KaleidaGraph®

Version 3.6

Note: For optimal viewing of this file, choose **File** > **Preferences** > **General** and select the **Display Page to Edge** option.

Introduction

Welcome to KaleidaGraph®. You have purchased a thoughtfully designed graphing tool which provides the most powerful visual displays of quantitative information available in either the Windows or Macintosh environment. Through our documentation, you will learn how to develop accurate journal quality graphs that mathematically transform even the most complex data into elegant but functional graphical displays.

About this Manual

This manual contains detailed cross-platform information about the KaleidaGraph application. It provides step-by-step instructions for the Windows and Macintosh platforms. Wherever there is a difference between the two platforms, it is noted in the text.

The manual assumes that you are familiar with basic operations, such as clicking, double-clicking, and dragging. It also assumes familiarity with the components of the user interface, such as menus, windows, dialogs, buttons, and check boxes. If you are not familiar with these terms, review the documentation that came with your computer.

Organization of the Manual

This manual contains examples and procedures for common tasks, such as entering data, creating plots, and applying curve fits. The following is a brief overview of the chapters and appendixes in the manual:

- Chapter 1, “Getting Started,” discusses system requirements, installation, and several basic concepts, including opening files, getting help, and quitting the program.
- Chapter 2, “Guided Tour of KaleidaGraph,” provides a tutorial for generating a data set, creating and editing Scatter and Column plots, and placing the plots in the layout window.
- Chapter 3, “Working with Data Windows,” discusses how to navigate, make selections, add and delete rows and columns, and format columns in the data window.
- Chapter 4, “Working with Data,” discusses how to enter, import, edit, sort, and export data in the data window.
- Chapter 5, “Analyzing Data,” discusses how to view column statistics, bin data, and analyze data using a number of parametric and nonparametric statistical tests
- Chapter 6, “Working with Formula Entry,” discusses how to use the Formula Entry window to perform calculations in the active data window.
- Chapter 7, “Creating Plots,” discusses the different plot types, creating a plot, and generating similar plots.
- Chapter 8, “Working with Plots,” discusses how to modify the plot, including the axes, ticks, grids, markers, and plot color.
- Chapter 9, “Working with Plot Tools and Objects,” discusses the plot tools available in KaleidaGraph, in addition to creating and editing text labels and shapes.
- Chapter 10, “Working with Curve Fits and Error Bars,” discusses how to add curve fits and error bars to a plot.

- Chapter 11, “Importing and Exporting Graphics,” discusses how to import graphic objects into a plot and export the plots to a printer, file, or another program.
- Chapter 12, “Working with the Layout Window,” discusses how to use the layout window to place multiple plots on a single page.
- The Appendixes contain information on the program commands, settings files, and toolbox shortcuts, as well as general reference information.

Learning KaleidaGraph

If you are new to KaleidaGraph, it is recommended that you take the guided tour in Chapter 2. This tutorial guides you through the process of generating a data set, creating and editing two different plots, and placing the plots onto a single page. Other topics in the tutorial include modifying the legend, using Formula Entry, applying a General curve fit, and modifying data in a saved plot.

Once you start using KaleidaGraph, you can refer to the manual and the online help for step-by-step instructions to complete specific tasks. The online help also includes detailed information on the commands and dialogs available in the program.

Conventions

This manual uses the following conventions:

- Instructions for choosing commands are displayed in bold type. Levels are separated with a greater than symbol (>). For example: Choose **Gallery** > **Linear** > **Line**.
- Dialog buttons and options are displayed in bold type. For example: Click **OK**.
- Keys that you should press appear in bold type. If joined with a plus sign (+), press and hold the first key while you press the remaining keys. For example: Press **Ctrl**+**Period**.

Contacting Synergy Software

To receive technical support and upgrade information, either fill out and send in the registration card included with the software or access our web site and register online.

If you have any questions concerning KaleidaGraph, please contact us at:

Synergy Software
2457 Perkiomen Avenue
Reading, PA 19606-2049 USA

Phone: (1) 610-779-0522
Fax: (1) 610-370-0548

Internet addresses:
Sales/Upgrades: sales@synergy.com
Tech support: tech@synergy.com

Web sites: www.kaleidagraph.com
www.synergy.com

Please feel free to check our web site periodically for:

- Product descriptions
- Technical notes
- Frequently asked questions
- New release information
- Downloadable upgrade patches
- And other topics of interest

Getting Started

Chapter 1

This chapter lists the hardware and operating system requirements and explains how to install KaleidaGraph onto your computer. It also covers some basic concepts, including:

- Starting KaleidaGraph.
- Introducing the data and plot windows.
- Opening saved files.
- Opening KaleidaGraph files on another platform.
- Getting help.
- Setting preferences.
- Quitting the program.

1.1 System Requirements

Windows Version	Macintosh Version
• A Pentium PC or faster computer equipped with a CD-ROM drive. • Windows 98 or later (including Windows 2000, NT, ME, and XP). • 32 MB of RAM. Additional RAM is recommended and increases KaleidaGraph's performance. • 10-15 MB free hard disk space.	• A Power PC running System 8.6 or later. CarbonLib v1.6 will be installed if it is not present. • OS X users should have OS 10.1.3 or later. • 15 MB of free RAM is recommended. • 15-20 MB free hard disk space.

1.2 Installing KaleidaGraph

This section describes how to install the KaleidaGraph software on both the Windows and Macintosh platforms. A section containing uninstall instructions is also included.

1.2.1 Installing the Windows Version

The installer creates a new folder on your hard disk containing the KaleidaGraph software. If you are upgrading from an earlier version, your existing KaleidaGraph files are not affected.

To install KaleidaGraph on your hard disk:

1. Quit any other programs that are currently running.
2. Insert the KaleidaGraph CD-ROM. If AutoPlay is enabled on your system, the CDSetup program starts automatically.
3. If it is not already running, launch the **CDSetup** program and click **Install KaleidaGraph**. Alternately, you can run **KGSetup** to install the software.
4. Proceed through the Welcome screen, ReadMe information, and license agreement.
5. Specify where to install the KaleidaGraph program and its related files. To specify a different directory, click **Browse**. When you are finished, click **Next**.
6. Choose the type of installation you want to perform and click **Next** to install the software. The choices are:
   - **Typical** - installs all of the KaleidaGraph files.
   - **Compact** - installs only the files needed to run KaleidaGraph.
   - **Custom** - allows you to select which components to install.
7. The installer will display a window so that you can submit your registration information. The registration form can be submitted by email, postal mail, or fax. By registering the software, you will receive free technical support and upgrade information.
8. Once all of the program files have been installed, a message is displayed to let you know the installation of KaleidaGraph is complete.

1.2.2 Installing the Macintosh Version

The installer creates a new folder on your hard disk containing the KaleidaGraph software. If you are upgrading from an earlier version, your existing KaleidaGraph files are not affected.

To install KaleidaGraph on your hard disk:

1. Insert the KaleidaGraph CD-ROM.
2. Double-click the **KaleidaGraph 3.6 Installer** icon.
3. Proceed through the Welcome screen, license agreement, and ReadMe information to display the KaleidaGraph Installer dialog.
4. Choose the installation method (easy or custom) and specify where to place the KaleidaGraph folder.

   The Easy Install option automatically installs the following items:

   - The KaleidaGraph application.
   - An Examples folder containing example plots, data files, and curve fits.
   - The QuickHelp application (if it is not already installed), along with the KaleidaGraph Help and New Features documents.
   - A Manuals folder containing PDF versions of the KaleidaGraph documentation.
   - For OS 9 and earlier, CarbonLib v1.6 is also installed. If this file is already present, it is skipped.

   To control which files get installed, choose **Custom Install** from the pop-up menu.
5. Click **Install**. A KaleidaGraph folder is created on your hard disk and the files are installed. A Status dialog is displayed to keep you informed as the installation progresses.
6. The installer will display a window so that you can submit your registration information. The registration form can be submitted by email, postal mail, or fax. By registering the software, you will receive free technical support and upgrade information.
7. Once all of the program files have been installed, a message is displayed to let you know the installation of KaleidaGraph is complete.

Note: If you are updating from an earlier version, the default location for storing the settings files (KG Style, KG Macros, KG Layout, and KG Script) has changed. The settings files are now stored in the KaleidaGraph Preferences folder. The KaleidaGraph Preferences folder is located in the Preferences folder (in the users Library folder under OS X or in the System Folder for OS 9 and earlier). The installer automatically moves a copy of the settings files into the new location.

1.2.3 Uninstalling KaleidaGraph

Windows Version

To remove the KaleidaGraph software from your hard disk, perform one of the following:

- Choose **Start** > **Programs** > **KaleidaGraph 3.6** > **Remove KaleidaGraph 3.6**.
- Choose **Start** > **Settings** > **Control Panel** and then double-click **Add/Remove Programs**. In the dialog that appears, select **KaleidaGraph 3.6** and click **Add/Remove**.

Macintosh Version

To remove the KaleidaGraph software from your hard disk:

1. Quit any other programs that are currently running.
2. Insert the KaleidaGraph CD-ROM.
3. Double-click the **KaleidaGraph 3.6 Installer** icon.
4. Proceed through the Welcome screen, license agreement, and ReadMe information to display the KaleidaGraph Installer dialog.
5. Choose **Uninstall** from the pop-up menu and locate the folder that contains the KaleidaGraph program.
6. Click **Uninstall**.

1.3 KaleidaGraph Basics

This section introduces some of KaleidaGraph’s basic concepts, such as starting the program, opening files, and quitting the program. The various areas and terms associated with the data and plot windows are also explained.

1.3.1 Starting KaleidaGraph

To start KaleidaGraph, perform one of the following:

- **Windows:** Double-click the KGraph icon or choose **Start** > **Programs** > **KaleidaGraph 3.6** > **KGraph**.
- **Macintosh:** Double-click the KaleidaGraph icon.

The first time the program is started, the dialog in Figure 1-1 is displayed to personalize your copy of KaleidaGraph and enter your serial number and authorization code. If this is a new copy of the program, the serial number is located on the back cover of the manual, or on your order confirmation if you purchased KaleidaGraph electronically. If this is an update from an earlier version, your serial number has not changed

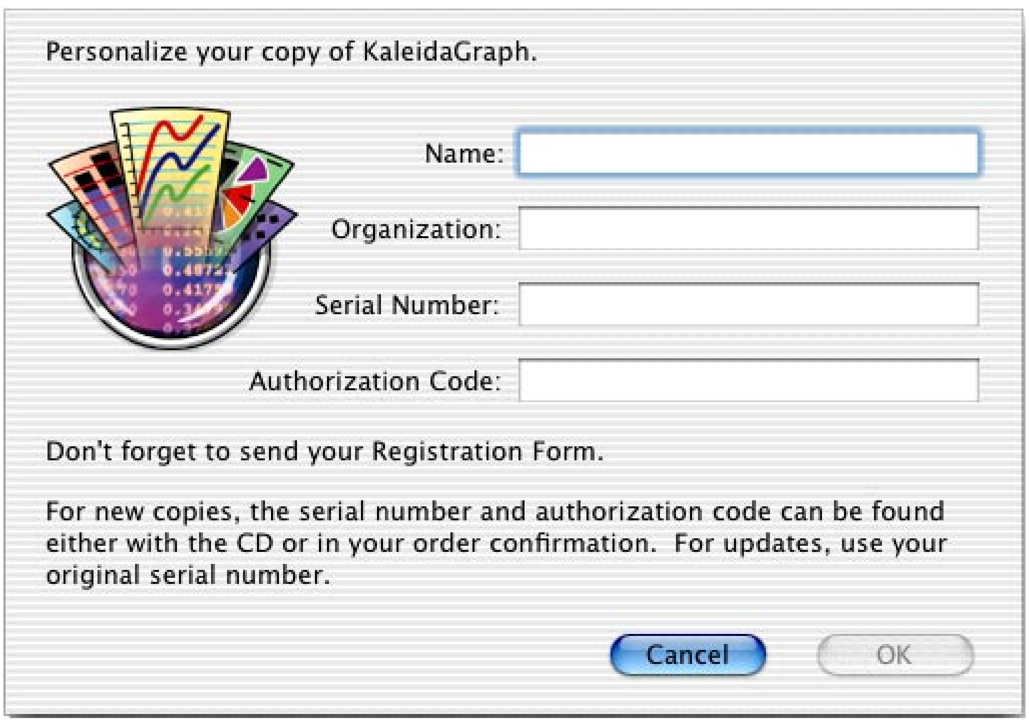


Figure 1-1 Startup dialog

After you click **OK**, two windows are displayed; a data window and the Formula Entry window. The next section contains an introduction to the data window. Refer to Chapter 6 for information on using the Formula Entry window to analyze and manipulate data in the active data window.

1.3.2 Data Window Overview

The data window, shown in Figure 1-2, contains a spreadsheet used to enter and store data for plotting and analysis. It consists of a number of cells organized in columns. A data window can contain a maximum of 1000 columns and 1 million rows, subject to memory limitations.

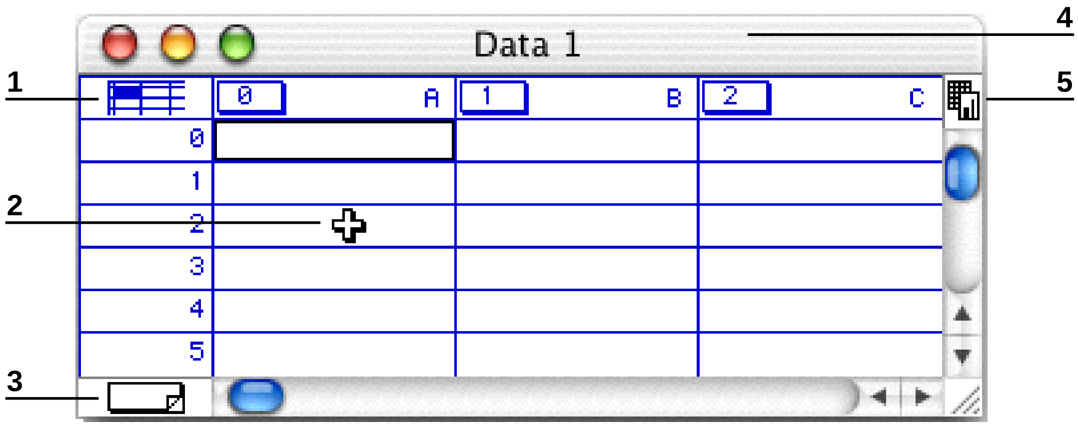


Figure 1-2 Data window

1. **Home Button**
   Click this button to return the data window to its origin (row 0, column 0). This is useful for quickly returning to the start of the window when viewing a different section of the data window.

2. **Cell Selection Cursor**
   This cursor is used to make selections in the data window.

3. **Posted Note**
   The Posted Note is used to enter information about the data in the active data window. Each data window has its own Posted Note. When information is stored in the Posted Note, its icon changes from an empty note to one with data in it.

4. **Title Bar**
   The title bar displays the name of the file. You can move the data window by dragging the title bar to a new location.

5. **Update Plot Button**
   Click this button to force an immediate update of the plot that is currently linked to this data window.

Each column in the data window can contain a different data type. You can change the data type for any of the columns by choosing **Data** > **Column Format**. Figure 1-3 shows a data window that contains columns formatted as Text, Time, and Float.

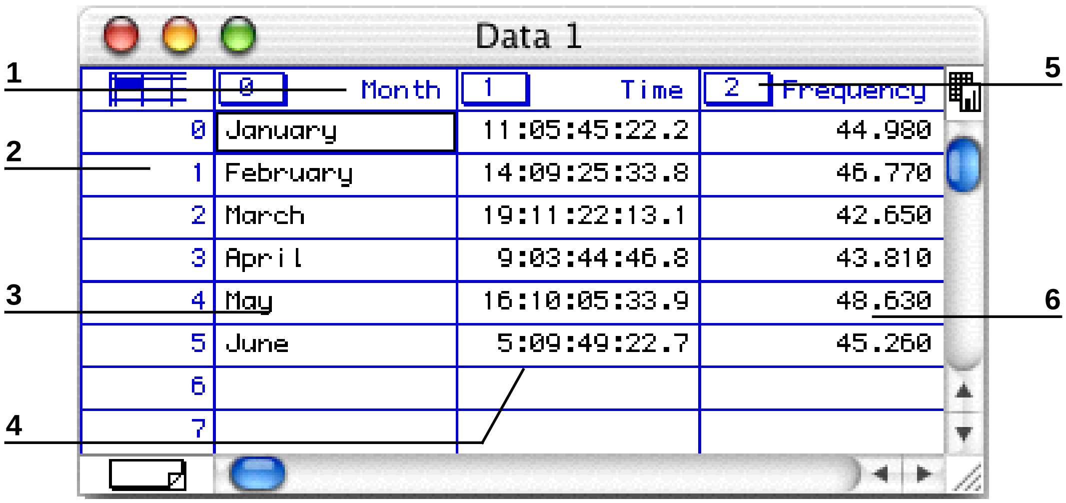


Figure 1-3 Data window elements

1. **Column Title**
   This is the title of the column (in this case **Month**). By default, the column titles are A, B, C, and so on for a new data window.

2. **Row Numbers**
   The row numbers are located on the left side of the data window.

3. **Text Column**
   This column displays data that has a **Text** format applied to it. Notice that the data in the text column is left-justified.

4. **Time Column**
   This column displays data that has a **Time** format applied to it. There are a number of options available for displaying time data.

5. **Column Numbers**
   Each column has a number associated with it. The column number is used to reference columns in the Macro Calculator and Formula Entry. Whenever a selection is made in the data window, the first column in the selection becomes column 0.

6. **Numeric Column**
   This column contains data using a Numeric (Float or Double) format. There are a number of options available for displaying numeric data.

1.3.3 Plot Window Overview

Plots are created by entering data into the data window and choosing a plot type from the **Gallery** menu. After selecting the data columns to be plotted, the plot is generated and displayed in the plot window. Figure 1-4 shows the components of the plot window.

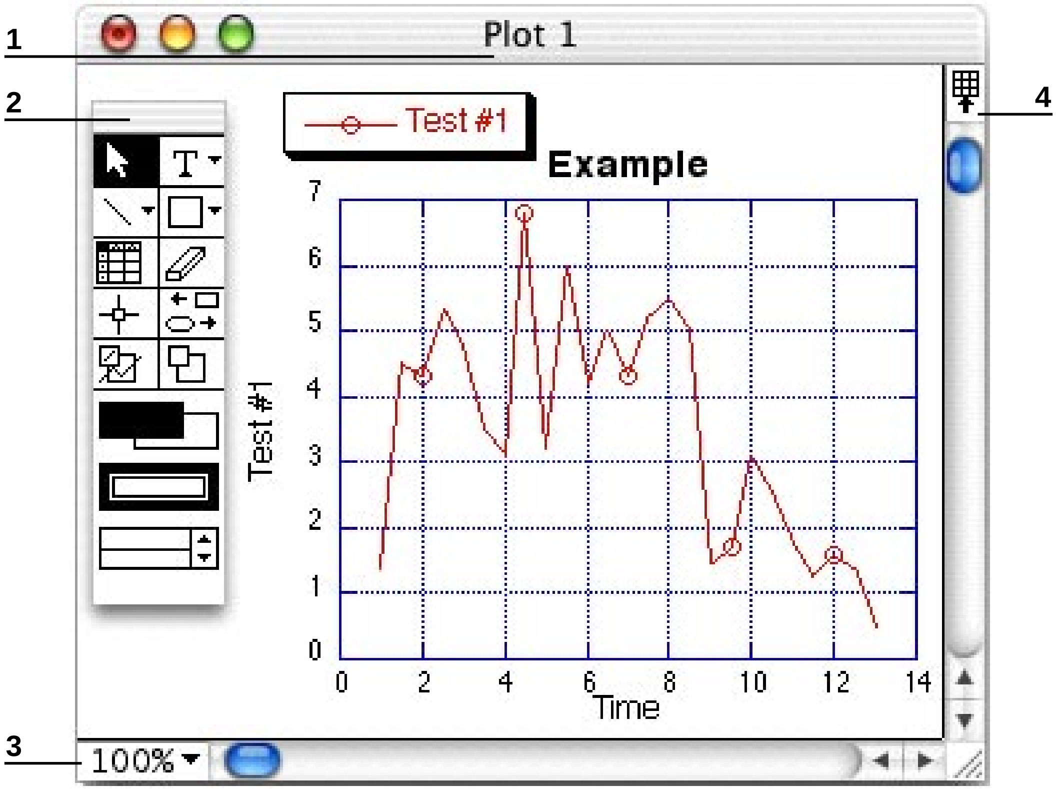


Figure 1-4 Plot window components

1. **Title Bar**
   The title bar displays the name of the file. You can move the plot window by clicking and dragging the title bar to a new location.

2. **Toolbox**
   The plot tools are located on a movable palette. The tools are used to create, modify, and enhance objects in the plot window.

3. **Zoom Setting**
   You can use this pop-up menu to change the view of the active plot window.

4. **Find Data Button**
   Clicking this button displays any data windows referenced by the plot. If the data is archived in the plot, the data windows are extracted and displayed.

Figure 1-5 shows the main elements found in a standard plot. There are other items that can be added to the plot, but the elements shown here are common to almost every kind of plot.

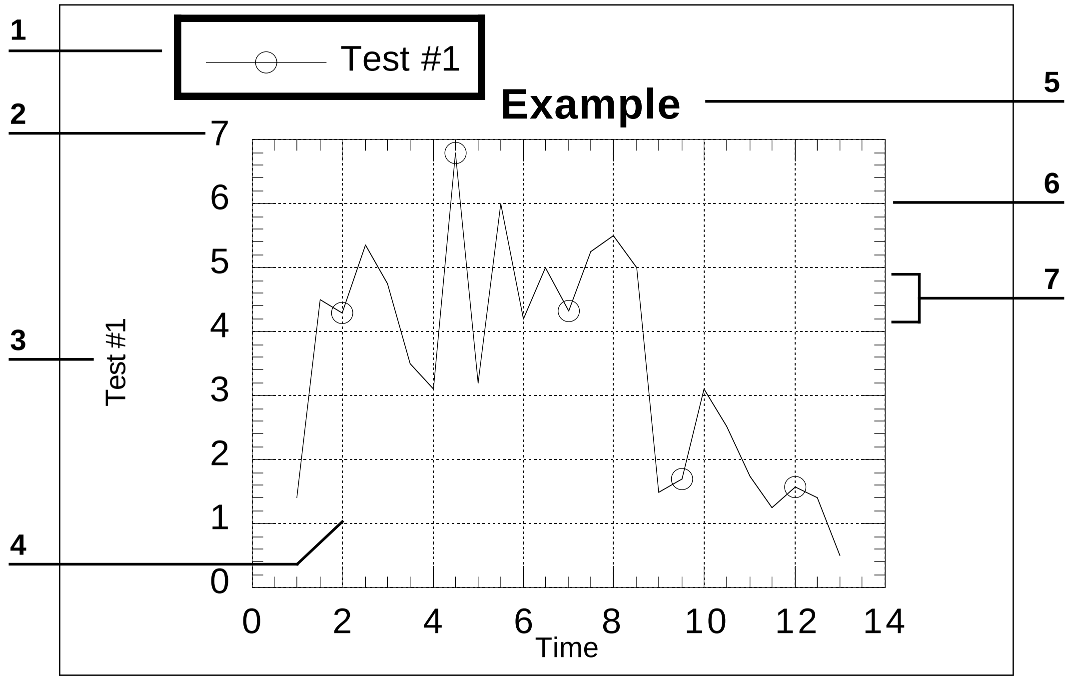


Figure 1-5 Plot elements

1. **Legend**
   The legend consists of a frame, the plot symbols, and the labels used to identify each variable in the plot. By default, the labels are taken from the names of the columns in the data window. The variables are listed in the order they are selected for plotting.

2. **Axis Labels**
   The axis labels display the values associated with each of the major tick marks on the plot.

3. **Axis Titles**
   The axis titles are taken from the column names of the data being plotted. If more than one variable is plotted on the same axis, the title is taken from the first column being plotted on that axis.

4. **Grid Lines**
   The grid lines can be used to display a horizontal or vertical line at each of the major and minor tick marks in the plot.

5. **Plot Title**
   The plot title is the name of the graph. By default, the plot title is taken from the name of the data window

6. **Major Tick Marks**
   The major tick marks are used to identify the major divisions along the axis.

7. **Minor Tick Marks**
   Minor ticks display the divisions between major tick marks.

1.3.4 Using the Toolbar

The Windows version of KaleidaGraph provides a toolbar for easy access to some commonly used commands. To use the toolbar, click a button to execute the appropriate command. Some of the buttons in the toolbar change when switching from a data window to a plot or layout window. The toolbars and the functions available are shown in Figure 1-6 and Figure 1-7.

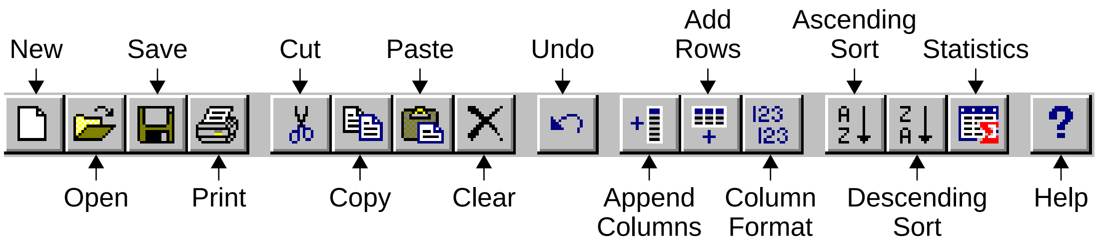


Figure 1-6 Data window toolbar

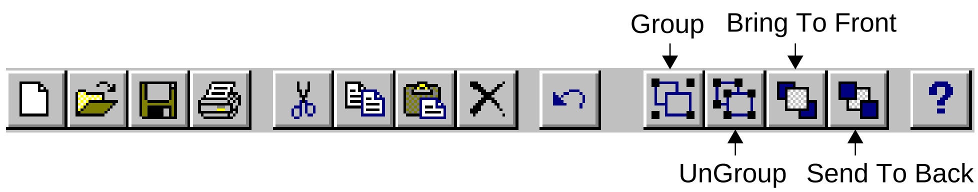


Figure 1-7 Plot and layout window toolbar

1.3.5 Using the Shortcut Menu

KaleidaGraph can display a shortcut menu containing commonly used commands when you are working in the data, plot, or layout window. The commands in the shortcut menu vary depending on what object is selected and where the pointer is when the shortcut menu is displayed.

