character to be used as the separator.

Minimum Fields. The minimum number of fields that must be specified for the control, if any.

Maximum Fields. The maximum number of fields that can be specified for the control, if any.

Required for Execution. Specifies whether a value is required in this control in order for execution to proceed. If **True** is specified, the **OK** and **Paste** buttons will be disabled until a value is specified for this control. If **False** is specified, the absence of a value in this control has no effect on the state of the **OK** and **Paste** buttons.

Variable Filter. Allows you to filter the set of fields that are displayed in the control. You can filter on field type and measurement level, and you can specify that multiple response sets are included in the field list. Click the ellipsis (...) button to open the Filter dialog. You can also open the Filter dialog by double-clicking the Field Chooser control on the canvas. See the topic "Filtering Variable Lists" on page 226 for more information.

Field Source. Specifies that another Field Chooser or Target List control is the source of fields for the current Field Chooser. When the Field Source property is not set, the source of fields is the current data source as determined by the Data Source property of the dialog. Click the ellipsis (...) button to open the dialog box and specify the field source.

Syntax. Specifies the command syntax that is generated and run by this control at run-time and can be inserted in the syntax template.

- You can specify any valid command syntax. For multi-line syntax or long syntax, click the ellipsis (...) button and enter your syntax in the Syntax Property dialog.
- The value `%%ThisValue%%` specifies the run-time value of the control, which is the list of fields. This is the default.

Enabling Rule. Specifies a rule that determines when the current control is enabled. Click the ellipsis (...) button to open the dialog box and specify the rule. The Enabling Rule property is visible only when other controls that can be used to specify an enabling rule exist on the canvas.

Note: The Field Chooser control is not available in compatibility mode.

Specifying the Field Source for a Field Chooser: The Field Source dialog box specifies the source of the fields that are displayed in the Field Chooser, and overrides the setting of the Data Source property for the dialog. The source can be any other Field Chooser or Target List control. You can choose to display the fields that are in the selected control or the fields, from the current data source, that are not in the selected control. The data source is determined by the Data Source property of the dialog.

Filtering Variable Lists

The Filter dialog box, associated with source list, target list, and field chooser controls, allows you to filter the types of variables from the active dataset that can appear in the lists. You can also specify whether multiple response sets associated with the active dataset are included. Numeric variables include all numeric formats except date and time formats.

Dataset Selector

The Dataset Selector control displays a list of the datasets that are currently open. You can use the Dataset Selector control to capture the name of a selected dataset. You can also use it with the Data Source property of the dialog to set the data source that is used by Source List and Field Chooser controls.

Identifier. The unique identifier for the control. This is the identifier to use when referencing the control in the syntax template.

Title. An optional title that appears with the control. For multi-line titles or long titles, click the ellipsis (...) button and enter your title in the Title Property dialog.

ToolTip. Optional ToolTip text that appears when the user hovers over the control.

Mnemonic Key. An optional character in the title to use as a keyboard shortcut to the control. The character appears underlined in the title. The shortcut is activated by pressing Alt+[mnemonic key]. The Mnemonic Key property is not supported on Mac.

Required for execution. Specifies whether a value is required in this control in order for execution to proceed. If **True** is specified, the **OK** and **Paste** buttons will be disabled until a value is specified for this control. If **False** is specified, the absence of a value in this control has no effect on the state of the **OK** and **Paste** buttons. The default is **False**.

Syntax. Specifies the command syntax that is generated by this control at run time and can be inserted in the syntax template.

- You can specify any valid command syntax. For multi-line syntax or long syntax, click the ellipsis (...) button and enter your syntax in the Syntax Property dialog.
- The value **%%ThisValue%%** specifies the run time value of the control, which is the name of the selected dataset. This is the default.

Enabling Rule. Specifies a rule that determines when the current control is enabled. Click the ellipsis (...) button to open the dialog box and specify the rule. The Enabling Rule property is visible only when other controls that can be used to specify an enabling rule exist on the canvas.

Note: Dataset Selector control is not available in compatibility mode.

Check Box

The Check Box control is a simple check box that can generate different command syntax for the checked versus the unchecked state. The Check Box control has the following properties:

Identifier. The unique identifier for the control. This is the identifier to use when referencing the control in the syntax template.

Title. An optional title that appears with the control. For multi-line titles or long titles, click the ellipsis (...) button and enter your title in the Title Property dialog.

ToolTip. Optional ToolTip text that appears when the user hovers over the control.

Mnemonic Key. An optional character in the title to use as a keyboard shortcut to the control. The character appears underlined in the title. The shortcut is activated by pressing Alt+[mnemonic key]. The Mnemonic Key property is not supported on Mac.

Default Value. The default state of the check box--checked or unchecked.

Checked/Unchecked Syntax. Specifies the command syntax that is generated when the control is checked and when it is unchecked. To include the command syntax in the syntax template, use the value of the Identifier property. The generated syntax, whether from the Checked Syntax or Unchecked Syntax property, will be inserted at the specified position(s) of the identifier. For example, if the identifier is *checkbox1*, then at run time, instances of %%checkbox1%% in the syntax template will be replaced by the value of the Checked Syntax property when the box is checked and the Unchecked Syntax property when the box is unchecked.

- You can specify any valid command syntax. For multi-line syntax or long syntax, click the ellipsis (...) button and enter your syntax in the Syntax Property dialog.

Enabling Rule. Specifies a rule that determines when the current control is enabled. Click the ellipsis (...) button to open the dialog box and specify the rule. The Enabling Rule property is visible only when other controls that can be used to specify an enabling rule exist on the canvas.

Combo Box

The Combo Box control allows you to create a drop-down list that can generate command syntax specific to the selected list item. It is limited to single selection. The Combo Box control has the following properties:

Identifier. The unique identifier for the control. This is the identifier to use when referencing the control in the syntax template.

Title. An optional title that appears with the control. For multi-line titles or long titles, click the ellipsis (...) button and enter your title in the Title Property dialog.

Title Position. Specifies the position of the title relative to the control. Values are Top and Left where Top is the default.

ToolTip. Optional ToolTip text that appears when the user hovers over the control.

List Items. Click the ellipsis (...) button to open the List Item Properties dialog box, which allows you to specify the list items of the control. You can also open the List Item Properties dialog by double-clicking the Combo Box control on the canvas.

Mnemonic Key. An optional character in the title to use as a keyboard shortcut to the control. The character appears underlined in the title. The shortcut is activated by pressing Alt+[mnemonic key].The Mnemonic Key property is not supported on Mac.

Editable. Specifies whether the Combo Box control is editable. When the control is editable, a custom value can be entered at run time.

Syntax. Specifies the command syntax that is generated by this control at run time and can be inserted in the syntax template.

- The value %%ThisValue%% specifies the run time value of the control and is the default. If the list items are manually defined, the run time value is the value of the Syntax property for the selected list item.

If the list items are based on a target list control, the run time value is the value of the selected list item. For multiple selection list box controls, the run time value is a blank-separated list of the selected items. See the topic "Specifying list items for combo boxes and list boxes" for more information.

- You can specify any valid command syntax. For multi-line syntax or long syntax, click the ellipsis (...) button and enter your syntax in the Syntax Property dialog.

Quote Handling. Specifies handling of quotation marks in the run time value of %%ThisValue%% when the Syntax property contains %%ThisValue%% as part of a quoted string. In this context, a quoted string is a string that is enclosed in single quotation marks or double quotation marks. Quote handling applies only to quotation marks that are the same type as the quotation marks that enclose %%ThisValue%%. The following types of quote handling are available.

Syntax

Quotation marks in the run time value of %%ThisValue%% that match the enclosing quotation marks are doubled. For example, if the Syntax property is '%%ThisValue%%' and the run time value of the combo box is Combo box's value, then the generated syntax is 'Combo box''s value'.

Python

Quotation marks in the run time value of %%ThisValue%% that match the enclosing quotation marks are escaped with the backslash character (\). For example, if the Syntax property is '%%ThisValue%%' and the run time value of the combo box is Combo box's value, then the generated syntax is 'Combo box\\'s value'. Note that quote handling is not done when %%ThisValue%% is enclosed in triple quotation marks.

R Quotation marks in the run time value of %%ThisValue%% that match the enclosing quotation marks are escaped with the backslash character (\). For example, if the Syntax property is '%%ThisValue%%' and the run time value of the combo box is Combo box's value, then the generated syntax is 'Combo box\\'s value'.

None Quotation marks in the run time value of %%ThisValue%% that match the enclosing quotation marks are retained with no modification.

Enabling Rule. Specifies a rule that determines when the current control is enabled. Click the ellipsis (...) button to open the dialog box and specify the rule. The Enabling Rule property is visible only when other controls that can be used to specify an enabling rule exist on the canvas.

Specifying list items for combo boxes and list boxes: The List Item Properties dialog box allows you to specify the list items of a combo box or list box control.

Manually defined values. Allows you to explicitly specify each of the list items.

- **Identifier.** A unique identifier for the list item.
- **Name.** The name that appears in the list for this item. The name is a required field.
- **Default.** For a combo box, specifies whether the list item is the default item displayed in the combo box. For a list box, specifies whether the list item is selected by default.
- **Syntax.** Specifies the command syntax that is generated when the list item is selected.
- You can specify any valid command syntax. For multi-line syntax or long syntax, click the ellipsis (...) button and enter your syntax in the Syntax Property dialog.

Note: You can add a new list item in the blank line at the bottom of the existing list. Entering any of the properties other than the identifier will generate a unique identifier, which you can keep or modify. You can delete a list item by clicking on the *Identifier* cell for the item and pressing delete.

Values based on the contents of a target list or field chooser control. Specifies that the list items are dynamically populated with values associated with the variables in a selected target list or field chooser

control. Select an existing target list or field chooser control as the source of the list items, or enter the value of the Identifier property for a target list or field chooser control into the text area of the Target List or Field Chooser combo box. The latter approach allows you to enter the Identifier for a target list or field chooser control that you plan to add later.

- **Variable Names.** Populate the list items with the names of the variables in the specified target list or field chooser control.
- **Value Labels.** Populate the list items with the union of the value labels associated with the variables in the specified target list control. You can choose whether the command syntax generated by the associated combo box or list box control contains the selected value label or its associated value. The **Value Labels** option is not available for field choosers.
- **Custom Attribute.** Populate the list items with the union of the attribute values associated with variables in the target list control that contain the specified custom attribute. The **Custom Attribute** option is not available for field choosers.
- **Syntax.** Displays the Syntax property of the associated combo box or list box control, allowing you to make changes to the property. See the topic “Combo Box” on page 227 for more information.

List Box

The List Box control allows you to display a list of items that support single or multiple selection and generate command syntax specific to the selected item(s). The List Box control has the following properties:

Identifier. The unique identifier for the control. This is the identifier to use when referencing the control in the syntax template.

Title. An optional title that appears with the control. For multi-line titles or long titles, click the ellipsis (...) button and enter your title in the Title Property dialog.

ToolTip. Optional ToolTip text that appears when the user hovers over the control.

List Items. Click the ellipsis (...) button to open the List Item Properties dialog box, which allows you to specify the list items of the control. You can also open the List Item Properties dialog by double-clicking the List Box control on the canvas.

Mnemonic Key. An optional character in the title to use as a keyboard shortcut to the control. The character appears underlined in the title. The shortcut is activated by pressing Alt+[mnemonic key]. The Mnemonic Key property is not supported on Mac.

List Box Type. Specifies whether the list box supports single selection only or multiple selection. You can also specify that items are displayed as a list of check boxes.

Separator Type. Specifies the delimiter between the selected list items in the generated syntax. The allowed separators are a blank, a comma, and a plus sign (+). You can also enter an arbitrary single character to be used as the separator.

Minimum Selected. The minimum number of items that must be selected in the control, if any.

Maximum Selected. The maximum number of items that can be selected in the control, if any.

Syntax. Specifies the command syntax that is generated by this control at run time and can be inserted in the syntax template.

- The value `%%ThisValue%%` specifies the run time value of the control and is the default. If the list items are manually defined, the run time value is the value of the Syntax property for the selected list item. If the list items are based on a target list control, the run time value is the value of the selected list item. For multiple selection list box controls, the run time value is a list of the selected items, delimited

by the specified Separator Type (default is blank separated). See the topic “Specifying list items for combo boxes and list boxes” on page 228 for more information.

- You can specify any valid command syntax. For multi-line syntax or long syntax, click the ellipsis (...) button and enter your syntax in the Syntax Property dialog.

Quote Handling. Specifies handling of quotation marks in the run time value of %%ThisValue%% when the Syntax property contains %%ThisValue%% as part of a quoted string. In this context, a quoted string is a string that is enclosed in single quotation marks or double quotation marks. Quote handling applies only to quotation marks that are the same type as the quotation marks that enclose %%ThisValue%%. The following types of quote handling are available.

Syntax

Quotation marks in the run time value of %%ThisValue%% that match the enclosing quotation marks are doubled. For example, if the Syntax property is '%%ThisValue%%' and the selected list item is List item's value, then the generated syntax is 'List item''s value'.

Python

Quotation marks in the run time value of %%ThisValue%% that match the enclosing quotation marks are escaped with the backslash character (\). For example, if the Syntax property is '%%ThisValue%%' and the selected list item is List item's value, then the generated syntax is 'List item\'s value'. Note that quote handling is not done when %%ThisValue%% is enclosed in triple quotation marks.

- R** Quotation marks in the run time value of %%ThisValue%% that match the enclosing quotation marks are escaped with the backslash character (\). For example, if the Syntax property is '%%ThisValue%%' and the selected list item is List item's value, then the generated syntax is 'List item\'s value'.
- None** Quotation marks in the run time value of %%ThisValue%% that match the enclosing quotation marks are retained with no modification.

Enabling Rule. Specifies a rule that determines when the current control is enabled. Click the ellipsis (...) button to open the dialog box and specify the rule. The Enabling Rule property is visible only when other controls that can be used to specify an enabling rule exist on the canvas.

Text control

The Text control is a simple text box that can accept arbitrary input, and has the following properties:

Identifier. The unique identifier for the control. This is the identifier to use when referencing the control in the syntax template.

Title. An optional title that appears with the control. For multi-line titles or long titles, click the ellipsis (...) button and enter your title in the Title Property dialog.

Title Position. Specifies the position of the title relative to the control. Values are Top and Left where Top is the default.

ToolTip. Optional ToolTip text that appears when the user hovers over the control.

Mnemonic Key. An optional character in the title to use as a keyboard shortcut to the control. The character appears underlined in the title. The shortcut is activated by pressing Alt+[mnemonic key]. The Mnemonic Key property is not supported on Mac.

Text Content. Specifies the allowed contents. The value **Any** specifies that the contents are arbitrary. The value **Variable Name** specifies that the text box must contain a string that adheres to rules for IBM SPSS

Statistics variable names. See the topic “Variable names” on page 50 for more information. The value **New dataset name** specifies that the text box must contain a valid IBM SPSS Statistics dataset name and cannot be the name of a currently open dataset.

Default Value. The default contents of the text box.

Width. Specifies the width of the text area of the control in characters. The allowed values are positive integers. An empty value means that the width is automatically determined.

Required for execution. Specifies whether a value is required in this control in order for execution to proceed. If **True** is specified, the **OK** and **Paste** buttons will be disabled until a value is specified for this control. If **False** is specified, the absence of a value in this control has no effect on the state of the **OK** and **Paste** buttons. The default is **False**.

Syntax. Specifies the command syntax that is generated by this control at run time and can be inserted in the syntax template.

- You can specify any valid command syntax. For multi-line syntax or long syntax, click the ellipsis (...) button and enter your syntax in the Syntax Property dialog.
- The value `%%ThisValue%%` specifies the run time value of the control, which is the content of the text box. This is the default.
- If the Syntax property includes `%%ThisValue%%` and the run time value of the text box is empty, then the text box control does not generate any command syntax.

Quote Handling. Specifies handling of quotation marks in the run time value of `%%ThisValue%%` when the Syntax property contains `%%ThisValue%%` as part of a quoted string. In this context, a quoted string is a string that is enclosed in single quotation marks or double quotation marks. Quote handling applies only to quotation marks that are the same type as the quotation marks that enclose `%%ThisValue%%`. The following types of quote handling are available.

Syntax

Quotation marks in the run time value of `%%ThisValue%%` that match the enclosing quotation marks are doubled. For example, if the Syntax property is '`%%ThisValue%%`' and the run time value of the text control is Text box's value, then the generated syntax is 'Text box''s value'.

Python

Quotation marks in the run time value of `%%ThisValue%%` that match the enclosing quotation marks are escaped with the backslash character (\). For example, if the Syntax property is '`%%ThisValue%%`' and the run time value of the text control is Text box's value, then the generated syntax is 'Text box\\'s value'. Quote handling is not done when `%%ThisValue%%` is enclosed in triple quotation marks.

R

Quotation marks in the run time value of `%%ThisValue%%` that match the enclosing quotation marks are escaped with the backslash character (\). For example, if the Syntax property is '`%%ThisValue%%`' and the run time value of the text control is Text box's value, then the generated syntax is 'Text box\\'s value'.

None

Quotation marks in the run time value of `%%ThisValue%%` that match the enclosing quotation marks are retained with no modification.

Enabling Rule. Specifies a rule that determines when the current control is enabled. Click the ellipsis (...) button to open the dialog box and specify the rule. The Enabling Rule property is visible only when other controls that can be used to specify an enabling rule exist on the canvas.

Number Control

The Number control is a text box for entering a numeric value, and has the following properties:

Identifier. The unique identifier for the control. This is the identifier to use when referencing the control in the syntax template.

Title. An optional title that appears with the control. For multi-line titles or long titles, click the ellipsis (...) button and enter your title in the Title Property dialog.

Title Position. Specifies the position of the title relative to the control. Values are Top and Left where Top is the default.

ToolTip. Optional ToolTip text that appears when the user hovers over the control.

Mnemonic Key. An optional character in the title to use as a keyboard shortcut to the control. The character appears underlined in the title. The shortcut is activated by pressing Alt+[mnemonic key]. The Mnemonic Key property is not supported on Mac.

Numeric Type. Specifies any limitations on what can be entered. A value of Real specifies that there are no restrictions on the entered value, other than it be numeric. A value of Integer specifies that the value must be an integer.

Spin Input. Specifies whether the control is displayed as a spinner. The default is False.

Increment. The increment when the control is displayed as a spinner.

Default Value. The default value, if any.

Minimum Value. The minimum allowed value, if any.

Maximum Value. The maximum allowed value, if any.

Width. Specifies the width of the text area of the control in characters. The allowed values are positive integers. An empty value means that the width is automatically determined.

Required for execution. Specifies whether a value is required in this control in order for execution to proceed. If **True** is specified, the **OK** and **Paste** buttons will be disabled until a value is specified for this control. If **False** is specified, the absence of a value in this control has no effect on the state of the **OK** and **Paste** buttons. The default is **False**.

Syntax. Specifies the command syntax that is generated by this control at run time and can be inserted in the syntax template.

- You can specify any valid command syntax. For multi-line syntax or long syntax, click the ellipsis (...) button and enter your syntax in the Syntax Property dialog.
- The value `%%ThisValue%%` specifies the run time value of the control, which is the numeric value. This is the default.
- If the Syntax property includes `%%ThisValue%%` and the run time value of the number control is empty, then the number control does not generate any command syntax.

Enabling Rule. Specifies a rule that determines when the current control is enabled. Click the ellipsis (...) button to open the dialog box and specify the rule. The Enabling Rule property is visible only when other controls that can be used to specify an enabling rule exist on the canvas.

Date control

The Date control is a spinner control for specifying date/time values, which include dates, times, and datetimes. The Date control has the following properties:

Identifier. The unique identifier for the control. This is the identifier to use when referencing the control in the syntax template.

Title. An optional title that appears with the control. For multi-line titles or long titles, click the ellipsis (...) button and enter your title in the Title Property dialog.

Title Position. Specifies the position of the title relative to the control. Values are Top and Left where Top is the default.

ToolTip. Optional ToolTip text that appears when the user hovers over the control.

Mnemonic Key. An optional character in the title to use as a keyboard shortcut to the control. The character appears underlined in the title. The shortcut is activated by pressing Alt+[mnemonic key]. The Mnemonic Key property is not supported on Mac.

Type. Specifies whether the control is for dates, times, or datetime values.

Date The control specifies a calendar date of the form yyyy-mm-dd. The default run time value is specified by the Default Value property.

Time The control specifies the time of day in the form hh:mm:ss. The default run time value is the current time of day.

Datetime

The control specifies a date and time of the form yyyy-mm-dd hh:mm:ss. The default run time value is the current date and time of day.

Default Value. The default run time value of the control when the type is Date. You can specify to display the current date or a particular date.