To display the shortcut menu, do one of the following:

- (Windows) Right-click.
- (Macintosh) Hold down **Control** while pressing the mouse button.

1.3.6 Opening KaleidaGraph Files

The **Open** and **Import** commands (**File** menu) can be used to open files previously saved from KaleidaGraph. The only difference between the two commands is that the Open command lists all files that can be opened by KaleidaGraph while the Import command only lists files that are the same type as what you select from the submenu.

The **File** menu contains a **Recent Files** command that lists the last eight data or plot files that have been opened or saved in KaleidaGraph. You can open one of the files in the list by choosing its file name from the submenu

For information on opening KaleidaGraph files created on another platform, refer to Section 1.3.7.

1.3.7 Sharing Files Across Platforms

KaleidaGraph has the ability to share files between the Macintosh and Windows platforms. The table below lists the file name extensions and file types required to do this successfully.

Kind of File	File Name Extension	Macintosh File Type
Data file	.QDA	QDAT
Plot file	.QPC	QPCT
Macro file	.EQN	EQNS
Layout file	.QPL	QPLY
Script file	.QSC	QSCP
Style file	.QST	QSTY

Mac to Windows

To take a Macintosh file and open it on a Windows computer, the proper file name extension must be added. For example, if you created a data file named **Results** in the Macintosh version, the file needs to be renamed **Results.QDA** to be recognized by the Windows version.

Under OS 9 and earlier, the Preferences dialog contains an option that helps when sharing files across platforms. If **Add Windows file extensions** is selected, the appropriate Windows file name extension is appended to the file name in the Save dialog. Under OS X, the file name extensions are added automatically

Windows to Mac

The method to open Windows files on a Macintosh depends on what version of KaleidaGraph is running on the Macintosh. The two cases are listed below:

- Version 3.0.8 or later - No extra steps are necessary. The Open dialog automatically lists files that have the appropriate file name extension and opens them correctly.
- Version 3.0.5 or earlier - For files coming across a network, the file type needs to be modified with a resource editor before KaleidaGraph can read these files correctly. For files stored on disks, the appropriate file conversion software (for example, PC Exchange) must be installed and configured, as shown in the preceding table. If the conversion software asks for the creator, QKPT should be entered.

Note: There are two AppleScript droplets available on our web site that make the task of sharing files much easier. By dragging a file or folder onto these programs, the appropriate file name extension or file type is automatically applied.

1.3.8 Getting Help

Windows Version

To get online help, do any of the following:

- Choose **Help** > **Contents**.
- Press **F1** within any dialog to display context sensitive help.
- Click **Help** in any dialog that contains a **Help** button.
- Click the  button on the toolbar.
- Choose **Start** > **Programs** > **KaleidaGraph 3.6** > **KGHelp**.

Note: If the Help dialog is not displayed, the Help file was not loaded during launch. This occurs if the **KGHelp** file is renamed or is not in the Help folder (within the KaleidaGraph folder). If necessary, you can perform a custom installation to reinstall the Help files.

Macintosh Version

To get online help, do any of the following:

- Choose **Help** > **KaleidaGraph Help**.
- Press the **Help** key on the keyboard.
- Click **Help** in any dialog that contains a **Help** button.

Note: If the Help dialog is not displayed, the KaleidaGraph Help file could not be found or the QuickHelp program was deleted. The KaleidaGraph Help file must be in the same folder as the KaleidaGraph program and it cannot be renamed. If necessary, you can perform a custom installation to reinstall the Help files.

Help Dialog

The Help dialog, shown in Figure 1-8, contains a table of contents for the topics available in the KaleidaGraph program. Double-click the topic and the dialog displays the information available for the selected topic. If you are unable to locate the desired information from the table of contents page, you can use the Index or search the entire Help system for a particular term.

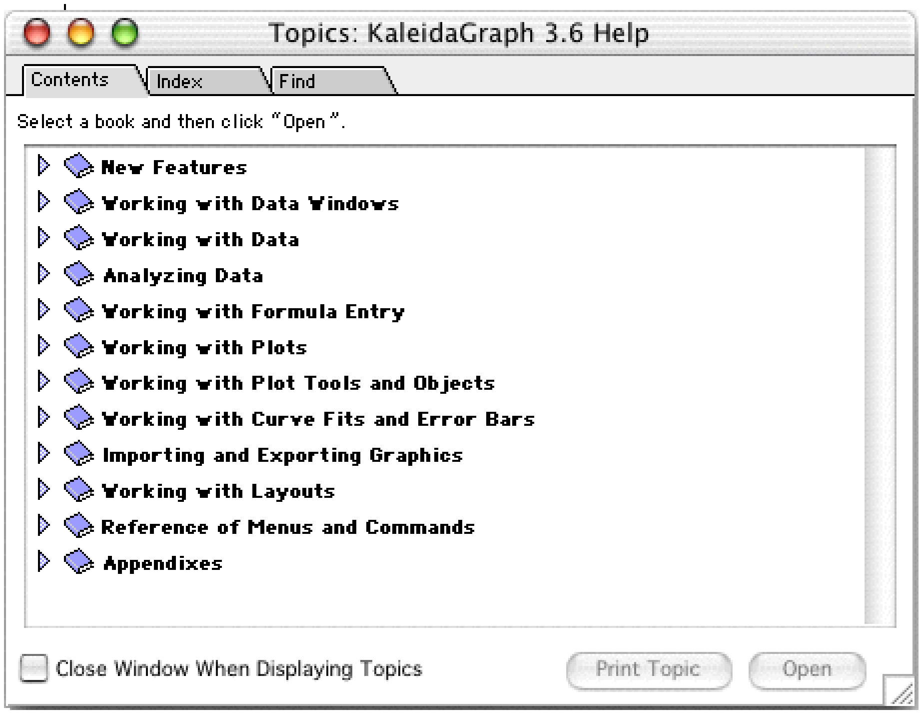


Figure 1-8 Help dialog

If you need information on using the Help system, choose the **How to Use Help** (Windows) or **Help with QuickHelp** (Macintosh) command.

1.3.9 Setting Preferences

KaleidaGraph provides a **Preferences** command in the **File** menu to specify what files are saved when you quit the program. The following dialog is displayed when you choose this command.

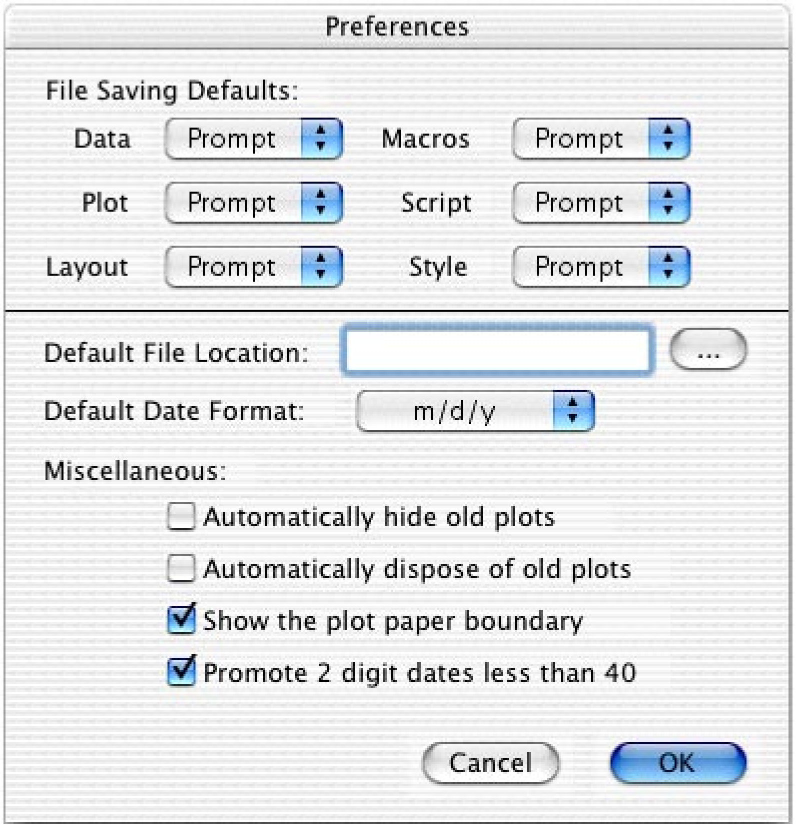


Figure 1-9 Preferences dialog

The pop-up menus in the upper section of the dialog control what happens to each of the different file types when quitting the program. For data and plot files, you can choose to **Always** save changes or to display a **Prompt** if there are any unsaved windows. If a data or plot window has not been saved previously, a Save dialog is displayed.

For the remaining types of files (Layout, Macros, Script, and Style), you can choose whether to display the **Prompt** dialog if any changes have occurred, **Always** save, or **Never** save the changes. For details on the information stored in each of these files, see Appendix B.

Note: The settings for this dialog are saved as part of the Style file. If you choose **Never** for the Style, you must manually overwrite the default Style file by choosing **File** > **Export** > **Style**.

1.3.10 Quitting KaleidaGraph

To close the KaleidaGraph program and return to the desktop, choose **File** > **Exit** (Windows) or **File** > **Quit** (Macintosh). KaleidaGraph saves any changes to the default Style, Macros, Script, and Layout files, depending on what is specified in the Preferences dialog. See Section 1.3.9 for details on setting the program preferences.

If **Prompt** is selected for any of the items in the Preferences dialog and changes have occurred to those particular items, a dialog similar to the one in Figure 1-10 is displayed.

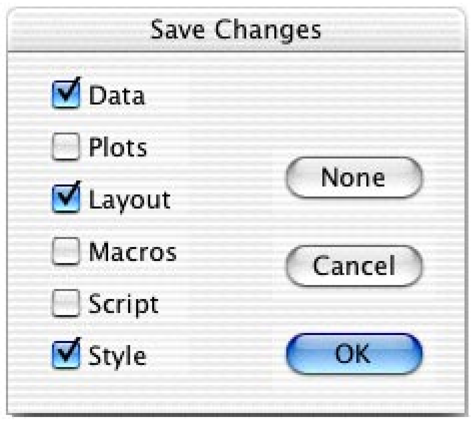


Figure 1-10 Save Changes dialog

Guided Tour of KaleidaGraph

Chapter 2

This chapter contains four major examples to guide you through the operation of KaleidaGraph. These examples show you how to:

- Create a new data set, change the column titles and format, sort the data, and calculate simple statistics for the data.
- Create a Scatter plot, change the display of the variable, use a few plot tools, and add a curve fit and error bars.
- Create a Column plot from a saved data set, modify the axes, change the display of the axis labels, and add value labels above the columns.
- Display the plots from the preceding examples on the same page using the layout window.

Some optional examples are also included to show you how to perform common operations not covered in the main examples. The topics include editing the legend text and frame, using Formula Entry, applying a user-defined curve fit, and modifying data in a saved plot.

2.1 Generating a Sample Data Set

This example takes you through the process of typing data into the data window, changing the column titles and format, sorting the data, and calculating statistics on the raw data.

2.1.1 Entering Data

The first step is to type some data into the data window. To do this:

1. Type **4.3** into the first cell of column 0.
2. Press the **Enter** (Windows), **Return** (Macintosh), or **Down Arrow** key to move down to the next cell.
3. Type the values **2.9**, **4.8**, **3.2**, **3.9**, **3.5**, and **2.3** into column 0. After each value is entered, press the **Enter**, **Return**, or **Down Arrow** key to move down a row.
4. Click the cell at row 0, column 1.
5. Use the same method to type the data values **8.0**, **6.2**, **9.0**, **5.7**, **8.8**, **7.2**, and **4.9** into column 1.

2.1.2 Changing Column Titles and Format

Next we will use the Column Format dialog to change the titles and format for these two columns.

1. Double-click the column title of column 0 (or choose **Data** > **Column Format**).
2. Type **Time** into the field below the listing of column titles.
3. Click the name of the second column (**B**).
4. Type **Test 1** into the field below the listing of column titles.

This dialog can also be used to change the format of the data columns. The following steps change the display of the data so that each value only has one decimal place.

1. Click **Time**, press **Shift**, and click **Test 1**. Both of these entries should be highlighted.
2. From the **Format** pop-up menu, choose **Fixed**.
3. From the **Decimals** pop-up menu, choose **1**.
4. Click **OK** to apply the changes to the data window.

Your data window should resemble the one shown in Figure 2-1.

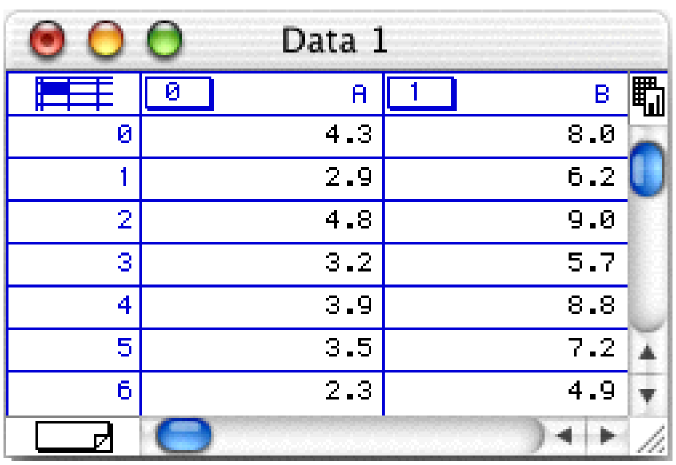


Figure 2-1 Sample data window

2.1.3 Sorting the Data

Now we will sort the data to get the values in ascending order (from low to high).

1. If it is not already selected, click the **Time** label in the data window to select the entire column.
2. Choose **Functions** > **Ascending Sort**.
3. Press **Shift** and click both the **Time** and **Test 1** entries.
4. Click **OK** to apply the changes to the data window.

2.1.4 Calculating Statistics

The final step is to calculate a number of standard statistics on one of the data columns.

1. Click the **Test 1** label in the data window to select the entire column.
2. Choose **Functions** > **Statistics** to display the statistics for the Test 1 data. The Statistics dialog provides a **Copy to Clipboard** button to export the results for use in a data, plot, or layout window
3. Click **OK**.

At this point, you can proceed to the next example or save your data by choosing **File** > **Save Data**.

2.2 Creating and Editing a Scatter Plot

This example uses the data from the preceding example to create a Scatter plot. This example shows how to change the marker type, size, and color, use the Identify and Data Selection tools, apply a Linear curve fit, display the curve fit equation, and add error bars.

2.2.1 Creating a Scatter Plot

Now, let's create a plot using the example data entered in the previous exercise.

1. Choose **Gallery** > **Linear** > **Scatter**. The Variable Selection dialog is displayed. Notice that the name of the data file and its column titles are displayed in this dialog.
2. Select **Time** as the X variable and **Test 1** as the Y variable by clicking the appropriate buttons. Figure 2-2 shows what the Variable Selection dialog should look like at this point.

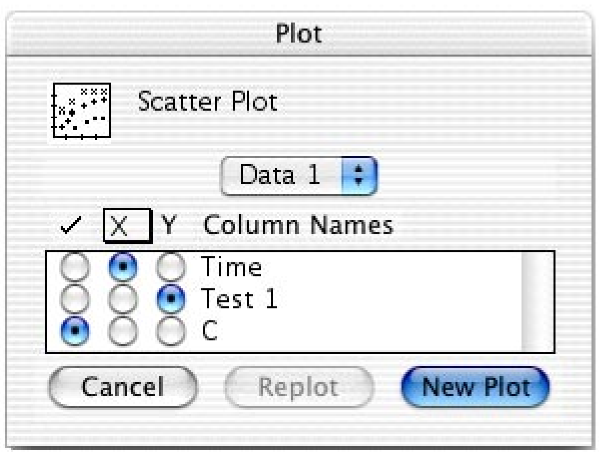


Figure 2-2 Variable Selection dialog

3. Click **New Plot** to create a Scatter plot.

The X variable you selected is the independent variable and the Y variable is the dependent variable. By default, the X variable is plotted on the horizontal axis and the Y variable is plotted on the vertical axis.

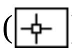
The title of the plot is taken from the name of the data window. The X and Y axis titles are taken from the column titles of the variables being plotted. The Y variable title is also used in the legend.

2.2.2 Changing Plot Style

Now that the graph has been created, it can be modified very easily. For example, let's change how the data is represented on the plot. You will use the Plot Style dialog to change the marker type, size, and color.

1. Triple-click the marker displayed in the legend (or choose **Plot** > **Plot Style**).
2. Select a different marker to represent the variable on the plot. The markers are displayed on the left side of the dialog. The first six markers in the left column are transparent; all of the others are opaque
3. Change the value in the **Marker Size** field to 18 and select a different color from the color palette.
4. Click **OK** and the plot is redrawn to reflect the changes that have been made.

2.2.3 Using the Identify Tool

Now we will use the Identify tool () on the toolbox to display the coordinates of the data.

1. Select the Identify tool by either clicking it or pressing **I** on your keyboard.
2. Once the tool is selected, click one of the data points. The X and Y coordinates are displayed in the upper-left corner of the plot window.

It is also possible to leave the coordinates directly on the plot. To do this:

- Press **Alt** (Windows) or **Option** (Macintosh) as you release the mouse button. This places a label containing the coordinates to the right of the point.

2.2.4 Applying a Linear Curve Fit

You can quickly and easily fit a curve to a set of data points. To add a curve fit to the plot:

1. Choose **Curve Fit** > **Linear**. This displays a dialog to select which variables to fit with the Least Squares Error method.
2. Select a variable to be fit (in this case **Test 1**) by clicking its check box.
3. Click **OK**. The curve fit is calculated and the curve fit line is drawn on the plot. By default, the curve fit results will also be displayed on the plot. If the equation is not displayed, turn on **Display Equation** in the **Plot** menu.

The position of the equation can be changed using the Selection Arrow.

1. Click the Selection Arrow on the toolbox.
2. Drag the equation to a new position.
3. When the move is complete, click anywhere else in the window and the object handles disappear.
4. You can use the same technique to move the legend.

At this point, your plot should resemble Figure 2-3.

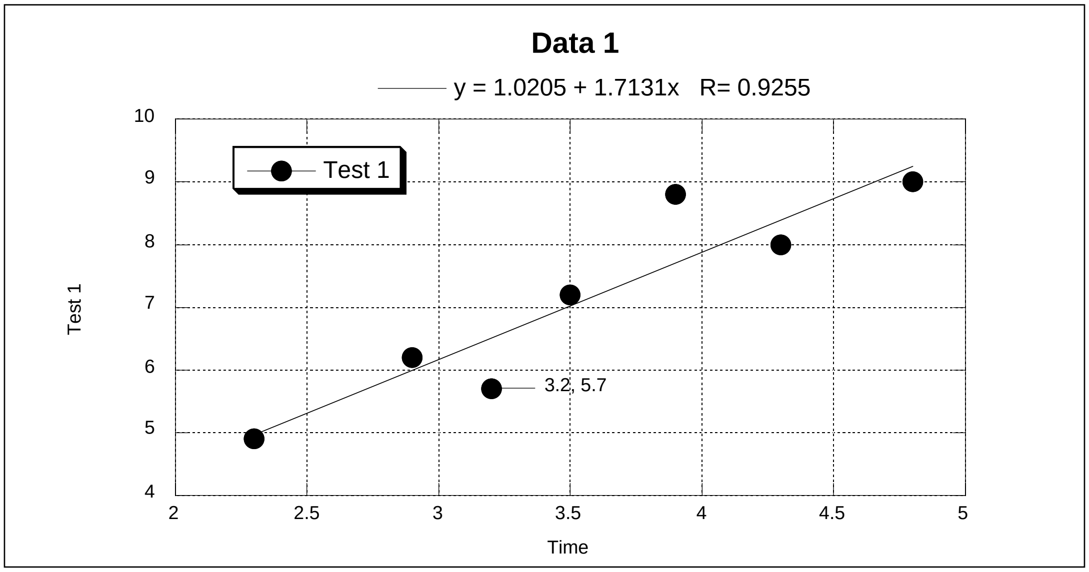


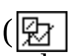
Figure 2-3 Sample Scatter plot

2.2.5 Exporting the Results of the Curve Fit

Once a curve fit is applied, you can copy the values of the curve fit line to the data window. These values are appended after the existing data in your data window. The first column will be a series of X values. The number of X values will be equal to the number of curve fit points specified in the Curve Fit Options dialog (**Format** menu). The second column will contain the values from the curve fit at each of these locations.

1. Reselect **Linear** from the **Curve Fit** menu. A Curve Fit Selections dialog appears with a drop-down arrow under **View**.
2. Click the drop-down arrow and choose **Copy Curve Fit to Data Window** from the pop-up menu.
3. Click **OK** to return to the plot window.

2.2.6 Using the Data Selection Tool

Now we will use the Data Selection tool () to graphically remove a point from the plot. The Data Selection tool operates by enclosing a region of the plot in a polygon. Any data points outside the polygon are removed from the plot. By pressing **Alt** (Windows) or **Option** (Macintosh) as you make the polygon, the data inside the polygon can be eliminated.

1. Select the Data Selection tool by either clicking it or pressing **S** on your keyboard.
2. Once the tool is selected, press **Alt** (Windows) or **Option** (Macintosh) and create a polygon around the data point in the lower-left corner of the plot window. Once the polygon is complete, the point is removed and the curve fit is recalculated.
3. Double-click the Data Selection tool to return the plot to its original state.

2.2.7 Adding Error Bars

The last modification to the plot will be the addition of error bars. Error bars enable you to illustrate the amount of error for the plotted data.

1. Choose **Plot** > **Error Bars** to display the Error Bar Variables dialog.
2. Click the check box in the **Y Err** column to add vertical error bars. The Error Bar Settings dialog is displayed to choose the type of error.
3. From the pop-up menu, choose **Standard Error** for the error type. The dialog should look like the one in Figure 2-4.

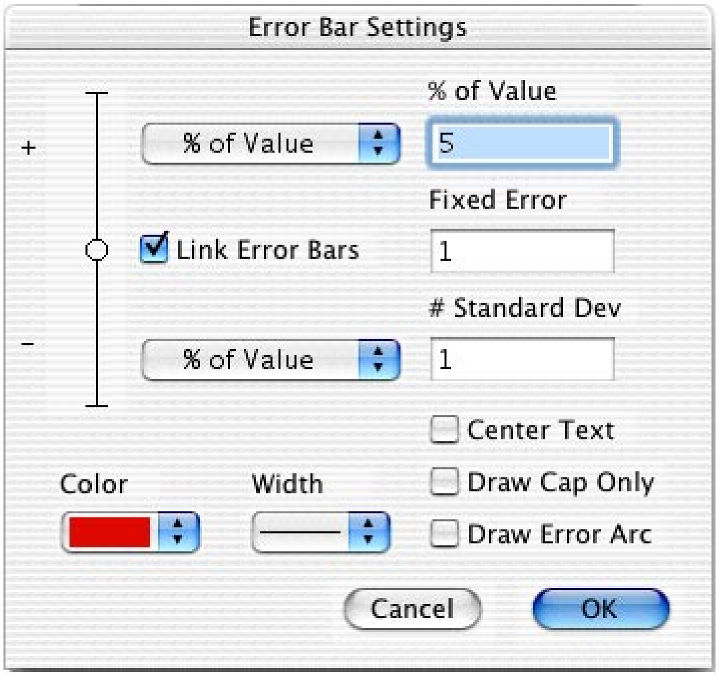


Figure 2-4 Error Bar Settings dialog

4. Click **OK** to return to the Error Bar Selection dialog.
5. Click **Plot** to add the error bars to the plot. The error bars represent the standard error of the entire data column.

The finished plot is shown in Figure 2-5.

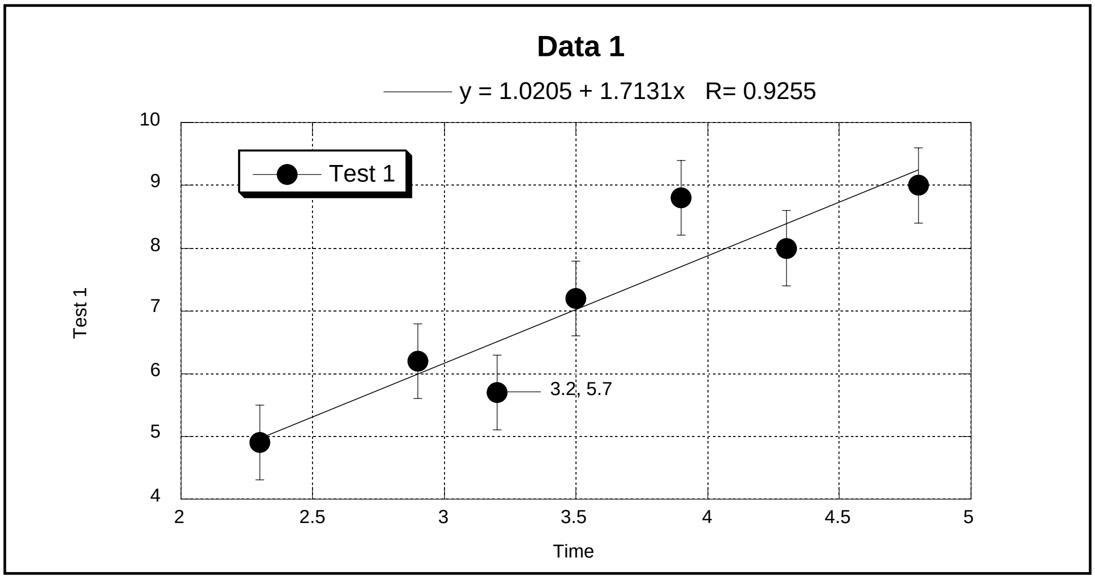


Figure 2-5 Finished Scatter plot

You have just created a customized plot. You can continue on to the next example or you can save the plot by choosing **File** > **Save Graph**. If you save the graph, a copy of the data window is saved with the plot (in the same file). The process of opening a saved plot and extracting the data is covered in Section 2.5.4.

2.3 Creating and Editing a Column Plot

This example uses a Column plot to show how to adjust major and minor ticks, axis labels, fill patterns, column spacing, plot color, and label rotation, in addition to displaying values above the columns.

2.3.1 Opening a Saved Data File

We will begin this example by opening a saved data set.

1. Choose **File** > **Open**.
2. Locate and open the **Data** folder, which is located in the **Examples** folder.
3. Double-click the **Housing Starts** file.

2.3.2 Creating a Column Plot

Now, let's create a plot using this data.

1. Choose **Gallery** > **Bar** > **Column**. The Variable Selection dialog is displayed.
2. Select **Month** as the X variable and **1966(K)** as the Y variable by clicking the appropriate buttons.
3. Click **New Plot** to create a Column plot.

2.3.3 Changing Axis Options

The first set of changes will be made in the Axis Options dialog. This dialog contains the majority of the settings for the axes, grid lines, tick marks, and axis labels.

The first change is to remove the vertical grid lines.

1. Triple-click the X axis (or choose **Plot** > **Axis Options**) to display the dialog in Figure 2-6.

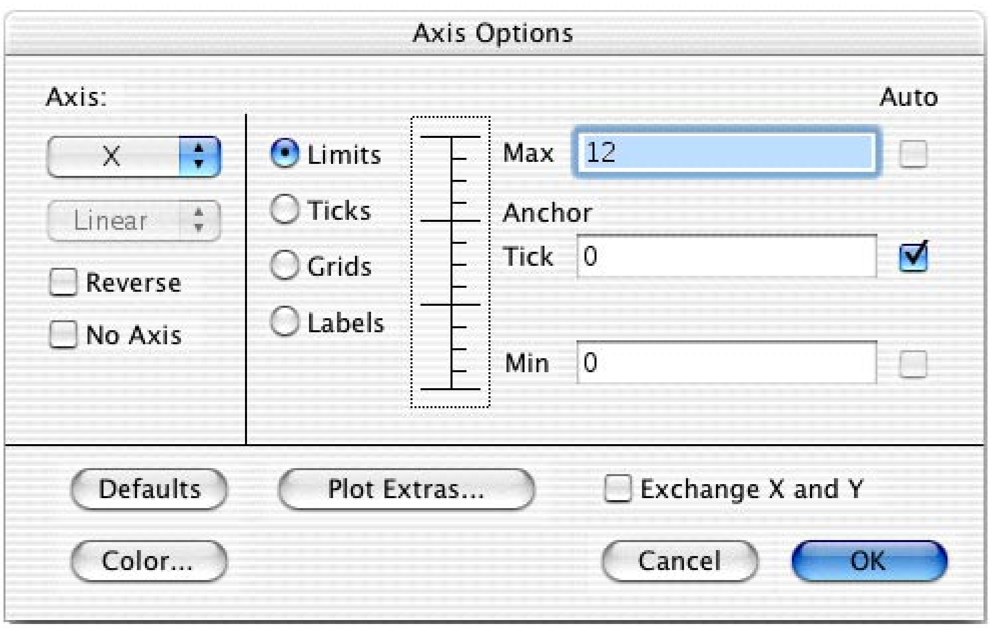


Figure 2-6 Axis Options dialog

2. Click **Grids**. The dialog changes to show the options that can be selected for the major and minor grids.
3. Choose **None** from the pop-up menu to the right of **Major**.

The next change is to remove the tick marks on the X axis.

1. Click **Ticks**. The dialog changes to show the options that can be selected for the major and minor ticks.
2. Choose **None** from the pop-up menus below both **Major** and **Minor**.

The next change also involves the tick marks, but this time on the Y axis.

1. Select the **Y** axis from the pop-up menu under **Axis**.
2. Choose **Out** from the pop-up menus below both **Major** and **Minor**.

The last step is to change the maximum Y axis limit from 140 to 160.

1. Click **Limits**. The dialog changes to show the options that can be selected for the limits.
2. Change the value in the **Max** field from 140 to 160.
3. Click **OK** to update the plot with all of the changes that were made while in the Axis Options dialog

2.3.4 Changing the Fill Pattern

Now you can change the fill pattern for the columns using the Plot Style dialog.

1. Triple-click the small square in the legend which is filled with the same pattern as the columns (or choose **Plot** > **Plot Style**).
2. You can now select a different fill pattern for the columns. Click one of the fill patterns and a frame appears around that pattern to show that it is selected.
3. Click **OK**.

2.3.5 Increasing the Column Offset

The next step is to increase the amount of space between the columns.

1. Choose **Format** > **Plot Extras**.
2. Change the **Column Offset** percentage from 20 to 40%.
3. Click **OK** to update the plot.

At this point, your plot should resemble the one shown in Figure 2-7.

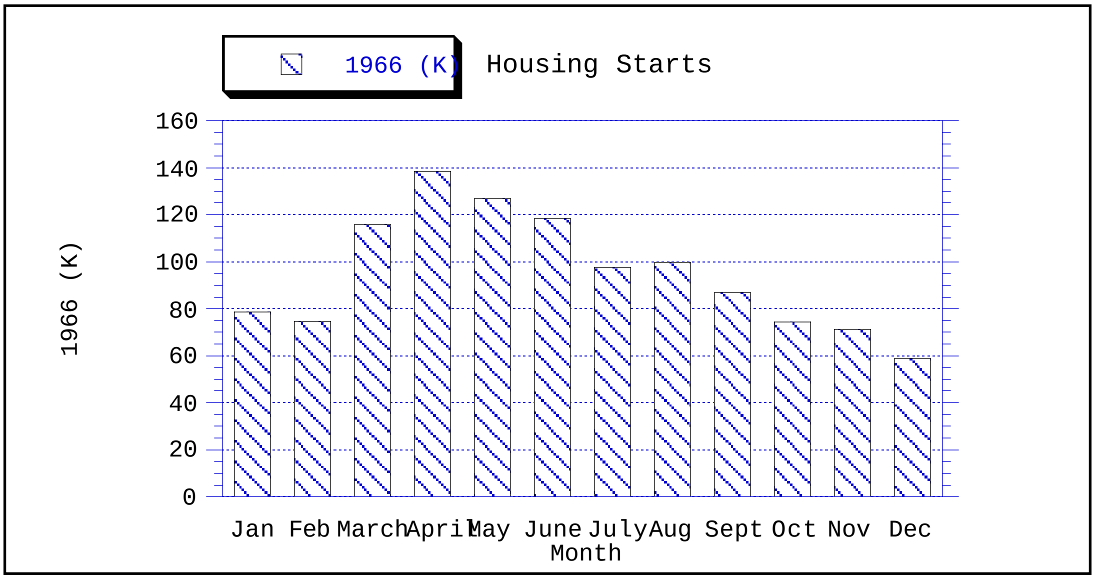


Figure 2-7 Sample Column plot

2.3.6 Changing Plot Color

The next step is to add some color to the interior of the plot. By default, plots are created without interior and background colors. To select an interior color:

1. Click any of the four axes to select the plot. Two sets of icons are displayed at the bottom of the toolbox. The first icon displays two overlapping rectangles which control the interior and background color of the plot.
2. Click the rectangle in the foreground and select one of the lighter colors from the color palette that appears. The selected color is used to fill the interior of the plot.

2.3.7 Editing Text Labels

The following steps remove the X axis title, resize the Y axis title, and rotate the X axis labels.

1. Click the X axis title, **Month**, and press **Backspace** (Windows) or **Delete** (Macintosh).
2. Click the Y axis title, **1966 (K)**. Drag any one of the four object handles to increase the font size of the label. It is also possible to change the font size by double-clicking the text label.
3. Double-click one of the X axis labels. Notice that this dialog has its own set of menus.
4. Choose **Format** > **90 Degree Rotation**.
5. Choose **Format** > **Right Justify** so that the rotated labels line up evenly.
6. Click **OK** to return to the plot window.
7. Drag one of the X axis labels closer to the axis. You can also use the arrow keys to move selected objects one pixel at a time in the specified direction (10 pixels if you press **Shift** while using the arrow keys).

2.3.8 Adding Value Labels

The last step is to display the value of each column. To do this, turn on **Add Values** in the **Plot** menu. The values are placed at the top of each column. The values can be moved as a group by dragging them to a new location.

The Column plot is now complete.

2.4 Laying out Plots for Printing

This example shows how to use the layout window to place the plots created in the previous examples on a single page.

Note: The following steps assume the two plots from the previous examples are still available. If you saved the plots and quit before reaching this example, use the **Open** command (**File** menu) to open these plots. If you do not have them any longer, you can open any two plots from the **Plots** folder in the **Examples** folder.

2.4.1 Placing Plots in the Layout

To place plots into the layout window:

1. Choose **Windows** > **Show Layout** > **KG Layout**. If no layout has been created previously, an empty layout window is displayed.
2. Use the **Select Plot** command (**Layout** menu) to select the two plots that were created in the previous examples. At this point, do not worry about their overall placement.

2.4.2 Arranging Plots in the Layout

To arrange the plots in the layout window:

1. Choose **Layout** > **Arrange Layout**. The Arrange Layout dialog allows you to enter the number of rows and columns to divide the layout window into equal sections.
2. The default settings (two rows and one column) are sufficient for this example, so click **OK**. Notice that the layout window is divided into two equal sections and the plots are automatically resized and placed into these sections.

2.4.3 Modifying and Exporting the Layout

It is possible to display more than just plots in the layout window. The plot tools are available, enabling you to add text and other objects to the layout. Various graphic images can be imported into the layout window. Also, a background pattern and frame can be added to the layout using the **Set Background** command (**Layout** menu).

The following steps explain how to add a text label to the layout window:

1. Select the Text tool (T) from the toolbox. You can select this tool by either clicking it or pressing **T** on your keyboard.
2. Click anywhere in the layout window. The Edit String dialog will appear.

3. Enter some text into this dialog. KaleidaGraph supports fully-stylized text, so feel free to highlight various portions of the text string you have entered and make changes to the font, font size, style, and color. Any changes you make only affect the selected portion of the text string.
4. Once you are finished making changes, click **OK** to add the text label to the layout window. You can move the label to a new position using either the Text tool or the Selection Arrow.
5. You can now print the layout by choosing **File** > **Print Layout**.
6. Close the layout window by choosing **File** > **Close**.

2.5 Additional Examples

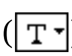
This section contains four optional examples to show you some of the finer points of KaleidaGraph. Unlike the major examples you completed earlier, you do not have to follow the additional examples in any order. You can select those topics which are relevant to the way you will be using KaleidaGraph to get a greater feel for the program.

2.5.1 Editing the Legend

This example shows how to edit the legend frame and text. The attributes of the legend frame are controlled by the bottom three icons on the toolbox. The steps in this example use the last icon in the toolbox, which is divided into two sections: a Line Style icon on the left and a Line Width icon on the right.

1. Open the **Sample Plot** file, which is located in the **Plots** folder in the **Examples** folder.
2. Click the legend to select it.
3. From the toolbox, click the Line Width icon (the up and down arrows) and choose Hairline from the pop-up menu. Notice that the legend frame changes from a shadow box to a hairline width line.
4. Now click the Line Style icon (the one to the left of the up and down arrows) and select one of the dashed lines from the pop-up menu. Notice that the line surrounding the legend now contains the dashed pattern you selected.
5. Finally, choose **None** from the Line Style pop-up menu. This removes the legend frame completely

Now we can edit the text inside the legend.

1. Select the Text tool () from the toolbox. You can select this tool by either clicking it or pressing **T** on your keyboard.
2. Double-click any of the three labels inside the legend. A dialog is displayed to modify the text.
3. Delete the text in this dialog and type any information you like. Feel free to change the font, size, and style as well.
4. Click **OK** to return to the plot and see the change.

The changes you made only affect this one label. If you use the Selection Arrow instead of the Text tool, you can change the attributes of all legend items at once. However, you cannot edit the text with the Selection Arrow.

2.5.2 Using Formula Entry

This example shows how to use the Formula Entry window, shown in Figure 2-8, to operate on the data window. Details on executing a multi-line formula are also included.

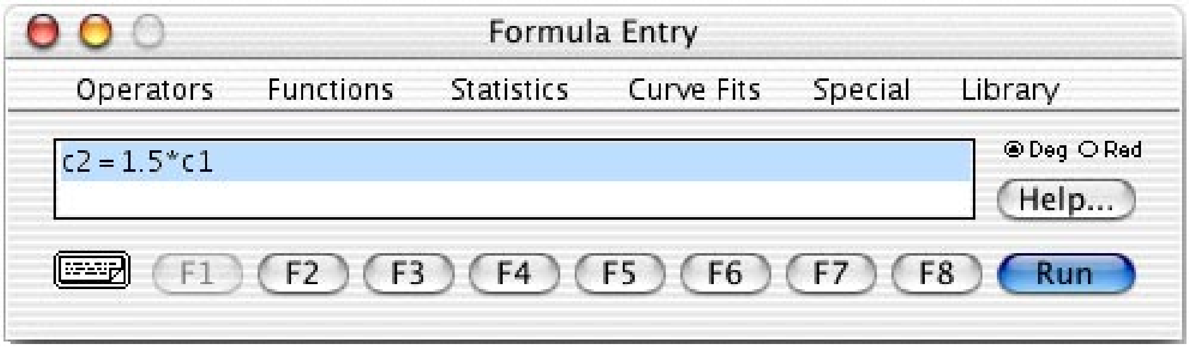


Figure 2-8 Formula Entry window

Formula Entry Overview

The Formula Entry window is a very powerful tool for data analysis. Use Formula Entry to enter equations (functions) that generate and manipulate data in the frontmost data window. The results of a formula can be placed in a data column, a single cell, or a memory location.

Memory locations and column numbers can be used in formulas. Memory locations range from 0 to 99 and need to be preceded by an **m** when used in a formula (m15, m35, and so on).

Column numbers range from 0 to 999 and need to be preceded by a **c** when used in a formula (c10, c55, and so on). Column numbers are displayed in a box at the top of the column. Please note that when a selection is made in the data window, the first column in the selection becomes column 0.

The following are a few examples of basic formulas along with a description of each:

c2=c0+c1; Adds the first two columns together and stores the results in column 2.
c1=c0/1000; Divides column 0 by 1000 and stores the results in column 1.
c2=cos(c0); Calculates the cosine of column 0 and stores the results in column 2.

Executing Individual Formulas

Let's get started by running a few formulas and seeing their effects on the data window. In the steps that follow, you can press **Enter** (Windows) or **Return** (Macintosh) instead of clicking **Run**.

1. Choose **File** > **New** to display an empty data window.
2. Choose **Windows** > **Formula Entry**. By default, the F1 button is selected. The F1–F8 buttons can be used to store common formulas, however, we recommend that you leave F1 for general use and store your formulas in F2–F8.
3. Click **F2**, type **c0=index() + 1**, and click **Run**. This creates a series from 1 to 100 in column 0.
4. Click **F3**, type **c1=log(c0)**, and click **Run**. This function calculates the logarithm of each value in column 0 and stores the results in column 1.
5. Click **F4**, type **c2=c1^2**, and click **Run**. This formula squares each value in column 1 and stores the results in column 2.

6. Click **F5**, type **cell(0,3)=csum(c2)**, and click **Run**. This formula calculates the total sum of the values in column 2 and stores the result in the cell at row 0, column 3. You need to click the right scroll arrow to see the result of this formula.

Executing Multi-line Formulas

It is not necessary to enter and execute each formula individually. KaleidaGraph has a method to enter multiple formulas and execute them all at once.

To the left of the F1 button is a Posted Note button ( ). Clicking this button displays a text editor. You can enter multiple formulas into the editor and run them all at once by clicking **Run**. The formulas must be on separate lines and each must be terminated with a semicolon.

Let’s try using the same formulas from before, but this time executing them using the Posted Note window:

1. Choose **File** > **New** to display an empty data window.
2. Choose **Windows** > **Formula Entry**.
3. Click the Posted Note button in the Formula Entry window to display a text editor.
4. Type the following formulas into the Posted Note window. Note that each formula ends with a semicolon and appears on a separate line.

```
c0=index() + 1;
c1=log(c0);
c2=c1^2;
cell(0,3)=csum(c2);
```

5. After the formulas are entered, choose **File** > **Close** to return to the Formula Entry window. A message is displayed in the Formula Entry window telling you to click Run to execute the Formula Posted Note.
6. Click **Run** to execute all of the formulas at once.

As you can see, this is a very convenient method to execute multiple formulas at once. Using this method, you can save the formulas as a text file that can be opened at a later time within the Posted Note dialog.

2.5.3 Applying a General (user-defined) Curve Fit

This example takes you through the process of opening a saved plot and applying a user-defined curve fit. KaleidaGraph’s General curve fit is based on the Levenberg-Marquardt algorithm. You can solve up to nine unknown parameters during the fitting process.

Opening a Saved Plot

We will start by opening a saved plot.

1. Choose **File** > **Open**.
2. Locate and open the **Plots** folder, which is located in the **Examples** folder.
3. Double-click the **Inhibition Plot** file.

Applying a General Curve Fit

Now we are ready to apply a General curve fit. The following steps will take you through the process of applying a Sigmoidal curve fit to the data. The equation is of the form **y = a + (b - a) / (1 + x / c)**.

1. Choose **Curve Fit** > **General** > **fit1**. This will display the Curve Fit Selections dialog.
2. Click **Define** to display the Curve Fit Definition dialog, shown in Figure 2-9.

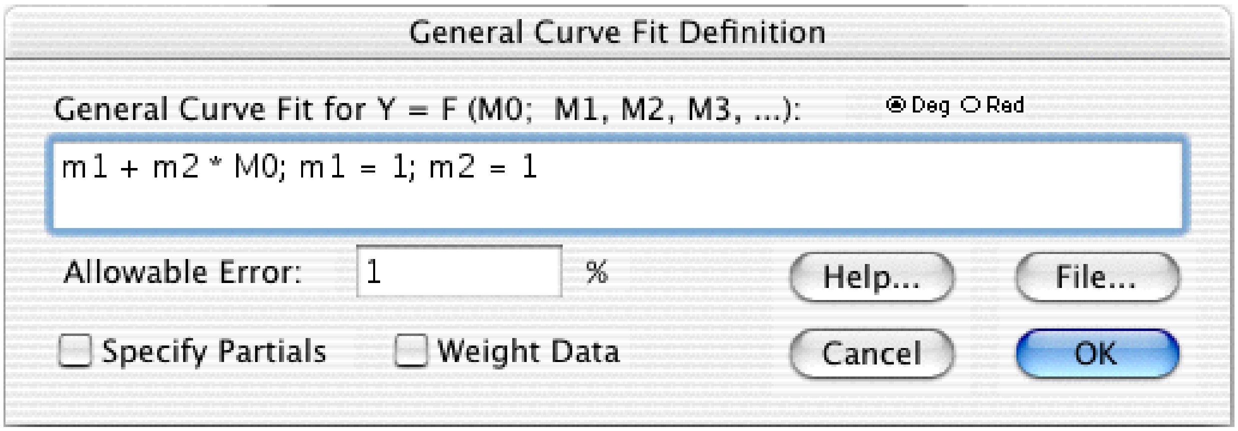


Figure 2-9 Curve Fit Definition dialog

3. Type **m1+(m2-m1) / (1+x/m3);m1=1;m2=100;m3=1** into the field provided and click **OK**. The information that appears after the curve fit definition represents the initial guesses for the unknown parameters in the equation.
4. Click the check box in front of **% Inhibition**. This indicates to KaleidaGraph that you want to apply a curve fit to this variable.
5. Click **OK** and the curve fit is calculated and displayed on the plot.
6. A table should appear containing the results of the fit. If it is not displayed automatically, turn on **Display Equation** in the **Plot** menu. The table lists the values of the unknown parameters along with the standard error of these values. It should be read as the parameter value +/– the standard error. The Chi Square and R values are also displayed as part of the curve fit results.
7. Feel free to move the table to a new location using the Selection Arrow. If you would prefer to hide the table, turn off **Display Equation** in the **Plot** menu.

Changing the Appearance of the Curve Fit Line

The final step will be to change the line style and width of the curve fit line.

1. Triple-click the marker in the legend (or choose **Plot** > **Plot Style**).
2. Click **Curve Fit**. This will allow you to control the appearance of the curve fit line.
3. Select a different line style and line width for the curve fit. Click **OK** to apply the changes.

   Note: Depending on the line width selected, you may not notice a difference on the screen. However, you will notice a difference when the plot is printed.

2.5.4 Modifying Data in a Saved Plot

This example shows how to modify a data point in a saved plot and have the plot and any curve fits automatically updated.

Extracting the Data

To get started, we need to open a saved plot and extract the data. To do this:

1. Choose **File** > **Open** and open the **Sample Plot** file (located in the **Plots** folder, within the **Examples** folder).
2. With the plot frontmost, choose **Plot** > **Extract Data**. The original data used to create the plot is displayed. The title of the window begins with the same name as the original data file. Additionally, a date and time stamp is appended to the name, identifying when the data was archived in the plot.

Changing the Data and Updating the Plot

Now we can make changes to the data and have the plot updated.

1. Turn on **Auto Link** in the **Plot** menu. When this command is active, you can make changes to the data and have the plot automatically updated after each individual change.
2. Change the value in the first row of column 1 from 78.5 to 100.
3. Use one of the arrow keys to move to another cell to activate the Auto Link feature. The plot and curve fit are automatically updated to reflect the modified data value.

Changing Multiple Data Points

If you need to add or modify multiple data points, it may be more efficient to use the **Update Plot** command (**Plot** menu) because Auto Link causes the plot to update after each change. In this case, turn off **Auto Link**, make any changes to the data, and choose the **Update Plot** command. The plot is updated to reflect all of the changes at once.

Working with Data Windows

Chapter 3

This chapter explains how to:

- Create a new data window.
- Use the keyboard to move through the data window.
- Select rows, columns, and a range of cells in the data window.
- Add, insert, and delete rows and columns.
- Edit column titles.
- Change the data format and width of columns.
- Use the Posted Note feature to make notes about the data.

3.1 Hiding, Displaying, and Closing Data Windows

A maximum of 160 data windows can be open at any one time, subject to memory limitations. You can view a list of all open data windows by choosing **Windows** > **Show Data**. Data windows that are currently displayed on the screen have their names in normal text. Names of hidden data windows appear in italic.

3.1.1 Creating a New Data Window

A blank data window can be created by choosing **File** > **New**. Any defaults that have been set using the **Column Format** command are used to create the new window. By default, data windows are created 10 columns by 100 rows in size and named **Data 1**, **Data 2**, and so on.

3.1.2 Changing the Active Window

To make a data window active, click anywhere in the desired data window or choose **Windows** > **Show Data** and select its name.

3.1.3 Hiding Data Windows

A data window can be hidden by choosing **Windows** > **Hide Window** and selecting its name. The next window in the list becomes active.

3.1.4 Displaying Data Windows

A hidden data window can be displayed using the **Show Data** command (**Windows** menu). The names of hidden data windows appear in italic. Select the name of the desired data window and it is brought to the front

3.1.5 Closing Data Windows

You can use the **Close All Data** command (**Windows** menu) to close all of the open data windows at once. Individual data windows can be closed by choosing **File** > **Close** or clicking the close box of each window. A dialog may be displayed, asking if you want to save the data window before closing.

Note: If you do not want to save the data, this dialog can be avoided by pressing **Shift** while either clicking the close box or choosing **File** > **Close/NoSave**.

3.2 Moving Around the Data Window

There are three methods available for moving around in the data window. You can use the mouse, the keyboard, or the **Go To Cell** command.

3.2.1 Using the Mouse

One of the easiest and most common methods to move around the data window is to use the mouse. With the mouse, you can move to any cell in the data window. To move to a different section of the data window, use the horizontal and vertical scroll bars.

You can also use the Home button ( ) in the upper-left corner of the data window to return the window to its origin (row 0, column 0). This is useful for quickly returning to the start of the data window.

3.2.2 Using the Keyboard

There are a number of keys on the keyboard that let you move from one cell to another. When the edge of the viewing area is reached, the data window scrolls to display the next row or column. The keys and their effects are listed in the following table.

Key	Direction
Left/Right Arrow	Move one cell left/right.
Up/Down Arrow	Move one cell up/down.
Tab	Move one cell to the right.
Shift+Tab	Move one cell to the left.
Return	Move down one cell.
Shift+Return	Move up one cell.
Enter	Windows: Same as the Return key. Macintosh: Move one cell in the same direction as the last move.
Page Up/Down	Move one window view up/down.
Home	Move to top of column.
End	Move to bottom of column.

3.2.3 Using the Go To Cell Command

The **Go To Cell** command (**Data** menu) can be used to view a specific cell in the data window. Choosing this command displays the dialog in Figure 3-1 to enter the row and column numbers of the desired cell. When you click **OK**, the data window automatically scrolls to make that cell position visible.

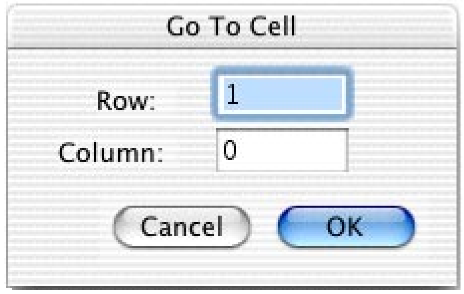


Figure 3-1 Go To Cell dialog

3.3 Switching Between Overwrite and Insert Mode

KaleidaGraph provides two commands, **Overwrite Mode** and **Insert Mode**, that control what happens when you try to edit or replace a data value. The active mode is preceded by a check mark in the **Data** menu.

If Overwrite Mode is active, cells are automatically selected when using the mouse or keyboard to move to a cell. Typing data into the current cell replaces any existing data.

If Insert Mode is active, clicking a cell does not automatically select the cell. Instead, a cursor is placed in the cell to edit the current value. To select a cell in Insert Mode, double-click the active cell.

3.4 Making Selections in the Data Window

This section explains how to select rows, columns, a range of cells, and all of the cells in the data window.

3.4.1 Selecting Rows

A single row can be selected by clicking its row number.

To select a range of rows, do one of the following:

- Click the number of the first or last row that you want and drag to complete the selection.
- Click the first row to be included, press **Shift**, and click the last row. You can use the scroll bars to move to the last row.

3.4.2 Selecting Columns

A single column can be selected by clicking the column title.

To select a range of columns, do one of the following:

- Click the column title of the first or last column and drag to complete the selection.
- Click the first column to be included, press **Shift**, and click the last column. You can use the scroll bars to move to the last column.

3.4.3 Selecting a Range of Cells

To select a range of cells, drag the mouse until all of the cells you want are selected. If some of the cells are not visible, keep dragging and the data window will scroll automatically.

To select a large block of data, click the first cell of the block, press **Shift**, and click the cell in the opposite corner of the block. You can use the scroll bars to move to the second cell.

3.4.4 Selecting the Entire Window

To select all of the cells in the data window, choose **Edit** > **Select All**.