Syntax. Specifies the command syntax that is generated by this control at run time and can be inserted in the syntax template.

- You can specify any valid command syntax. For multi-line syntax or long syntax, click the ellipsis (...) button and enter your syntax in the Syntax Property dialog.
- The value %%ThisValue%% specifies the run time value of the control. This is the default.

Enabling Rule. Specifies a rule that determines when the current control is enabled. Click the ellipsis (...) button to open the dialog box and specify the rule. The Enabling Rule property is visible only when other controls that can be used to specify an enabling rule exist on the canvas.

Note: The Date control is not available in compatibility mode.

Secured Text

The Secured Text control is a text box that masks user entry with asterisks.

Identifier. The unique identifier for the control. This is the identifier to use when referencing the control in the syntax template.

Title. An optional title that appears with the control. For multi-line titles or long titles, click the ellipsis (...) button and enter your title in the Title Property dialog.

Title Position. Specifies the position of the title relative to the control. Values are Top and Left where Top is the default.

ToolTip. Optional ToolTip text that appears when the user hovers over the control.

Mnemonic Key. An optional character in the title to use as a keyboard shortcut to the control. The character appears underlined in the title. The shortcut is activated by pressing Alt+[mnemonic key]. The Mnemonic Key property is not supported on Mac.

Encrypt passed value. Specifies whether the value in the generated command syntax is encrypted. True specifies that the value is encrypted. The default is False, which specifies that the value is not encrypted. Encrypted values can be decrypted only by IBM SPSS Statistics commands that can process encrypted passwords, such as the GET and SAVE commands.

Width. Specifies the width of the text area of the control in characters. The allowed values are positive integers. An empty value means that the width is automatically determined.

Required for execution. Specifies whether a value is required in this control in order for execution to proceed. If **True** is specified, the **OK** and **Paste** buttons will be disabled until a value is specified for this control. If **False** is specified, the absence of a value in this control has no effect on the state of the **OK** and **Paste** buttons. The default is **False**.

Syntax. Specifies the command syntax that is generated by this control at run time and can be inserted in the syntax template.

- You can specify any valid command syntax. For multi-line syntax or long syntax, click the ellipsis (...) button and enter your syntax in the Syntax Property dialog.
- The value `%%ThisValue%%` specifies the run time value of the control, which is the content of the text box. This is the default.
- If the Syntax property includes `%%ThisValue%%` and the run time value of the secured text control is empty, then the secured text control does not generate any command syntax.

Quote Handling. Specifies handling of quotation marks in the run time value of `%%ThisValue%%` when the Syntax property contains `%%ThisValue%%` as part of a quoted string. In this context, a quoted string is a string that is enclosed in single quotation marks or double quotation marks. Quote handling applies only to quotation marks that are the same type as the quotation marks that enclose `%%ThisValue%%` and only when Encrypt passed value=False. The following types of quote handling are available.

Syntax

Quotation marks in the run time value of `%%ThisValue%%` that match the enclosing quotation marks are doubled. For example, if the Syntax property is '`%%ThisValue%%`' and the run time value of the control is Secured Text's value, then the generated syntax is 'Secured Text''s value'.

Python

Quotation marks in the run time value of `%%ThisValue%%` that match the enclosing quotation marks are escaped with the backslash character (\). For example, if the Syntax property is '`%%ThisValue%%`' and the run time value of the control is Secured Text's value, then the generated syntax is 'Secured Text\'s value'. Quote handling is not done when `%%ThisValue%%` is enclosed in triple quotation marks.

R Quotation marks in the run time value of `%%ThisValue%%` that match the enclosing quotation marks are escaped with the backslash character (\). For example, if the Syntax property is '`%%ThisValue%%`' and the run time value of the control is Secured Text's value, then the generated syntax is 'Secured Text\'s value'.

None Quotation marks in the run time value of `%%ThisValue%%` that match the enclosing quotation marks are retained with no modification.

Enabling Rule. Specifies a rule that determines when the current control is enabled. Click the ellipsis (...) button to open the dialog box and specify the rule. The Enabling Rule property is visible only when other controls that can be used to specify an enabling rule exist on the canvas.

Note: The Secured Text control is not available in compatibility mode.

Static Text Control

The Static Text control allows you to add a block of text to your dialog, and has the following properties:

Identifier. The unique identifier for the control.

Title. The content of the text block. For multiple lines or long lines of text, click the ellipsis (...) button and enter your text in the Title Property dialog.

Enabling Rule. Specifies a rule that determines when the current control is enabled. Click the ellipsis (...) button to open the dialog box and specify the rule. The Enabling Rule property is visible only when other controls that can be used to specify an enabling rule exist on the canvas.

Color Picker

The Color Picker control is a user interface for specifying a color and generating the associated RGB value. The Color Picker control has the following properties:

Identifier. The unique identifier for the control. This is the identifier to use when referencing the control in the syntax template.

Title. An optional title that appears with the control. For multi-line titles or long titles, click the ellipsis (...) button and enter your title in the Title Property dialog.

Title Position. Specifies the position of the title relative to the control. Values are Top and Left where Top is the default.

ToolTip. Optional ToolTip text that appears when the user hovers over the control.

Mnemonic Key. An optional character in the title to use as a keyboard shortcut to the control. The character appears underlined in the title. The shortcut is activated by pressing Alt+[mnemonic key]. The Mnemonic Key property is not supported on Mac.

Syntax. Specifies the command syntax that is generated by this control at run time and can be inserted in the syntax template.

- You can specify any valid command syntax. For multi-line syntax or long syntax, click the ellipsis (...) button and enter your syntax in the Syntax Property dialog.
- The value %%ThisValue%% specifies the run time value of the control, which is the RGB value of the selected color. The RGB value is represented as a blank separated list of integers in the following order: R value, G value, B value.

Enabling Rule. Specifies a rule that determines when the current control is enabled. Click the ellipsis (...) button to open the dialog box and specify the rule. The Enabling Rule property is visible only when other controls that can be used to specify an enabling rule exist on the canvas.

Note: The Color Picker control is not available in compatibility mode.

Table Control

The Table control creates a table with a fixed number of columns and a variable number of rows that are added at run time. The Table control has the following properties:

Identifier. The unique identifier for the control. This is the identifier to use when referencing the control in the syntax template.

Title. An optional title that appears with the control. For multi-line titles or long titles, click the ellipsis (...) button and enter your title in the Title Property dialog.

ToolTip. Optional ToolTip text that appears when the user hovers over the control.

Mnemonic Key. An optional character in the title to use as a keyboard shortcut to the control. The character appears underlined in the title. The shortcut is activated by pressing Alt+[mnemonic key].The Mnemonic Key property is not supported on Mac.

Reorder Buttons. Specifies whether move up and move down buttons are added to the table. These buttons are used at run time to reorder the rows of the table.

Table Columns. Click the ellipsis (...) button to open the Table Columns dialog box, where you specify the columns of the table.

Minimum Rows. The minimum number of rows that must be in the table.

Maximum Rows. The maximum number of rows that can be in the table.

Required for Execution. Specifies whether a value is required in this control in order for execution to proceed. If **True** is specified, the **OK** and **Paste** buttons will be disabled until a value is specified for this control. If **False** is specified, the absence of a value in this control has no effect on the state of the **OK** and **Paste** buttons.

Syntax. Specifies the command syntax that is generated by this control at run time and can be inserted in the syntax template.

- The value `%%ThisValue%%` specifies the run time value of the control and is the default. The run time value is a blank-separated list of the syntax that is generated by each column in the table, starting with the leftmost column. If the Syntax property includes `%%ThisValue%%` and none of the columns generate syntax, then the table as a whole does not generate any syntax.
- You can specify any valid command syntax. For multi-line syntax or long syntax, click the ellipsis (...) button and enter your syntax in the Syntax Property dialog.

Enabling Rule. Specifies a rule that determines when the current control is enabled. Click the ellipsis (...) button to open the dialog box and specify the rule. The Enabling Rule property is visible only when other controls that can be used to specify an enabling rule exist on the canvas.

Note: The Table control is not available in compatibility mode.

Specifying columns for table controls: The Table Columns dialog box specifies the properties of the columns of the Table control.

Identifier. A unique identifier for the column.

Column Name. The name of the column as it appears in the table.

Contents. Specifies the type of data for the column. The value **Real** specifies that there are no restrictions on the entered value, other than it is numeric. The value **Integer** specifies that the value must be an integer. The value **Any** specifies that there are no restrictions on the entered value. The value **Variable Name** specifies that the value must meet the requirements for a valid variable name in IBM SPSS Statistics.

Default Value. The default value for this column, if any, when new rows are added to the table at run time.

Separator Type. Specifies the delimiter between the values of the column, in the generated syntax. The allowed separators are a blank, a comma, and a plus sign (+). You can also enter an arbitrary single character to be used as the separator.

Quoted. Specifies whether each value in the column is enclosed in double quotation marks in the generated syntax.

Quote Handling. Specifies handling of quotation marks in cell entries for the column when the Quoted property is true. Quote handling applies only to double quotation marks in cell values. The following types of quote handling are available.

Syntax

Double quotation marks in cell values are doubled. For example, if the cell value is This "quoted" value then the generated syntax is "This ""quoted"" value".

Python

Double quotation marks in cell values are escaped with the backslash character (\). For example, if the cell value is This "quoted" value then the generated syntax is "This \"quoted\" value".

R Double quotation marks in cell values are escaped with the backslash character (\). For example, if the cell value is This "quoted" value then the generated syntax is "This \\\"quoted\\\" value".

None Double quotation marks in cell values are retained with no modification.

Width(chars). Specifies the width of the column in characters. The allowed values are positive integers.

Syntax. Specifies the command syntax that is generated by this column at run time. The generated syntax for the table as a whole is a blank-separated list of the syntax that is generated by each column in the table, starting with the leftmost column.

- You can specify any valid command syntax. For multi-line syntax or long syntax, click the ellipsis (...) button and enter your syntax in the Syntax Property dialog.
- The value %%ThisValue%% specifies the run time value of the column, which is a list of the values in the column, delimited by the specified separator.
- If the Syntax property for the column includes %%ThisValue%% and the run time value of the column is empty, then the column does not generate any syntax.

Note: You can add a row for a new Table column in the blank line at the bottom of the existing list in the Table Columns dialog. Entering any of the properties other than the identifier generates a unique identifier, which you can keep or modify. You can delete a Table column by clicking the identifier cell for the Table column and pressing delete.

Link to Control

You can link a Table control to a Field Chooser control. When a Table control is linked to a Field Chooser, there is a row in the table for each field in the Field Chooser. Rows are added to the table by adding fields to the Field Chooser. Rows are deleted from the table by removing fields from the Field Chooser. A linked Table control can be used, for example, to specify properties of fields that are selected in a Field Chooser.

To enable linking, the table must have a column with Variable Name for the Contents property and there must be at least one Field Chooser control on the canvas.

To link a Table control to a Field Chooser, specify the Field Chooser from the list of Available controls in the Link to Control group on the Table Columns dialog box. Then, select the table column, referred to as the **Linked column**, that defines the link. When the table is rendered, the linked column displays the current fields in the Field Chooser. You can link only to multi-field Field Choosers.

Item Group

The Item Group control is a container for other controls, allowing you to group and control the syntax generated from multiple controls. For example, you have a set of check boxes that specify optional settings for a subcommand, but only want to generate the syntax for the subcommand if at least one box is checked. This is accomplished by using an Item Group control as a container for the check box controls. The following types of controls can be contained in an Item Group: field chooser, dataset selector, check box, combo box, list box, text control, number control, date control, secured text, static text, color picker, table control, radio group, and file browser. The Item Group control has the following properties:

Identifier. The unique identifier for the control. This is the identifier to use when referencing the control in the syntax template.

Title. An optional title for the group. For multi-line titles or long titles, click the ellipsis (...) button and enter your title in the Title Property dialog.

Syntax. Specifies the command syntax that is generated by this control at run time and can be inserted in the syntax template.

- You can specify any valid command syntax. For multi-line syntax or long syntax, click the ellipsis (...) button and enter your syntax in the Syntax Property dialog.
- You can include identifiers for any controls contained in the item group. At run time the identifiers are replaced with the syntax generated by the controls.
- The value `%%ThisValue%%` generates a blank-separated list of the syntax generated by each control in the item group, in the order in which they appear in the group (top to bottom). This is the default. If the Syntax property includes `%%ThisValue%%` and no syntax is generated by any of the controls in the item group, then the item group as a whole does not generate any command syntax.

Enabling Rule. Specifies a rule that determines when the current control is enabled. Click the ellipsis (...) button to open the dialog box and specify the rule. The Enabling Rule property is visible only when other controls that can be used to specify an enabling rule exist on the canvas.

Radio Group

The Radio Group control is a container for a set of radio buttons, each of which can contain a set of nested controls. The Radio Group control has the following properties:

Identifier. The unique identifier for the control. This is the identifier to use when referencing the control in the syntax template.

Title. An optional title for the group. For multi-line titles or long titles, click the ellipsis (...) button and enter your title in the Title Property dialog.

ToolTip. Optional ToolTip text that appears when the user hovers over the control.

Radio Buttons. Click the ellipsis (...) button to open the Radio Group Properties dialog box, which allows you to specify the properties of the radio buttons as well as to add or remove buttons from the group. The ability to nest controls under a given radio button is a property of the radio button and is set in the Radio Group Properties dialog box. Note that you can also open the Radio Group Properties dialog by double-clicking the Radio Group control on the canvas.

Syntax. Specifies the command syntax that is generated by this control at run time and can be inserted in the syntax template.

- You can specify any valid command syntax. For multi-line syntax or long syntax, click the ellipsis (...) button and enter your syntax in the Syntax Property dialog.

- The value `%%ThisValue%%` specifies the run time value of the radio button group, which is the value of the Syntax property for the selected radio button. This is the default. If the Syntax property includes `%%ThisValue%%` and no syntax is generated by the selected radio button, then the radio button group does not generate any command syntax.

Enabling Rule. Specifies a rule that determines when the current control is enabled. Click the ellipsis (...) button to open the dialog box and specify the rule. The Enabling Rule property is visible only when other controls that can be used to specify an enabling rule exist on the canvas.

Defining radio buttons: The Radio Button Group Properties dialog box allows you to specify a group of radio buttons.

Identifier. A unique identifier for the radio button.

Column Name. The name that appears next to the radio button. The name is a required field.

ToolTip. Optional ToolTip text that appears when the user hovers over the control.

Mnemonic Key. An optional character in the name to use as a mnemonic. The specified character must exist in the name.

Nested Group. Specifies whether other controls can be nested under this radio button. The default is false. When the nested group property is set to true, a rectangular drop zone is displayed, nested and indented, under the associated radio button. The following controls can be nested under a radio button: field chooser, dataset selector, check box, combo box, list box, text control, number control, date control, secured text, static text, color picker, table control, and file browser.

Default. Specifies whether the radio button is the default selection.

Enabling Rule. Specifies a rule that determines when the current control is enabled. Click the ellipsis (...) button to open the dialog box and specify the rule. The Enabling Rule property is visible only when other controls that can be used to specify an enabling rule exist on the canvas.

Syntax. Specifies the command syntax that is generated when the radio button is selected.

- You can specify any valid command syntax. For multi-line syntax or long syntax, click the ellipsis (...) button and enter your syntax in the Syntax Property dialog.
- For radio buttons containing nested controls, the value `%%ThisValue%%` generates a blank-separated list of the syntax generated by each nested control, in the order in which they appear under the radio button (top to bottom).

You can add a new radio button in the blank line at the bottom of the existing list. Entering any of the properties other than the identifier will generate a unique identifier, which you can keep or modify. You can delete a radio button by clicking on the *Identifier* cell for the button and pressing delete.

Check Box Group

The Check Box Group control is a container for a set of controls that are enabled or disabled as a group, by a single check box. The following types of controls can be contained in a Check Box Group: field chooser, dataset selector, check box, combo box, list box, text control, number control, date control, secured text, static text, color picker, table control, radio group, and file browser. The Check Box Group control has the following properties:

Identifier. The unique identifier for the control. This is the identifier to use when referencing the control in the syntax template.

Title. An optional title for the group. For multi-line titles or long titles, click the ellipsis (...) button and enter your title in the Title Property dialog.

Checkbox Title. An optional label that is displayed with the controlling check box. Supports \n to specify line breaks.

ToolTip. Optional ToolTip text that appears when the user hovers over the control.

Mnemonic Key. An optional character in the title to use as a keyboard shortcut to the control. The character appears underlined in the title. The shortcut is activated by pressing Alt+[mnemonic key]. The Mnemonic Key property is not supported on Mac.

Default Value. The default state of the controlling check box--checked or unchecked.

Checked/Unchecked Syntax. Specifies the command syntax that is generated when the control is checked and when it is unchecked. To include the command syntax in the syntax template, use the value of the Identifier property. The generated syntax, whether from the Checked Syntax or Unchecked Syntax property, will be inserted at the specified position(s) of the identifier. For example, if the identifier is *checkboxgroup1*, then at run time, instances of %%checkboxgroup1%% in the syntax template will be replaced by the value of the Checked Syntax property when the box is checked and the Unchecked Syntax property when the box is unchecked.

- You can specify any valid command syntax. For multi-line syntax or long syntax, click the ellipsis (...) button and enter your syntax in the Syntax Property dialog.
- You can include identifiers for any controls contained in the check box group. At run time the identifiers are replaced with the syntax generated by the controls.
- The value %%ThisValue%% can be used in either the Checked Syntax or Unchecked Syntax property. It generates a blank-separated list of the syntax generated by each control in the check box group, in the order in which they appear in the group (top to bottom).
- By default, the Checked Syntax property has a value of %%ThisValue%% and the Unchecked Syntax property is blank.

Enabling Rule. Specifies a rule that determines when the current control is enabled. Click the ellipsis (...) button to open the dialog box and specify the rule. The Enabling Rule property is visible only when other controls that can be used to specify an enabling rule exist on the canvas.

File Browser

The File Browser control consists of a text box for a file path and a browse button that opens a standard IBM SPSS Statistics dialog to open or save a file. The File Browser control has the following properties:

Identifier. The unique identifier for the control. This is the identifier to use when referencing the control in the syntax template.

Title. An optional title that appears with the control. For multi-line titles or long titles, click the ellipsis (...) button and enter your title in the Title Property dialog.

Title Position. Specifies the position of the title relative to the control. Values are Top and Left where Top is the default.

ToolTip. Optional ToolTip text that appears when the user hovers over the control.

Mnemonic Key. An optional character in the title to use as a keyboard shortcut to the control. The character appears underlined in the title. The shortcut is activated by pressing Alt+[mnemonic key]. The Mnemonic Key property is not supported on Mac.

File System Operation. Specifies whether the dialog launched by the browse button is appropriate for opening files or for saving files. A value of Open indicates that the browse dialog validates the existence of the specified file. A value of Save indicates that the browse dialog does not validate the existence of the specified file.

Browser Type. Specifies whether the browse dialog is used to select a file (Locate File) or to select a folder (Locate Folder).

File Filter. Click the ellipsis (...) button to open the File Filter dialog box, which allows you to specify the available file types for the open or save dialog. By default, all file types are allowed. Note that you can also open the File Filter dialog by double-clicking the File Browser control on the canvas.

File System Type. In distributed analysis mode, this specifies whether the open or save dialog browses the file system on which IBM SPSS Statistics Server is running or the file system of your local computer. Select **Server** to browse the file system of the server or **Client** to browse the file system of your local computer. The property has no effect in local analysis mode.

Required for execution. Specifies whether a value is required in this control in order for execution to proceed. If **True** is specified, the **OK** and **Paste** buttons will be disabled until a value is specified for this control. If **False** is specified, the absence of a value in this control has no effect on the state of the **OK** and **Paste** buttons. The default is **False**.

Default. The default value of the control.

Syntax. Specifies the command syntax that is generated by this control at run time and can be inserted in the syntax template.

- You can specify any valid command syntax. For multi-line syntax or long syntax, click the ellipsis (...) button and enter your syntax in the Syntax Property dialog.
- The value `%%ThisValue%%` specifies the run time value of the text box, which is the file path, specified manually or populated by the browse dialog. This is the default.
- If the Syntax property includes `%%ThisValue%%` and the run time value of the text box is empty, then the file browser control does not generate any command syntax.

Enabling Rule. Specifies a rule that determines when the current control is enabled. Click the ellipsis (...) button to open the dialog box and specify the rule. The Enabling Rule property is visible only when other controls that can be used to specify an enabling rule exist on the canvas.

File type filter: The File Filter dialog box allows you to specify the file types displayed in the Files of type and Save as type drop-down lists for open and save dialogs accessed from a File System Browser control. By default, all file types are allowed.