3.5 Adding and Deleting Rows and Columns

By default, data windows are created with 10 columns and 100 rows. It is very easy to increase the number of rows and columns in the data window.

3.5.1 Adding Rows and Columns

Adding Rows

To add more rows to the data window, choose **Data** > **Add Rows**. This command displays the dialog in Figure 3-2 to enter the number of rows to be added to the data window. The number entered is rounded up to the next multiple of 100.

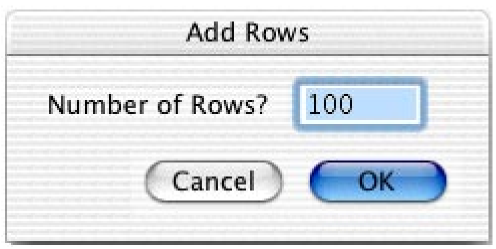


Figure 3-2 Add Rows dialog

Note: If the active cell is in the last row of the data window, pressing the **Enter** (Windows), **Return** (Macintosh), or **Down Arrow** key automatically adds 100 rows to the data window.

Adding Columns

To add more columns to the data window, do one of the following:

- Choose **Data** > **Append Columns** and enter the number of columns to be appended to the data window.
- Press the **Right Arrow** key (if the active cell is in the last column of the data window).
- Click the **Add** button in the Column Format dialog (**Data** menu).

3.5.2 Inserting Rows and Columns

Inserting Rows

Rows can be inserted anywhere in the data window. A row can be inserted in a single column or in a range of columns.

To insert a row:

1. Make a selection in the data window. The row is inserted above the selection.
2. Choose **Data** > **Insert Row**. A row of blank cells is inserted into the data window.

Inserting Columns

To insert a column into the data window:

1. Select a column in the data window.
2. Choose **Data** > **Insert Column**. A blank column is inserted before the selected column in the data window.

Columns can also be inserted by pressing **Alt** (Windows) or **Option** (Macintosh) and clicking a column. A new column is inserted before the selected column.

3.5.3 Deleting Rows and Columns

Deleting Rows

Rows can be deleted from the data window by making a selection and choosing **Data** > **Delete Row**. The selection is deleted and the remaining data shifts up to take the place of the deleted data.

Deleting Columns

Columns can be deleted from the data window by making a selection and choosing **Data** > **Delete Column**. The selection is deleted and the remaining data shifts over to take the place of the deleted data.

3.6 Formatting Data Columns

Each column in the data window can contain numeric, date, time, or text data. KaleidaGraph supports six different data types in the data window: Float, Double, Integer, Date, Time, and Text. Most of these data types also have several formats available for displaying the data. An individual column can contain only one data format. The properties of each column are controlled by the settings in the Column Format dialog, shown in Figure 3-3.

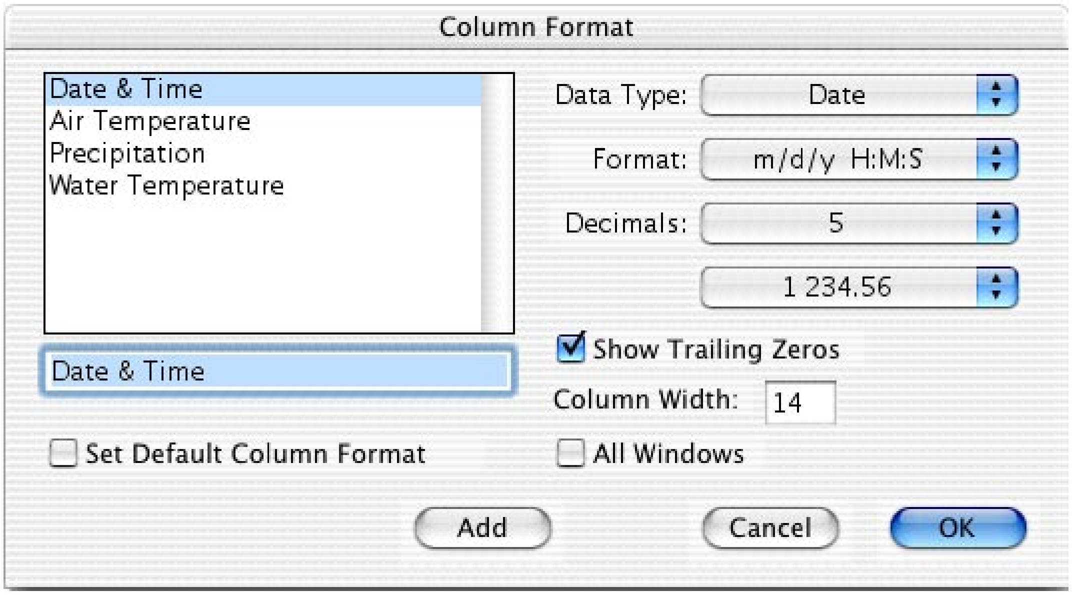


Figure 3-3 Column Format dialog

3.6.1 Editing Column Titles

To change the title of a column:

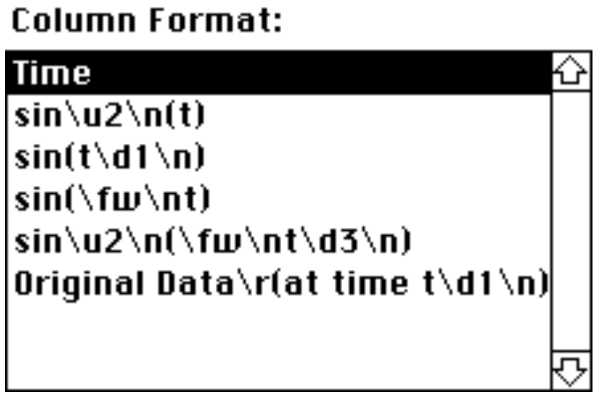
1. Double-click the title of the column or select the column and choose **Data** > **Column Format**. Either method displays the Column Format dialog. The column that you select in the data window is preselected in this dialog.
2. Change the column name in the text field. The change takes effect as soon as a different column is selected or the dialog is closed by clicking **OK**.

Adding Superscripts, Subscripts, Alternate Fonts, and Line Breaks to Column Titles

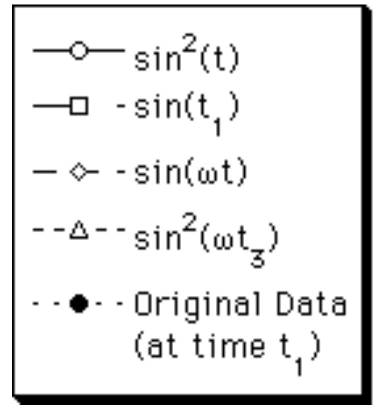
Superscripts, subscripts, and an alternate font can be used in the column titles by preceding the characters to be altered with **\u** (up), **\d** (down), and **\f** (alternate font), respectively. To return to the default attributes, use **\n** (normal).

Multi-line titles can be created by placing **\r** where the line break should occur. Any characters following **\r** will be placed on a separate line.

Note: The column titles in the data window do not display the superscripts, subscripts, alternate fonts, or line breaks. The \n, \f, \u, \d, and \r characters are mixed in with the normal text. However, when you create a plot, these characters are converted to obtain the desired results, as shown in the following figures.



Column title list



Plot legend

3.6.2 Changing the Data Format

There are several options in the Column Format dialog that control how the data is displayed. You can change the data format on a single column or several columns at once.

To change the format of a column:

1. Double-click the title of the column or select the column and choose **Data** > **Column Format**. Either method displays the Column Format dialog. The column that you select in the data window is preselected in this dialog.
2. Select one or more columns to be formatted. Groups of columns can be selected by clicking one title and dragging to another title.

   Windows: To select nonadjacent columns, press **Shift** or **Ctrl** and click each column name. To deselect a column, press **Ctrl** and click the title.

   Macintosh: To select nonadjacent columns, press **Shift** or ⌘ and click each column name. To deselect a column, press ⌘ and click the title.
3. Make selections for the data type, format, number of significant digits, trailing zeros, and decimal separator.
4. When you are finished, you can select more columns to format or click **OK** to return to the data window.

3.6.3 Changing Column Widths

To change the width of the columns in the data window:

1. Double-click the title of a column or select a column and choose **Data** > **Column Format**. Either method displays the Column Format dialog.
2. Change the value in the **Column Width** field. The value in this field defines the column width for all columns in the active data window. There is no way to set different column widths in the same data window.
3. To apply the current column width to all open data windows, select the **All Windows** check box.
4. When you are finished, click **OK** to return to the data window.

3.7 Making Notes about the Data

Each data window in KaleidaGraph provides a text editor where you can make notes about the data set. The text editor is displayed by either choosing **Data** > **Posted Note** or clicking the Posted Note button ( ) in the data window. This information is unique for each data window and is saved as part of the data file.

Working with Data

Chapter 4

This chapter explains how to:

- Enter data.
- Generate a data series.
- Import text files.
- Cut, copy, paste, and clear data.
- Sort data.
- Exchange rows and columns.
- Execute macros.
- Save and print data.

4.1 Entering Data

You can enter numeric, date, time, or text data into the data window. Before you enter data, make sure you know how to move around the data window and how to change the format of a column. Moving around the data window is covered in Section 3.2; changing the data format is covered in Section 3.6.

4.1.1 Entering Numeric Data

The Column Format dialog (**Data** menu) provides three data types for displaying numeric data. They are:

- **Float -** Accurate for numbers containing up to seven digits.
- **Double** - Accurate for numbers containing up to 16 digits.
- **Integer** - Displays the numbers as integers.

To enter numeric data:

1. Double-click the title of a column or select a column and choose **Data** > **Column Format**. Verify that the correct data type and format are selected for the column.
2. After you leave the Column Format dialog, use the mouse or arrow keys to select a cell.
3. Type the data value. If you make a mistake, press **Backspace** (Windows) or **Delete** (Macintosh) to remove incorrect characters.

   Note: To enter numbers in scientific notation, enter the number, followed by an **e** and the power (for example, 1.23e–4).
4. Press **Enter** (Windows) or **Return** (Macintosh) to move down one row, or use the arrow keys to move around the data window.
5. Repeat steps 3 and 4 until all of the data is entered.

Note: If KaleidaGraph detects any characters that are not associated with the selected data type, a beep sounds when you move to the next cell. Examples of incorrect characters for numeric data include: text, multiple decimal points, and separators for date and time numbers.

4.1.2 Entering Date or Time Data

To enter date or time data:

1. Double-click the title of a column or select a column and choose **Data** > **Column Format**.
2. Choose either **Date** or **Time** from the **Data Type** pop-up menu.
3. Choose the desired format from the **Format** pop-up menu and click **OK**.
4. Use the mouse or arrow keys to select a cell.
5. Type the date or time data. If you make a mistake, press **Backspace** (Windows) or **Delete** (Macintosh) to remove incorrect characters.

   Note: When entering two digit years less than 40, the **Promote 2 digit dates less than 40** option in the Preferences dialog determines whether the dates are interpreted as 19xx or 20xx.
6. Press **Enter** (Windows) or **Return** (Macintosh) to move down one row, or use the arrow keys to move around the data window.
7. Repeat steps 5 and 6 until all of the data is entered.

If you import date or time data from a file, the format must closely match the actual format shown to enable KaleidaGraph to recognize the column properly. However, if you are typing the data or pasting the data from the Clipboard (to a column that has been set to a date or time format), variations from the exact format are allowed. Valid separators for the date and time formats include: slashes (/), colons (:), commas, and spaces.

When importing dates from another program, the **Default Date Format** setting in the Preferences dialog (**File** menu) determines how the dates are interpreted by KaleidaGraph.

4.1.3 Entering Text Data

To enter text data:

1. Double-click the title of a column or select a column and choose **Data** > **Column Format**.
2. Choose **Text** from the **Data Type** pop-up menu and click **OK**.
3. Use the mouse or arrow keys to select a cell.
4. Enter the text data. A maximum of 50 characters can be entered for each text label. If you make a mistake, press **Backspace** (Windows) or **Delete** (Macintosh) to remove incorrect characters.
5. Press **Enter** (Windows) or **Return** (Macintosh) to move down one row, or use the arrow keys to move around the data window.
6. Repeat steps 4 and 5 until all of the data is entered.

Note: Any data that is entered into a Text column appears left-justified.

4.1.4 Missing Values

There may be times when data values are missing in a series of data. In the data window, KaleidaGraph represents a missing data value with an empty cell, regardless of the data type.

When plotting, if the independent variable (X) contains missing values, KaleidaGraph searches for the first and last cells that contain data and plots all values in between, ignoring any empty cells. If the dependent variable (Y) contains missing values, the empty cells are ignored when plotting.

There is an option in the Plot Extras dialog (**Format** menu) that controls how missing values are treated in Line, Double Y, Double X, Double XY, High/Low, Step, X-Y Probability, and Polar plots. When **Missing Data Breaks** is selected, a missing data point in an X or Y variable causes a break in the line being plotted. Otherwise, the line is continuous and the missing data points are ignored.

4.2 Generating a Data Series

There are several methods that can be used to generate a series of data. Some of the default macros that come with KaleidaGraph can be used to create a data series. The Formula Entry window contains commands which can also be used to place a series into a column. The only problem is these methods do not give you very much control over the type of series that is generated.

The one method that does give you complete control is the **Create Series** command from the **Functions** menu. This command allows you to specify an initial value, an increment, a multiplier, and a final value for the series.

You can create a numeric series in any column that contains a Float, Double, or Integer data format. A date or time series can be created in any column that contains a Date or Time format.

Note: It is not possible to generate a series in a Text column.

To create a series:

1. Select a column or a range of columns in the data window.
2. Choose **Functions** > **Create Series**. A dialog similar to one of the following appears, based on the format of the column.

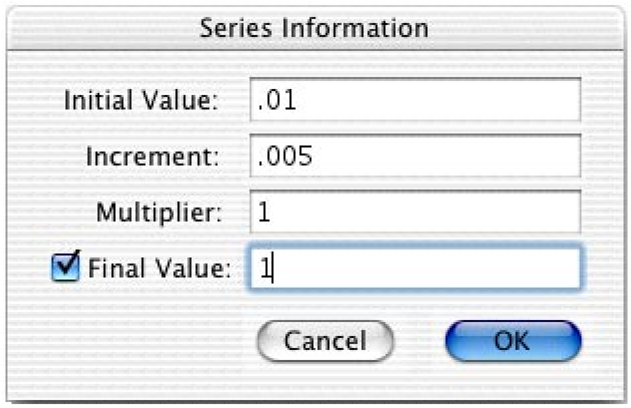


Figure 4-1 Create Series dialog (Numeric format)

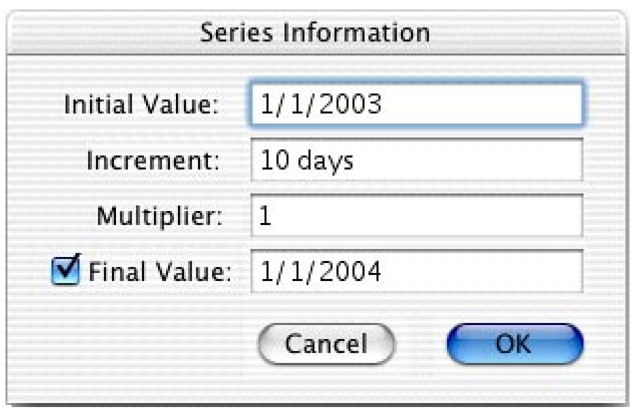


Figure 4-2 Create Series dialog (Date format)

3. Enter values into the appropriate fields of the dialog and click **OK** to generate the series.

4.3 Importing Text Files

KaleidaGraph can open text files that are formatted in a number of different ways. This allows you to read data created in another program and use that data in KaleidaGraph. The only requirements are that the data must be saved as a text file and the information must be arranged in a repeating pattern.

4.3.1 Text File Input Format Dialog

Whenever you select a text file to be read into KaleidaGraph, the dialog in Figure 4-3 is displayed. This dialog allows you to:

- Preview the file.
- Select the delimiter.

- Indicate the number of delimiters between columns.
- Specify the number of lines to skip at the beginning of a file.
- Select whether or not to read titles into the data window.

Note: If you are using the Macintosh version of KaleidaGraph, this dialog is not displayed when opening text files saved from KaleidaGraph. To display the dialog, press **Option** when opening these text files.

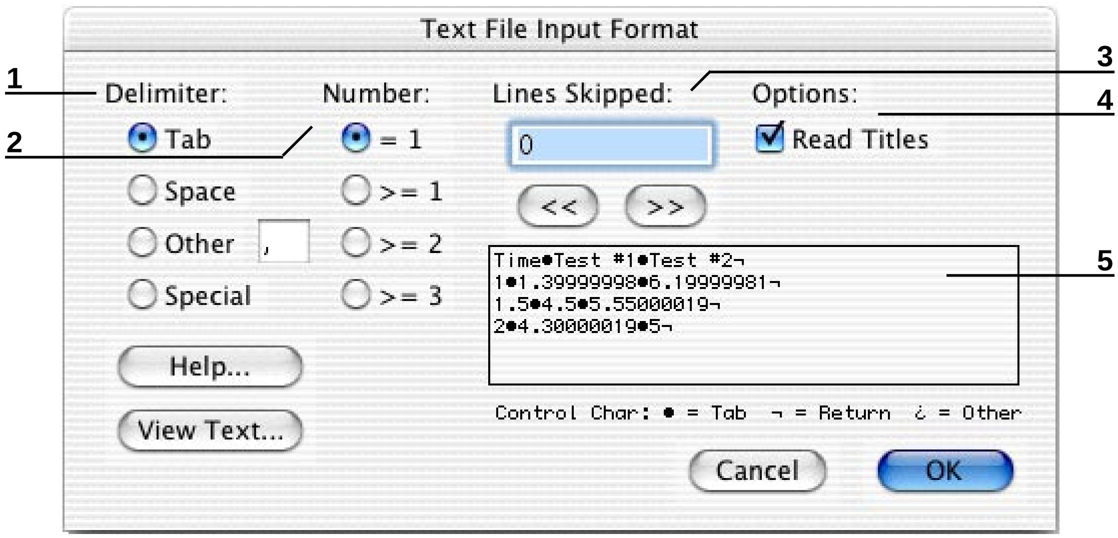


Figure 4-3 Text File Input Format dialog

The settings in the Text File Input Format dialog are discussed below:

1. Delimiter:

Delimiters are characters that separate the data values in a text file. The two common delimiters, **Tab** and **Space**, assume that the data contains numbers, dates, or text, and that it is organized in columns, separated by either tabs or spaces. A delimiter can also be user-defined by selecting **Other** and typing the desired delimiter. This delimiter is set to a comma by default.

The **Special** format can be used to control how the data file is imported into KaleidaGraph. The Special format is covered in Section 4.3.4.

2. Number:

This setting determines the number of delimiters (tabs, spaces, or user-defined) that are present between data values. The four options are described below:

- Select **= 1** if a single delimiter separates the data values.
- Select **>= 1** if one or more delimiters separate the data values.
- Select **>= 2** if two or more delimiters separate the data values. If less than two delimiters are present, they are treated as part of the same column.
- Select **>= 3** if three or more delimiters separate the data values. If less than three delimiters are present, they are treated as part of the same column.

3. **Lines Skipped:**
This setting determines the number of lines at the beginning of the file which are skipped before interpreting the rest of the data. As this number changes, the data displayed in the preview window adjusts to reflect the change. The skipped lines are automatically placed into the Posted Note, unless you are merging a text file with a data set that already contains information in the Posted Note.

4. **Options:**
When the **Read Titles** check box is selected, the first line in the preview window is used for the column titles.

5. **Preview Window**
This window displays 40 characters from the first four lines of the text file. To see a larger sample of the file, click **View Text**.

4.3.2 Importing Missing Data Points

Missing data points can be entered from text files. If **=1** is selected for the **Number** setting in the Text File Input Format dialog, whenever KaleidaGraph detects two consecutive delimiters, an empty cell is placed in the data window.

A period can also be used to represent an empty data cell. There is a mainframe program called SAS that uses the period as a default character for empty data cells.

4.3.3 Example - Reading Basic Formats

This section shows how to import a simple tab-delimited text file. One of the text files that comes with KaleidaGraph is used as an example.

To import this example data set:

1. Choose **File** > **Open**.
2. Locate and open the **Data** folder, which is located in the **Examples** folder.
3. Double-click the file **Text Example**. When KaleidaGraph determines that the file is a text file, the dialog in Figure 4-4 appears.

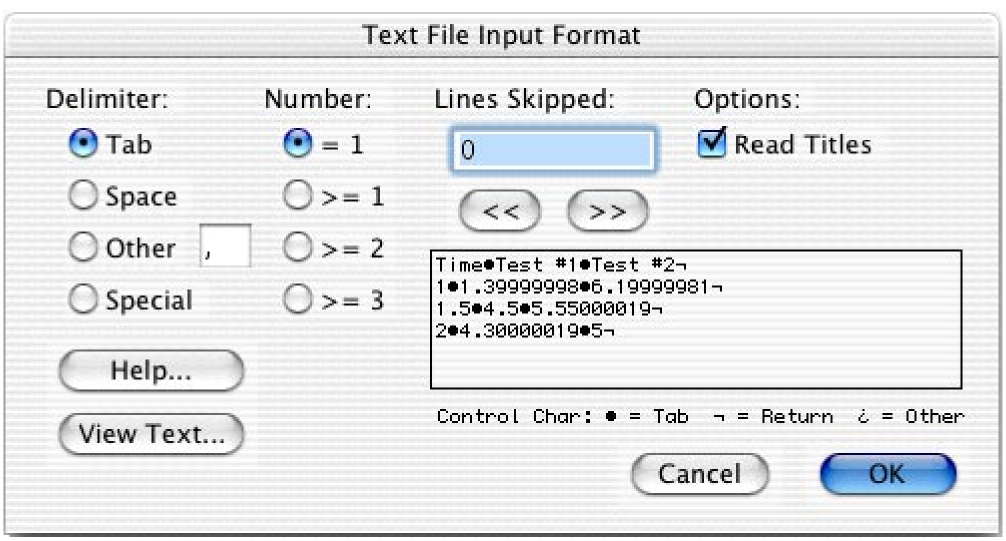
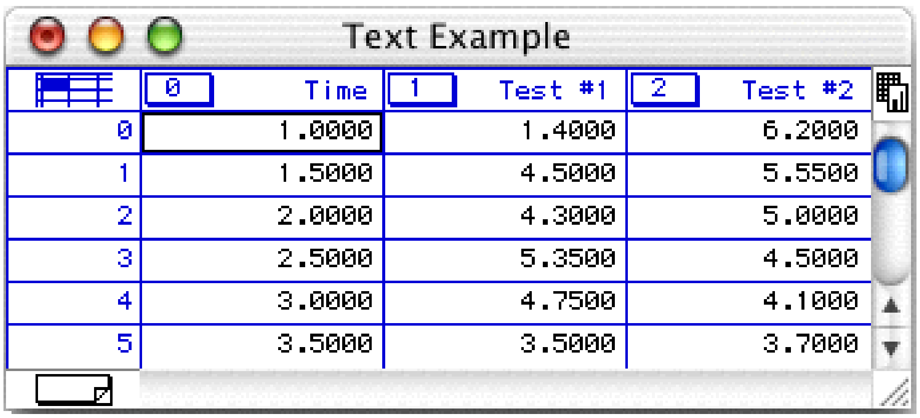


Figure 4-4 Text File Input Format dialog

Looking at the preview window of this dialog, you can see that:

- This particular file is tab-separated.
- A single tab is present between the values.
- None of the lines need to be skipped.
- The first line of the file should be used for the column titles.

4. Make the settings shown in the previous dialog and click **OK**. The text file is opened into a new data window. Figure 4-5 shows the data window that results from importing the example data set.


Text Example

	0 Time	1 Test #1	2 Test #2
0	1.0000	1.4000	6.2000
1	1.5000	4.5000	5.5500
2	2.0000	4.3000	5.0000
3	2.5000	5.3500	4.5000
4	3.0000	4.7500	4.1000
5	3.5000	3.5000	3.7000

Figure 4-5 Resulting data window

4.3.4 Using the Special Format

If the delimiter is set to **Special**, the dialog expands, as shown in Figure 4-6. The Special format requires that you define the format of the text file by specifying which fields to read and which fields to skip.

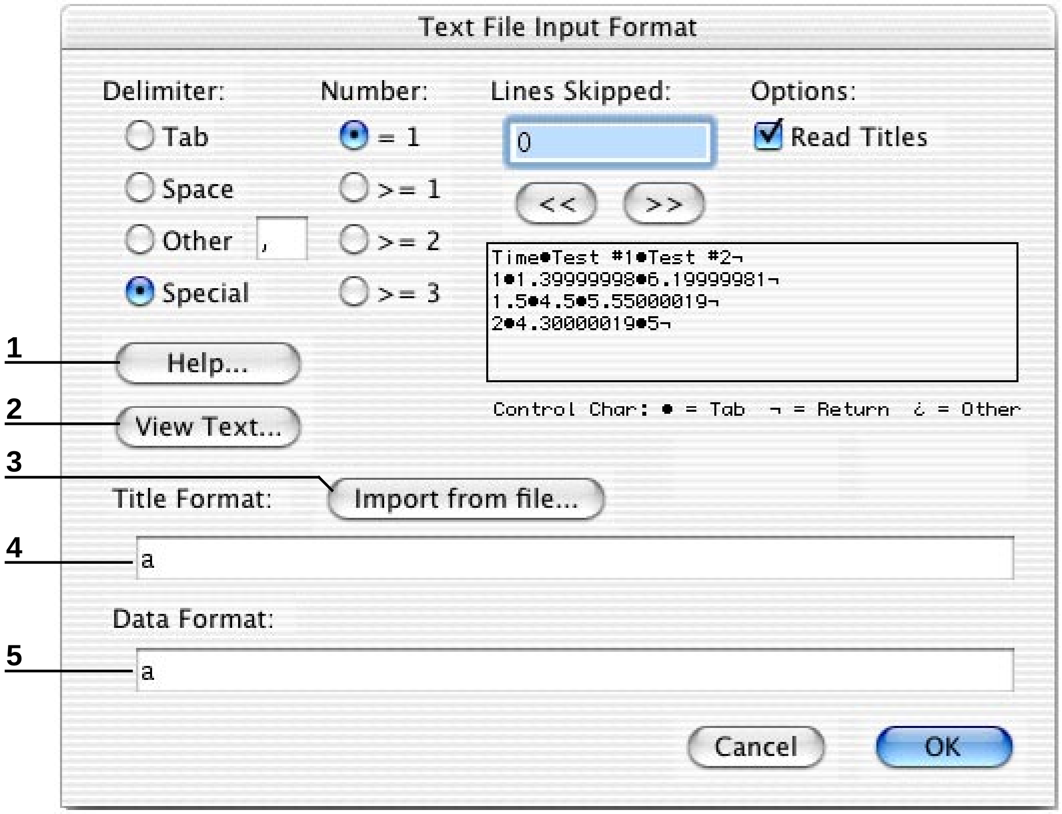


Figure 4-6 Expanded Text File Input Format dialog

1. **Help...**
   Clicking **Help** displays the definitions for the format control characters. These characters are used in the **Title Format** and **Data Format** fields. Some of these definitions display a lowercase letter and an uppercase letter enclosed in parentheses. The lowercase letter should be used to read that type of value. The uppercase letter should be used to skip that type of value.

2. **View Text...**
   Clicking **View Text** displays a larger portion of the text file. This is useful when determining the format of a text file. The sole purpose of this window is to view the data. It is not possible to edit the text in this window before importing it into the data window. The only commands you should use are **Close** and **Cancel**. There is no difference between the functionality of these commands.

3. **Import from file...**
   Clicking this button displays a text editor which can be used to enter, open, and save frequently used formats. The first line in the editor defines the title format and the second line defines the data format.

4. **Title Format:**
   The **Title Format** field specifies which labels are to be skipped or read into the data window. It is only necessary to enter a format in this field if **Read Titles** is selected. The title format does not have to be the same as the data format; however, you must read one title for each column being imported.

5. **Data Format:**
   This field defines which data is to be skipped or read into the data window.

4.3.5 Example - Reading Special Formats

This section shows an example of importing a text file that contains unwanted data values. One of the text files that comes with KaleidaGraph is used as an example.

To import this example data set:

1. Choose **File** > **Open**.
2. Locate and open the **Data** folder, which is located in the **Examples** folder.
3. Double-click the file **Measurements**. When KaleidaGraph determines that the file is a text file, the Text File Input Format dialog appears.
4. Click the **Special** button. The dialog expands so you can enter the title and data formats. The dialog should look like Figure 4-7 at this point.

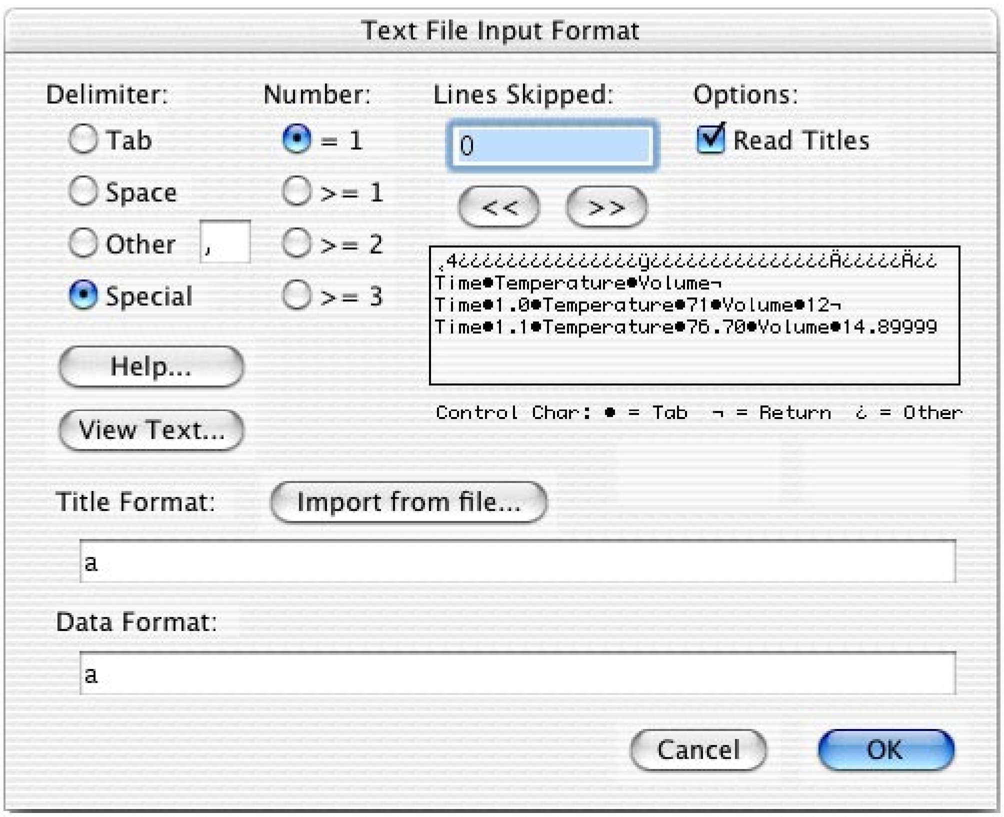


Figure 4-7 Expanded Text File Input Format dialog

Looking at the preview window of this dialog, you can see that:

- The file begins with an unusable line which must be skipped.
- There is a title line containing three labels.
- The data is spread across six columns. Three of the columns contain labels and should be skipped. The other three contain values which should be read as one fixed width field (Time values) and two variable width fields (Temperature and Volume values).

5. Using the above information, you can now specify the format of the titles and the data. Figure 4-8 shows how the dialog should appear after making the following selections.
   - **Lines Skipped** should be 1 to skip the first line.
   - **Read Titles** should be selected to read in the column titles.
   - The **Title Format** should be either **a a a** or **3(a)**. Either one of these formats reads in three labels, separated by white space.
   - The **Data Format** should be either **A f3 A v A v** or **A f3 2(A v)**. Using either one of these formats, the three columns of labels are skipped and only the three columns of data are read into the data window.

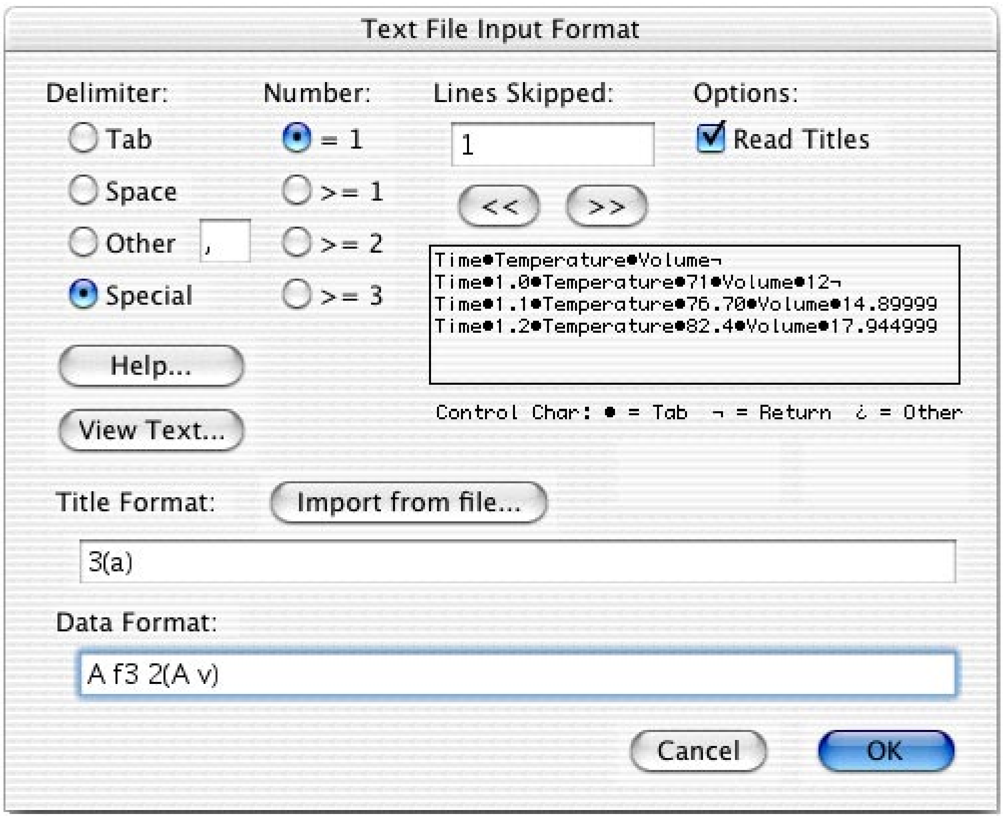


Figure 4-8 Finished Text File Input Format dialog

6. Click **OK** to import the text file. The resulting data window is shown in Figure 4-9.

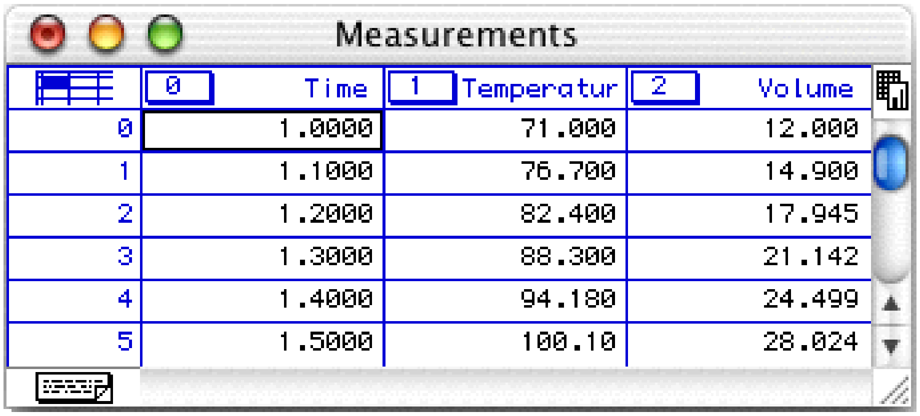


Figure 4-9 Resulting data window

4.3.6 Special Format Examples

The following table contains some sample data format strings that may be used for the **Special** format.

Data Format	Effect
w v v v	Skips any white space at the beginning of each row before reading in the variable width numbers.
v v v c	Reads the first three columns of variable width numbers and skips everything else.
v V v c	Reads the first and third columns of variable width numbers and skips everything else.
4(v) c 4(v)	Takes a data set with four columns and places it into eight columns in the data window.
v v v \	Takes a data set with any number of columns containing variable width numbers and places it into three columns in the data window.
3 (y f6 b b)	Repeats the pattern in the parentheses three times. After executing this format, the data window would contain 12 columns.
u, 8(v)	Reads eight comma-separated columns containing variable width numbers.

4.4 Cutting, Copying, Pasting, and Clearing Data

There are several commands in the **Edit** menu that can be used to edit the contents of the data window. Using these commands, you can cut, copy, paste, and clear data selections.

4.4.1 Binary Mode/Text Mode Setting

This command controls whether the **Cut** and **Copy** commands use binary or text format to copy large amounts of data to the Clipboard. When copying 10000 cells or less, a binary and text representation of the data are copied to the Clipboard. This occurs regardless of which mode is currently active.

Binary editing is much faster than text editing and allows you to quickly move large amounts of data. Use Text mode if you need to copy more than 10000 cells and paste them into the plot window or another program.

To view the active mode, click the **Edit** menu. The active mode is displayed in the menu and has a check mark in front of it. To switch between modes, choose the active mode.

4.4.2 Cutting Data

Choose **Edit** > **Cut** to remove the data selection from the window and place it on the Clipboard, replacing the previous contents, if any.

4.4.3 Copying Data

Choose **Edit** > **Copy** to place a copy of the data selection on the Clipboard, replacing the previous contents, if any.

4.4.4 Pasting Data

Choose **Edit** > **Paste** to place the Clipboard's contents into the data window. You can also paste data into the plot window, layout window, or another program.

4.4.5 Clearing Data

Choose **Edit** > **Clear** to delete all of the cells in the selection without disturbing the contents of the Clipboard. To clear a cell's contents without deleting the cell, use the **Clear Data** command, the **Backspace** key (Windows), or the **Delete** key (Macintosh).

4.4.6 Including Column Titles when Editing Data

To include the column titles when editing, press **Shift** while choosing the **Cut**, **Copy**, **Paste**, or **Clear** command from the **Edit** menu.

4.5 Sorting Data

KaleidaGraph can perform ascending and descending sorts on a column, a range of columns, or any data selection. Sorts can be applied to selections that contain any data format.

The sort commands are located in the **Functions** menu. The **Ascending Sort** command sorts the data from low value to high value. The **Descending Sort** command sorts from high value to low value. Both of these commands use the same dialog, shown in Figure 4-10. This dialog allows you to select which columns to reorder during the sort. Any columns you select before entering this dialog are preselected for you.

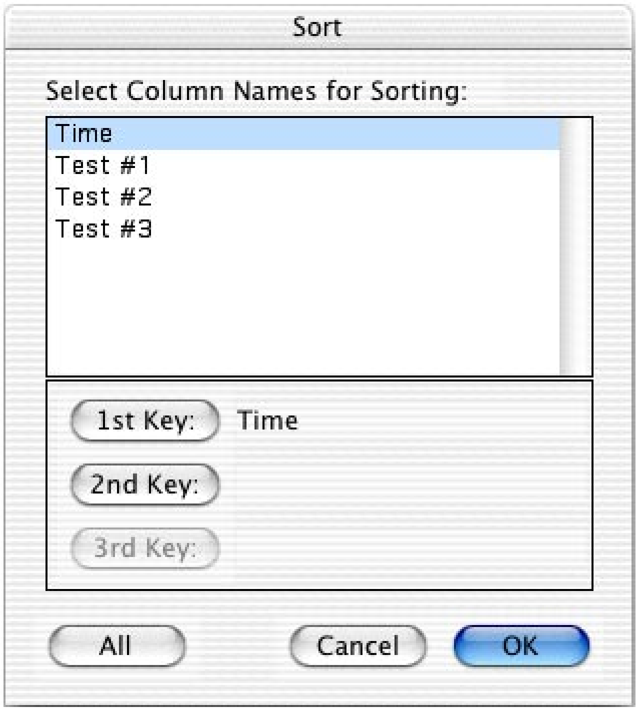


Figure 4-10 Sort dialog

To sort a range of data:

1. Select the data to be sorted. The selection can contain a few cells, columns, or the entire data window
2. Choose either **Ascending Sort** or **Descending Sort** from the **Functions** menu. The Sort dialog appears, with the names of any selected columns already highlighted. The first column in the selection is automatically assigned as the **1st Key**.
3. To perform a multi-level sort, click the name of the column to be used as the second sort key and click the **2nd Key** button. Repeat this step for the **3rd Key**, if necessary.
4. Select all of the columns to be sorted.
   - To select individual columns, press **Shift**, **Ctrl** (Windows), or ⌘ (Macintosh) and click the column name.
   - To select a range of columns, drag with the mouse.
   - To select every column in the data window, click **All**.
   - To deselect a column, press **Ctrl** (Windows) or ⌘ (Macintosh) and click the column name.
5. Click **OK** to perform the sort.

4.6 Transposing Data

You can exchange rows and columns in the data window. When you use the **Transpose** command, the columns are converted to rows and the rows to columns. For example, if a selection contains four columns and 128 rows, it contains 128 columns and four rows after transposing the data.

To switch rows and columns:

1. Make a selection in the data window.
2. Choose **Functions** > **Transpose**. The selected data is transposed and displayed in the data window

Figure 4-11 shows a sample data window. Figure 4-12 shows the same data window after selecting all of the columns and using the **Transpose** command. Notice that the data originally in column 0 is now in row 0 after being transposed.

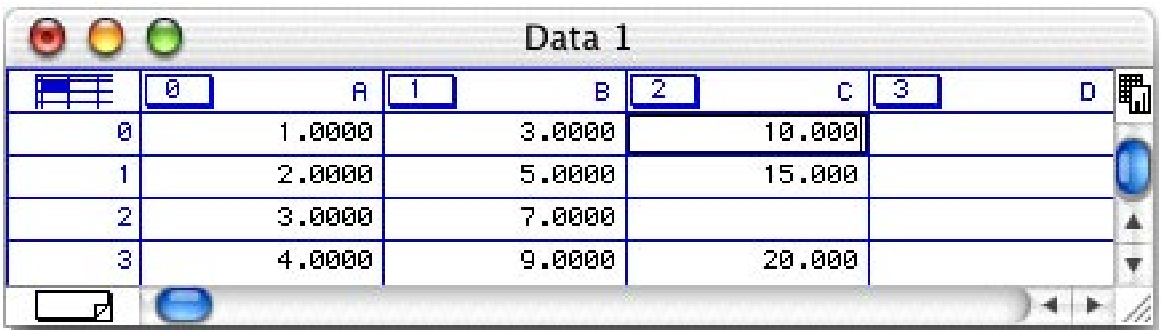


Figure 4-11 Before choosing Transpose

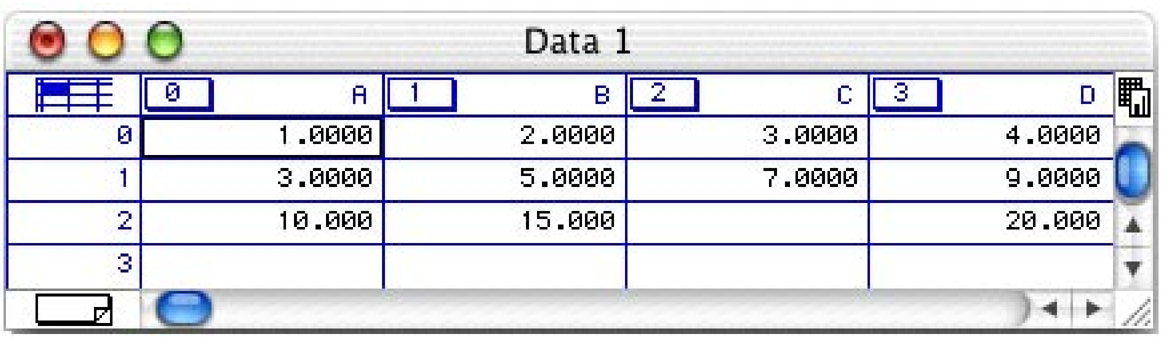


Figure 4-12 After choosing Transpose

Note: An error message is displayed if either of the following occurs:

- You attempt to transpose a selection containing text data.
- Your selection contains more than 1000 rows, because the data window is limited to a maximum of 1000 columns.

4.7 Masking Data

Masking data provides a way to leave data in the data window but not have it used in any calculations, curve fits, or plots. Any data that is masked is covered with a red stipple pattern in the data window. Figure 4-13 shows a data window containing masked data cells.

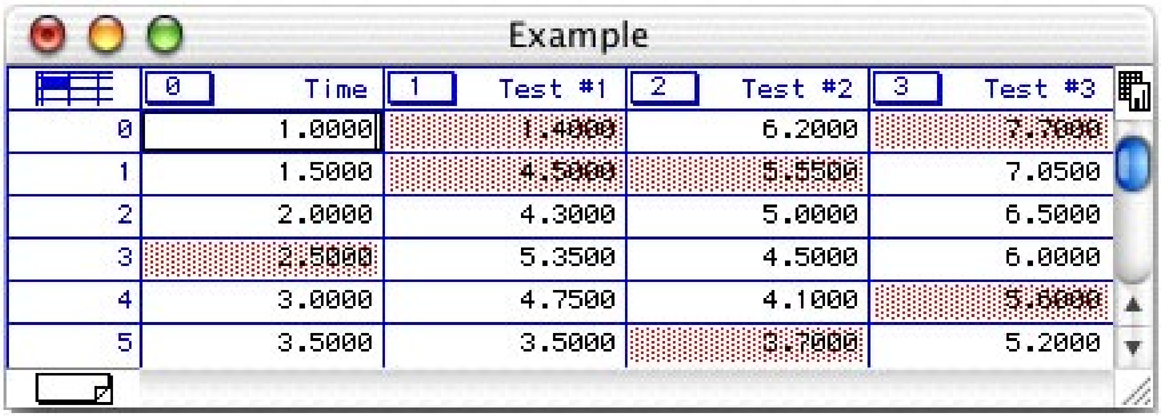


Figure 4-13 Data window containing masked data

Data can be masked using any of the following: the **Mask** command (**Functions** menu), the Data Selection tool from the toolbox, macros, and Formula Entry. Using Formula Entry to mask data is covered in Section 6.4. The other methods are discussed below.

Note: The **Unmask** command (**Functions** menu) can be used to unmask selected data cells.

Mask Command

To mask data with the **Mask** command:

1. Select the cells to be masked.
2. Choose **Functions** > **Mask**. A stipple pattern appears in each of the masked data cells.

Data Selection Tool

You can use the Data Selection tool to select a portion of the plotted data for further plotting and analysis. The data that is removed from the plot becomes masked in the data window. The use of this tool is discussed in Section 9.4.

Note: Double-clicking this tool unmasks the data and returns the plot to its original state.

Macros

KaleidaGraph contains three default macros in the **Macros** menu that mask data upon execution. These macros are **Filter**, **Simplify**, and **Invert Mask**. Refer to the online help for information on these macros.

4.8 Executing Macros

Macros can be used to perform a wide variety of data operations, including those required for data analysis. KaleidaGraph contains several default macros in the **Macros** menu. Some of these macros are used to create data series and others are used to manipulate and analyze data. For more information about a particular macro or the Macro Calculator, refer to the online help. There is also a PDF file on the CD-ROM that explains how to write macros and provides some example macros.

To execute a macro:

1. Open the data window to be operated on by the macro. A sample data set is shown in Figure 4-14.

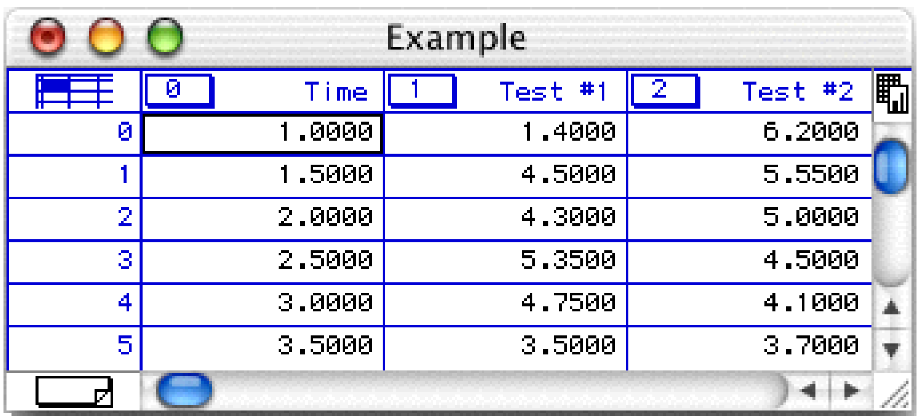


Figure 4-14 Sample data window

2. Choose a macro from the **Macros** menu. The Integrate - Area macro is used for this example. Choosing a macro from the menu automatically causes it to be executed. Some macros may prompt for specific information during execution. The following dialogs are displayed during execution of the Integrate - Area macro.

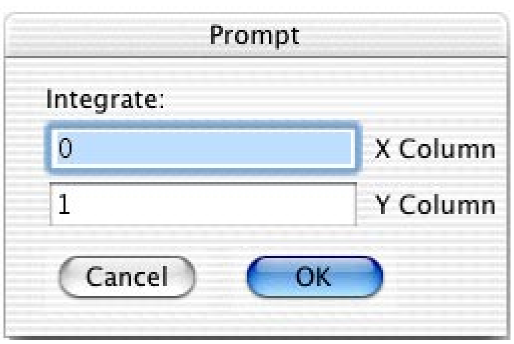


Figure 4-15 First input dialog

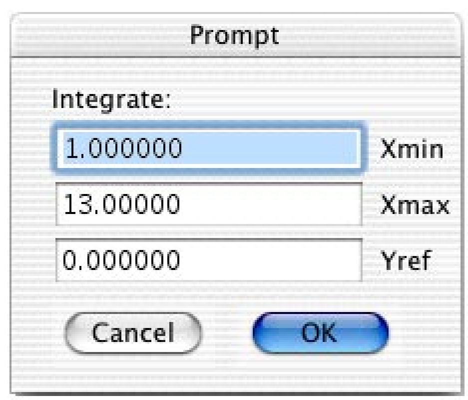


Figure 4-16 Second input dialog

3. Type the appropriate values into any dialogs that appear. Click **OK** to continue execution or click **Cancel** to stop the macro. Figure 4-17 shows the results of executing the Integrate - Area macro.

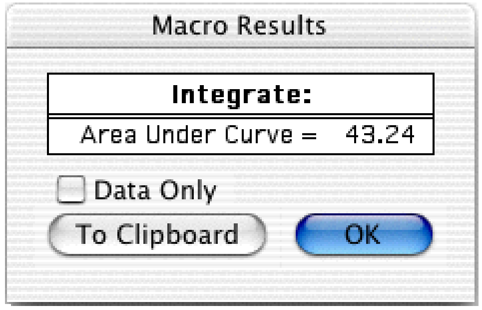


Figure 4-17 Macro results

4.9 Exporting Data

KaleidaGraph provides two methods for exporting data files. You can save the data file to disk (in either binary or text format) or you can print the data file.

4.9.1 Saving Data

To save the data to disk, choose **Save Data** or **Save Data As** from the **File** menu when the data window is active. If the **Save Data** command is used on a window that has not been saved before, a Save dialog appears, allowing you to specify the location and file name.

The **Save Data** and **Save Data As** commands always save the data in binary format, because this is considerably faster and more compact than text format. Binary files can only be read by KaleidaGraph.

If text format is desired, choose **File** > **Export** > **Tab-Delimited Text**. A Save dialog appears, allowing you to specify the location and file name for the text file.

4.9.2 Printing Data

To print the data set, choose **File** > **Print Data**. The Print dialog and its options depend on the printer and operating system.

If nothing is selected in the data window, KaleidaGraph prints the entire data set. A maximum of 15 columns can be printed on each page. Decreasing the width of the columns and scaling the output does not increase this limit.