To specify file types not explicitly listed in the dialog box:

1. Select Other.
2. Enter a name for the file type.
3. Enter a file type using the form `*.suffix`--for example, `*.xls`. You can specify multiple file types, each separated by a semicolon.

Tab

The Tab control adds a tab to the dialog. Any of the other controls can be added to the new tab. The Tab control has the following properties:

Identifier. The unique identifier for the control.

Title. The title of the tab.

Position. Specifies the position of the tab on the dialog, relative to the other tabs on the dialog.

Enabling Rule. Specifies a rule that determines when the current control is enabled. Click the ellipsis (...) button to open the dialog box and specify the rule. The Enabling Rule property is visible only when other controls that can be used to specify an enabling rule exist on the canvas.

Note: The Tab control is not available in compatibility mode.

Sub-dialog button

The Sub-dialog Button control specifies a button for launching a sub-dialog and provides access to the Dialog Builder for the sub-dialog. The Sub-dialog Button has the following properties:

Identifier. The unique identifier for the control.

Title. The text that is displayed in the button.

ToolTip. Optional ToolTip text that appears when the user hovers over the control.

Sub-dialog. Click the ellipsis (...) button to open the Custom Dialog Builder for the sub-dialog. You can also open the builder by double-clicking on the Sub-dialog button.

Mnemonic Key. An optional character in the title to use as a keyboard shortcut to the control. The character appears underlined in the title. The shortcut is activated by pressing Alt+[mnemonic key]. The Mnemonic Key property is not supported on Mac.

Enabling Rule. Specifies a rule that determines when the current control is enabled. Click the ellipsis (...) button to open the dialog box and specify the rule. The Enabling Rule property is visible only when other controls that can be used to specify an enabling rule exist on the canvas.

Note: The Sub-dialog Button control cannot be added to a sub-dialog.

Dialog properties for a sub-dialog: To view and set properties for a sub-dialog:

1. Open the sub-dialog by double-clicking on the button for the sub-dialog in the main dialog, or single-click the sub-dialog button and click the ellipsis (...) button for the Sub-dialog property.
2. In the sub-dialog, click on the canvas in an area outside of any controls. With no controls on the canvas, the properties for a sub-dialog are always visible.

Sub-dialog Name. The unique identifier for the sub-dialog. The Sub-dialog Name property is required.

Note: If you specify the Sub-dialog Name as an identifier in the Syntax Template--as in %%My Sub-dialog Name%%--it will be replaced at run-time with a blank-separated list of the syntax generated by each control in the sub-dialog, in the order in which they appear (top to bottom and left to right).

Title. Specifies the text to be displayed in the title bar of the sub-dialog box. The Title property is optional but recommended.

Help File. Specifies the path to an optional help file for the sub-dialog. This is the file that will be launched when the user clicks the Help button on the sub-dialog, and may be the same help file specified for the main dialog. Help files must be in HTML format. See the description of the Help File property for Dialog Properties for more information.

Specifying an Enabling Rule for a Control

You can specify a rule that determines when a control is enabled. For example, you can specify that a radio group is enabled when a target list is populated. The available options for specifying the enabling rule depend on the type of control that defines the rule.

Target List or Field Chooser

You can specify that the current control is enabled when a Target List or Field Chooser is populated with at least one field (Non-empty). You can alternately specify that the current control is enabled when a Target List or Field Chooser is not populated (Empty).

Check Box or Check Box Group

You can specify that the current control is enabled when a Check Box or Check Box Group is checked. You can alternately specify that the current control is enabled when a Check Box or Check Box Group is not checked.

Combo Box or Single-Select List Box

You can specify that the current control is enabled when a particular value is selected in a Combo Box or Single-Select List Box. You can alternately specify that the current control is enabled when a particular value is not selected in a Combo Box or Single-Select List Box.

Multi-Select List Box

You can specify that the current control is enabled when a particular value is among the selected values in a Multi-Select List Box. You can alternately specify that the current control is enabled when a particular value is not among the selected values in a Multi-Select List Box.

Radio Group

You can specify that the current control is enabled when a particular radio button is selected. You can alternately specify that the current control is enabled when a particular radio button is not selected.

Controls for which an enabling rule can be specified have an associated Enabling Rule property.

Note:

- Enabling rules apply whether the control that defines the rule is enabled or not. For example, consider a rule that specifies that a radio group is enabled when a target list is populated. The radio group is then enabled whenever the target list is populated, regardless of whether the target list is enabled.
- When a Tab control is disabled, all controls on the tab are disabled regardless of whether any of those controls have an enabling rule that is satisfied.
- When a Check Box Group is disabled, all controls in the group are disabled regardless of whether the controlling check box is checked.

Extension Properties

The Extension Properties dialog specifies information about the current extension within the Custom Dialog Builder for Extensions, such as the name of the extension and the files in the extension.

Note: The Extension Properties dialog does not apply in Compatibility mode.

- All custom dialogs that are created in the Custom Dialog Builder for Extensions are part of an extension.
- Fields on the Required tab of the Extension Properties dialog must be specified before you can install an extension and the custom dialogs that are contained in it.

To specify the properties for an extension, from the menus in the Custom Dialog Builder for Extensions choose:

Extension > Properties

Required properties of extensions

Name A unique name to associate with the extension. It can consist of up to three words and is not case sensitive. Characters are restricted to seven-bit ASCII. To minimize the possibility of name conflicts, you may want to use a multi-word name, where the first word is an identifier for your organization, such as a URL.

Summary

A short description of the extension that is intended to be displayed as a single line.

Version

A version identifier of the form x.x.x, where each component of the identifier must be an integer, as in 1.0.0. Zeros are implied if not provided. For example, a version identifier of 3.1 implies 3.1.0. The version identifier is independent of the IBM SPSS Statistics version.

Minimum SPSS Statistics Version

The minimum version of SPSS Statistics required to run the extension.

Files The Files list displays the files that are currently included in the extension. Click **Add** to add files to the extension. You can also remove files from the extension and you can extract files to a specified folder.

- Enhanced custom dialogs have a filetype of .cfe and compatible custom dialogs have a filetype of .spd. An extension can contain multiple .cfe files but it can contain only one .spd file.
- An extension must at least include a custom dialog specification (.cfe or .spd) file, or an XML specification file for an extension command. If an XML specification file is included, then the extension must include at least one Python, R, or Java code file--specifically, a file of type .py, .pyc, .pyo, .R, .class, or .jar.
- Translation files for components of the extension are added from the Localization settings on the Optional tab.
- You can add a readme file to the extension. Specify the filename as ReadMe.txt. Users can access the readme file from the dialog that displays the details for the extension. You can include localized versions of the readme file, specified as ReadMe_<language identifier>.txt, as in ReadMe_fr.txt for a French version.

Optional properties of extensions

General properties

Description

A more detailed description of the extension than that provided for the **Summary** field. For example, you might list the major features available with the extension.

Date An optional date for the current version of the extension. No formatting is provided.

Author

The author of the extension. You might want to include an email address.

Links A set of URLs to associate with the extension; for example, the author's home page. The format of this field is arbitrary so be sure to delimit multiple URLs with spaces, commas, or some other reasonable delimiter.

Keywords

A set of keywords with which to associate the extension.

Platform

Information about any restrictions that apply to using the extension on particular operating system platforms.

Dependencies

Maximum SPSS Statistics Version

The maximum version of IBM SPSS Statistics on which the extension can be run.

Integration Plug-in for Python required

Specifies whether the Integration Plug-in for Python is required.

- If the Python implementation code is specified to run in Python 3, then select **Python 3** for the Python Version.
- If the extension requires Python modules that are not explicitly included in the extension, then enter the names in the Required Python Modules control. Any such modules should be contributed to the IBM SPSS Predictive Analytics community (<https://developer.ibm.com/>)

`predictiveanalytics/`). To add the first module, click anywhere in the Required Python Modules control to highlight the entry field. Pressing **Enter**, with the cursor in a given row, will create a new row. You delete a row by selecting it and pressing **Delete**.

The user of the extension is responsible for downloading any required Python modules and copying them to a location that is specified for extension commands, as shown in the output from the SHOW EXTPATHS command. Alternatively, the modules can be copied to a location on the Python search path, such as the Python site-packages directory.

Integration Plug-in for R required

Specifies whether the Integration Plug-in for R is required.

If the extension requires any R packages from the CRAN package repository, then enter the names of those packages in the Required R Packages control. Names are case sensitive. To add the first package, click anywhere in the Required R Packages control to highlight the entry field. Pressing **Enter**, with the cursor in a given row, will create a new row. You delete a row by selecting it and pressing **Delete**. When the extension is installed, IBM SPSS Statistics will check if the required R packages are installed and attempt to download and install any that are missing.

Required Extensions

Enter the names of any extensions that are required by the current extension. To add the first extension, click anywhere in the Required Extensions control to highlight the entry field. Pressing **Enter**, with the cursor in a particular row, creates a new row. You delete a row by selecting it and pressing **Delete**. The user of the extensions is responsible for installing any required extensions.

Note: There is no option for specifying the Integration Plug-in for Java because it is always installed with IBM SPSS Statistics.

Localization

Custom Dialogs

You can add translated versions of the properties file (specifies all strings that appear in the dialog) for a custom dialog within the extension. To add translations for a particular dialog, select the dialog, click **Add Translations**, and select the folder that contains the translated versions. All translated files for a particular dialog must be in the same folder. For instructions on creating the translation files, see the topic “Creating Localized Versions of Custom Dialogs” on page 250.

Translation Catalogues Folder

You can include a folder containing translation catalogues. This allows you to provide localized messages and localized output from Python or R programs that implement an extension command included in the extension. You can also provide localized versions of the **Summary** and **Description** fields for the extension that are displayed when end users view the extension details from the Extension Hub. The set of all localized files for an extension must be in a folder named `lang`. Browse to the `lang` folder that contains the localized files and select that folder.

For information on localizing output from Python and R programs, see the topics on the Integration Plug-in for Python and the Integration Plug-in for R, in the Help system.

To provide localized versions of the **Summary** and **Description** fields, create a file named `<extension name>_<language-identifier>.properties` for each language for which a translation is being provided. At run time, if the `.properties` file for the current user interface language cannot be found, the values of the **Summary** and **Description** fields that are specified on the Required and Optional tabs are used.

- `<extension name>` is the value of the **Name** field for the extension, with any spaces replaced by underscore characters.
- `<language-identifier>` is the identifier for a particular language. Identifiers for the languages that are supported by IBM SPSS Statistics are shown in what follows.

For example, the French translations for an extension named `MYORG MYSTAT` are stored in the file `MYORG_MYSTAT_fr.properties`.

The .properties file must contain the following two lines, which specify the localized text for the two fields:

```
Summary=<localized text for Summary field>
Description=<localized text for Description field>
```

- The keywords Summary and Description must be in English and the localized text must be on the same line as the keyword, and with no line breaks.
- The file must be in ISO 8859-1 encoding. Characters that cannot be directly represented in this encoding must be written with Unicode escapes ("\\u").

The lang folder that contains the localized files must have a subfolder named <language-identifier> that contains the localized .properties file for a particular language. For example, the French .properties file must be in the lang/fr folder.

Language Identifiers

de. German

en. English

es. Spanish

fr. French

it. Italian

ja. Japanese

ko. Korean

pl. Polish

pt_BR. Brazilian Portuguese

ru. Russian

zh_CN. Simplified Chinese

zh_TW. Traditional Chinese

Managing custom dialogs

The Custom Dialog Builder for Extensions allows you to manage custom dialogs, within extensions that are created by you or by other users. Custom dialogs must be installed before they can be used.

Note: If you are working in compatibility mode, then see the topic “Managing custom dialogs in compatibility mode” on page 248.

Opening an extension that contains custom dialogs

You can open an extension bundle file (.spe) that contains the specifications for one or more custom dialogs or you can open an installed extension. You can modify any of the enhanced dialogs in the extension and save or install the extension. Installing the extension installs the dialogs that are contained in the extension. Saving the extension saves changes that were made to any of the dialogs in the extension.

To open an extension bundle file, from the menus in the Custom Dialog Builder for Extensions choose:

File > Open

To open an installed extension, from the menus in the Custom Dialog Builder for Extensions choose:

File > Open Installed

Note: If you are opening an installed extension to modify it, choosing **File > Install** reinstalls it, replacing the existing version.

Saving to an extension bundle file

Saving the extension that is open in the Custom Dialog Builder for Extensions also saves the custom dialogs that are contained in the extension. Extensions are saved to an extension bundle file (.spe).

From the menus in the Custom Dialog Builder for Extensions, choose:

File > Save

Installing an extension

Installing the extension that is open in the Custom Dialog Builder for Extensions also installs the custom dialogs that are contained in the extension. Installing an existing extension replaces the existing version, which includes replacing all custom dialogs in the extension that were already installed.

To install the currently open extension, from the menus in the Custom Dialog Builder for Extensions choose:

File > Install

By default, extensions are installed to a general user-writable location for your operating system. For more information, see the topic “[Installation locations for extensions](#)” on page 214.

Uninstalling an extension

From the menus in the Custom Dialog Builder for Extensions, choose:

File > Uninstall

Uninstalling an extension uninstalls all custom dialogs that are contained in the extension. You can also uninstall extensions from the Extension Hub.

Converting a compatible custom dialog to an enhanced custom dialog

If the currently open extension contains a compatible custom dialog (.spd) file, you can convert the compatible dialog to an enhanced dialog. The original compatible version of the custom dialog is retained in the extension.

From the menus in the Custom Dialog Builder for Extensions, choose:

Extension > Convert Compatible Dialog

Adding a custom dialog to an extension

You can add a new enhanced custom dialog to an extension.

From the menus in the Custom Dialog Builder for Extensions, choose:

Extension > New Dialog

Switching between multiple custom dialogs in an extension

If the current extension contains multiple custom dialogs, you can switch between them.

From the menus in the Custom Dialog Builder for Extensions, choose:

Extension > Edit Dialog and select the custom dialog that you want to work with.

Note: You cannot edit compatible custom dialog (.spd) files in the Custom Dialog Builder for Extensions. To modify a compatible custom dialog, you must use the Custom Dialog Builder in Compatibility mode. If you do not have a separate copy of the compatible custom dialog, you can extract it using the Extension Properties dialog, which is accessed from **Extension>Properties** within the Custom Dialog Builder for Extensions.

Creating a new extension

When you create a new extension in the Custom Dialog Builder for Extensions, a new empty custom dialog is added to the extension.

To create a new extension, from the menus in the Custom Dialog Builder for Extensions choose:

File > New

Managing custom dialogs in compatibility mode

In compatibility mode, you create and modify custom dialogs that are compatible with all releases of IBM SPSS Statistics and are referred to as *compatible custom dialogs*.

To open the Custom Dialog Builder in compatibility mode, from the menus choose:

Extensions > Utilities > Custom Dialog Builder (Compatibility mode)...

Opening a compatible custom dialog

You can open a compatible custom dialog package (.spd) file containing the specifications for a compatible custom dialog or you can open an installed compatible custom dialog.

To open a compatible custom dialog package file, from the menus in the Custom Dialog Builder choose:

File > Open

To open an installed compatible custom dialog, from the menus in the Custom Dialog Builder choose:

File > Open Installed

Note: If you open an installed dialog and modify it, choosing **File > Install** reinstalls it, replacing the existing version.

Saving to a compatible custom dialog package file

You can save the specifications for a compatible custom dialog to an external file. Specifications are saved to a compatible custom dialog package (.spd) file.

From the menus in the Custom Dialog Builder, choose:

File > Save

Converting a compatible custom dialog to an enhanced custom dialog

You can convert a dialog that is open in compatibility mode to an enhanced dialog to take advantage of the features that are available for enhanced dialogs. The enhanced version of the dialog is then part of a new extension.

From the menus in the Custom Dialog Builder, choose:

File > Convert to enhanced

Note: The conversion to an enhanced dialog cannot be undone.

Installing a compatible custom dialog

You can install the dialog that is open in the Custom Dialog Builder or you can install a dialog from a compatible custom dialog package (.spd) file. Reinstalling an existing dialog replaces the existing version.

To install the currently open dialog, from the menus in the Custom Dialog Builder choose:

File > Install

To install from a compatible custom dialog package file, from the IBM SPSS Statistics menus choose:

Extensions > Utilities > Install Custom Dialog (Compatibility mode)...

For Windows 7 and later, and Mac, dialogs are installed to a general user-writable location. To view the current installation locations for custom dialogs, run the following command syntax: SHOW EXTPATHS.

For Linux, and by default, installing a dialog requires write permission to the IBM SPSS Statistics installation directory.

If you do not have write permissions to the required location or would like to store installed dialogs elsewhere, you can specify one or more alternate locations by defining the *SPSS_CDIALOGS_PATH* environment variable. When present, the paths specified in *SPSS_CDIALOGS_PATH* take precedence over the default location. Custom dialogs will be installed to the first writable location. Note that Mac users may also utilize the *SPSS_CDIALOGS_PATH* environment variable. For multiple locations, separate each with a semicolon on Windows and a colon on Linux and Mac. The specified locations must exist on the target machine. After setting *SPSS_CDIALOGS_PATH*, you must restart IBM SPSS Statistics for the changes to take effect.

To create the *SPSS_CDIALOGS_PATH* environment variable on Windows, from the Control Panel:

Windows 7

1. Select User Accounts.
2. Select Change my environment variables.
3. Click **New**, enter *SPSS_CDIALOGS_PATH* in the Variable name field and the path(s) in the Variable value field.

Windows 8 and Later

1. Select System.
2. Select the Advanced tab, which is accessed from Advanced system settings, and click **Environment Variables**.

3. In the User variables section, click **New**, enter *SPSS_CDIALOGS_PATH* in the Variable name field and the path(s) in the Variable value field.

Sharing a compatible custom dialog as an extension

You can create an extension that includes a compatible custom dialog so that you can share the dialog with other users. See the topic “Creating and editing extension bundles” on page 252 for more information.

Uninstalling a compatible custom dialog

From the menus in the Custom Dialog Builder, choose:

File > Uninstall

Creating a new compatible custom dialog

To create a new compatible custom dialog, from the menus in the Custom Dialog Builder choose:

File > New

Custom Dialogs for Extension Commands

Extension commands are user-defined IBM SPSS Statistics commands that are implemented in the Python programming language, R or Java. Once deployed to an instance of IBM SPSS Statistics, an extension command is run in the same manner as any built-in IBM SPSS Statistics command. You can use the Custom Dialog Builder to create a dialog for an extension command by specifying the dialog's syntax template so that it generates the command syntax for the extension command.

When you build your custom dialog in the Custom Dialog Builder for Extensions, you add the files for the extension command (the XML file that specifies the syntax of the extension command and the implementation files that are written in Python, R, or Java) to the extension that contains your dialog. In the Custom Dialog Builder for Extensions, you add files to an extension from the Extension Properties dialog, which is accessed from **Extension > Properties**. You can then install the extension or save it to an extension bundle (.spe) file so that you can share it with other users.

If you are working in Compatibility mode, you first save the custom dialog to a compatible custom dialog package (.spd) file. You then create a new extension bundle and add the .spd file and the files for the extension command (the XML file that specifies the syntax of the extension command and the implementation files that are written in Python, R, or Java) to the extension bundle. You can then install the extension bundle or share it with other users. See the topic “Creating and editing extension bundles” on page 252 for more information.

Creating Localized Versions of Custom Dialogs

You can create localized versions of custom dialogs for any of the languages supported by IBM SPSS Statistics. You can localize any string that appears in a custom dialog and you can localize the optional help file.

To localize dialog strings

You must create a copy of the properties file associated with the custom dialog for each language that you plan to deploy. The properties file contains all of the localizable strings associated with the dialog.

Extract the custom dialog file (.cfe) from the extension by selecting the file in the Extension Properties dialog (within the Custom Dialog Builder for Extensions) and clicking **Extract**. Then, extract the contents of the .cfe file. A .cfe file is simply a .zip file. The extracted contents of a .cfe file includes a properties

file for each supported language, where the name of the file for a particular language is given by <Dialog Name>_<language identifier>.properties (see language identifiers in the table that follows).

Note: If you are working in Compatibility mode, then save the custom dialog to an external .spd file and extract the contents of the .spd file. An .spd file is simply a .zip file. Make a copy of the <Dialog Name>.properties file for each language that you plan to deploy and rename it to <Dialog Name>_<language identifier>.properties. Use the language identifiers in the table that follows. For example, if the dialog name is *mydialog* and you want to create a Japanese version of the dialog, then the localized properties file must be named *mydialog_ja.properties*.

1. Open each properties file, that you plan to translate, with a text editor that supports UTF-8, such as Notepad on Windows, or theTextEdit application on Mac. Modify the values associated with any properties that need to be localized, but do not modify the names of the properties. Properties associated with a specific control are prefixed with the identifier for the control. For example, the ToolTip property for a control with the identifier *options_button* is *options_button_tooltip_LABEL*. Title properties are simply named <*identifier*>_LABEL, as in *options_button_LABEL*.
2. Add the localized versions of the properties files back to the custom dialog file (.cfe) from the Localization settings on the Optional tab of the Extension Properties dialog. For more information, see the topic "Optional properties of extensions" on page 244. If you are working with an .spd file, then the localized properties files must be manually added back to the .spd file.

When the dialog is launched, IBM SPSS Statistics searches for a properties file whose language identifier matches the current language, as specified by the Language drop-down on the General tab in the Options dialog box. If no such properties file is found, the default file <Dialog Name>.properties is used.

To localize the help file

1. Make a copy of the help file associated with the custom dialog and localize the text for the language you want.
2. Rename the copy to <Help File>_<language identifier>.htm, using the language identifiers in the table below. For example, if the help file is *myhelp.htm* and you want to create a German version of the file, then the localized help file should be named *myhelp_de.htm*.

Store all localized versions of the help file in the same directory as the non-localized version. When you add the non-localized help file from the Help File property of Dialog Properties, the localized versions are automatically added to the dialog.

If there are supplementary files such as image files that also need to be localized, then you must manually modify the appropriate paths in the main help file to point to the localized versions. Supplementary files, including localized versions, must be manually added to the custom dialog (.cfe or .spd) file. See the previous section titled "To localize dialog strings" for information about accessing and manually modifying custom dialog files.

When the dialog is launched, IBM SPSS Statistics searches for a help file whose language identifier matches the current language, as specified by the Language drop-down on the General tab in the Options dialog box. If no such help file is found, the help file specified for the dialog (the file specified in the Help File property of Dialog Properties) is used.

Language Identifiers

de. German

en. English

es. Spanish

fr. French

it. Italian

ja. Japanese

ko. Korean

pl. Polish

pt_BR. Brazilian Portuguese

ru. Russian

zh_CN. Simplified Chinese

zh_TW. Traditional Chinese

Note: Text in custom dialogs and associated help files is not limited to the languages supported by IBM SPSS Statistics. You are free to write the dialog and help text in any language without creating language-specific properties and help files. All users of your dialog will then see the text in that language.

Creating and editing extension bundles

You can create or edit any extension bundle from the Create Extension Bundle or Edit Extension Bundle dialog.

- If you are creating or editing an extension bundle that contains an enhanced custom dialog, then you might prefer to use the Custom Dialog Builder for Extensions. With the Custom Dialog Builder for Extensions, you can create or modify an enhanced custom dialog at the same time as you create or edit the extension that contains the dialog. And you can edit installed extensions from within the Custom Dialog Builder for Extensions.
- If your extension bundle does not contain an enhanced custom dialog, then you must use the Create Extension Bundle or Edit Extension Bundle dialog to create or edit the extension bundle. For example, if you are creating an extension that includes a compatible custom dialog package (.spd) file and no enhanced dialogs, or an extension that consists only of an extension command, then you must use the Create Extension Bundle dialog.

Creating an extension bundle

1. From the menus, choose:
Extensions > Utilities > Create Extension Bundle...
2. Enter values for all fields on the Required tab.
3. Enter values for any fields on the Optional tab that are needed for your extension.
4. Specify a target file for the extension bundle.
5. Click **Save** to save the extension bundle to the specified location. This action closes the Create Extension Bundle dialog box.

Editing an extension bundle

1. From the menus, choose:
Extensions > Utilities > Edit Extension Bundles...
2. Open the extension bundle.
3. Modify values for any fields on the Required tab.
4. Modify values for any fields on the Optional tab.
5. Specify a target file for the extension bundle.

6. Click **Save** to save the extension bundle to the specified location. This action closes the Edit Extension Bundle dialog box.

For detailed information about fields on the Required tab and Optional tab, see the topics “Required properties of extensions” on page 243 and “Optional properties of extensions” on page 244.

Chapter 21. Production jobs

Production jobs run IBM SPSS Statistics in an automated fashion. The program runs unattended and ends after the last command runs. You can also schedule the production job to run automatically at scheduled times. Production jobs are useful if you often run the same set of time-consuming analyses, such as weekly reports.

You can run production jobs in two different ways:

Interactively. The program runs unattended in a separate session on either your local computer or a remote server. Your local computer must remain on (and connected to the remote server, if applicable) until the job is complete.

In the background on a server. The program runs in a separate session on a remote server. Your local computer does not have to remain on and does not have to remain connected to the remote server. You can disconnect and retrieve the results later.

Note: Running a production job on a remote server requires access to a server that is running IBM SPSS Statistics Server.

Creating and running production jobs

To create and run a production job:

1. From the menus, choose:
Utilities > Production Job
2. Click **New** to create a new production job.
or
3. Select a production job to run or modify from the list. Click **Browse** to change the directory location for the files that appears in the list.
Note: Production Facility job files (.spp) created in releases before release 16.0 do not run in release 16.0 or later. A conversion utility is available to convert Windows and Macintosh Production Facility job files to production jobs (.spj). For more information, see the topic “Converting Production Facility files” on page 261.
4. Specify one or more command syntax files to include in the job. Click the plus sign (+) icon to select command syntax files.
5. Select the output file name, location, and format.
6. Click **Run** to run the production job interactively or in the background on a server.

Default encoding

By default, IBM SPSS Statistics runs in Unicode mode. You can run production jobs in Unicode mode or the current locale encoding. The encoding affects how data and syntax files are read. For more information, see the topic “General options” on page 195.

- **Unicode (UTF-8).** The production job runs in Unicode mode. By default, text data files and command syntax files are read as Unicode UTF-8. You can specify a code page encoding for text data files with the ENCODING subcommand on the GET DATA command. You can specify a code page encoding for syntax files with the ENCODING subcommand on the INCLUDE or INSERT command.
 - **Local encoding for syntax files.** If a syntax file does not contain a UTF-8 byte order mark, read syntax files as the current locale encoding. This setting overrides any ENCODING specification on INCLUDE or INSERT. It also ignores any code page identifier in the file.

- **Local encoding.** The production job runs in the current locale encoding. Unless a different encoding is explicitly specified on a command that reads text data (for example, GET DATA), text data files are read in the current locale encoding. Syntax file with a Unicode UTF-8 byte order mark are read as Unicode UTF-8. All other syntax files are read in the current locale encoding.

Syntax files

Production jobs use command syntax files to tell IBM SPSS Statistics what to do. A command syntax file is a simple text file containing command syntax. You can use the syntax editor or any text editor to create the file. You can also generate command syntax by pasting dialog box selections into a syntax window. See the topic Chapter 14, "Working with Command Syntax," on page 165 for more information.

If you include multiple command syntax files, the files are concatenated together in the order they appear in the list and run as a single job.

Syntax format. Controls the form of the syntax rules used for the job.

- **Interactive.** Each command must end with a period. Periods can appear anywhere within the command, and commands can continue on multiple lines, but a period as the last nonblank character on a line is interpreted as the end of the command. Continuation lines and new commands can start anywhere on a new line. These are the "interactive" rules in effect when you select and run commands in a syntax window.
- **Batch.** Each command must start at the beginning of a new line (no blank spaces before the start of the command), and continuation lines must be indented at least one space. If you want to indent new commands, you can use a plus sign, dash, or period as the first character at the start of the line and then indent the actual command. The period at the end of the command is optional. This setting is compatible with the syntax rules for command files included with the INCLUDE command.

Note: Do *not* use the Batch option if your syntax files contain GGRAPH command syntax that includes GPL statements. GPL statements will only run under interactive rules.

Error Processing. Controls the treatment of error conditions in the job.

- **Continue processing after errors.** Errors in the job do not automatically stop command processing. The commands in the production job files are treated as part of the normal command stream, and command processing continues in the normal fashion.
- **Stop processing immediately.** Command processing stops when the first error in a production job file is encountered. This is compatible with the behavior of command files included with the INCLUDE command.

Output

These options control the name, location, and format of the production job results. The following format options are available:

- **Viewer file (.spv).** Results are saved in IBM SPSS Statistics Viewer format in the specified file location. You can store to disk or to an IBM SPSS Collaboration and Deployment Services Repository. Storing to an IBM SPSS Collaboration and Deployment Services Repository requires the Statistics Adapter.
- **Web Reports (.spw).** Results are stored to an IBM SPSS Collaboration and Deployment Services Repository. This requires the Statistics Adapter.
- **Word/RTF.** Pivot tables are exported as Word tables with all formatting attributes intact (for example, cell borders, font styles, and background colors). Text output is exported as formatted RTF. Charts, tree diagrams, and model views are included in PNG format. Note that Microsoft Word might not display extremely wide tables properly.
- **Excel.** Pivot table rows, columns, and cells are exported as Excel rows, columns, and cells, with all formatting attributes intact (for example, cell borders, font styles, and background colors). Text output is exported with all font attributes intact. Each line in the text output is a row in the Excel file, with the

entire contents of the line in a single cell. Charts, tree diagrams, and model views are included in PNG format. Output can be exported as *Excel 97-2004* or *Excel 2007 and higher*.

- **HTML.** Pivot tables are exported as HTML tables. Text output is exported as preformatted HTML. Charts, tree diagrams, and model views are embedded in the document in the selected graphic format. A browser compatible with HTML 5 is required for viewing output that is exported in HTML format.
- **Web Report.** A web report is an interactive document that is compatible with most browsers. Many of the interactive features of pivot tables available in the Viewer are also available in web reports. You can also export a web report as an IBM Cognos Active Report.
- **Portable Document Format.** All output is exported as it appears in Print Preview, with all formatting attributes intact.
- **PowerPoint file.** Pivot tables are exported as Word tables and are embedded on separate slides in the PowerPoint file, with one slide for each pivot table. All formatting attributes of the pivot table are retained (for example, cell borders, font styles, and background colors). Charts, tree diagrams, and model views are exported in TIFF format. Text output is not included.
Export to PowerPoint is available only on Windows operating systems.
- **Text.** Text output formats include plain text, UTF-8, and UTF-16. Pivot tables can be exported in tab-separated or space-separated format. All text output is exported in space-separated format. For charts, tree diagrams, and model views, a line is inserted in the text file for each graphic, indicating the image file name.

Print Viewer file on completion. Sends the final Viewer output file to the printer on completion of the production job. This option is not available when running a production job in the background on a remote server.

HTML options

Table Options. No table options are available for HTML format. All pivot tables are converted to HTML tables.

Image Options. The available image types are: EPS, JPEG, TIFF, PNG, and BMP. On Windows operating systems, EMF (enhanced metafile) format is also available. You can also scale the image size from 1% to 200%.

PowerPoint options

Table Options. You can use the Viewer outline entries as slide titles. Each slide contains a single output item. The title is formed from the outline entry for the item in the outline pane of the Viewer.

Image Options. You can scale the image size from 1% to 200%. (All images are exported to PowerPoint in TIFF format.)

Note: PowerPoint format is only available on Windows operating systems and requires PowerPoint 97 or later.

PDF options

Embed bookmarks. This option includes bookmarks in the PDF document that correspond to the Viewer outline entries. Like the Viewer outline pane, bookmarks can make it much easier to navigate documents with a large number of output objects.

Embed fonts. Embedding fonts ensures that the PDF document will look the same on all computers. Otherwise, if some fonts used in the document are not available on the computer being used to view (or print) the PDF document, font substitution may yield suboptimal results.

Text options

Table Options. Pivot tables can be exported in tab-separated or space-separated format. For space-separated format, you can also control:

- **Column Width.** Autofit does not wrap any column contents, and each column is as wide as the widest label or value in that column. Custom sets a maximum column width that is applied to all columns in the table, and values that exceed that width wrap onto the next line in that column.
- **Row/Column Border Character.** Controls the characters used to create row and column borders. To suppress display of row and column borders, enter blank spaces for the values.

Image Options. The available image types are: EPS, JPEG, TIFF, PNG, and BMP. On Windows operating systems, EMF (enhanced metafile) format is also available. You can also scale the image size from 1% to 200%.

Production jobs with OUTPUT commands

Production jobs honor OUTPUT commands, such as OUTPUT SAVE, OUTPUT ACTIVATE, and OUTPUT NEW. OUTPUT SAVE commands executed during the course of a production job will write the contents of the specified output documents to the specified locations. This is in addition to the output file created by the production job. When using OUTPUT NEW to create a new output document, it is recommended that you explicitly save it with the OUTPUT SAVE command.

A production job output file consists of the contents of the active output document as of the end of the job. For jobs containing OUTPUT commands, the output file may not contain all output created in the session. For example, suppose the production job consists of a number of procedures followed by an OUTPUT NEW command, followed by more procedures but no more OUTPUT commands. The OUTPUT NEW command defines a new active output document. At the end of the production job, it will contain output from only the procedures executed after the OUTPUT NEW command.

Runtime values

Runtime values defined in a production job file and used in a command syntax file simplify tasks such as running the same analysis for different data files or running the same set of commands for different sets of variables. For example, you could define the runtime value @datafile to prompt you for a data file name each time you run a production job that uses the string @datafile in place of a file name in the command syntax file.

- Runtime value substitution uses the macro facility (DEFINE-!ENDDEFINE) to create string substitution values.
- Runtime values in command syntax files are ignored if they are enclosed in quotation marks. If the runtime value needs to be quoted, select **Quote Value**. If the runtime value is only part of a quoted string, you can include the runtime value in a macro with the !UNQUOTE and !EVAL parameters.

Symbol. The string in the command syntax file that triggers the production job to prompt the user for a value. The symbol name must begin with an @ sign and must conform to variable naming rules. See the topic “Variable names” on page 50 for more information.

Default Value. The value that the production job supplies by default if you don't enter a different value. This value is displayed when the production job prompts you for information. You can replace or modify the value at runtime. If you don't provide a default value, don't use the silent keyword when running the production job with command line switches, unless you also use the -symbol switch to specify runtime values. See the topic “Running production jobs from a command line” on page 260 for more information.

User Prompt. The descriptive label that is displayed when the production job prompts you to enter information. For example, you could use the phrase “What data file do you want to use?” to identify a field that requires a data filename.

Quote Value. Encloses the default value or the value entered by the user in quotes. For example, file specifications should be enclosed in quotes.

Command syntax file with user prompt symbols

```
GET FILE @datafile. /*check the Quote value option to quote file specifications.  
FREQUENCIES VARIABLES=@varlist. /*do not check the Quote value option
```

Using a macro to replace part of a string value

If the entire replacement string is enclosed in quotes, you can use the **Quote Value** option. If the replacement string is only part of a quoted string, you can include the runtime value in a macro, using the !UNQUOTE and !EVAL functions.

```
DEFINE !LabelSub()  
VARIABLE LABELS Var1  
!QUOTE(!concat(!UNQUOTE('First part of label - '), !UNQUOTE(!EVAL(@replace)), !UNQUOTE(' - rest of label'))).  
!ENDDEFINE.  
!LabelSub.
```

Run options

You can run production jobs in two different ways:

Interactively. The program runs unattended in a separate session on either your local computer or a remote server. Your local computer must remain on (and connected to the remote server, if applicable) until the job is complete.

In the background on a server. The program runs in a separate session on a remote server. Your local computer does not have to remain on and does not have to remain connected to the remote server. You can disconnect and retrieve the results later.

Note: Running a production job on a remote server requires access to a server that is running IBM SPSS Statistics Server.

Statistics server. If you select to run the production job in the background on a remote server, you must specify the server on which it will run. Click **Select Server** to specify the server. This applies only to jobs run in the background on a remote server, not jobs run interactively on a remote server.

Server login

Use the Server login dialog to add and modify remote servers and to select the server to use to run the current production job. Remote servers usually require a user ID and password, and a domain name may also be necessary. Contact your system administrator for information about available servers, a user ID and password, domain names, and other connection information.

If your site is running IBM SPSS Collaboration and Deployment Services 3.5 or later, you can click **Search...** to view a list of servers that are available on your network. If you are not logged on to a IBM SPSS Collaboration and Deployment Services Repository, you will be prompted to enter connection information before you can view the list of servers.

Adding and Editing Server Login Settings

Use the Server Login Settings dialog box to add or edit connection information for remote servers for use in distributed analysis mode.

Contact your system administrator for a list of available servers, port numbers for the servers, and additional connection information. Do not use the Secure Socket Layer unless instructed to do so by your administrator.

Server Name. A server "name" can be an alphanumeric name that is assigned to a computer (for example, NetworkServer) or a unique IP address that is assigned to a computer (for example, 202.123.456.78).

Port Number. The port number is the port that the server software uses for communications.

Description. You can enter an optional description to display in the servers list.

Connect with Secure Socket Layer. Secure Socket Layer (SSL) encrypts requests for distributed analysis when they are sent to the remote server. Before you use SSL, check with your administrator. For this option to be enabled, SSL must be configured on your desktop computer and the server.

User prompts

A production job prompts you for values whenever you run a production job that contains defined runtime symbols. You can replace or modify the default values that are displayed. Those values are then substituted for the runtime symbols in all command syntax files associated with the production job.

Background job status

The background job status tab displays the status of production jobs that have been submitted to run in the background on a remote server.

Server name. Displays the name of the currently selected remote server. Only jobs submitted to that server are displayed in the list. To display jobs submitted to a different server, click **Select Server**.

Job status information. Includes production job name, current job status, and start and end time.

Refresh. Updates the job status information.

Get job output. Retrieves the output from the selected production job. The output for each job resides on the server the job was run on; so you must switch to the status for that server to select the job and retrieve the output. This button is disabled if the job status is *Running*.

Cancel job. Cancels the selected production job. This button is only enabled if the job status is *Running*.

Remove job. Removes the selected production job. This removes the job from the list and removes the associated files from the remote server. This button is disabled if the job status is *Running*.

Note: Background job status does not reflect the status of any jobs run interactively on a remote server.

Running production jobs from a command line

Command line switches enable you to schedule production jobs to run automatically at certain times, using scheduling utilities available on your operating system. The basic form of the command line argument is:

```
stats filename.spj -production
```

Depending on how you invoke the production job, you may need to include directory paths for the *stats* executable file (located in the directory in which the application is installed) and/or the production job file.

You can run production jobs from a command line with the following switches:

-production [prompt|silent]. Start the application in production mode. The **prompt** and **silent** keywords specify whether to display the dialog box that prompts for runtime values if they are specified in the job.

The prompt keyword is the default and shows the dialog box. The silent keyword suppresses the dialog box. If you use the silent keyword, you can define the runtime symbols with the -symbol switch. Otherwise, the default value is used. The -switchserver and -singleseat switches are ignored when using the -production switch.

-symbol <values>. List of symbol-value pairs used in the production job. Each symbol name starts with @. Values that contain spaces should be enclosed in quotes. Rules for including quotes or apostrophes in string literals may vary across operating systems, but enclosing a string that includes single quotes or apostrophes in double quotes usually works (for example, "'a quoted value'"). The symbols must be defined in the production job using the Runtime Values tab. See the topic "Runtime values" on page 258 for more information.

-background. Run the production job in the background on a remote server. Your local computer does not have to remain on and does not have to remain connected to the remote server. You can disconnect and retrieve the results later. You must also include the -production switch and specify the server using the -server switch.

To run production jobs on a remote server, you also need to specify the server login information:

-server <inet:hostname:port> or -server <ssl:hostname:port>. The name or IP address and port number of the server. Windows only.

-user <name>. A valid user name. If a domain name is required, precede the user name with the domain name and a backslash (\). Windows only.

-password <password>. The user's password.

Example

```
stats \production_jobs\prodjob1.spj -production silent -symbol @datafile /data/July_data.sav
```

- This example assumes that you are running the command line from the installation directory, so no path is required for the *stats* executable file.
- This example also assumes that the production job specifies that the value for @datafile should be quoted (Quote Value checkbox on the Runtime Values tab), so no quotes are necessary when specifying the data file on the command line. Otherwise, you would need to specify something like "'/data/July_data.sav'" to include quotes with the data file specification, since file specifications should be quoted in command syntax.
- The directory path for the location of the production job uses the Windows back slash convention. On Macintosh and Linux, use forward slashes. The forward slashes in the quoted data file specification will work on all operating systems since this quoted string is inserted into the command syntax file and forward slashes are acceptable in commands that include file specifications (for example, GET FILE, GET DATA, SAVE) on all operating systems.
- The silent keyword suppresses any user prompts in the production job, and the -symbol switch inserts the quoted data file name and location wherever the runtime symbol @datafile appears in the command syntax files included in the production job.