Analyzing Data

Chapter 5

This chapter explains how to:

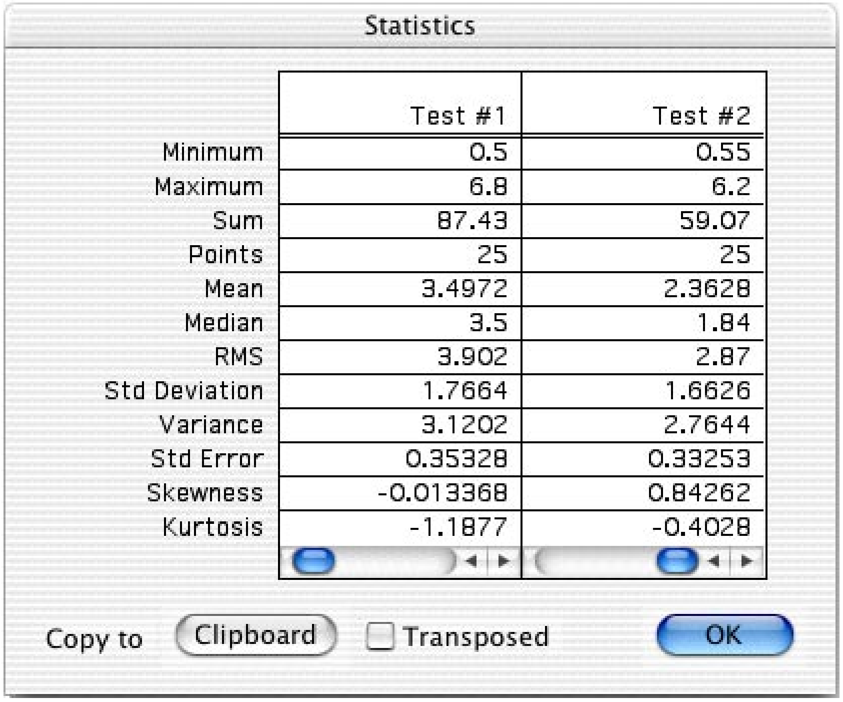
- View column statistics
- Place data into bins and export the results to create a Histogram, Step plot, or Spike plot.
- Perform tests to analyze single data samples.
- Perform tests to analyze two or more groups.
- Perform tests to analyze repeated measurements.
- Use a post hoc test after running a one way or one way repeated measures ANOVA.

5.1 Viewing Column Statistics

KaleidaGraph provides 12 different statistics to help analyze data. These statistics are automatically calculated for each column in the data window. If you need to calculate statistics across multiple columns of data, see Section 6.3 for details on using Formula Entry to do this. The equations used to calculate the statistics are listed in Section D.2.

To view statistics on a data set:

1. Select the data for which you want the statistics calculated. If you do not make a selection, the statistics are calculated for each column in the data window.
2. Choose **Functions** > **Statistics**. The Statistics dialog, shown in Figure 5-1, is displayed. This dialog has a split-screen display so that any two columns can be compared.
3. Click **Copy to Clipboard** if you want to paste this information into a data, plot, or layout window. You can also select the **Transposed** check box so that when the data is copied, each statistic becomes a column and the different variables become rows.
4. When you are finished, click **OK**.



	Test #1	Test #2
Minimum	0.5	0.55
Maximum	6.8	6.2
Sum	87.43	59.07
Points	25	25
Mean	3.4972	2.3628
Median	3.5	1.84
RMS	3.902	2.87
Std Deviation	1.7664	1.6626
Variance	3.1202	2.7644
Std Error	0.35328	0.33253
Skewness	-0.013368	0.84262
Kurtosis	-1.1877	-0.4028

Figure 5-1 Statistics dialog

5.2 Binning Data

Binning data involves counting the number of data points that fall within a certain range. The binned data distribution can be exported in any one of three formats for plotting or for annotating a plot.

5.2.1 Binning the Data

To bin the data:

1. Select the columns containing the variables to be binned.
2. Choose **Functions** > **Bin Data**. A dialog similar to Figure 5-2 appears.

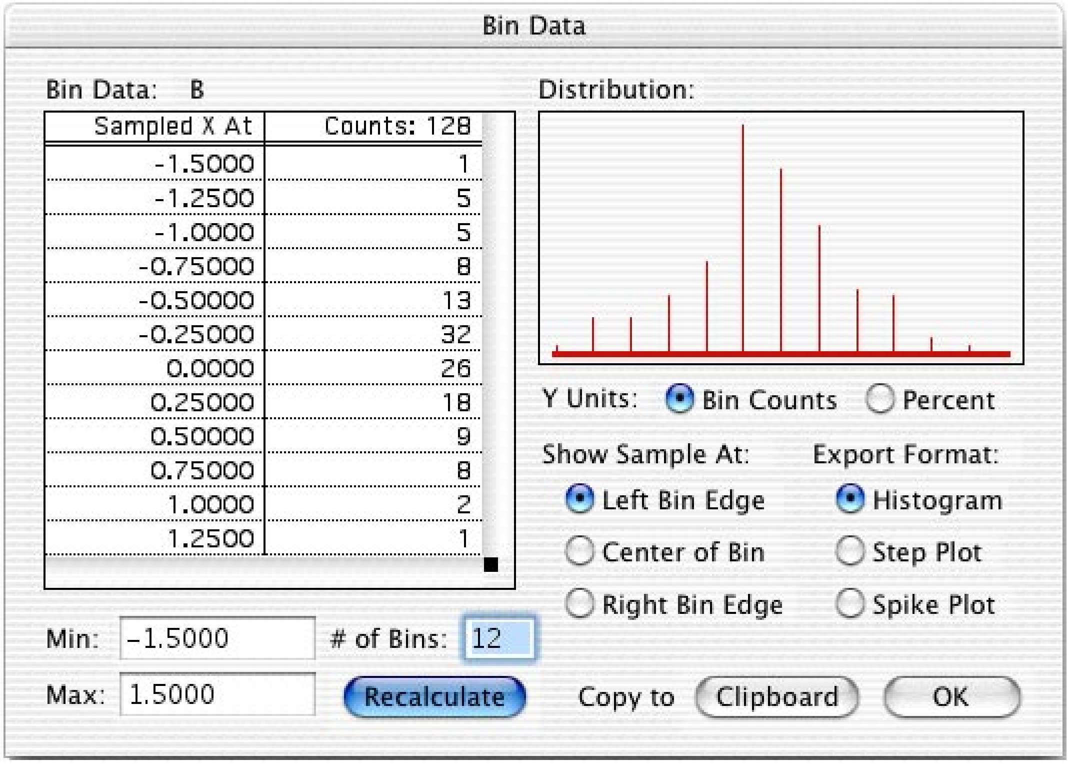


Figure 5-2 Bin Data dialog

3. Bin the variables by specifying the minimum and maximum values, number of bins, Y units, and where to show the counts. If you make any changes to the minimum, maximum, or number of bins, click **Recalculate** to update the count.
4. Select the appropriate **Export Format** for the type of plot to be created.
5. Click **Copy to Clipboard**.
6. Click **OK**.

5.2.2 Pasting the Results from the Bin Data Dialog

Once the data is binned, it can be pasted into a new data window.

1. Choose **File** > **New** to create a blank data window.
2. Choose **Edit** > **Paste**. The results obtained from binning the data are placed into the data window. Figure 5-3 shows binning results which were exported with a Histogram format.

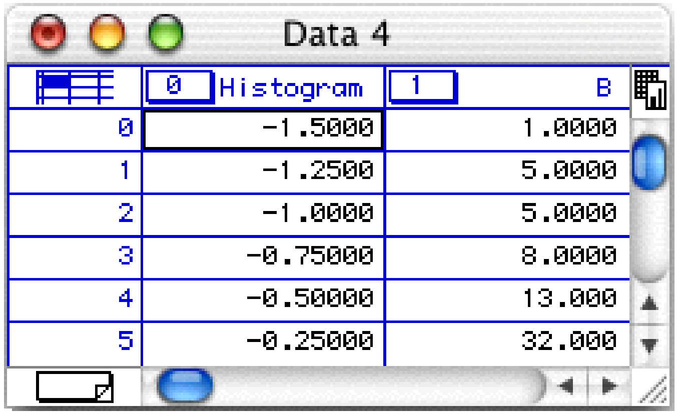


Figure 5-3 Results of binning the data

5.2.3 Creating the Plot

1. Choose the proper plot type from the **Gallery** menu.
   - **Histogram** - For a single variable, choose **Gallery** > **Bar** > **Column**. For multiple variables, choose **Gallery** > **Bar** > **Stack Column**.
   - **Step Plot** - Choose **Gallery** > **Linear** > **Line**.
   - **Spike Plot** - Choose **Gallery** > **Linear** > **High/Low**.
2. Select the variables to be plotted.
   - **Histogram** - Select **Histogram X** for the X variable and one or more binned columns for the Y variables.
   - **Step Plot** - Select **Step X** for the X variable and one or more binned columns for the Y variables
   - **Spike Plot** - Select **Spike X** for the X variable, plus a binned column and **Zero** for the Y variables.
3. Click **New Plot**. The following three figures show the different plot types that can be created. All three were generated from the same original data set, using the same settings in the Bin Data dialog. The only difference was the Export Format.

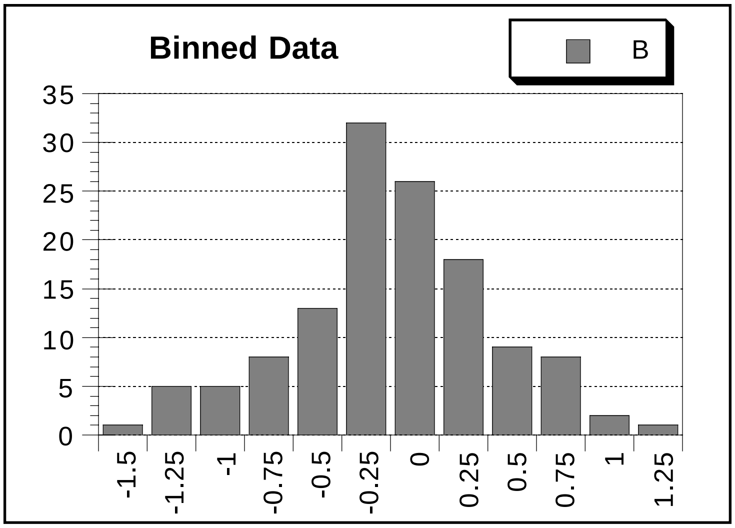


Figure 5-4 Histogram plot of binned data

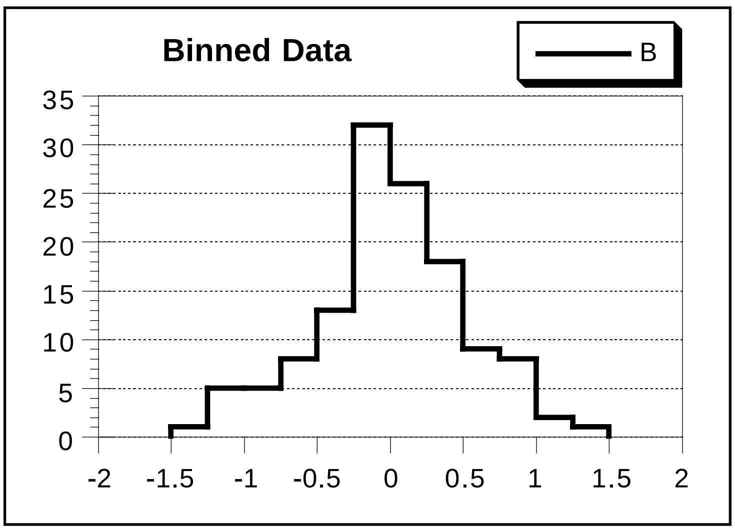


Figure 5-5 Step plot of binned data

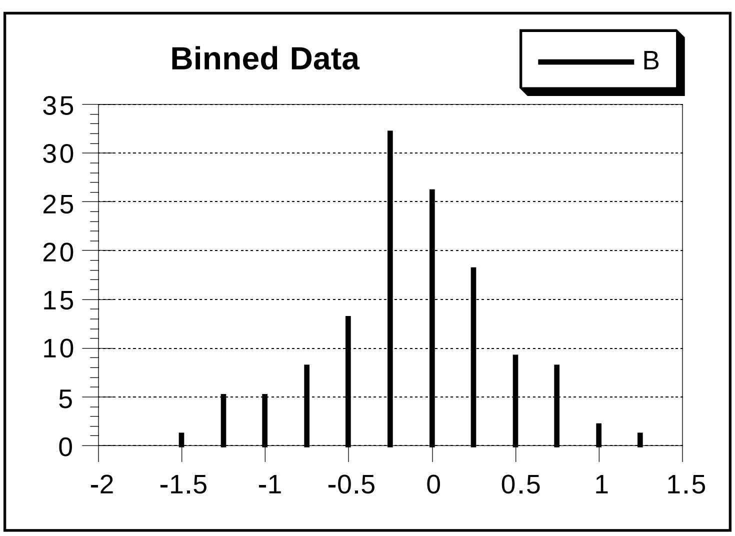


Figure 5-6 Spike plot of binned data

5.3 Choosing a Statistical Test

KaleidaGraph can be used to perform a variety of statistical tests. The type of test to use depends on the number of samples and the nature of the data.

- Use a **single sample test** to analyze one sample to a known or hypothesized value.
- Use a **group comparison test** to analyze two or more different samples for statistically significant differences.
- Use a **repeated measures test** to analyze the differences in two or more matched samples (such as the same individuals before and after one or more treatments).

The table below shows the proper test to use for each experiment and the type of data collected.

Type of Experiment	**Type of Data**	
	Numeric (normally distributed with equal variances)	Rank, score, or numeric (not normally distributed and/or with unequal variances)
Compare one sample to a hypothetical value	One sample t-test	Wilcoxon signed rank test
Compare two paired samples	Paired t-test	Wilcoxon matched pairs test
Compare two unpaired samples	Unpaired t-test	Wilcoxon-Mann-Whitney test
Compare three or more matched samples	One way or Two way repeated measures ANOVA	Friedman test
Compare three or more unmatched samples	One way or Two way ANOVA	Kruskal-Wallis test

5.4 Parametric and Nonparametric Tests

The statistical tests in KaleidaGraph can be divided into two categories: parametric and nonparametric.

Parametric tests assume the samples are drawn from a population that is normally distributed with equal variances. Examples of parametric tests include the t-test and analysis of variance (ANOVA). In general, you should use a parametric test if you are confident that the samples are drawn from a normally distributed population.

Nonparametric tests (or distribution-free tests) do not make any assumptions about the distribution of the data. These tests rank the data from low to high and analyze the rank. Examples of nonparametric tests include the Wilcoxon, Kruskal-Wallis, and Friedman tests. In general, you should use a nonparametric test if the data represents a rank or score, or if the measurements are drawn from a population that is not normally distributed

5.5 Analyzing One Sample

KaleidaGraph provides two tests that can be used to determine whether the mean or median of a single sample differs significantly from a hypothetical value. The two tests are listed below:

- Use a **single sample t-test** if the sample was taken from a normally distributed population. This is a parametric test that compares the mean of a single sample to a known or hypothesized value.
- Use a **Wilcoxon signed rank test** if the sample was taken from a non-normally distributed population or if the data consists of ranks or scores. This is a nonparametric test that compares the median of a single sample to a known or hypothesized value.

5.5.1 Entering Data for One Sample Tests

The single sample t-test and the Wilcoxon signed rank test both require that the observations for the sample be entered into a single data column. If the data includes a grouping variable, it can be included. However, the grouping variable will not be used in either of these analyses.

5.5.2 Performing a Single Sample t-Test

Use this test when you want to compare the mean of a sample to a known or hypothesized value.

As part of the results, KaleidaGraph calculates a t probability (P) value. This value determines if there is a statistically significant difference between the mean of the sample and the hypothetical mean. If this value is below a certain level (usually 0.05), the conclusion is that there is a difference between the two. For more information on the results of a single sample t-test, refer to the online help.

To perform a single sample t-test:

1. Choose **Functions** > **Student t** to display a dialog similar to Figure 5-7.

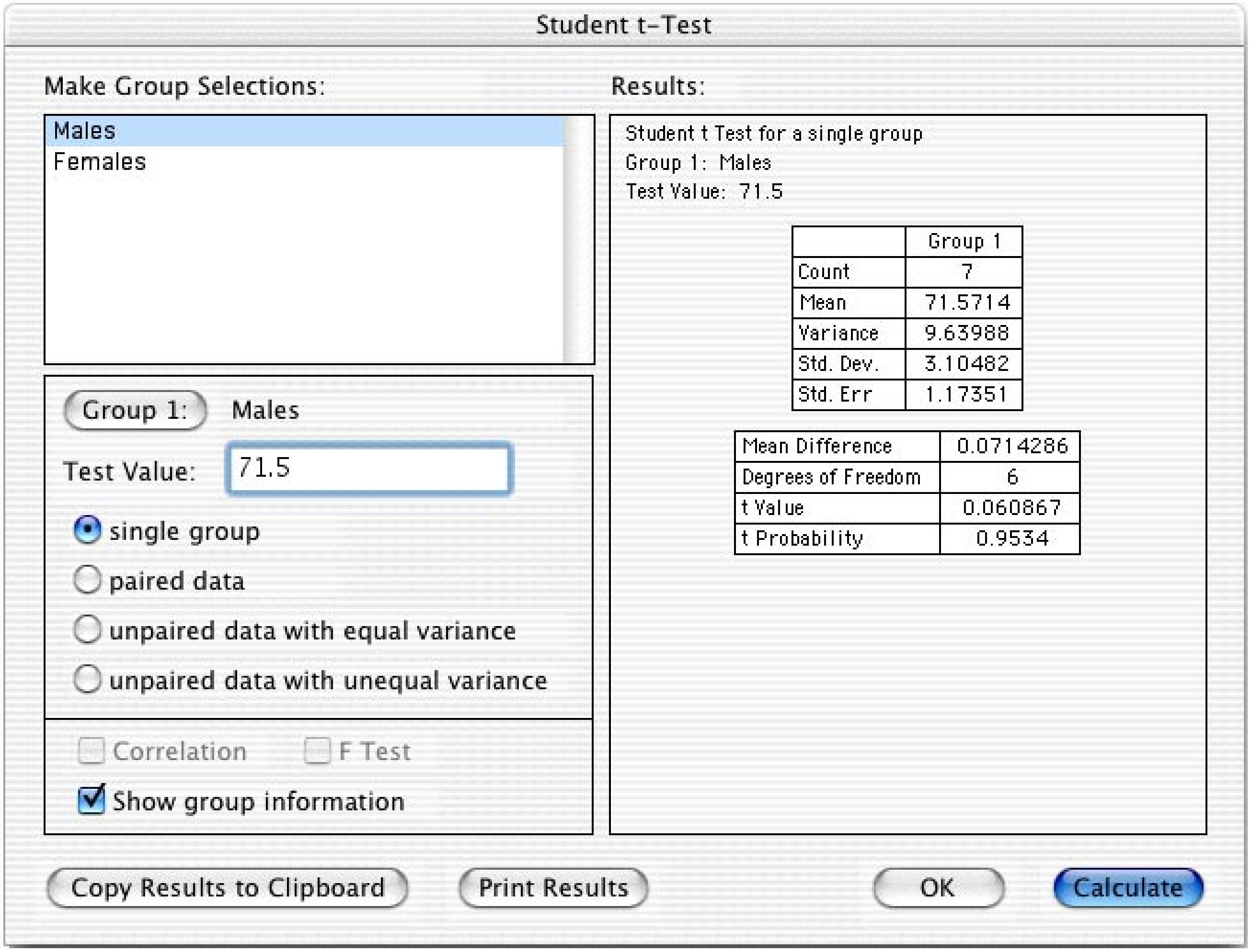


Figure 5-7 Student t-Test dialog

2. Click the **single group** button.
3. Select the column to be tested by either clicking the column name and pressing **Group 1** or double-clicking the column name.
4. Enter the known or hypothesized value in the **Test Value** field.
5. Click **Calculate**. The results of the test are displayed on the right side of the dialog.
6. To export the results, click **Copy Results to Clipboard**. You can then paste the results into a data, plot, or layout window.
7. To print the results, click the **Print Results** button.
8. Click **OK** to return to the data window.

5.5.3 Performing a Wilcoxon Signed Rank Test

Use this test to compare the median of a single sample to a known or hypothesized value.

This is a nonparametric test that does not require that the samples be drawn from a normally distributed population. If you know that the sample is normally distributed, use a single sample t-test instead.

As part of the results, KaleidaGraph calculates a P value. This value determines if there is a statistically significant difference between the median of the sample and the hypothetical median. If this value is below a certain level (usually 0.05), the conclusion is that there is a difference between the two. For more information on the results of a Wilcoxon signed rank test, refer to the online help.

To perform a Wilcoxon signed rank test:

1. Choose **Functions** > **Wilcoxon** to display a dialog similar to Figure 5-8.

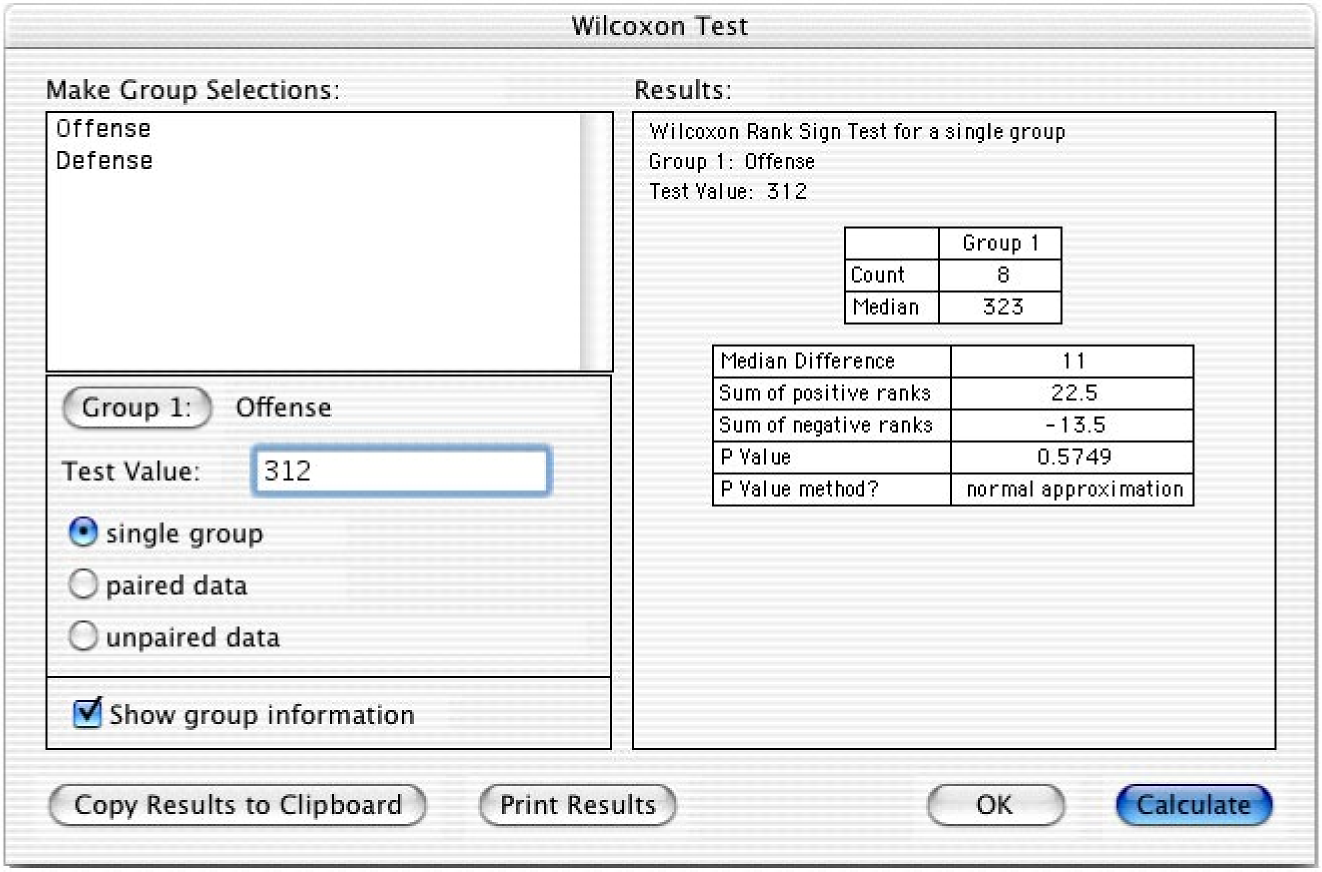


Figure 5-8 Wilcoxon Test dialog

2. Click the **single group** button.
3. Select the column to be tested by either clicking the column name and pressing **Group 1** or double-clicking the column name.
4. Enter the known or hypothesized value in the **Test Value** field.
5. Click **Calculate**. The results of the test are displayed on the right side of the dialog.
6. To export the results, click **Copy Results to Clipboard**. You can then paste the results into a data, plot, or layout window.
7. To print the results, click the **Print Results** button.
8. Click **OK** to return to the data window.

5.6 Analyzing Two or More Groups

KaleidaGraph provides five tests that can be used to analyze two or more different samples for a significant difference in the mean or median values. The proper test to use depends on the number of samples and the distribution of the data.

Comparing Two Groups

If data was collected from two different groups of subjects (for example, male and female manatees), use one of the following group comparison tests:

- Use the **unpaired t-test** if the samples were taken from normally distributed populations with equal variances. This is a parametric test that compares the sample data directly.
- Use the **Wilcoxon-Mann-Whitney test** if the samples were taken from non-normally distributed populations or if the data consists of ranks or scores. This is a nonparametric test that ranks the data, and analyzes the ranks instead of the raw data.

Comparing Three or More Groups

If the samples were taken from normally distributed populations with equal variances, use one of the following group comparison tests:

- Use a **one way ANOVA** to compare the effect of a single factor on the mean of three or more groups (for example, the effect of three drugs on a group of test animals).
- Use a **two way ANOVA** to compare the effect of two different factors on the mean of three or more groups (for example, the effect of three drugs given at different times during the day to a group of test animals).

If the samples were taken from non-normally distributed populations or if the data consists of ranks or scores, use the **Kruskal-Wallis test**.

Note: If you find a statistically significant difference using the one way ANOVA, you can use one of the post hoc tests to determine which groups are different.