Converting Production Facility files

Production Facility job files (.spp) created in releases prior to 16.0 will not run in release 16.0 or later. For Windows and Macintosh Production Facility job files created in earlier releases, you can use *prodconvert*, located in the installation directory, to convert those files to new production job files (.spj). Run *prodconvert* from a command window using the following specifications:

```
[installpath]\prodconvert [filepath]\filename.spp
```

where *[installpath]* is the location of the folder in which IBM SPSS Statistics is installed and *[filepath]* is the folder t the original production job file is located. A new file with the same name but with the extension

.spj is created in the same folder as the original file. (Note: If the path contains spaces, enclose each path and file specification in double quotes. On Macintosh operating systems, use forward slashes instead of back slashes.)

Limitations

- WMF and EMF chart formats are not supported. PNG format is used in place of these formats.
- The export options **Output Document (No Charts)**, **Charts Only**, and **Nothing** are not supported. All output objects supported by the selected format are included.
- Remote server settings are ignored. To specify remote server settings for distributed analysis, you need to run the production job from a command line, using command line switches to specify the server settings. See the topic “Running production jobs from a command line” on page 260 for more information.
- Publish to Web settings are ignored.

Chapter 22. Output Management System

The Output Management System (OMS) provides the ability to automatically write selected categories of output to different output files in different formats. Formats include: Word, Excel, PDF, IBM SPSS Statistics data file format (.sav), Viewer file format (.spv), web report format (.spw), XML, HTML, and text. See the topic “OMS options” on page 266 for more information.

To use the Output Management System Control Panel

1. From the menus choose:

Utilities > OMS Control Panel...

You can use the control panel to start and stop the routing of output to various destinations.

- Each OMS request remains active until explicitly ended or until the end of the session.
- A destination file that is specified on an OMS request is unavailable to other procedures and other applications until the OMS request is ended.
- While an OMS request is active, the specified destination files are stored in memory (RAM), so active OMS requests that write a large amount of output to external files may consume a large amount of memory.
- Multiple OMS requests are independent of each other. The same output can be routed to different locations in different formats, based on the specifications in different OMS requests.
- The order of the output objects in any particular destination is the order in which they were created, which is determined by the order and operation of the procedures that generate the output.

Limitations

- For Output XML format, the specification for the Headings output type has no effect. If any output from a procedure is included, the procedure title output is included.
- If the OMS specification results in nothing other than Headings objects or a Notes tables being included for a procedure, then nothing is included for that procedure.

Adding new OMS requests

1. Select the output types (tables, charts, etc.) that you want to include. See the topic “Output object types” on page 264 for more information.
2. Select the commands to include. If you want to include all output, select all items in the list. See the topic “Command identifiers and table subtypes” on page 265 for more information.
3. For commands that produce pivot table output, select the specific table types to include.
The list displays only the tables that are available in the selected commands; any table type that is available in one or more of the selected commands is displayed in the list. If no commands are selected, all table types are displayed. See the topic “Command identifiers and table subtypes” on page 265 for more information.
4. To select tables based on text labels instead of subtypes, click **Labels**. See the topic “Labels” on page 266 for more information.
5. Click **Options** to specify the output format (for example, IBM SPSS Statistics data file, XML, or HTML). By default, Output XML format is used. See the topic “OMS options” on page 266 for more information.
6. Specify an output destination:
 - **File.** All selected output is routed to a single file.

- **Based on object names.** Output is routed to multiple destination files based on object names. A separate file is created for each output object, with a filename based on either table subtype names or table labels. Enter the destination folder name.
- **New dataset.** For IBM SPSS Statistics data file format output, you can route the output to a dataset. The dataset is available for subsequent use in the same session but is not saved unless you explicitly save it as a file prior to the end of the session. This option is available only for IBM SPSS Statistics data file format output. Dataset names must conform to variable-naming rules. See the topic “Variable names” on page 50 for more information.

7. Optionally:

- Exclude the selected output from the Viewer. If you select **Exclude from Viewer**, the output types in the OMS request will not be displayed in the Viewer window. If multiple active OMS requests include the same output types, the display of those output types in the Viewer is determined by the most recent OMS request that contains those output types. See the topic “Excluding output display from the viewer” on page 269 for more information.
- Assign an ID string to the request. All requests are automatically assigned an ID value, and you can override the system default ID string with a descriptive ID, which can be useful if you have multiple active requests that you want to identify easily. ID values that you assign cannot start with a dollar sign (\$).

Tips for selecting multiple items in a list

The following tips are for selecting multiple items in a list:

- Press Ctrl+A to select all items in a list.
- Use Shift+click to select multiple contiguous items.
- Use Ctrl+click to select multiple noncontiguous items.

To end and delete OMS requests

Active and new OMS requests are displayed in the Requests list, with the most recent request at the top. You can change the widths of the information columns by clicking and dragging the borders, and you can scroll the list horizontally to see more information about a particular request.

An asterisk (*) after the word **Active** in the *Status* column indicates an OMS request that was created with command syntax that includes features that are not available in the Control Panel.

To end a specific, active OMS request:

1. In the Requests list, click any cell in the row for the request.
2. Click **End**.

To end all active OMS requests:

1. Click **End All**.

To delete a new request (a request that has been added but is not yet active):

1. In the Requests list, click any cell in the row for the request.
2. Click **Delete**.

Note: Active OMS requests are not ended until you click **OK**.

Output object types

There are different types of output objects:

Charts. This includes charts created with the Chart Builder, charting procedures, and charts created by statistical procedures (for example, a bar chart created by the Frequencies procedure).

Headings. Text objects that are labeled *Title* in the outline pane of the Viewer.

Logs. Log text objects. Log objects contain certain types of error and warning messages. Depending on your Options settings (Edit menu, Options, Viewer tab), log objects may also contain the command syntax that is executed during the session. Log objects are labeled *Log* in the outline pane of the Viewer.

Models. Output objects displayed in the Model Viewer. A single model object can contain multiple views of the model, including both tables and charts.

Tables. Output objects that are pivot tables in the Viewer (includes Notes tables). Tables are the only output objects that can be routed to IBM SPSS Statistics data file (.sav) format.

Texts. Text objects that aren't logs or headings (includes objects labeled *Text Output* in the outline pane of the Viewer).

Trees. Tree model diagrams that are produced by SPSS Statistics Professional Edition or the Decision Trees option.

Warnings. Warning objects contain certain types of error and warning messages.

Command identifiers and table subtypes

Command identifiers

Command identifiers are available for all statistical and charting procedures and any other commands that produce blocks of output with their own identifiable heading in the outline pane of the Viewer. These identifiers are usually (but not always) the same or similar to the procedure names on the menus and dialog box titles, which are usually (but not always) similar to the underlying command names. For example, the command identifier for the Frequencies procedure is "Frequencies," and the underlying command name is also the same.

There are, however, some cases where the procedure name and the command identifier and/or the command name are not all that similar. For example, all of the procedures on the Nonparametric Tests submenu (from the Analyze menu) use the same underlying command, and the command identifier is the same as the underlying command name: Npar Tests.

Table subtypes

Table subtypes are the different types of pivot tables that can be produced. Some subtypes are produced by only one command; other subtypes can be produced by multiple commands (although the tables may not look similar). Although table subtype names are generally descriptive, there can be many names to choose from (particularly if you have selected a large number of commands); also, two subtypes may have very similar names.

To find command identifiers and table subtypes

When in doubt, you can find command identifiers and table subtype names in the Viewer window:

1. Run the procedure to generate some output in the Viewer.
2. Right-click the item in the outline pane of the Viewer.
3. Choose **Copy OMS Command Identifier** or **Copy OMS Table Subtype**.
4. Paste the copied command identifier or table subtype name into any text editor (such as a Syntax Editor window).

Labels

As an alternative to table subtype names, you can select tables based on the text that is displayed in the outline pane of the Viewer. You can also select other object types based on their labels. Labels are useful for differentiating between multiple tables of the same type in which the outline text reflects some attribute of the particular output object, such as the variable names or labels. There are, however, a number of factors that can affect the label text:

- If split-file processing is on, split-file group identification may be appended to the label.
- Labels that include information about variables or values are affected by your current output label options settings (Edit menu, Options, Output Labels tab).
- Labels are affected by the current output language setting (Edit menu, Options, General tab).

To specify labels to use to identify output objects

1. In the Output Management System Control Panel, select one or more output types and then select one or more commands.
2. Click **Labels**.
3. Enter the label *exactly* as it appears in the outline pane of the Viewer window. (You can also right-click the item in the outline, choose **Copy OMS Label**, and paste the copied label into the Label text field.)
4. Click **Add**.
5. Repeat the process for each label that you want to include.
6. Click **Continue**.

Wildcards

You can use an asterisk (*) as the last character of the label string as a wildcard character. All labels that begin with the specified string (except for the asterisk) will be selected. This process works only when the asterisk is the last character, because asterisks can appear as valid characters inside a label.

OMS options

You can use the OMS Options dialog box to:

- Specify the output format.
- Specify the image format (for HTML and Output XML output formats).
- Specify what table dimension elements should go into the row dimension.
- Include a variable that identifies the sequential table number that is the source for each case (for IBM SPSS Statistics data file format).

To specify OMS options

1. Click **Options** in the Output Management System Control Panel.

Format

Excel. Excel 97-2004 and Excel 2007 and higher formats. Pivot table rows, columns, and cells are exported as Excel rows, columns, and cells, with all formatting attributes intact -- for example, cell borders, font styles, and background colors. Text output is exported with all font attributes intact. Each line in the text output is a row in the Excel file, with the entire contents of the line contained in a single cell. Charts, tree diagrams, and model views are included in PNG format.

HTML. Output objects that would be pivot tables in the Viewer are converted to HTML tables. Text output objects are tagged <PRE> in the HTML. Charts, tree diagrams, and model views are embedded in the document in the selected format.

Output XML. XML that conforms to the *spss-output* schema.

PDF. Output is exported as it would appear in Print Preview, with all formatting attributes intact. The PDF file includes bookmarks that correspond to the entries in the Viewer outline pane.

IBM SPSS Statistics Data File. This format is a binary file format. All output object types other than tables are excluded. Each column of a table becomes a variable in the data file. To use a data file that is created with OMS in the same session, you must end the active OMS request before you can open the data file. See the topic “Routing output to IBM SPSS Statistics data files” on page 269 for more information.

Text. Space-separated text. Output is written as text, with tabular output aligned with spaces for fixed-pitch fonts. Charts, tree diagrams, and model views are excluded.

Tabbed Text. Tab-delimited text. For output that is displayed as pivot tables in the Viewer, tabs delimit table column elements. Text block lines are written as is; no attempt is made to divide them with tabs at useful places. Charts, tree diagrams, and model views are excluded.

Viewer File. This is the same format used when you save the contents of a Viewer window.

Web Report File. This output file format is designed for use with Predictive Enterprise Services. It is essentially the same as the IBM SPSS Statistics Viewer format except that tree diagrams are saved as static images.

Word/RTF. Pivot tables are exported as Word tables with all formatting attributes intact—for example, cell borders, font styles, and background colors. Text output is exported as formatted RTF. Charts, tree diagrams, and model views are included in PNG format.

Graphics Images

For HTML and Output XML formats, you can include charts, tree diagrams, and model views as image files. A separate image file is created for each chart and/or tree.

- For HTML document format, standard tags are included in the HTML document for each image file.
- For Output XML document format, the XML file contains a chart element with an ImageFile attribute of the general form <chart imageFile="filepath/filename"/> for each image file.
- Image files are saved in a separate subdirectory (folder). The subdirectory name is the name of the destination file, without any extension and with _files appended to the end. For example, if the destination file is *julydata.htm*, the images subdirectory will be named *julydata_files*.

Format. The available image formats are PNG, JPG, and BMP.

Size. You can scale the image size from 10% to 200%.

Include Imagemaps. For HTML document format, this option creates image map ToolTips that display information for some chart elements, such as the value of the selected point on a line chart or bar on a bar chart.

Table Pivots

For pivot table output, you can specify the dimension element(s) that should appear in the columns. All other dimension elements appear in the rows. For IBM SPSS Statistics data file format, table columns become variables, and rows become cases.

- If you specify multiple dimension elements for the columns, they are nested in the columns in the order in which they are listed. For IBM SPSS Statistics data file format, variable names are constructed by nested column elements. See the topic “Variable names in OMS-generated data files” on page 270 for more information.
- If a table doesn't contain any of the listed dimension elements, all dimension elements for that table will appear in the rows.
- Table pivots that are specified here have no effect on tables that are displayed in the Viewer.

Each dimension of a table--row, column, layer--may contain zero or more elements. For example, a simple two-dimensional crosstabulation contains a single row dimension element and a single column dimension element, each of which contains one of the variables that are used in the table. You can use either positional arguments or dimension element "names" to specify the dimension elements that you want to put into the column dimension.

All dimensions in rows. Creates a single row for each table. For IBM SPSS Statistics format data files, this means each table is a single case, and all the table elements are variables.

List of positions. The general form of a positional argument is a letter indicating the default position of the element--C for column, R for row, or L for layer--followed by a positive integer indicating the default position within that dimension. For example, R1 would indicate the outermost row dimension element.

- To specify multiple elements from multiple dimensions, separate each dimension with a space—for example, R1 C2.
- The dimension letter followed by ALL indicates all elements in that dimension in their default order. For example, CALL is the same as the default behavior (using all column elements in their default order to create columns).
- CALL RALL LALL (or RALL CALL LALL, and so on) will put all dimension elements into the columns. For IBM SPSS Statistics data file format, this creates one row/case per table in the data file.

List of dimension names. As an alternative to positional arguments, you can use dimension element "names," which are the text labels that appear in the table. For example, a simple two-dimensional crosstabulation contains a single row dimension element and a single column dimension element, each with labels based on the variables in those dimensions, plus a single layer dimension element labeled *Statistics* (if English is the output language).

- Dimension element names may vary, based on the output language and/or settings that affect the display of variable names and/or labels in tables.
- Each dimension element name must be enclosed in single or double quotation marks. To specify multiple dimension element names, include a space between each quoted name.

The labels that are associated with the dimension elements may not always be obvious.

To see all dimension elements and their labels for a pivot table

1. Activate (double-click) the table in the Viewer.
2. From the menus choose:
View > Show All
or
3. If the pivoting trays aren't displayed, from the menus choose:
Pivot > Pivoting Trays

The element labels are displayed in the pivoting trays.

Logging

You can record OMS activity in a log in XML or text format.

- The log tracks all new OMS requests for the session but does not include OMS requests that were already active before you requested a log.
- The current log file ends if you specify a new log file or if you deselect (clear) **Log OMS activity**.

How to specify OMS logging

To specify OMS logging:

1. Click **Logging** in the Output Management System Control Panel.

Excluding output display from the viewer

The **Exclude from Viewer** check box affects all output that is selected in the OMS request by suppressing the display of that output in the Viewer window. This process is often useful for production jobs that generate a lot of output and when you don't need the results in the form of a Viewer document (.spv file). You can also use this functionality to suppress the display of particular output objects that you simply never want to see, without routing any other output to some external file and format.

To suppress the display of certain output objects without routing other output to an external file:

1. Create an OMS request that identifies the unwanted output.
2. Select **Exclude from Viewer**.
3. For the output destination, select **File**--but leave the File field blank.
4. Click **Add**.

The selected output will be excluded from the Viewer while all other output will be displayed in the Viewer in the normal fashion.

Note: This setting has no effect on OMS output saved to external formats or files, including the Viewer SPV and SPW formats. It also has no effect on output saved to SPV format in a batch job executed with the Batch Facility (available with IBM SPSS Statistics Server).

Routing output to IBM SPSS Statistics data files

A data file in IBM SPSS Statistics format consists of variables in the columns and cases in the rows, which is essentially the format in which pivot tables are converted to data files:

- Columns in the table are variables in the data file. Valid variable names are constructed from the column labels.
- Row labels in the table become variables with generic variable names (*Var1*, *Var2*, *Var3*, and so on) in the data file. The values of these variables are the row labels in the table.
- Three table-identifier variables are automatically included in the data file: *Command_*, *Subtype_*, and *Label_*. All three are string variables. The first two variables correspond to the command and subtype identifiers. See the topic “Command identifiers and table subtypes” on page 265 for more information. *Label_* contains the table title text.
- Rows in the table become cases in the data file.

Data files created from multiple tables

When multiple tables are routed to the same data file, each table is added to the data file in a fashion that is similar to merging data files by adding cases from one data file to another data file (Data menu, Merge Files, Add Cases).

- Each subsequent table will always add cases to the data file.
- If column labels in the tables differ, each table may also add variables to the data file, with missing values for cases from other tables that don't have an identically labeled column.

- If any tables do not have the same number of row elements as the other tables, no data file will be created. The number of rows doesn't have to be the same; the number of row *elements* that become variables in the data file must be the same. For example, a two-variable crosstabulation and a three-variable crosstabulation contain different numbers of row elements, because the “layer” variable is actually nested within the row variable in the default three-variable crosstabulation display.

Controlling column elements to control variables in the data file

In the Options dialog box of the Output Management System Control Panel, you can specify which dimension elements should be in the columns and therefore will be used to create variables in the generated data file. This process is equivalent to pivoting the table in the Viewer.

For example, the Frequencies procedure produces a descriptive statistics table with statistics in the rows, while the Descriptives procedure produces a descriptive statistics table with statistics in the columns. To include both table types in the same data file in a meaningful fashion, you need to change the column dimension for one of the table types.

Because both table types use the element name "Statistics" for the statistics dimension, we can put the statistics from the Frequencies statistics table in the columns simply by specifying "Statistics" (in quotation marks) in the list of dimension names in the OMS Options dialog box.

Some of the variables will have missing values, because the table structures still aren't exactly the same with statistics in the columns.

Variable names in OMS-generated data files

OMS constructs valid, unique variable names from column labels:

- Row and layer elements are assigned generic variable names—the prefix *Var* followed by a sequential number.
- Characters that aren't allowed in variable names (spaces, parentheses, etc.) are removed. For example, “This (Column) Label” would become a variable named *ThisColumnNameLabel*.
- If the label begins with a character that is allowed in variable names but not allowed as the first character (for example, a number), “@” is inserted as a prefix. For example, “2nd” would become a variable named @*2nd*.
- Column labels that would result in duplicate variable names are resolved by appending an underscore and a sequential letter. For example, the second instance of “Count” would become a variable named *Count_A*.
- Underscores or periods at the end of labels are removed from the resulting variable names. The underscores at the end of the automatically generated variables *Command_*, *Subtype_*, and *Label_* are not removed.
- If more than one element is in the column dimension, variable names are constructed by combining category labels with underscores between category labels. Group labels are not included. For example, if *VarB* is nested under *VarA* in the columns, you would get variables like *CatA1_CatB1*, not *VarA_CatA1_VarB_CatB1*.

OXML table structure

Output XML (OXML) is XML that conforms to the *spss-output* schema. For a detailed description of the schema, see the Output Schema section of the Help system.

- OMS command and subtype identifiers are used as values of the *command* and *subType* attributes in OXML. An example is as follows:


```
<command text="Frequencies" command="Frequencies" ...>
  <pivotTable text="Gender" label="Gender" subType="Frequencies" ...>
```
- OMS command and subType attribute values are not affected by output language or display settings for variable names/labels or values/value labels.

- XML is case sensitive. A subType attribute value of "frequencies" is *not* the same as a subType attribute value of "Frequencies."
- All information that is displayed in a table is contained in attribute values in OXML. At the individual cell level, OXML consists of "empty" elements that contain attributes but no "content" other than the content that is contained in attribute values.
- Table structure in OXML is represented row by row; elements that represent columns are nested within the rows, and individual cells are nested within the column elements:

```
<pivotTable...>
<dimension axis='row'...>
  <dimension axis='column'...>
    <category...>
      <cell text='...' number='...' decimals='...' />
    </category>
    <category...>
      <cell text='...' number='...' decimals='...' />
    </category>
  </dimension>
</dimension>
...
</pivotTable>
```

The preceding example is a simplified representation of the structure that shows the descendant/ancestor relationships of these elements. However, the example does not necessarily show the parent/child relationships, because there are typically intervening nested element levels.

The following example shows a simple frequency table and the complete output XML representation of that table.