5.6.1 Entering Data for Group Comparison Tests

The group comparison tests in KaleidaGraph require that the data be entered in one of two ways:

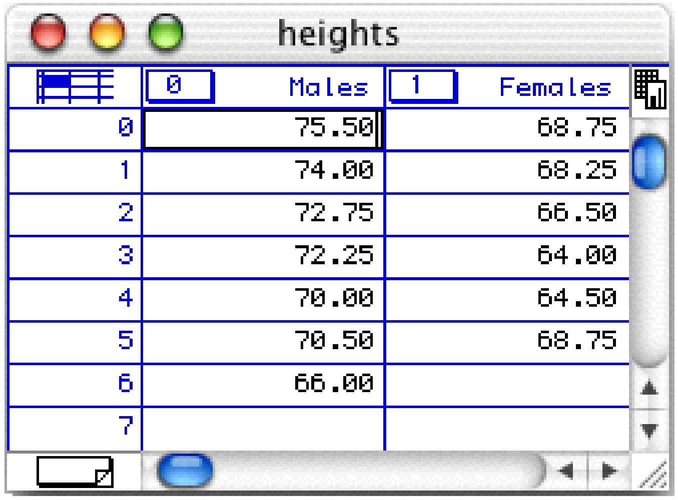
- as raw data, where the data for each group is entered into a separate column.
- as indexed data, where one of the columns is a factor column or grouping variable, with the corresponding data points in one or more columns.

You can use raw data for all of the tests except the two way ANOVA. The two way ANOVA requires one or two factor columns and one or more data columns.

Note: The groups do not have to be the same size.

The following examples show how raw and indexed data should be entered for the various group comparison tests.

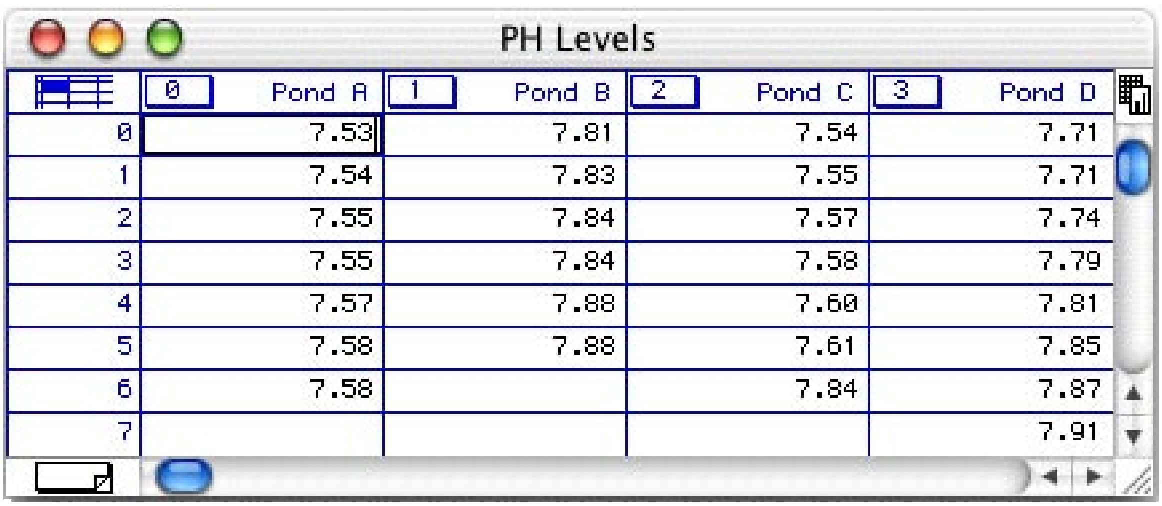
Raw Data Examples - Group Comparisons



heights

	0 Males	1 Females
0	75.50	68.75
1	74.00	68.25
2	72.75	66.50
3	72.25	64.00
4	70.00	64.50
5	70.50	68.75
6	66.00	
7		

Unpaired t-test and Wilcoxon-Mann-Whitney test



PH Levels

	0 Pond A	1 Pond B	2 Pond C	3 Pond D
0	7.53	7.81	7.54	7.71
1	7.54	7.83	7.55	7.71
2	7.55	7.84	7.57	7.74
3	7.55	7.84	7.58	7.79
4	7.57	7.88	7.60	7.81
5	7.58	7.88	7.61	7.85
6	7.58		7.84	7.87
7				7.91

One way ANOVA and Kruskal-Wallis test

Indexed Data Examples - Group Comparisons

heights

	0 Sex	1 Height
0	male	75.50
1	female	68.75
2	male	74.00
3	female	68.25
4	male	72.75
5	female	66.50
6	male	72.25
7	female	64.00

Unpaired t-test and Wilcoxon-Mann-Whitney test

PH Levels

	0 Ponds	1 PH
0	Pond A	7.53
1	Pond B	7.81
2	Pond C	7.54
3	Pond D	7.71
4	Pond A	7.54
5	Pond B	7.83
6	Pond C	7.55
7	Pond D	7.71

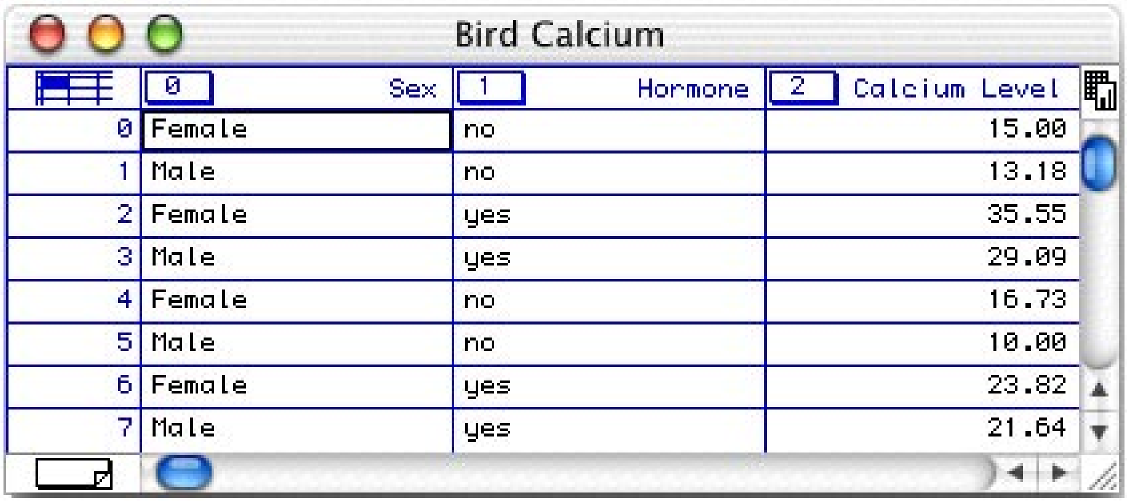
One way ANOVA and Kruskal-Wallis test

Sales Stats

	0 Promotion	1 1st Qtr	2 2nd Qtr	3 3rd Qtr	4 4th Qtr
0	A	958	1047	933	947
1	A	1005	1122	986	1022
2	A	351	436	339	356
3	A	549	632	512	545
4	A	730	784	707	722
5	B	780	897	718	803
6	B	229	275	202	256
7	B	883	964	817	845

Two way ANOVA
(one factor column and four columns containing the data values)

Indexed Data Examples - Group Comparisons



Bird Calcium

	0 Sex	1 Hormone	2 Calcium Level
0	Female	no	15.00
1	Male	no	13.18
2	Female	yes	35.55
3	Male	yes	29.09
4	Female	no	16.73
5	Male	no	10.00
6	Female	yes	23.82
7	Male	yes	21.64

Two way ANOVA
(two factor columns and one column containing the data values)

5.6.2 Performing an Unpaired t-Test

Use this test to compare the means of two independent samples. This test can be performed on columns of different length, since no relationship is assumed between the samples. As a special case, if Group 1 is assigned a Text column, it is used as a grouping variable (for example, gender) to separate the data in the second column into two groups.

As part of the results, KaleidaGraph calculates a t probability (P) value. This value determines if there is a statistically significant difference between the two means. If this value is below a certain level (usually 0.05), the conclusion is that there is a difference between the two group means. For more information on the results of an unpaired t-test, refer to the online help.

To perform an unpaired t-test:

1. Choose **Functions** > **Student t** to display the Student t-Test dialog.
2. Click the **unpaired data with equal variance** or **unpaired data with unequal variance** button.
3. Select the columns to be tested by either clicking the column name and pressing a Group button or double-clicking the column names. If you are using a Text column as a grouping variable, it must be assigned to Group 1.
4. Click the **F Test** check box if you want KaleidaGraph to calculate the F and F probability values. The F probability determines if the two groups have different variances. A small F probability value (usually less than 0.05) indicates that the two groups have significantly different variances.
5. Click **Calculate**. The results of the test are displayed on the right side of the dialog.
6. To export the results, click **Copy Results to Clipboard**. You can then paste the results into a data, plot, or layout window.
7. To print the results, click the **Print Results** button.
8. Click **OK** to return to the data window.

5.6.3 Performing a Wilcoxon-Mann-Whitney Test

Use this test to see if two different samples have been drawn from the same population. This test can be performed on columns of different length, since no relationship is assumed between the samples. As a special case, if Group 1 is assigned a Text column, it is used as a grouping variable (for example, gender) to separate the data in the second column into two groups.

This test assumes the samples are not normally distributed with the same variances. If you know that the samples are normally distributed, use an unpaired t-test instead. If there are more than two samples to compare, use the Kruskal-Wallis test.

As part of the results, KaleidaGraph calculates a P value. This value determines if there is a statistically significant difference between the medians of the two groups. If this value is below a certain level (usually 0.05), the conclusion is that there is a difference between the two. For more information on the results of a Wilcoxon-Mann-Whitney test, refer to the online help.

To perform a Wilcoxon-Mann-Whitney test:

1. Choose **Functions** > **Wilcoxon** to display the Wilcoxon Test dialog.
2. Click the **unpaired data** button.
3. Select the columns to be tested by either clicking the column name and pressing a Group button or double-clicking the column names. If you are using a Text column as a grouping variable, it must be assigned to Group 1.
4. Click **Calculate**. The results of the test are displayed on the right side of the dialog.
5. To export the results, click **Copy Results to Clipboard**. You can then paste the results into a data, plot, or layout window.
6. To print the results, click the **Print Results** button.
7. Click **OK** to return to the data window.

5.6.4 Performing a One Way ANOVA

Use this test when you want to see if the means of three or more different groups are affected by a single factor. This test is the same as the unpaired t-test, except that more than two groups can be compared.

This is a parametric test that assumes the samples are drawn from normally distributed populations with equal variances. If you know that the samples are not normally distributed, use the Kruskal-Wallis test. If there are only two samples to compare, use an unpaired t-test.

As part of the results, KaleidaGraph calculates F and P values. For more information on the results of a one way ANOVA, refer to the online help.

- **F value** - This value is the ratio of the groups mean square over the error mean square. If this value is close to 1.0, you can conclude that there are no significant differences between the groups. If this value is large, you can conclude that one or more of the samples was drawn from a different population. To determine which groups are different, use one of the post hoc tests.
- **P value** - This value determines if there is a statistically significant difference between the groups. If this value is below a certain level (usually 0.05), the conclusion is that there is a difference between the groups.

To perform a one way ANOVA:

1. Choose **Functions** > **ANOVA** to display a dialog similar to Figure 5-9.

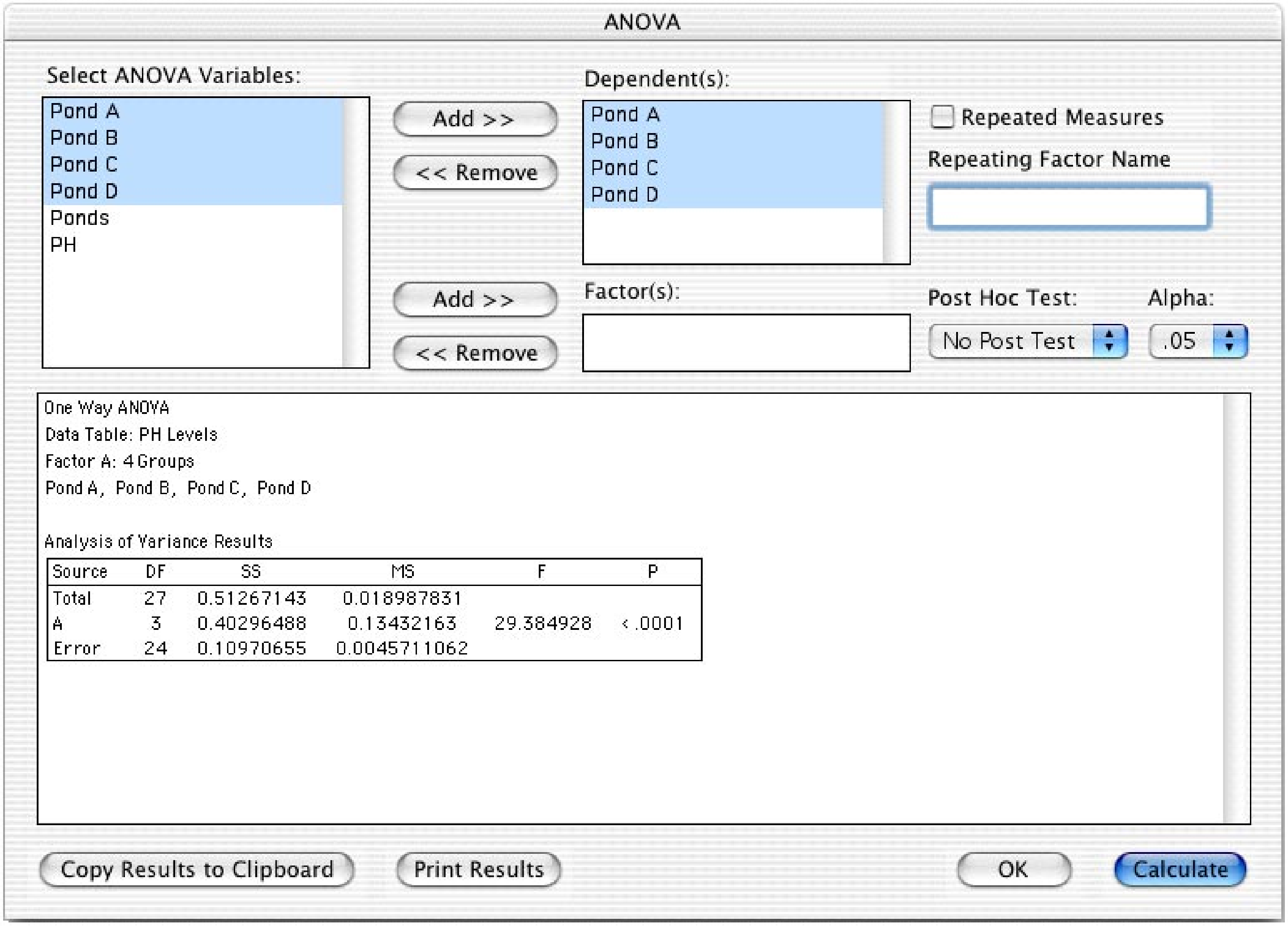


Figure 5-9 ANOVA dialog

2. Assign the data columns containing the measured values to the Dependent(s) list.
3. If you are using indexed data, assign the factor column to the Factor(s) list. In this case, only one dependent column should be used. If you assign multiple columns to the Dependent(s) list, a two way ANOVA will be performed.
4. Click **Calculate**. The results of the test are displayed in the bottom half of the dialog.
5. If you find a statistically significant difference, you can use one of the post hoc tests to determine which groups are different and the size of the difference. For more information on the post hoc tests, refer to Section 5.8.
6. To export the results, click **Copy Results to Clipboard**. You can then paste the results into a data, plot, or layout window.
7. To print the results, click the **Print Results** button.
8. Click **OK** to return to the data window.

5.6.5 Performing a Two Way ANOVA

Use this test when you want to see if the means of three or more different groups are affected by two factors. A two way ANOVA tests three hypotheses:

- There is no difference among the levels of the first factor.
- There is no difference among the levels of the second factor.
- There is no interaction between the factors.

This is a parametric test that assumes the samples are drawn from normally distributed populations with equal variances. If you want to see the effect of only one factor on your data, use a one way ANOVA. KaleidaGraph does not have a nonparametric equivalent to the two way ANOVA.

As part of the results, KaleidaGraph calculates F and P values. For more information on the results of a two way ANOVA, refer to the online help.

- **F value** - The F value is calculated for each of the factors and the interaction. For each factor, this value is the ratio of the mean square of the factor over the error mean square. For the interaction, it is the ratio of the interaction mean square over the error mean square.

  If the F value is close to 1.0, you can conclude that there are no significant differences between the factor levels or that there is no interaction between the factors. If this value is large, you can conclude that one or more of the samples for that factor or combination of factors was drawn from a different population.

- **P value** - This value determines if there is a statistically significant difference between the groups. If this value is below a certain level (usually 0.05), the conclusion is that there is a difference between the groups.

To perform a two way ANOVA:

1. Choose **Functions** > **ANOVA** to display the ANOVA dialog.
2. Assign the data columns containing the measured values to the Dependent(s) list.
3. Assign one or two factor columns to the Factor(s) list. If two factor columns are assigned, only one dependent column may be used.
4. Click **Calculate**. The results of the test are displayed in the bottom half of the dialog.
5. To export the results, click **Copy Results to Clipboard**. You can then paste the results into a data, plot, or layout window.
6. To print the results, click the **Print Results** button.
7. Click **OK** to return to the data window.

5.6.6 Performing a Kruskal-Wallis Test

Use this test when you want to see if three or more groups are affected by a single factor.

This is a nonparametric test that does not require that the samples be drawn from a normally distributed population. If you know that the samples are normally distributed, use the one way ANOVA. If there are only two samples to compare, use the Wilcoxon-Mann-Whitney test.

As part of the results, KaleidaGraph calculates a P value. This value determines if there is a statistically significant difference between the medians of the groups. If this value is below a certain level (usually 0.05), the conclusion is that there is a difference between the groups. For more information on the results of a Kruskal-Wallis test, refer to the online help.

To perform a Kruskal-Wallis test:

1. Choose **Functions** > **Kruskal-Wallis** to display a dialog similar to Figure 5-10.

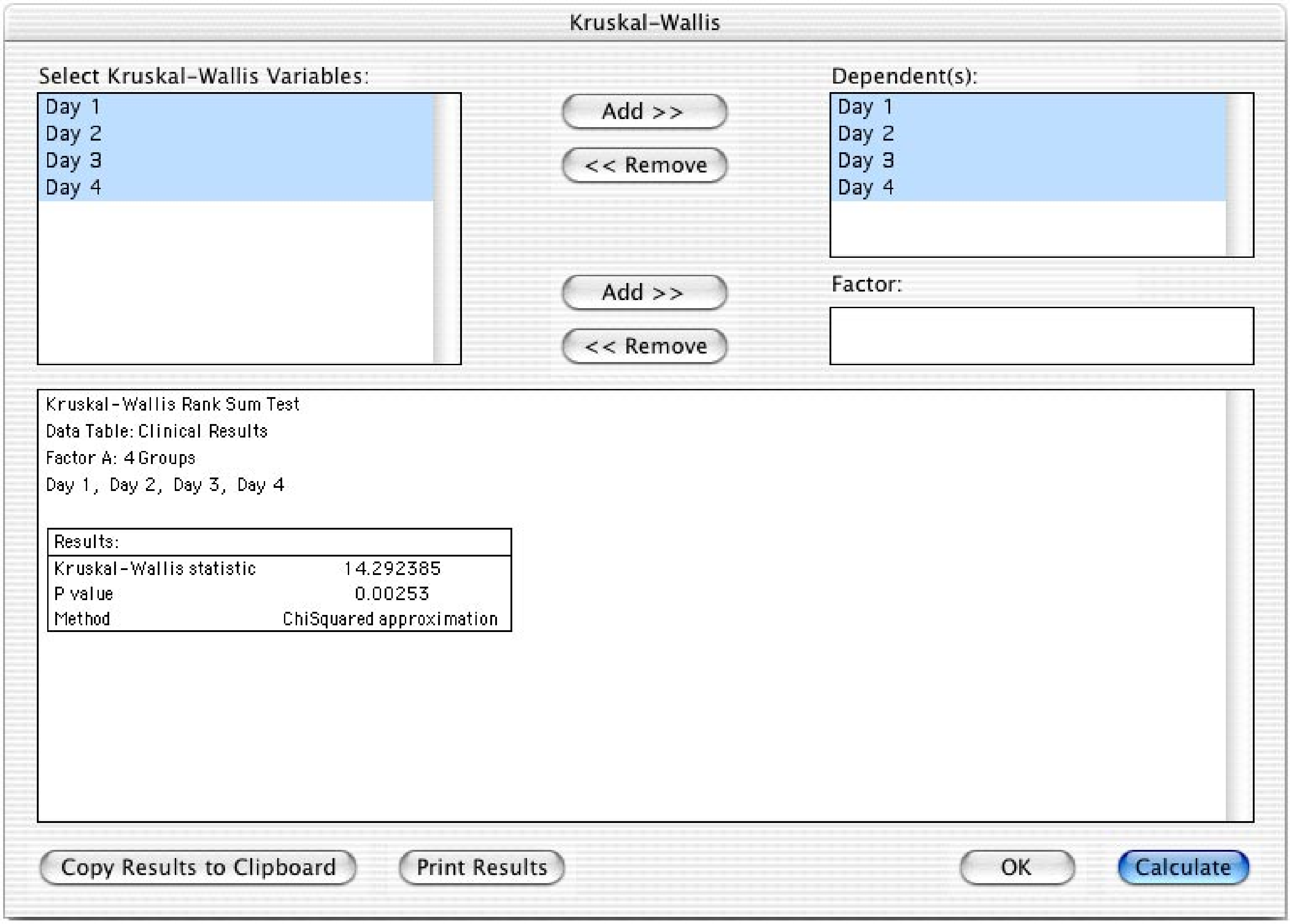


Figure 5-10 Kruskal-Wallis dialog

2. Assign the data columns containing the measured values to the Dependent(s) list.
3. If you are using indexed data, assign the factor column to the Factor(s) list. In this case, only one dependent column may be used.
4. Click **Calculate**. The results of the test are displayed in the bottom half of the dialog.
5. To export the results, click **Copy Results to Clipboard**. You can then paste the results into a data, plot, or layout window.
6. To print the results, click the **Print Results** button.
7. Click **OK** to return to the data window.

5.7 Analyzing Repeated Measurements

KaleidaGraph provides five tests that can be used to analyze two or more matched samples for a significant difference in the mean or median values. The proper test to use depends on the number of samples and the distribution of the data.

Comparing Two Groups

If data was collected from the same group of subjects (for example, the gas mileage for a number of cars before and after receiving a tune-up), use one of the following repeated measures tests:

- Use the **paired t-test** if the samples were taken from normally distributed populations with equal variances. This is a parametric test that compares the sample data directly.
- Use the **Wilcoxon matched pairs test** if the samples were taken from non-normally distributed populations or if the data consists of ranks or scores. This is a nonparametric test that ranks the data, and analyzes the ranks instead of the raw data.

Comparing Three or More Groups

If the samples were taken from normally distributed populations with equal variances, use one of the following repeated measures tests:

- Use a **one way repeated measures ANOVA** to compare the effect of a single factor on the mean of three or more matched groups (for example, comparing the effects of a drug on a group of subjects before, during, and after the treatment).
- Use a **two way repeated measures ANOVA** to compare the effect of two different factors on the mean of three or more matched groups (for example, comparing the effects of a drug on males and females before, during, and after the treatment).

If the samples were taken from non-normally distributed populations or if the data consists of ranks or scores, use the **Friedman test**.

Note: If you find a statistically significant difference using the one way repeated measures ANOVA, you can use one of the post hoc tests to determine which groups are different.

5.7.1 Entering Data for Repeated Measurements

The repeated measurement tests in KaleidaGraph require that the data be entered as raw data, where the data for each treatment is entered into a separate column.

The only exception is the two way repeated measures ANOVA, which requires indexed data. In this case, one of the columns is a factor column or grouping variable, with the corresponding data points in two or more columns.

Note: Each group being analyzed must have the same number of observations.

The following examples show how the data should be entered for the various repeated measurement tests.

Data Examples - Repeated Measurements

Heart Rate

	0 Before	1 After
0	78.000	81.000
1	81.000	83.000
2	75.000	76.000
3	84.000	87.000
4	72.000	76.000
5	80.000	82.000
6	76.000	79.000
7	74.000	76.000

Paired t-test and Wilcoxon matched pairs test

Clinical Results

	0 Day 1	1 Day 2	2 Day 3	3 Day 4
0	20.000	24.000	28.000	28.000
1	15.000	18.000	23.000	24.000
2	18.000	19.000	24.000	23.000
3	26.000	26.000	30.000	30.000
4	22.000	24.000	28.000	26.000
5	19.000	21.000	27.000	25.000
6	23.000	24.000	26.000	26.000
7	22.000	22.000	23.000	25.000

One way repeated measures ANOVA and Friedman test

Data Examples - Repeated Measurements

Sales Stats

	0 Promotion	1 Before	2 During	3 After
0	A	958	1047	933
1	A	1005	1122	986
2	A	351	436	339
3	A	549	632	512
4	B	730	784	707
5	B	780	897	718
6	B	229	275	202
7	B	883	964	817

Two way repeated measures ANOVA

5.7.2 Performing a Paired t-Test

Use this test to compare means on the same or related subjects over time or under different conditions. This test requires columns of equal length. Any missing or extra values are skipped when running this test.

As part of the results, KaleidaGraph calculates a t probability (P) value. This value determines if there is a statistically significant difference between the two means. If this value is below a certain level (usually 0.05), the conclusion is that there is a difference between the two group means. For more information on the results of a paired t-test, refer to the online help.

To perform a paired t-test:

1. Choose **Functions** > **Student t** to display the Student t-Test dialog.
2. Click the **paired data** button.
3. Select the columns to be tested by either clicking the column name and pressing a Group button or double-clicking the column names.
4. Click the **Correlation** check box if you want KaleidaGraph to calculate the correlation and correlation probability values. The correlation probability determines if there is a correlation between the two groups. A small correlation probability (usually less than 0.05) indicates that the two groups are significantly correlated.
5. Click **Calculate**. The results of the test are displayed on the right side of the dialog.
6. To export the results, click **Copy Results to Clipboard**. You can then paste the results into a data, plot, or layout window.
7. To print the results, click the **Print Results** button.
8. Click **OK** to return to the data window.

5.7.3 Performing a Wilcoxon Matched Pairs Test

Use this test to compare the median values on the same or related subjects over time or under different conditions. This test requires columns of equal length. Any missing or extra values are skipped when running this test.

This test assumes the samples are not normally distributed with the same variances. If you know that the samples are normally distributed, use a paired t-test instead. If there are more than two samples to compare, use the Friedman test.