Table 24. Simple frequency table

Gender		Frequency	Percent	Valid Percent	Cumulative Percent
Valid	Female	216	45.6	45.6	45.6
	Male	258	54.4	54.4	100.0
	Total	474	100.0	100.0	

```

<?xml version="1.0" encoding="UTF-8" ?>
<outputTree outputTree xmlns="http://xml.spss.com/spss/oms"
  xmlns:xsi="http://www.w3.org/2001/XMLSchema-instance"
  xsi:schemaLocation="http://xml.spss.com/spss/oms
  http://xml.spss.com/spss/oms/spss-output-1.0.xsd">
<command text="Frequencies" command="Frequencies">
  <displayTableValues="label" displayOutlineValues="label" />
  <pivotTable text="Gender" label="Gender" subType="Frequencies">
    <varName="gender" variable="true">
      <dimension axis="row" text="Gender" label="Gender" />
      <varName="gender" variable="true">
        <group text="Valid">
          <group hide="true" text="Dummy">
            <category text="Female" label="Female" string="f" />
            <varName="gender" />
            <dimension axis="column" text="Statistics" />
            <category text="Frequency">
              <cell text="216" number="216" />
            </category>
            <category text="Percent">
              <cell text="45.6" number="45.569620253165" decimals="1" />
            </category>
            <category text="Valid Percent">
              <cell text="45.6" number="45.569620253165" decimals="1" />
            </category>
            <category text="Cumulative Percent">
              <cell text="45.6" number="45.569620253165" decimals="1" />
            </category>
          </dimension>
        </group>
        <category text="Male" label="Male" string="m" varName="gender">
          <dimension axis="column" text="Statistics" />
          <category text="Frequency">
            <cell text="258" number="258" />
          </category>
          <category text="Percent">
            <cell text="54.4" number="54.430379746835" decimals="1" />
          </category>
          <category text="Valid Percent">
            <cell text="54.4" number="54.430379746835" decimals="1" />
          </category>
          <category text="Cumulative Percent">
            <cell text="100.0" number="100" decimals="1" />
          </category>
        </dimension>
      </group>
    </pivotTable>
  </command>
</outputTree>

```

Figure 2. Output XML for the simple frequency table

As you may notice, a simple, small table produces a substantial amount of XML. That's partly because the XML contains some information that is not readily apparent in the original table, some information that might not even be available in the original table, and a certain amount of redundancy.

- The table contents as they are (or would be) displayed in a pivot table in the Viewer are contained in text attributes. An example is as follows:

```
<command text="Frequencies" command="Frequencies"....>
```

- Text attributes can be affected by both output language and settings that affect the display of variable names/labels and values/value labels. In this example, the text attribute value will differ, depending on the output language, whereas the command attribute value remains the same, regardless of output language.

- Wherever variables or values of variables are used in row or column labels, the XML will contain a text attribute and one or more additional attribute values. An example is as follows:

```
<dimension axis="row" text="Gender" label="Gender" varName="gender">
...<category text="Female" label="Female" string="f" varName="gender">
```

- For a numeric variable, there would be a number attribute instead of a string attribute. The label attribute is present only if the variable or values have defined labels.

- The <cell> elements that contain cell values for numbers will contain the text attribute and one or more additional attribute values. An example is as follows:

```
<cell text="45.6" number="45.569620253165" decimals="1"/>
```

The number attribute is the actual, unrounded numeric value, and the decimals attribute indicates the number of decimal positions that are displayed in the table.

- Because columns are nested within rows, the category element that identifies each column is repeated for each row. For example, because the statistics are displayed in the columns, the element <category text="Frequency"> appears three times in the XML: once for the male row, once for the female row, and once for the total row.

OMS identifiers

The OMS Identifiers dialog box is designed to assist you in writing OMS command syntax. You can use this dialog box to paste selected command and subtype identifiers into a command syntax window.

To use the oms identifiers dialog box

- From the menus choose:

Utilities > OMS Identifiers...

- Select one or more command or subtype identifiers. (Use Ctrl+click to select multiple identifiers in each list.)

- Click **Paste Commands** and/or **Paste Subtypes**.

- The list of available subtypes is based on the currently selected command(s). If multiple commands are selected, the list of available subtypes is the union of all subtypes that are available for any of the selected commands. If no commands are selected, all subtypes are listed.
- The identifiers are pasted into the designated command syntax window at the current cursor location. If there are no open command syntax windows, a new syntax window is automatically opened.
- Each command and/or subtype identifier is enclosed in quotation marks when pasted, because OMS command syntax requires these quotation marks.
- Identifier lists for the COMMANDS and SUBTYPES keywords must be enclosed in brackets, as in:

```
/IF COMMANDS=['Crosstabs' 'Descriptives']
SUBTYPES=['Crosstabulation' 'Descriptive Statistics']
```

Copying OMS identifiers from the viewer outline

You can copy and paste OMS command and subtype identifiers from the Viewer outline pane.

- In the outline pane, right-click the outline entry for the item.
- Choose **Copy OMS Command Identifier** or **Copy OMS Table Subtype**.

This method differs from the OMS Identifiers dialog box method in one respect: The copied identifier is not automatically pasted into a command syntax window. The identifier is simply copied to the clipboard, and you can then paste it anywhere you want. Because command and subtype identifier

values are identical to the corresponding command and subtype attribute values in Output XML format (OXML), you might find this copy/paste method useful if you write XSLT transformations.

Copying OMS labels

Instead of identifiers, you can copy labels for use with the LABELS keyword. Labels can be used to differentiate between multiple graphs or multiple tables of the same type in which the outline text reflects some attribute of the particular output object, such as the variable names or labels. There are, however, a number of factors that can affect the label text:

- If split-file processing is on, split-file group identification may be appended to the label.
- Labels that include information about variables or values are affected by the settings for the display of variable names/labels and values/value labels in the outline pane (Edit menu, Options, Output Labels tab).
- Labels are affected by the current output language setting (Edit menu, Options, General tab).

To copy OMS labels

1. In the outline pane, right-click the outline entry for the item.
2. Choose **Copy OMS Label**.

As with command and subtype identifiers, the labels must be in quotation marks, and the entire list must be enclosed in square brackets, as in:

```
/IF LABELS=['Employment Category' 'Education Level']
```

Chapter 23. Scripting Facility

The scripting facility allows you to automate tasks, including:

- Opening and saving data files.
- Exporting charts as graphic files in a variety of formats.
- Customizing output in the Viewer.

The available scripting languages depend on your platform. For Windows, the available scripting languages are Basic, which is installed with the Core system, and the Python programming language. For all other platforms, scripting is available with the Python programming language.

To use scripting with the Python programming language, you need IBM SPSS Statistics - Essentials for Python, which is installed by default with your IBM SPSS Statistics product.

Default Script Language

The default script language determines the script editor that is launched when new scripts are created. It also specifies the default language whose executable will be used to run autoscripts. On Windows, the default script language is Basic. You can change the default language from the Scripts tab in the Options dialog. See the topic “Script options” on page 205 for more information.

Sample Scripts

A number of scripts are included with the software, in the *Samples* subdirectory of the directory where IBM SPSS Statistics is installed. You can use these scripts as they are or you can customize them to your needs.

To Create a New Script

1. From the menus choose:

File > New > Script

The editor associated with the default script language opens.

To Run a Script

1. From the menus choose:
Utilities > Run Script...
2. Select the script you want.
3. Click **Run**.

Python scripts can be run in a number of ways, other than from Utilities>Run Script. See the topic “Scripting with the Python Programming Language” on page 277 for more information.

To Edit a Script

1. From the menus choose:
File > Open > Script...
2. Select the script you want.
3. Click **Open**.

The script is opened in the editor associated with the language in which the script is written.

Autoscripts

Autoscripts are scripts that run automatically when triggered by the creation of specific pieces of output from selected procedures. For example, you can use an autoscript to automatically remove the upper diagonal and highlight correlation coefficients below a certain significance whenever a Correlations table is produced by the Bivariate Correlations procedure.

Autoscripts can be specific to a given procedure and output type or apply to specific output types from different procedures. For example, you might have an autoscript that formats the ANOVA tables produced by One-Way ANOVA as well as ANOVA tables produced by other statistical procedures. On the other hand, Frequencies produces both a frequency table and a table of statistics, and you might choose to have a different autoscript for each.

Each output type for a given procedure can only be associated with a single autoscript. You can, however, create a base autoscript that is applied to all new Viewer items prior to the application of any autoscripts for specific output types. See the topic “Script options” on page 205 for more information.

The Scripts tab in the Options dialog box (accessed from the Edit menu) displays the autoscripts that have been configured on your system and allows you to set up new autoscripts or modify the settings for existing ones. Optionally, you can create and configure autoscripts for output items directly from the Viewer.

Events that Trigger Autoscripts

The following events can trigger autoscripts:

- Creation of a pivot table
- Creation of a Notes object
- Creation of a Warnings object

You can also use a script to trigger an autoscript indirectly. For example, you could write a script that invokes the Correlations procedure, which in turn triggers an autoscript registered to the resulting Correlations table.

Creating Autoscripts

You can create an autoscript by starting with the output object that you want to serve as the trigger--for instance, a frequency table.

1. In the Viewer, select the object that will trigger the autoscript.

2. From the menus choose:

Utilities > Create/Edit AutoScript...

If the selected object does not have an associated autoscript, an Open dialog prompts you for the location and name of a new script.

3. Browse to the location where the new script will be stored, enter a file name and click **Open**. The editor for the default script language opens. You can change the default script language from the Scripts tab on the Options dialog. See the topic “Script options” on page 205 for more information.

4. Type the code.

For help with converting custom Sax Basic autoscripts used in pre-16.0 versions, see “Compatibility with Versions Prior to 16.0” on page 279.

Note: By default, the executable associated with the default script language will be used to run the autoscript. You can change the executable from the Scripts tab in the Options dialog.

If the selected object is already associated with an autoscript, the script is opened in the script editor associated with the language in which the script is written.

Associating Existing Scripts with Viewer Objects

You can use existing scripts as autoscripts by associating them with a selected object in the Viewer--for instance, a frequency table.

1. In the Viewer, select an object to associate with an autoscript (multiple Viewer objects can trigger the same autoscript, but each object can only be associated with a single autoscript).
2. From the menus choose:
Utilities > Associate AutoScript...
If the selected object does not have an associated autoscript, the Select Autoscript dialog opens.
3. Browse for the script you want and select it.
4. Click **Apply**.

If the selected object is already associated with an autoscript, you are prompted to verify that you want to change the association. Clicking **OK** opens the Select Autoscript dialog.

Optionally, you can configure an existing script as an autoscript from the Scripts tab in the Options dialog box. The autoscript can be applied to a selected set of output types or specified as the base autoscript that is applied to all new Viewer items. See the topic "Script options" on page 205 for more information.

Scripting with the Python Programming Language

IBM SPSS Statistics provides two separate interfaces for programming with the Python language on Windows, Linux, Mac OS, and for IBM SPSS Statistics Server. Use of these interfaces requires the IBM SPSS Statistics - Integration Plug-in for Python, which is installed by default with your IBM SPSS Statistics product. For help getting started with the Python programming language, see the Python tutorial, available at <http://docs.python.org/tut/tut.html>.

Python Scripts

Python scripts make use of the interface exposed by the Python SpssClient module. They operate on user interface and output objects and can also run command syntax. For instance, you would use a Python script to customize a pivot table.

- Python scripts are run from Utilities>Run Script, from the Python editor launched from IBM SPSS Statistics (accessed from File>Open>Script), or from an external Python process, such as a Python IDE or the Python interpreter.
- Python scripts can be run as autoscripts.
- Python scripts run on the machine where the IBM SPSS Statistics client is running.

Complete documentation of the IBM SPSS Statistics classes and methods available for Python scripts can be found in the Scripting Guide for IBM SPSS Statistics, available under Integration Plug-in for Python in the Help system.

Python Programs

Python programs make use of the interface exposed by the Python spss module. They operate on the IBM SPSS Statistics processor and are used to control the flow of a command syntax job, read from and write to the active dataset, create new datasets, and create custom procedures that generate their own pivot table output.

- Python programs are run from command syntax within BEGIN PROGRAM-END PROGRAM blocks, or from an external Python process, such as a Python IDE or the Python interpreter.

- Python programs cannot be run as autoscripts.
- In distributed analysis mode (available with IBM SPSS Statistics Server), Python programs execute on the computer where IBM SPSS Statistics Server is running.

More information about Python programs, including complete documentation of the IBM SPSS Statistics functions and classes available for them, can be found in the documentation for the Python Integration Package for IBM SPSS Statistics, available under Integration Plug-in for Python in the Help system.

Running Python Scripts and Python programs

Both Python scripts and Python programs can be run from within IBM SPSS Statistics or from an external Python process, such as a Python IDE or the Python interpreter.

Python Scripts

Python Script Run from IBM SPSS Statistics. You can run a Python script from Utilities>Run Script or from the Python script editor which is launched when opening a Python file (.py) from File>Open>Script. Scripts run from the Python editor that is launched from IBM SPSS Statistics operate on the IBM SPSS Statistics client that launched the editor. This allows you to debug your Python code from a Python editor.

Python Script Run from an External Python Process. You can run a Python script from any external Python process, such as a Python IDE that is not launched from IBM SPSS Statistics, or the Python interpreter. The script will attempt to connect to an existing IBM SPSS Statistics client. If more than one client is found, a connection is made to the most recently launched one. If an existing client is not found, the Python script starts up a new instance of the IBM SPSS Statistics client. By default, the Data Editor and Viewer are invisible for the new client. You can choose to make them visible or work in invisible mode with datasets and output documents.

Python Programs

Python Program Run from Command Syntax. You can run a Python program by embedding Python code within a BEGIN PROGRAM-END PROGRAM block in command syntax. The command syntax can be run from the IBM SPSS Statistics client or from the IBM SPSS Statistics Batch Facility--a separate executable provided with IBM SPSS Statistics Server.

Python Program Run from an External Python Process. You can run a Python program from any external Python process, such as a Python IDE or the Python interpreter. In this mode, the Python program starts up a new instance of the IBM SPSS Statistics processor without an associated instance of the IBM SPSS Statistics client. You can use this mode to debug your Python programs using the Python IDE of your choice.

Invoking Python Scripts from Python Programs and Vice Versa

Python Script Run from Python Program. You can run a Python script from a Python program by importing the Python module containing the script and calling the function in the module that implements the script. You can also call Python script methods directly from within a Python program. These features are not available when running a Python program from an external Python process or when running a Python program from the IBM SPSS Statistics Batch Facility (available with IBM SPSS Statistics Server).

Python Autoscript Triggered from Python Program. A Python script specified as an autoscript will be triggered when a Python program executes the procedure containing the output item associated with the autoscript. For example, you associate an autoscript with the Descriptive Statistics table generated by the Descriptives procedure. You then run a Python program that executes the Descriptives procedure. The Python autoscript will be executed.

Python Program Run from Python Script. Python scripts can run command syntax, which means they can run command syntax containing Python programs.

Limitations and Warnings

- Running a Python program from the Python editor launched by IBM SPSS Statistics will start up a new instance of the IBM SPSS Statistics processor and will not interact with the instance of IBM SPSS Statistics that launched the editor.
- Python programs are not intended to be run from Utilities>Run Script.
- Python programs cannot be run as autoscripts.
- The interfaces exposed by the `spss` module cannot be used in a Python script.

Script Editor for the Python Programming Language

For the Python programming language, the default editor is IDLE, which is provided with Python. IDLE provides an integrated development environment (IDE) with a limited set of features. Many IDE's are available for the Python programming language. For instance, on Windows you might choose to use the freely available PythonWin IDE.

To change the script editor for the Python programming language:

1. Open the file `clientscriptingcfg.ini`, located in the directory where IBM SPSS Statistics is installed. Note: `clientscriptingcfg.ini` must be edited with a UTF-16 aware editor, such as SciTE on Windows or the TextEdit application on Mac.
2. Under the section that is labeled [Python2] or [Python3], change the value of EDITOR_PATH to point to the file that starts the editor. The specifications for Python 2 and Python 3 are independent.
3. In that same section, change the value of EDITOR_ARGS to handle any arguments that need to be passed to the editor. If no arguments are required, remove any existing values.

Scripting in Basic

Scripting in Basic is available on Windows only and is installed with the Core system. Extensive online help for scripting in Basic is available from the IBM SPSS Statistics Basic Script Editor. The editor can be accessed from File>New>Script when the default script language (set from the Scripts tab on the Options dialog) is set to Basic (the system default on Windows). It is also accessed from File>Open>Script, by choosing Basic (wwd;sbs) in the Files of type list.

Note: For Windows 7 and higher, accessing the online help for Basic scripting requires the Windows Help program (WinHlp32.exe), which may not be present on your system. If you cannot view the online help, then contact Microsoft for instructions on how to obtain the Windows Help program (WinHlp32.exe).

Compatibility with Versions Prior to 16.0

Obsolete Methods and Properties

A number of automation methods and properties are obsolete for version 16.0 and above. In terms of general features, this includes all objects associated with interactive graphs, the Draft Document object, and methods and properties associated with maps. For additional information, see "Release Notes for Version 16.0" in the help system provided with the IBM SPSS Statistics Basic Script Editor. The IBM SPSS Statistics-specific help is accessed from Help>IBM SPSS Statistics Objects Help, in the script editor.

Global Procedures

Prior to version 16.0, the scripting facility included a global procedures file. For version 16.0 and above the scripting facility does not use a global procedures file, although the pre-16.0 version of `Global.sbs` (renamed `Global.wwd`) is installed for backwards compatibility.

To migrate a pre-16.0 version of a script that called functions in the global procedures file, add the statement '#Uses "<install dir>\Samples\Global.wwd" to the declarations section of the script, where <install dir> is the directory where IBM SPSS Statistics is installed. '#Uses is a special comment recognized by the Basic script processor. If you're not sure if a script uses the global procedures file, you should add the '#Uses statement. You can also use '\$Include: instead of '#Uses.

Legacy Autoscripts

Prior to version 16.0, the scripting facility included a single autoscript file containing all autoscripts. For version 16.0 and above there is no single autoscript file. Each autoscript is now stored in a separate file and can be applied to one or more output items, in contrast to pre-16.0 versions where each autoscript was specific to a particular output item.

Some of the autoscripts installed with pre-16.0 versions are available as a set of separate script files located in the *Samples* subdirectory of the directory where IBM SPSS Statistics is installed. They are identified by a filename ending in *Autoscript*, with a file type of *wwd*. By default, they are not associated with any output items. The association is done from the Scripts tab of the Options dialog. See the topic "Script options" on page 205 for more information.

Any custom autoscripts used in pre-16.0 versions must be manually converted and associated with one or more output items, from the Scripts tab of the Options dialog. The conversion process involves the following steps:

1. Extract the subroutine specifying the autoscript from the legacy *Autoscript.sbs* file and save it as a new file with an extension of *wwd* or *sbs*. The name of the file is arbitrary.
2. Change the name of the subroutine to *Main* and remove the parameter specification, keeping track of which parameters are required by the script, such as a pivot table that triggers the autoscript.
3. Use the *scriptContext* object (always available) to get the values required by the autoscript, such as the output item that triggered the autoscript.
4. From the Scripts tab of the Options dialog, associate the script file with the output object.

To illustrate the converted code, consider the autoscript *Descriptives_Table_DescriptiveStatistics_Create* from the legacy *Autoscript.sbs* file.

```
Sub Descriptives_Table_DescriptiveStatistics_Create
    (objPivotTable As Object,objOutputDoc As Object,lngIndex As Long)
    'Autoscript
    'Trigger Event: DescriptiveStatistics Table Creation after running
    '    Descriptives procedure.

    'Purpose: Swaps the Rows and Columns in the currently active pivot table.
    'Assumptions: Selected Pivot Table is already activated.
    'Effects: Swaps the Rows and Columns in the output
    'Inputs: Pivot Table, OutputDoc, Item Index

    Dim objPivotManager As ISpssPivotMgr
    Set objPivotManager=objPivotTable.PivotManager
    objPivotManager.TransposeRowsWithColumns

End Sub
```

Following is the converted script:

```
Sub Main

    'Purpose: Swaps the Rows and Columns in the currently active pivot table.
    'Effects: Swaps the Rows and Columns in the output

    Dim objOutputItem As ISpssItem
    Dim objPivotTable as PivotTable
    Set objOutputItem = scriptContext.GetOutputItem()
    Set objPivotTable = objOutputItem.ActivateTable

    Dim objPivotManager As ISpssPivotMgr
```

```

Set objPivotManager = objPivotTable.PivotManager
objPivotManager.TransposeRowsWithColumns
objOutputItem.Deactivate
End Sub

```

- Notice that nothing in the converted script indicates which object the script is to be applied to. The association between an output item and an autoscript is set from the Scripts tab of the Options dialog and maintained across sessions.
- `scriptContext.GetOutputItem` gets the output item (an `ISpssItem` object) that triggered the autoscript.
- The object returned by `scriptContext.GetOutputItem` is not activated. If your script requires an activated object, you'll need to activate it, as done in this example with the `ActivateTable` method. When you're finished with any table manipulations, call the `Deactivate` method.

For version 16.0, there is no distinction between scripts that are run as autoscripts and scripts that aren't run as autoscripts. Any script, appropriately coded, can be used in either context. See the topic "The `scriptContext` Object" for more information.

Note: To trigger a script from the application creation event, see "Startup Scripts" on page 282.

Script Editor

For version 16.0 and above the script editor for Basic no longer supports the following pre-16.0 features:

- The Script, Analyze, Graph, Utilities, and Add-Ons menus.
- The ability to paste command syntax into a script window.

The IBM SPSS Statistics Basic Script Editor is a standalone application that is launched from within IBM SPSS Statistics via `File>New>Script`, `File>Open>Script`, or `Utilities>Create/Edit AutoScript` (from a Viewer window). It allows you to run scripts against the instance of IBM SPSS Statistics from which it was launched. Once opened, the editor will remain open after exiting IBM SPSS Statistics, but scripts that use IBM SPSS Statistics objects will no longer run.

File Types

For version 16.0 and above, the scripting facility will continue to support running and editing scripts with a file type of `sbs`. By default, new Basic scripts created with the IBM SPSS Statistics Basic Script Editor have a file type of `wwd`.

Using External COM Clients

For version 16.0 and above, the program identifier for instantiating IBM SPSS Statistics from an external COM client is `SPSS.Application16`. Application objects should be declared as `spsswinLib.Application16`. For example:

```

Dim objSpssApp As spsswinLib.Application16
Set objSpssApp=CreateObject("SPSS.Application16")

```

To connect to a running instance of the IBM SPSS Statistics client from an external COM client, use:

```

Dim objSpssApp As spsswinLib.Application16
Set objSpssApp=GetObject("", "SPSS.Application16")

```

If more than one client is running, `GetObject` will connect to the most recently launched one.

Note: For post-16.0 versions, the identifier is still `Application16`.

The `scriptContext` Object

Detecting When a Script is Run as an Autoscript

Using the `scriptContext` object, you can detect when a script is being run as an autoscript. This allows you to code a script so that it functions in either context (autoscript or not). This trivial script illustrates the approach.

```
Sub Main
    If scriptContext Is Nothing Then
        MsgBox "I'm not an autoscript"
    Else
        MsgBox "I'm an autoscript"
    End If
End Sub
```

- When a script is not run as an autoscript, the `scriptContext` object will have a value of `Nothing`.
- Given the `If-Else` logic in this example, you would include your autoscript-specific code in the `Else` clause. Any code that is not to be run in the context of an autoscript would be included in the `If` clause. Of course you can also include code that is to be run in either context.

Getting Values Required by Autoscripts

The `scriptContext` object provides access to values required by an autoscript, such as the output item that triggered the current autoscript.

- The `scriptContext.GetOutputItem` method returns the output item (an `ISpssItem` object) that triggered the current autoscript.
- The `scriptContext.GetOutputDoc` method returns the output document (an `ISpssOutputDoc` object) associated with the current autoscript.
- The `scriptContext.GetOutputItemIndex` method returns the index, in the associated output document, of the output item that triggered the current autoscript.

Note: The object returned by `scriptContext.GetOutputItem` is not activated. If your script requires an activated object, you'll need to activate it--for instance, with the `ActivateTable` method. When you're finished with any manipulations, call the `Deactivate` method.

Startup Scripts

You can create a script that runs at the start of each session and a separate script that runs each time you switch servers. For Windows you can have versions of these scripts in both Python and Basic. For all other platforms the scripts can only be in Python.

- The startup script must be named `StartClient_.py` for Python or `StartClient_.wwd` for Basic.
- The script that runs when switching servers must be named `StartServer_.py` for Python or `StartServer_.wwd` for Basic.
- The scripts must be located in the `scripts` directory of the installation directory--located at the root of the installation directory for Windows and Linux, and under the `Contents` directory in the application bundle for Mac. Note that regardless of whether you are working in distributed mode, all scripts (including the `StartServer_` scripts) must reside on the client machine.
- On Windows, if the `scripts` directory contains both a Python and a Basic version of `StartClient_` or `StartServer_` then both versions are executed. The order of execution is the Python version followed by the Basic version.
- If your system is configured to start up in distributed mode, then at the start of each session any `StartClient_` scripts are run followed by any `StartServer_` scripts. *Note:* The `StartServer_` scripts also run each time you switch servers, but the `StartClient_` scripts only run at the start of a session.

Example

This is an example of a `StartServer_` script that maps a drive letter to a shared network resource specified by a UNC identifier. This allows users working in distributed mode to access data files on the network resource from the Open Remote File dialog box.

```
#StartServer_.py
import SpssClient
SpssClient.StartClient()
SpssClient.RunSyntax(r""" HOST COMMAND=['net use y: \\myserver\data']. """)
SpssClient.StopClient()
```

The `SpssClient.RunSyntax` method is used to run a `HOST` command that calls the Windows `net use` command to perform the mapping. When the `StartServer_` script runs, IBM SPSS Statistics is in distributed mode so the `HOST` command runs on the IBM SPSS Statistics Server machine.

Chapter 24. TABLES and IGRAPH Command Syntax Converter

If you have command syntax files that contain TABLES syntax that you want to convert to CTABLES syntax and/or IGRAPH syntax that you want to convert to GGRAPH syntax, a simple utility program is provided to help you get started with the conversion process. There are, however, significant functionality differences between TABLES and CTABLES and between IGRAPH and GGRAPH. It is likely that you will find that the utility program cannot convert some of your TABLES and IGRAPH syntax jobs or may generate CTABLES and GGRAPH syntax that produces tables and graphs that do not closely resemble the originals produced by the TABLES and IGRAPH commands. For most tables, you can edit the converted syntax to produce a table closely resembling the original.

The utility program is designed to:

- Create a new syntax file from an existing syntax file. The original syntax file is not altered.
- Convert only TABLES and IGRAPH commands in the syntax file. Other commands in the file are not altered.
- Retain the original TABLES and IGRAPH syntax in commented form.
- Identify the beginning and end of each conversion block with comments.
- Identify TABLES and IGRAPH syntax commands that could not be converted.
- Convert command syntax files that follow either interactive or production mode syntax rules.

This utility cannot convert commands that contain errors. The following other limitations also apply.

TABLES Limitations

The utility program may convert TABLES commands incorrectly under some circumstances, including TABLES commands that contain:

- Parenthesized variable names with the initial letters "sta" or "lab" in the TABLES subcommand if the variable is parenthesized by itself—for example, var1 by (statvar) by (labvar). These will be interpreted as the (STATISTICS) and (LABELS) keywords.
- SORT subcommands that use the abbreviations A or D to indicate ascending or descending sort order. These will be interpreted as variable names.

The utility program cannot convert TABLES commands that contain:

- Syntax errors.
- OBSERVATION subcommands that refer to a range of variables using the TO keyword (for example, var01 TO var05).
- String literals broken into segments separated by plus signs (for example, TITLE "My" + "Title").
- Macro calls that, in the absence of macro expansion, would be invalid TABLES syntax. Since the converter does not expand the macro calls, it treats them as if they were simply part of the standard TABLES syntax.

The utility program will not convert TABLES commands contained in macros. All macros are unaffected by the conversion process.

IGRAPH Limitations

IGRAPH changed significantly in release 16. Because of these changes, some subcommands and keywords in IGRAPH syntax created before that release may not be honored. See the IGRAPH section in the *Command Syntax Reference* for the complete revision history.

The conversion utility program may generate additional syntax that it stores in the `INLINETEMPLATE` keyword within the `GGGRAPH` syntax. This keyword is created only by the conversion program. Its syntax is not intended to be user-editable.

Using the Conversion Utility Program

The conversion utility program, `SyntaxConverter.exe`, can be found in the installation directory. It is designed to run from a command prompt. The general form of the command is:

```
syntaxconverter.exe [path]/inputfilename.sps [path]/outputfilename.sps
```

You must run this command from the installation directory.

If any directory names contain spaces, enclose the entire path and filename in quotation marks, as in:

```
syntaxconverter.exe /myfiles/oldfile.sps "/new files/newfile.sps"
```

Interactive versus Production Mode Command Syntax Rules

The conversion utility program can convert command files that use interactive or production mode syntax rules.

Interactive. The interactive syntax rules are:

- Each command begins on a new line.
- Each command ends with a period (.).

Production mode. The Production Facility and commands in files accessed via the `INCLUDE` command in a different command file use production mode syntax rules:

- Each command must begin in the first column of a new line.
- Continuation lines must be indented at least one space.
- The period at the end of the command is optional.

If your command files use production mode syntax rules and don't contain periods at the end of each command, you need to include the command line switch `-b` (or `/b`) when you run `SyntaxConverter.exe`, as in:

```
syntaxconverter.exe -b /myfiles/oldfile.sps /myfiles/newfile.sps
```

SyntaxConverter Script (Windows Only)

On Windows, you can also run the syntax converter with the script `SyntaxConverter.wvd`, located in the `Samples` directory of the installation directory.

1. From the menus choose:
Utilities > Run Script...
2. Navigate to the `Samples` directory and select `SyntaxConverter.wvd`.

This will open a simple dialog box where you can specify the names and locations of the old and new command syntax files.

Chapter 25. Encrypting data files, output documents, and syntax files

You can protect confidential information stored in a data file, an output document, or a syntax file by encrypting the file with a password. Once encrypted, the file can only be opened by providing the password. The option to encrypt a file is provided on the Save As dialogs for data files, output documents, and syntax files. You can also encrypt a data file when sorting it and saving the sorted file.

- Passwords cannot be recovered if they are lost. If the password is lost the file cannot be opened.
- Passwords are limited to 10 characters and are case-sensitive.

Creating strong passwords

- Use eight or more characters.
- Include numbers, symbols and even punctuation in your password.
- Avoid sequences of numbers or characters, such as "123" and "abc", and avoid repetition, such as "111aaa".
- Do not create passwords that use personal information such as birthdays or nicknames.
- Periodically change the password.

Modifying encrypted files

- If you open an encrypted file, make modifications to it and choose File > Save, the modified file will be saved with the same password.
- You can change the password on an encrypted file by opening the file, repeating the steps for encrypting it, and specifying a different password in the Encrypt File dialog box.
- You can save an unencrypted version of an encrypted data file or output document by opening the file, choosing File > Save As and deselecting **Encrypt file with password** in the associated Save As dialog box. For an encrypted syntax file, select **Syntax** from the Save as type drop-down list to save an unencrypted version of the file.

Note: Encrypted data files and output documents cannot be opened in versions of IBM SPSS Statistics prior to version 21. Encrypted syntax files cannot be opened in versions prior to version 22.

Notices

This information was developed for products and services offered in the US. This material might be available from IBM in other languages. However, you may be required to own a copy of the product or product version in that language in order to access it.

IBM may not offer the products, services, or features discussed in this document in other countries. Consult your local IBM representative for information on the products and services currently available in your area. Any reference to an IBM product, program, or service is not intended to state or imply that only that IBM product, program, or service may be used. Any functionally equivalent product, program, or service that does not infringe any IBM intellectual property right may be used instead. However, it is the user's responsibility to evaluate and verify the operation of any non-IBM product, program, or service.

IBM may have patents or pending patent applications covering subject matter described in this document. The furnishing of this document does not grant you any license to these patents. You can send license inquiries, in writing, to:

*IBM Director of Licensing
IBM Corporation
North Castle Drive, MD-NC119
Armonk, NY 10504-1785
US*

For license inquiries regarding double-byte (DBCS) information, contact the IBM Intellectual Property Department in your country or send inquiries, in writing, to:

*Intellectual Property Licensing
Legal and Intellectual Property Law
IBM Japan Ltd.
19-21, Nihonbashi-Hakozakicho, Chuo-ku
Tokyo 103-8510, Japan*

INTERNATIONAL BUSINESS MACHINES CORPORATION PROVIDES THIS PUBLICATION "AS IS" WITHOUT WARRANTY OF ANY KIND, EITHER EXPRESS OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF NON-INFRINGEMENT, MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE. Some jurisdictions do not allow disclaimer of express or implied warranties in certain transactions, therefore, this statement may not apply to you.

This information could include technical inaccuracies or typographical errors. Changes are periodically made to the information herein; these changes will be incorporated in new editions of the publication. IBM may make improvements and/or changes in the product(s) and/or the program(s) described in this publication at any time without notice.

Any references in this information to non-IBM websites are provided for convenience only and do not in any manner serve as an endorsement of those websites. The materials at those websites are not part of the materials for this IBM product and use of those websites is at your own risk.

IBM may use or distribute any of the information you provide in any way it believes appropriate without incurring any obligation to you.

Licensees of this program who wish to have information about it for the purpose of enabling: (i) the exchange of information between independently created programs and other programs (including this one) and (ii) the mutual use of the information which has been exchanged, should contact:

*IBM Director of Licensing
IBM Corporation
North Castle Drive, MD-NC119
Armonk, NY 10504-1785
US*

Such information may be available, subject to appropriate terms and conditions, including in some cases, payment of a fee.

The licensed program described in this document and all licensed material available for it are provided by IBM under terms of the IBM Customer Agreement, IBM International Program License Agreement or any equivalent agreement between us.

The performance data and client examples cited are presented for illustrative purposes only. Actual performance results may vary depending on specific configurations and operating conditions.

Information concerning non-IBM products was obtained from the suppliers of those products, their published announcements or other publicly available sources. IBM has not tested those products and cannot confirm the accuracy of performance, compatibility or any other claims related to non-IBM products. Questions on the capabilities of non-IBM products should be addressed to the suppliers of those products.

Statements regarding IBM's future direction or intent are subject to change or withdrawal without notice, and represent goals and objectives only.

This information contains examples of data and reports used in daily business operations. To illustrate them as completely as possible, the examples include the names of individuals, companies, brands, and products. All of these names are fictitious and any similarity to actual people or business enterprises is entirely coincidental.

COPYRIGHT LICENSE:

This information contains sample application programs in source language, which illustrate programming techniques on various operating platforms. You may copy, modify, and distribute these sample programs in any form without payment to IBM, for the purposes of developing, using, marketing or distributing application programs conforming to the application programming interface for the operating platform for which the sample programs are written. These examples have not been thoroughly tested under all conditions. IBM, therefore, cannot guarantee or imply reliability, serviceability, or function of these programs. The sample programs are provided "AS IS", without warranty of any kind. IBM shall not be liable for any damages arising out of your use of the sample programs.

Each copy or any portion of these sample programs or any derivative work, must include a copyright notice as follows:

© your company name) (year). Portions of this code are derived from IBM Corp. Sample Programs.

© Copyright IBM Corp. _enter the year or years_. All rights reserved.

Trademarks

IBM, the IBM logo, and ibm.com are trademarks or registered trademarks of International Business Machines Corp., registered in many jurisdictions worldwide. Other product and service names might be trademarks of IBM or other companies. A current list of IBM trademarks is available on the web at "Copyright and trademark information" at www.ibm.com/legal/copytrade.shtml.

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

Intel, Intel logo, Intel Inside, Intel Inside logo, Intel Centrino, Intel Centrino logo, Celeron, Intel Xeon, Intel SpeedStep, Itanium, and Pentium are trademarks or registered trademarks of Intel Corporation or its subsidiaries in the United States and other countries.

Linux is a registered trademark of Linus Torvalds in the United States, other countries, or both.

Microsoft, Windows, Windows NT, and the Windows logo are trademarks of Microsoft Corporation in the United States, other countries, or both.

UNIX is a registered trademark of The Open Group in the United States and other countries.

Java and all Java-based trademarks and logos are trademarks or registered trademarks of Oracle and/or its affiliates.

Index

A

Access (Microsoft) 15
active file 40, 41
 caching 41
 creating a temporary active file 41
 virtual active file 40
active window 1
adding group labels 138
aggregating data 107
 aggregate functions 108
 variable names and labels 108
algorithms 5
alignment 54, 122, 196
 in Data Editor 54
 output 122, 196
alternating row colors
 pivot tables 144
aspect ratio 201
attributes
 custom variable attributes 55
automated output modification 157, 158,
 160, 161, 162
automated production 255
automatic output modification 157
autoscripts 205, 276
 associating with viewer objects 277
Basic 281
 creating 276
 trigger events 276

B

background color 145
banding 77
bi-directional text 199
binning 77
Blom estimates 91
BMP files 126, 132
 exporting charts 126, 132
bookmarks
 syntax editor 171
borders 144, 148
 displaying hidden borders 148
break variables
 in Aggregate Data 107
breakpoints
 syntax editor 170

C

caching 41
 active file 41
captions 145, 146
cases 60, 112
 finding duplicates 76
 finding in Data Editor 61
 inserting new cases 60
 restructuring into variables 112
 selecting subsets 109, 110, 111
 sorting 101

cases (*continued*)
 weighting 111
categorical data 69
 converting interval data to discrete
 categories 77
cell properties 145
cells in pivot tables 141, 144, 147
 formats 144
 hiding 141
 selecting 148
 showing 141
 widths 147
centered moving average function 98
centering output 122, 196
character encoding 199
Chart Builder
 gallery 181
Chart Editor
 properties 182
chart options 201
charts 121, 126, 149, 201
 aspect ratio 201
 creating from pivot tables 149
 exporting 126
 hiding 121
 missing values 184
 overview 181
 size 184
 templates 184, 201
 wrapping panels 184
Cognos
 exporting to Cognos TM1 36
 reading Cognos Business Intelligence
 data 19
 reading Cognos TM1 data 21
Cognos active report 128
collapsing categories 77
color coding
 syntax editor 169
colors in pivot tables 144
column edit mode 174
column width 54, 142, 148, 203
 controlling default width 203
 controlling maximum width 142
 controlling width for wrapped
 text 142
 in Data Editor 54
 pivot tables 148
columns 148
 changing width in pivot tables 148
 selecting in pivot tables 148
COMMA format 51, 52
comma-delimited files 10
command identifiers 265
command language 165
command line switches 260
 Production jobs 260
command syntax 165, 176, 209, 210, 255
 accessing Command Syntax
 Reference 5

command syntax (*continued*)
 adding to menus 209
 journal file 177, 178
 output log 166
 pasting 166
 Production jobs rules 255
 running 176
 running with toolbar buttons 210
 syntax rules 165
command syntax editor 167
 auto-completion 169
 bookmarks 167, 171
 breakpoints 167, 170, 176
 color coding 169
 command spans 167
 commenting or uncommenting
 text 172
 duplicate lines 173
 formatting syntax 172, 173, 174, 175
 indenting syntax 172
 joining syntax 173
 line numbers 167
 moving syntax 174
 multiple views/panes 167
 options 205
 removing empty syntax 173
 shortcuts 175
 trimming leading and trailing
 spaces 175
command syntax files 176, 177
comparing datasets 38
compatible custom dialog 216
computing variables 83
 computing new string variables 84
conditional transformations 83
continuation text 144
 for pivot tables 144
controlling number of rows to
 display 142
copy special 124
copying and pasting output into other
 applications 124
counting occurrences 85
Crosstabs
 fractional weights 111
CSV format
 reading data 10
 saving data 24
CTABLES
 converting TABLES command syntax
 to CTABLES 285
cumulative sum function 98
currency formats 200
custom attributes 55
custom currency formats 51, 200
Custom Dialog Builder 216
 check box 226
 check box group 239
 color picker 235
 combo box 227
 combo box list items 228

Custom Dialog Builder (*continued*)
compatible custom dialog package
(spd) files 248
converting to enhanced dialogs 248
custom dialogs for extension
 commands 250
data source 219
dataset selector 226
date control 232
dialog properties 218
enabling rules 242
extension bundle files 246
field chooser 224
field source 226
file browser 240
file type filter 241
filtering variable lists 226
help file 218
installing compatible custom
 dialogs 248
installing extensions containing
 dialogs 246
item group control 238
layout rules 220
list box 229
list box list items 228
localizing dialogs and help files 250
menu location 219
modifying compatible custom
 dialogs 248
modifying dialogs in installed
 extensions 246
number control 231
opening compatible custom dialog
 package files 248
opening extensions containing
 dialogs 246
preview 222
radio group 238
radio group buttons 239
saving compatible custom
 dialogs 248
saving extensions containing
 dialogs 246
secured text 233
source list 223
static text control 235
sub-dialog button 242
sub-dialog properties 242
syntax template 220
tab 241
table control 235
table control columns 236
target list 223
text control 230
Custom Dialog Builder for
 Extensions 216
custom tables
 converting TABLES command syntax
 to CTABLES 285
custom variable attributes 55

D

data dictionary
 applying from another file 73

Data Editor 49, 50, 54, 58, 59, 60, 61, 63,
64, 209
 alignment 54
 changing data type 61
 column width 54
 data value restrictions 59
Data View 49
 defining variables 50
 descriptive statistics 62
 descriptive statistics options 200
 display options 63
 editing data 59, 60
 entering data 58
 entering non-numeric data 59
 entering numeric data 58
 filtered cases 63
 inserting new cases 60
 inserting new variables 60
 moving variables 61
 multiple open data files 65, 195
 multiple views/panes 63
 printing 64
 roles 54
 sending data to other
 applications 209
 Variable View 49
data entry 58
data files 7, 10, 23, 24, 29, 41, 45, 112
 adding comments 191
 dictionary information 23
 encrypting 29
 file information 23
 flipping 103
 improving performance for large
 files 41
 multiple open data files 65, 195
 opening 7
 protecting 40
 remote servers 45
 restructuring 112
 saving 23, 24
 saving output as IBM SPSS Statistics
 data files 263
 saving subsets of variables 29
 text 10
 transposing 103
DATA LIST 40
 versus GET DATA command 40
data transformations 197
 computing variables 83
 conditional transformations 83
 delaying execution 197
 functions 84
 ranking cases 90
 recoding values 86, 87, 88, 89
 string variables 84
 time series 96, 97
data types 51, 52, 61, 200
 changing 61
 custom currency 51, 200
 defining 51
 display formats 52
 input formats 52
Data View 49
databases 14, 15, 16, 17, 18, 19
 adding new fields to a table 34
databases (*continued*)
 appending records (cases) to a
 table 34
 conditional expressions 17
 converting strings to numeric
 variables 19
 creating a new table 34
 creating relationships 16
 defining variables 19
 Microsoft Access 15
 parameter queries 17, 18
 Prompt for Value 18
 random sampling 17
 reading 14, 15, 16
 replacing a table 34
 replacing values in existing fields 33
 saving 30
 saving queries 19
 selecting a data source 15
 selecting data fields 16
 specifying criteria 17
 SQL syntax 19
 table joins 16
 updating 30
 verifying results 19
 Where clause 17
datasets
 comparing 38
 renaming 66
date format variables 51, 52, 197
 add or subtract from date/time
 variables 92
 create date/time variable from set of
 variables 92
 create date/time variable from
 string 92
 extract part of date/time variable 92
date formats
 two-digit years 197
date variables
 defining for time series data 96
dBASE files 7, 9, 24
 reading 7, 9
 saving 24
default file locations 204
defining variables 50, 51, 53, 54, 55, 67
 applying a data dictionary 73
 copying and pasting attributes 54, 55
 data types 51
 missing values 53
 templates 54, 55
 value labels 53, 67
 variable labels 53
Delete lines 172
deleting multiple EXECUTES in syntax
 files 177, 178
deleting output 122
descriptive statistics
 Data Editor 62
designated window 1
dialog boxes 192, 195
 defining variable sets 192
 displaying variable labels 2, 195
 displaying variable names 2, 195
 reordering target lists 193
 using variable sets 192
 variable display order 195

dialog boxes (*continued*)
 variable icons 2
 dictionary 23
 difference function 98
 disk space 40, 41
 temporary 40, 41
 display formats 52
 display order 137
 distributed mode 43, 45, 46, 259
 available procedures 46
 data file access 45
 relative paths 46
 DOLLAR format 51, 52
 DOT format 51, 52
 duplicate cases (records)
 finding and filtering 76
 Duplicate lines 172, 173

E

editing data 59, 60
 enhanced custom dialog 216
 ensemble viewer 153
 automatic data preparation 155
 component model accuracy 155
 component model details 155
 model summary 154
 predictor frequency 154
 predictor importance 154
 entering data 58, 59
 non-numeric 59
 numeric 58
 using value labels 59
 environment variables 204
 SPSSTMPDIR 204
 EPS files 126, 132
 exporting charts 126, 132
 Excel files 7, 24, 209
 adding menu item to send data to
 Excel 209
 opening 7
 reading 8
 saving 24
 saving value labels instead of
 values 24
 Excel format
 exporting output 126, 129, 266
 excluding output from Viewer with
 OMS 269
 EXECUTE (command)
 pasted from dialog boxes 177, 178
 Exit column edit mode 172
 export data 24
 exporting
 models 153
 exporting charts 126, 132, 255
 automated production 255
 exporting data 209
 adding menu items to export
 data 209
 exporting output 126, 130, 131
 Excel format 126, 129, 266
 HTML 127
 HTML format 126
 OMS 263
 PDF format 126, 130, 266
 PowerPoint format 126

exporting output (*continued*)
 text format 266
 web report 128
 Word format 126, 129, 266
 extension bundles
 batch installation 216
 creating extension bundles 252
 installing extension bundles 214
 installing on Statistics Server 216
 extension commands
 custom dialogs 250
 extensions 211
 extension details 213
 finding and installing new
 extensions 212
 installing updates to extensions 212
 removing extensions 212
 viewing installed extensions 212

grouping variables (*continued*)
 creating 112

H

headers 133
 Help windows 5
 hiding 121, 141, 209
 captions 146
 dimension labels 141
 footnotes 146
 procedure results 121
 rows and columns 141
 titles 141
 toolbars 209
 hiding (excluding) output from the
 Viewer with OMS 269
 hiding variables
 Data Editor 192
 dialog lists 192
 HTML 126, 127
 exporting output 126, 127

I

IBM SPSS Data Collection data
 saving 36
 IBM SPSS Statistics data file format
 routing output to a data file 266, 269
 icons
 in dialog boxes 2
 IGRAPH
 converting IGRAPH to
 GGRAPH 285
 import data 7, 14
 imputations
 finding in Data Editor 61
 inner join 16
 input formats 52
 inserting group labels 138
 interactive output 126

J

Join lines 172, 173
 journal file 204
 JPEG files 126, 132
 exporting charts 126, 132
 justification 122, 196
 output 122, 196

K

keyed table 104

L

labels 138
 deleting 138
 inserting group labels 138
 vs. subtype names in OMS 266
 LAG (function) 98
 lag function 86
 language
 changing output language 139

language settings 199

layers 133, 140, 141, 142, 144

creating 140

displaying 140, 141

in pivot tables 140

printing 133, 142, 144

lead function 86, 98

legacy tables 149

level of measurement 51, 69

defining 51

line breaks

variable and value labels 53

local encoding 176, 177

logging in to a server 43

Lotus 1-2-3 files 7, 24, 209

adding menu item to send data to

Lotus 209

opening 7

saving 24

M

measurement level 51, 69

default measurement level 197

defining 51

icons in dialog boxes 2

unknown measurement level 70

measurement system 195

memory 195

menus 209

customizing 209

merging data files

dictionary information 104

files with different cases 103

files with different variables 104

renaming variables 104

metafiles 126

exporting charts 126

Microsoft Access 15

missing values 53, 184

charts 184

defining 53

in functions 84

replacing in time series data 99

scoring models 186

string variables 53

model viewer

split models 155

Model Viewer 151

models 151

activating 151

copying 152

exporting 153

interacting 151

merging model and transformation files 188

Model Viewer 151

models supported for export and scoring 185

printing 152

properties 152

scoring 185

Move lines down 172, 174

Move lines up 172, 174

moving rows and columns 137

multiple open data files 65, 195

suppressing 66

multiple response sets

defining 71

multiple categories 71

multiple dichotomies 71

multiple views/panes

Data Editor 63

syntax editor 167

ordinal (*continued*)

measurement level 51, 69

outer join 16

outline 122, 123

changing levels 123

collapsing 122

expanding 122

in Viewer 122

output 121, 122, 124, 126, 134, 196

alignment 122, 196

centering 122, 196

changing output language 139

closing output items 124

copying 121

deleting 121, 122

encrypting 134

exporting 126

hiding 121

interactive 126

modifying 157, 158, 160, 161, 162

moving 121

pasting into other applications 124

saving 134

showing 121

Viewer 121

Output Management System (OMS) 263, 273

output object types

in OMS 264

OXML 273

O

OMS 263, 273

command identifiers 265

controlling table pivots 266, 270

Excel format 266

excluding output from the

Viewer 269

IBM SPSS Statistics data file

format 266, 269

output object types 264

PDF format 266

SAV file format 266, 269

table subtypes 265

text format 266

using XSLT with OXML 273

variable names in SAV files 270

Word format 266

XML 266, 270

online Help 5

Statistics Coach 2

opening files 7, 9, 10, 14, 15

controlling default file locations 204

data files 7

dBASE files 7, 9

Excel files 7

Lotus 1-2-3 files 7

spreadsheet files 7

Stata files 9

SYSTAT files 7

tab-delimited files 7

text data files 10

options 195, 196, 197, 200, 201, 203, 204,

205

charts 201

currency 200

data 197

descriptive statistics in Data

Editor 200

general 195

language 199

output labels 200

pivot table look 203

scripts 205

syntax editor 205

temporary directory 204

two-digit years 197

Variable View 199

Viewer 196

ordinal 51

P

page numbering 134

page setup 133, 134

chart size 134

headers and footers 133

pane splitter

Data Editor 63

syntax editor 167

pasting output into other

applications 124

PDF

exporting output 126, 130

PDF format

exporting output 266

performance 41

caching data 41

pivot tables 121, 124, 126, 133, 137, 138,

140, 141, 142, 143, 144, 147, 148, 149, 203

alignment 145

alternating row colors 144

background color 145

borders 144

captions 145, 146

cell formats 144

cell properties 145

cell widths 147

changing display order 137

changing the look 141

continuation text 144

controlling number of rows to

display 143

controlling table breaks 148

creating charts from tables 149

default column width

adjustment 203

pivot tables (*continued*)
 default look for new tables 203
 deleting group labels 138
 displaying hidden borders 148
 editing 137
 exporting as HTML 126
 fast pivot tables 203
 fonts 145
 footnote properties 143
 footnotes 145, 146, 147
 general properties 142
 grid lines 148
 grouping rows or columns 138
 hiding 121
 inserting group labels 138
 inserting rows and columns 139
 language 139
 layers 140
 legacy tables 149
 manipulating 137
 margins 145
 moving rows and columns 137
 pasting as tables 124
 pasting into other applications 124
 pivoting 137
 printing large tables 148
 printing layers 133
 properties 142
 render tables faster 203
 rotating labels 138
 scaling to fit page 142, 144
 selecting rows and columns 148
 showing and hiding cells 141
 sorting rows 138
 transposing rows and columns 138
 undoing changes 140
 ungrouping rows or columns 138
 using icons 137
 value labels 139
 variable labels 139
pivoting
 controlling with OMS for exported output 270
 PNG files 126, 132
 exporting charts 126, 132
 port numbers 43, 259
portable files
 variable names 24
 post-processing output 157, 158, 160, 161, 162
 PostScript files (encapsulated) 126, 132
 exporting charts 126, 132
 PowerPoint 130
 exporting output as PowerPoint 130
PowerPoint format
 exporting output 126
 printing 64, 133, 134, 142, 144, 148
 chart size 134
 charts 133
 controlling table breaks 148
 data 64
 headers and footers 133
 layers 133, 142, 144
 models 152
 page numbers 134
 pivot tables 133
 print preview 133

printing (*continued*)
 scaling tables 142, 144
 space between output items 134
 text output 133
prior moving average function 98
Production Facility 195
 converting files to production jobs 261
 using command syntax from journal file 195
Production jobs 255, 260
 command line switches 260
 converting Production Facility files 261
 exporting charts 255
 output files 255
 running multiple production jobs 260
 scheduling production jobs 260
 syntax rules 255
programming with command language 165
properties 142
 pivot tables 142
 tables 142
proportion estimates in Rank Cases 91
Python
 scripts 277

restructuring data (*continued*)
 creating index variables for variables to cases 116
 creating multiple index variables for variables to cases 117
 example of cases to variables 114
 example of one index for variables to cases 116
 example of two indices for variables to cases 116
 example of variables to cases 113
 options for cases to variables 119
 options for variables to cases 118
 overview 111
 selecting data for cases to variables 118
 selecting data for variables to cases 115
 sorting data for cases to variables 119
 types of restructuring 112
 variable groups for variables to cases 114
roles
 Data Editor 54
 rotating labels 138
 rows 148
 selecting in pivot tables 148
 running median function 98

R

random number seed 85
 random sample 17
 databases 17
 random number seed 85
 selecting 110
ranking cases 90
 fractional ranks 91
 percentiles 91
 Savage scores 91
 tied values 91
Rankit estimates 91
 recoding values 77, 86, 87, 88, 89
 remote servers 43, 45, 46, 259
 adding 43, 259
 available procedures 46
 data file access 45
 editing 43, 259
 logging in 43
 relative paths 46
Remove empty lines 172, 173
removing group labels 138
renaming datasets 66
reordering rows and columns 137
replacing missing values
 linear interpolation 99
 linear trend 99
 mean of nearby points 99
 median of nearby points 99
 series mean 99
restricted numeric format 51
restricting data 111, 112, 113, 114, 115, 116, 117, 118, 119, 120
 and weighted data 120
 creating a single index variable for variables to cases 117

S

sampling
 random sample 110
SAS files
 opening 7
 reading 7
 saving 24
SAV file format
 routing output to IBM SPSS Statistics data file 266
 routing output to IBM SPSS Statistics data files 269
Savage scores 91
saving charts 126, 132
 BMP files 126, 132
 EMF files 126
 EPS files 126, 132
 JPEG files 126, 132
 metafiles 126
 PICT files 126
 PNG files 132
 PostScript files 132
 TIFF files 132
saving files 23, 24
 controlling default file locations 204
 data files 24
 database file queries 19
 IBM SPSS Statistics data files 23
saving output 126, 130, 131
 Excel format 126, 129
 HTML 126, 127
 HTML format 126
 PDF format 126, 130
 PowerPoint format 126, 130
 text format 126, 131
 web report 128

saving output (*continued*)
 Word format 126, 129
 scale 51
 measurement level 51, 69
 scale variables
 binning to create categorical variables 77
 scaling
 pivot tables 142, 144
 scientific notation 51, 195
 suppressing in output 195
 scoring 185
 matching dataset fields to model fields 186
 merging model and transformation XML files 188
 missing values 186
 models supported for export and scoring 185
 scoring functions 187
 scripts 209, 210, 275
 adding to menus 209
 autoscripts 276
 Basic 279
 creating 275
 default language 205, 275
 editing 275
 languages 275
 Python 277
 running 275
 running with toolbar buttons 210
 startup scripts 282
 search and replace
 Viewer documents 123
 seasonal difference function 98
 select cases 109
 selecting cases 109
 based on selection criteria 110
 date range 111
 random sample 110
 range of cases 111
 time range 111
 selection methods 148
 selecting rows and columns in pivot tables 148
 servers 43, 259
 adding 43, 259
 editing 43, 259
 logging in 43
 names 43, 259
 port numbers 43, 259
 session journal 204
 Shift Values 86
 shortcuts 175
 showing 121, 141, 209
 captions 146
 dimension labels 141
 footnotes 146
 results 121
 rows or columns 141
 titles 141
 toolbars 209
 sizes 123
 in outline 123
 smoothing function 98
 sorting
 pivot table rows 138

sorting (*continued*)
 variables 102
 sorting cases 101
 sorting variables 102
 space-delimited data 10
 speed 41
 caching data 41
 spelling 57, 58
 dictionary 197
 split-file processing 109
 split-model viewer 155
 splitting tables 148
 controlling table breaks 148
 spp files
 converting to spj files 261
 spreadsheet files 7
 reading 9
 SPSSTMPDIR environment variable 204
 Stata files 9
 opening 7, 9
 reading 7
 saving 24
 Statistics Coach 2
 string format 51
 string variables 53, 59
 breaking up long strings in earlier releases 24
 computing new string variables 84
 entering data 59
 missing values 53
 recoding into consecutive integers 89
 style output 158
 subsets of cases
 random sample 110
 selecting 109, 110, 111
 subtitles
 charts 183
 subtypes 265
 vs. labels 266
 syntax 165, 176, 210, 255
 accessing Command Syntax Reference 5
 journal file 177, 178
 output log 166
 pasting 166
 Production jobs rules 255
 running 176
 running command syntax with toolbar buttons 210
 syntax rules 165
 Unicode command syntax files 176, 177
 syntax converter 285
 syntax editor 167
 auto-completion 169
 bookmarks 167, 171
 breakpoints 167, 170, 176
 color coding 169
 column edit mode 174
 command spans 167
 commenting or uncommenting text 172
 duplicate lines 173
 formatting syntax 172, 173, 174, 175
 indenting syntax 172
 joining syntax 173
 line numbers 167

syntax editor (*continued*)
 moving syntax 174
 multiple views/panes 167
 options 205
 removing empty syntax 173
 shortcuts 175
 trimming leading and trailing spaces 175
 syntax files
 encrypting 178
 SYSTAT files 7
 opening 7

T

T4253H smoothing 98
 tab-delimited files 7, 10, 24
 opening 7
 saving 24
 table breaks 148
 table chart 149
 table subtypes 265
 vs. labels 266
 TableLooks 141, 142
 applying 142
 creating 142
 tables 148
 alignment 145
 background color 145
 cell properties 145
 controlling table breaks 148
 converting TABLES command syntax to CTABLES 285
 fonts 145
 margins 145
 TABLES
 converting TABLES command syntax to CTABLES 285
 target lists 193
 templates 54, 55, 184, 201
 charts 184
 in charts 201
 using an external data file as a template 73
 variable definition 54, 55
 temporary active file 41
 temporary directory 204
 setting location in local mode 204
 SPSSTMPDIR environment variable 204
 temporary disk space 40, 41
 text 10, 123, 126, 131
 adding a text file to the Viewer 123
 adding to Viewer 123
 data files 10
 exporting output as text 126, 131, 266
 TIFF files 132
 exporting charts 126, 132
 time series data
 creating new time series variables 97
 data transformations 96
 defining date variables 96
 replacing missing values 99
 transformation functions 98
 titles 123
 adding to Viewer 123

titles (*continued*)
charts 183
TM1
 exporting to Cognos TM1 36
 reading Cognos TM1 data 21
Toggle column edit mode 172, 174
toolbars 209, 210
 creating 209, 210
 creating new tools 210
 customizing 209, 210
 displaying in different windows 210
 showing and hiding 209
transposing rows and columns 138
transposing variables and cases 103
trigger events 276
 autoscripts 276
Trim leading and trailing spaces 172, 175
Trim leading spaces 172, 175
Trim trailing spaces 172, 175
Tukey estimates 91

U

Unicode 7, 23, 199
Unicode command syntax files 176, 177
unknown measurement level 70
user-missing values 53

V

value labels 53, 59, 63, 67, 139, 200
 applying to multiple variables 70
 copying 70
 in Data Editor 63
 in merged data files 104
 in outline pane 200
 in pivot tables 200
 inserting line breaks 53
 saving in Excel files 24
 using for data entry 59
Van der Waerden estimates 91
variable attributes 54, 55
 copying and pasting 54, 55
 custom 55
variable information 191
variable labels 53, 139, 195, 200
 in dialog boxes 2, 195
 in merged data files 104
 in outline pane 200
 in pivot tables 200
 inserting line breaks 53
variable lists 193
 reordering target lists 193
variable names 50, 195
 generated by OMS 270
 in dialog boxes 2, 195
 mixed case variable names 50
 portable files 24
 rules 50
 truncating long variable names in earlier releases 24
 wrapping long variable names in output 50
variable pairs 112
 creating 112

variable sets 192
 defining 192
 using 192
Variable View 49
 customizing 56, 57, 199
variables 50, 60, 61, 112, 191, 192, 195
 defining 50
 defining variable sets 192
 definition information 191
 display order in dialog boxes 195
 finding in Data Editor 61
 inserting new variables 60
 moving 61
 recoding 86, 87, 88, 89
 renaming for merged data files 104
 restructuring into cases 112
 sorting 102
vertical label text 138
Viewer 121, 122, 123, 134, 196, 200
 changing outline font 123
 changing outline levels 123
 changing outline sizes 123
 collapsing outline 122
 deleting output 122
 display options 196
 displaying data values 200
 displaying value labels 200
 displaying variable labels 200
 displaying variable names 200
 excluding output types with OMS 269
 expanding outline 122
 find and replace information 123
 hiding results 121
 moving output 121
 outline 122
 outline pane 121
 results pane 121
 saving document 134
 search and replace information 123
 space between output items 134
virtual active file 40
Visual Bander 77

W

web report 128
 exporting output 128
weighted data 120
 and restructured data files 120
weighting cases 111
 fractional weights in Crosstabs 111
wide tables
 pasting into Microsoft Word 124
window splitter
 Data Editor 63
 syntax editor 167
windows 1
 active window 1
 designated window 1
Word format
 exporting output 126, 129, 266
 wide tables 126
wrapping 142
 controlling column width for wrapped text 142
variable and value labels 53

X

XML
 OXML output from OMS 273
 routing output to XML 266
 saving output as XML 263
 table structure in OXML 270
XSLT
 using with OXML 273

Y

years 197
 two-digit values 197

Z

z scores
 in Rank Cases 91

IBM[®]

Printed in USA