As part of the results, KaleidaGraph calculates a P value. This value determines if there is a statistically significant difference between the medians of the two groups. If this value is below a certain level (usually 0.05), the conclusion is that there is a difference between the two. For more information on the results of a Wilcoxon matched pairs test, refer to the online help.

To perform a Wilcoxon matched pairs test:

1. Choose **Functions** > **Wilcoxon** to display the Wilcoxon Test dialog.
2. Click the **paired data** button.
3. Select the columns to be tested by either clicking the column name and pressing a Group button or double-clicking the column names.
4. Click **Calculate**. The results of the test are displayed on the right side of the dialog.
5. To export the results, click **Copy Results to Clipboard**. You can then paste the results into a data, plot, or layout window.
6. To print the results, click the **Print Results** button.
7. Click **OK** to return to the data window.

5.7.4 Performing a One Way Repeated Measures ANOVA

Use this test when you want to see if the same group of individuals is affected by a single factor. This test is the same as the paired t-test, except that more than two treatments can be compared.

This is a parametric test that assumes the samples are drawn from normally distributed populations with equal variances. If you know that the samples are not normally distributed, use the Friedman test. If there are only two samples to compare, use a paired t-test.

Note: This test requires columns of equal length. If any missing or extra values are found, a one way ANOVA is performed on the data.

As part of the results, KaleidaGraph calculates F and P values. For more information on the results of a one way repeated measures ANOVA, refer to the online help.

- **F value** - The F value is calculated for both the treatment and the subject. For the treatment, this value is the ratio of the treatment mean square over the error mean square. For the subject, it is the ratio of the subject mean square over the error mean square.

  If this value is close to 1.0, you can conclude that there are no significant differences among treatments. If this value is large, you can conclude that the treatments have different effects. To determine which treatments are different, use one of the post hoc tests.

- **P value** - This value determines if there is a statistically significant difference between the treatments. If this value is below a certain level (usually 0.05), the conclusion is that there is a difference between the treatments.

To perform a one way repeated measures ANOVA:

1. Choose **Functions** > **ANOVA** to display the ANOVA dialog.
2. Assign the data columns containing the measured values to the Dependent(s) list.
3. Click the **Repeated Measures** check box.
4. Enter a label in the **Repeating Factor Name** field. This label will appear in the ANOVA results.
5. Click **Calculate**. The results of the test are displayed in the bottom half of the dialog.
6. If you find a statistically significant difference, you can use one of the post hoc tests to determine which groups are different and the size of the difference. For more information on the post hoc tests, refer to Section 5.8.
7. To export the results, click **Copy Results to Clipboard**. You can then paste the results into a data, plot, or layout window.
8. To print the results, click the **Print Results** button.
9. Click **OK** to return to the data window.

5.7.5 Performing a Two Way Repeated Measures ANOVA

Use this test when you want to see if the same group of individuals is affected by two factors. At least one of these factors is a repeated treatment on the same group of individuals. A two way repeated measures ANOVA tests three hypotheses:

- There is no difference among the levels or treatments of the first factor.
- There is no difference among the levels or treatments of the second factor.
- There is no interaction between the factors.

This is a parametric test that assumes the samples are drawn from normally distributed populations with equal variances. If you want to see the effect of only one factor on your groups, use a one way repeated measures ANOVA. KaleidaGraph does not have a nonparametric equivalent to the two way repeated measures ANOVA.

Note: Each group must have the same number of observations for this test.

As part of the results, KaleidaGraph calculates F and P values. For more information on the results of a two way repeated measures ANOVA, refer to the online help.

- **F value** - The F value is calculated for each of the factors, the interaction, and the subjects. If this value is close to 1.0, you can conclude that there are no significant differences among treatments. If this value is large, you can conclude that the treatments have different effects.
  - For the first factor, this value is the ratio of the factor mean square over the subject mean square
  - For the second factor, it is the ratio of the factor mean square over the error mean square.
  - For the interaction, it is the ratio of the interaction mean square over the error mean square.
  - For the subjects, it is the ratio of the subject mean square over the error mean square.
- **P value** - This value determines if there is a statistically significant difference between the treatments. If this value is below a certain level (usually 0.05), the conclusion is that there is a difference between the treatments.

To perform a two way repeated measures ANOVA:

1. Choose **Functions** > **ANOVA** to display the ANOVA dialog.
2. Assign the data columns containing the measured values to the Dependent(s) list.
3. Assign a factor column to the Factor(s) list.
4. Click the **Repeated Measures** check box.
5. Enter a label in the **Repeating Factor Name** field. This label will appear in the ANOVA results.
6. Click **Calculate**. The results of the test are displayed in the bottom half of the dialog.
7. To export the results, click **Copy Results to Clipboard**. You can then paste the results into a data, plot, or layout window.
8. To print the results, click the **Print Results** button.
9. Click **OK** to return to the data window.

5.7.6 Performing a Friedman Test

Use this test when you want to see if a single group of individuals is affected by a series of three or more treatments.

This is a nonparametric test that does not require that the samples be drawn from a normally distributed population. If you know that the samples are normally distributed, use the one way repeated measures ANOVA. If there are only two samples to compare, use the Wilcoxon matched pairs test.

Note: This test requires columns of equal length.

As part of the results, KaleidaGraph calculates two different P values. These values determine if there is a statistically significant difference between the groups. If the P values are below a certain level (usually 0.05), the conclusion is that there is a difference between the groups. For more information on the results of a Friedman test, refer to the online help.

To perform a Friedman test:

1. Choose **Functions** > **Friedman** to display a dialog similar to Figure 5-11.

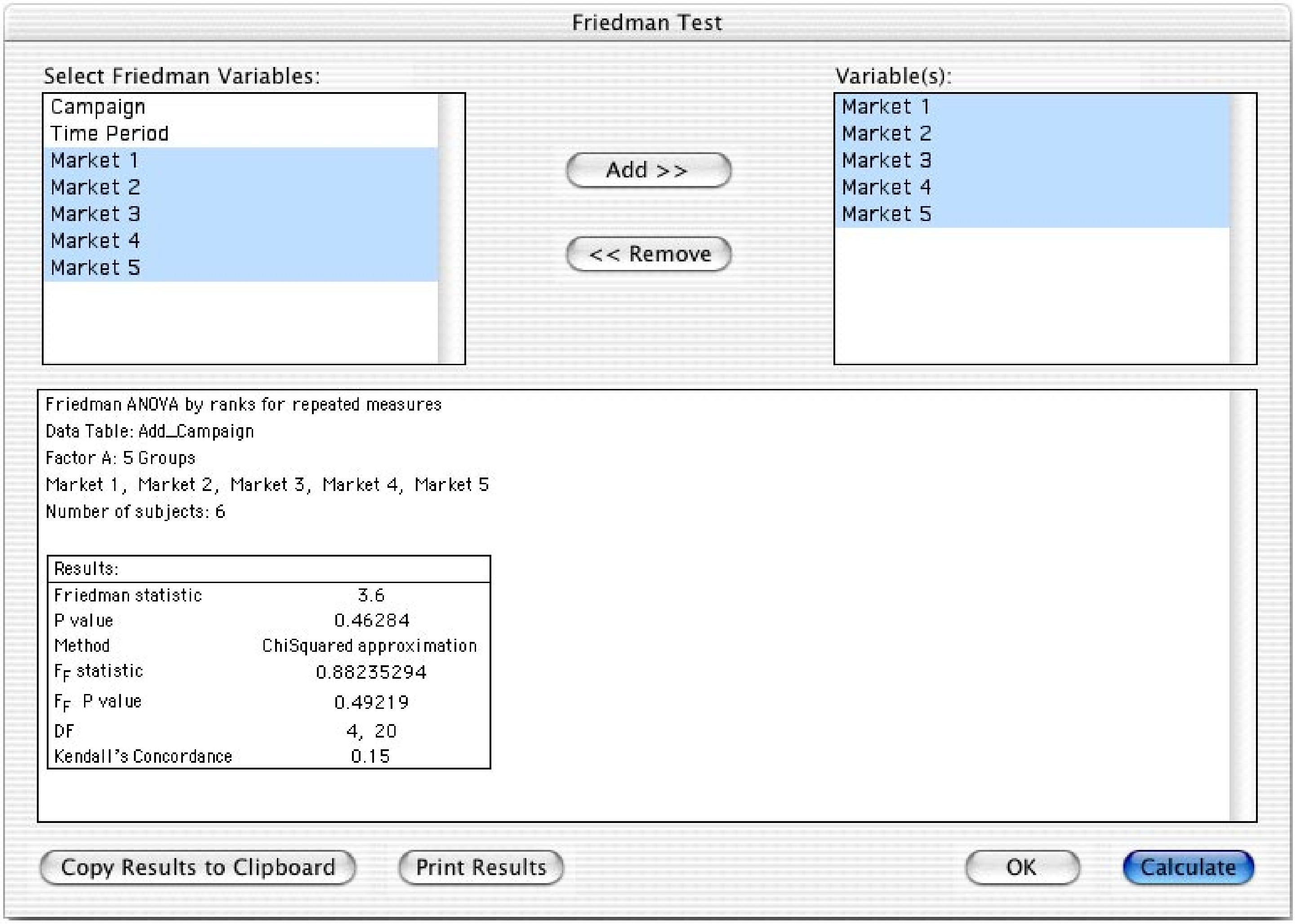


Figure 5-11 Friedman Test dialog

2. Assign the data columns containing the measured values to the Variable(s) list.
3. Click **Calculate**. The results of the test are displayed in the bottom half of the dialog.
4. To export the results, click **Copy Results to Clipboard**. You can then paste the results into a data, plot, or layout window.
5. To print the results, click the **Print Results** button.
6. Click **OK** to return to the data window.

5.8 Choosing a Post Hoc Test

Once you have performed a one way or one way repeated measures ANOVA and found a statistically significant difference, you can use one of the four post hoc tests provided by KaleidaGraph to determine which groups are different.

- Use the Tukey HSD, Bonferroni, or Holm test to compare all possible combinations of group pairs.
- Use the Dunnett test if one column represents control data, and you want to compare all other columns to the control column (but not to each other).

To perform a post hoc test:

1. Verify that One Way ANOVA or One Way ANOVA – Repeated Measures is listed on the first line in the ANOVA results. The post hoc tests cannot be used with the two way or two way repeated measures ANOVA.
2. Select the desired test from the **Post Hoc Test** pop-up menu.
3. If necessary, select a different significance level using the **Alpha** pop-up menu.
4. Click **Calculate**. The post hoc test results will appear below the ANOVA results.

5.8.1 Tukey HSD Test

The Tukey HSD (honestly significant difference) test is a pairwise comparison of every combination of group pairs. This test calculates the mean difference for each treatment or group pair, calculates a q test statistic for each pair, and displays the P value that corresponds to the q test statistic. The P value determines if there is a statistically significant difference between the treatments or groups being compared. If this value is below a certain level (usually 0.05), the conclusion is that there is a difference between the groups.

For more information on the results of a Tukey HSD test, refer to the online help.

5.8.2 Bonferroni Test

The Bonferroni test is a pairwise comparison of every combination of group pairs. This test calculates the mean difference for each treatment or group pair, calculates a t test statistic for each pair, and displays the P value that corresponds to the t test statistic. The P value determines if there is a statistically significant difference between the treatments or groups being compared. If this value is below a certain level (usually 0.05), the conclusion is that there is a difference between the groups.

For more information on the results of a Bonferroni test, refer to the online help.

5.8.3 Holm Test

The Holm test is a pairwise comparison of every combination of group pairs. This test calculates the mean difference for each treatment or group pair, calculates a t test statistic for each pair, and displays the two-sided P value that corresponds to the t test statistic.

The smallest P value is compared to the adjusted Alpha value. If the P value is greater than or equal to the adjusted Alpha, you can conclude that there is no significant difference for this test and any remaining comparisons. If the P value is less than the adjusted Alpha, there is a statistically significant difference for this test. A new adjusted Alpha value is calculated, and the tests continue with the next smallest P value.

For more information on the results of a Holm test, refer to the online help.

5.8.4 Dunnett Test

The Dunnett test is used to determine if the mean of one group, designated as a control, differs significantly from each of the means of the other groups. This test calculates the mean difference between the control and each treatment or group, calculates a q test statistic for each pair, and displays the P value that corresponds to the q test statistic. The P value determines if there is a statistically significant difference between the control and the group being compared. If this value is below a certain level (usually 0.05), the conclusion is that there is a difference between the groups.

For more information on the results of a Dunnett test, refer to the online help.

Working with Formula Entry

Chapter 6

The Formula Entry window is a very powerful tool for data analysis. Using this window, you can apply a wide range of algebraic functions to your data. It also provides a formula script feature to help automate certain functions in the program.

This chapter explains how to:

- Enter and execute formulas.
- Reference columns, cells, and memory locations in a formula.
- Create multi-line formulas.
- Calculate statistics.
- Linearly interpolate data.
- Obtain values from a curve fit.
- Create shorthand references for constants, variables, and functions in the library.
- Automate the process of creating plots using formula scripts.

6.1 Formula Entry Basics

This section provides the basic information you need to start working with the Formula Entry window. In addition to this section, you may also want to refer to Section 2.5.2, which guides you through the process of executing several formulas.

6.1.1 Formula Entry Window

Choose **Windows** > **Formula Entry** to display the window in Figure 6-1. Use the Formula Entry window to enter formulas that generate and manipulate data in the active data window. The various features of the Formula Entry window are discussed below.

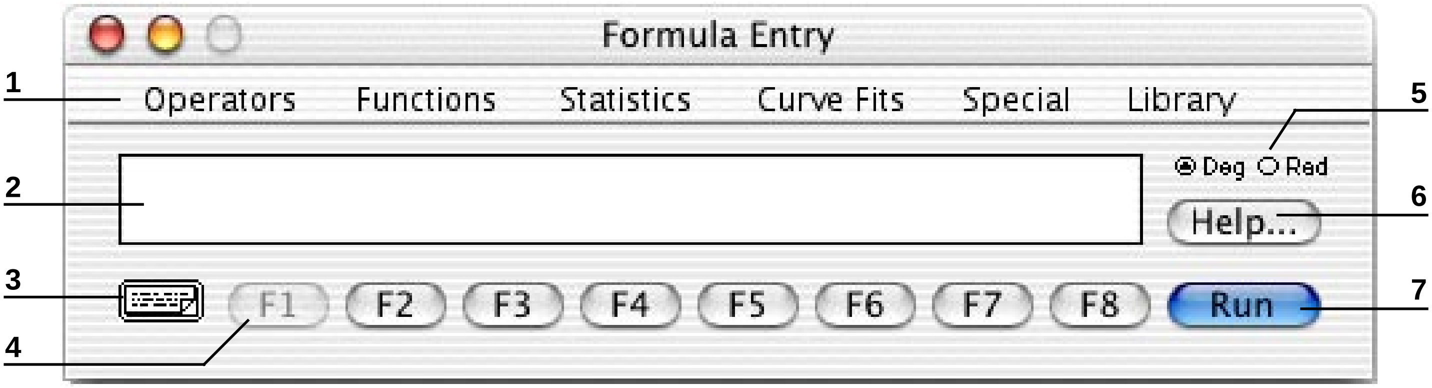


Figure 6-1 Formula Entry window

1. **Formula Entry Menu Bar**
   These menus contain all of the functions that are available in Formula Entry. The commands in these menus are described in Section 6.9.

2. **Text Field**
   Formulas are entered in this area of the window.

3. **Posted Note**
   This is an internal text editor where multiple functions can be strung together. See Section 6.2 for more information on multi-line formulas.

4. **Function Buttons**
   Using these buttons, you can assign frequently used formulas to the first eight function keys of your keyboard. This topic is discussed in Section 6.1.6.

5. **Degrees/Radians Buttons**
   These buttons determine whether the results of trigonometric functions are in degrees or radians.

6. **Help...**
   Click this button to display the Help dialog.

7. **Run**
   This button is used to execute the current formula. Formulas can also be executed by typing **Enter** (Windows) or **Return** (Macintosh). If a data window is not open, this button is unavailable.

6.1.2 Formula Entry Menus

The menus in Formula Entry contain all of the commands and operators that are available for use. You have the option of choosing the commands from the menu or typing them into the window. Each of the commands is discussed in detail in Section 6.9.

Operators Menu

The operators available within Formula Entry are listed below. Grouped items have the same precedence.

!	factorial
^	power
*	multiply
/	divide
%	modulo
+	add
–	subtract
<	less than
<=	less than or equal to
>	greater than
>=	greater than or equal to
==	equal to (used for comparisons)
!=	not equal to
&&	logical AND
||	logical OR
?:	conditional
( )	left and right parentheses
=	equal (used to store results)
[ ]	matrix operator

Functions Menu

The **Functions** menu contains general math functions. Examples of these commands include trigonometric functions (sine, cosine, and tangent), square root, and absolute value.

Statistics Menu

The **Statistics** menu contains the commands that are used to calculate statistics on data. Section 6.3 explains how to calculate statistics using these commands. The equations used to calculate these statistics are located in Section D.2.

Curve Fits Menu

The **Curve Fits** menu contains commands that can be used to place curve fit values into the data window. This allows you to find curve fit values at specific points that were not originally plotted. The procedure for doing this is discussed in Section 6.6.

Special Menu

The **Special** menu contains commands that use either a Boolean expression or a text string as part of their definition.

Library Menu

The **Library** menu displays all of the definitions in the current library. If the library is empty, this menu is also empty. For more information on the library, see Section 6.8.

6.1.3 Formula Structure

A formula calculates new values from existing data in the data window. A formula can be a simple mathematical operation, such as multiplying a column by a constant, or a more complex operation, such as calculating the root mean square of a specific group of data.

Formulas are created using a combination of column references, operators, and functions. Memory locations and constants can also be used in a formula. Formulas are usually constructed with the destination to the left of the equal sign (=) and the calculation to the right of the equal sign. One exception is the **name** function, which specifies the output column as one of the parameters.

The results of a formula can be stored in another column, a cell, or a memory register. If a destination is not specified, the results are displayed in a dialog similar to Figure 6-2. This is useful for formulas that result in a single value, such as the commands located in the **Statistics** menu.

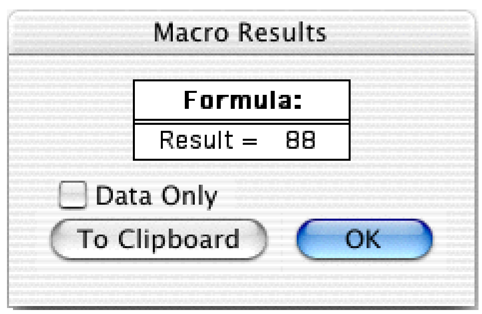


Figure 6-2 Result dialog

6.1.4 Referencing Columns, Cells, and Memory Registers

Column numbers range from 0 to 999 and need to be preceded by a **c** when used in a formula (c0, c1, and so on). Column numbers are displayed in a box at the top of the column and are referenced to the start of any data selection. Please note that when a selection is made in the data window, the first column in the selection becomes column 0.

Calculations involving columns are performed on a row-by-row basis. If two or more columns are used in a calculation, corresponding cells in the same row are used to determine the results. If empty or masked cells are encountered during the calculation, the corresponding cell in the destination column is empty.

Individual cells can be referenced in a formula using the **cell** function. The syntax of this function is **cell(row#, col#)**. Using this function, it is possible to store and retrieve data anywhere in the data window.

Memory registers range from 0 to 99 and need to be preceded by an **m** when used in a formula (m0, m1, and so on). Memory registers are useful in calculations that result in a single value. The value can be stored in a memory register and used at a later time in other calculations.

Some example formulas are shown below:

Example Formulas	
c1 = c0/1000	cell(0,5) = cell(0,0) – 11
c5 = (c1+c2) * (c3–c4)	m1 = npts(c0)
cell(0,2) = mean(c1)	m10 = cell(0,0)

6.1.5 Entering and Executing a Formula

This section provides the general instructions for entering and executing a formula in the Formula Entry window. Section 2.5.2 provides a detailed example of running several formulas and viewing their effect on the data window.

To enter and execute a formula:

1. Open the data window to be operated on by the formula.
2. Choose **Windows** > **Formula Entry**.
3. Enter the formula into the text field of Formula Entry. You can either type the operators and functions or choose them from the appropriate menus in the Formula Entry window.
4. Multiple formulas can be entered by separating each formula with a semicolon. The Posted Note can also be used to enter multiple formulas. This topic is covered in Section 6.2.
5. Click **Run** to execute the formula.

Note: The results of a formula are based on the data at the time the formula is executed. Changing the data does not cause the results to automatically recalculate. To update the results, the formula must be executed again.

6.1.6 Assigning Formulas to Function Buttons

You can store formulas in any of the eight function buttons in Formula Entry. Any formulas that are assigned to these buttons are saved in the Macros file. It is recommended that you store frequently-used formulas in F2–F8 and leave F1 for entering general formulas that do not need to be saved.

To assign a formula to a particular function button:

1. Select the appropriate function button in Formula Entry using one of the following methods:
   - Click the button in the Formula Entry window.
   - Press the corresponding function key on the keyboard.
   - Hold down **Ctrl** (Windows) or ⌘ (Macintosh) and press the number of the function button you want to select.
2. Enter the formula into the text field of Formula Entry.

6.2 Multi-Line Formulas

With the use of the Posted Note in Formula Entry, you can string formulas together and have them executed sequentially. Each formula should be entered on a separate line and end with a semicolon. This section provides the general instructions for entering and executing a multi-line formula. Section 2.5.2 provides a detailed example of executing a multi-line formula and viewing its effect on the data window.

To enter and execute a multi-line formula:

1. Open the data window to be operated on by the formula.
2. Choose **Windows** > **Formula Entry**.
3. Click the Posted Note button ( ) to display the text editor.
4. Enter the individual formulas into the Posted Note. Figure 6-3 shows a multi-line formula entered in the text editor. Notice that each formula is entered on a separate line and is terminated by a semicolon.

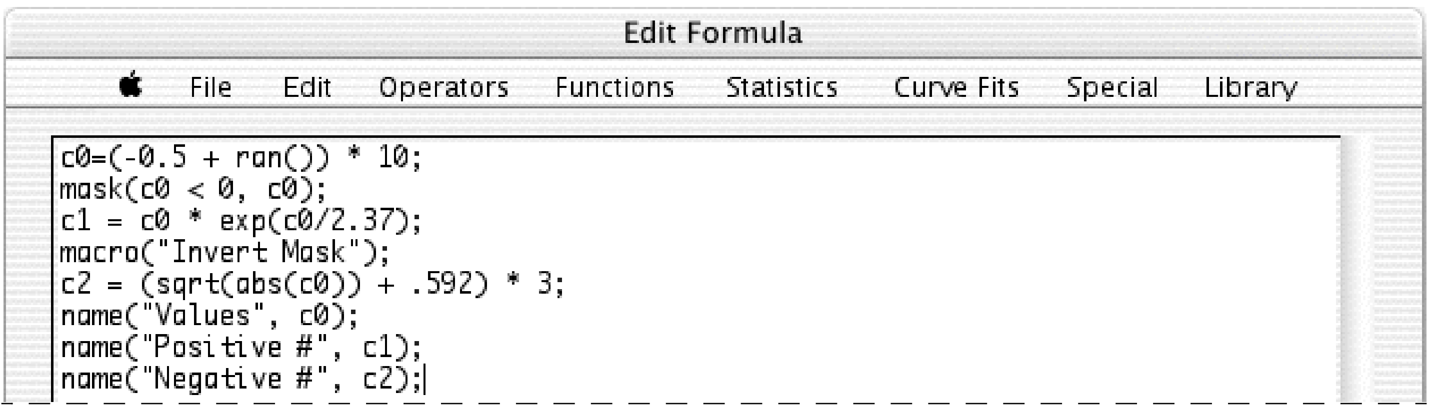


Figure 6-3 Posted Note with formula entered

5. When you are finished entering the formulas, choose **File** > **Close** to leave the editor. The Formula Entry window appears, displaying the message shown in Figure 6-4.

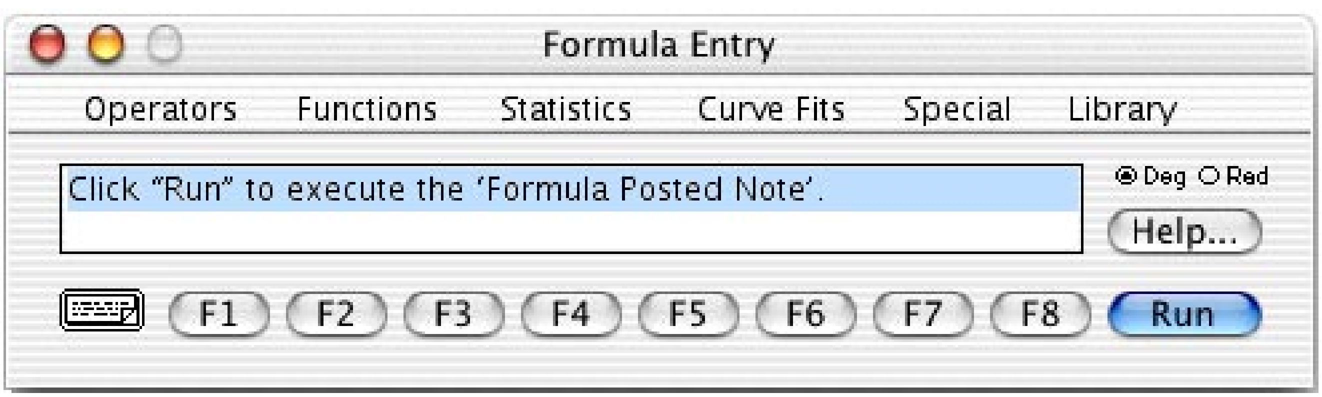


Figure 6-4 Formula Entry after exiting the Posted Note

6. Click **Run** to execute the formulas in the Posted Note.

6.2.1 Adding Comments

Comments can be added to a multi-line formula. Comments are used to make the formula more readable by describing the purpose of the program and the use of each individual formula.

Comments can be added anywhere in the Posted Note with the use of a semicolon. Anything after the semicolon to the first occurrence of a carriage return is treated as a comment. The following lines from the multi-line formula in Figure 6-3 have had comments added.

```
c0 = (–0.5 + ran()) * 10;    Fill c0 with random numbers between –5 and 5.
macro("Invert Mask");    Execute the Invert Mask macro from the Macros menu.
name("Values", c0);   Change the title of c0 to "Values."
```

6.2.2 Saving Multi-Line Formulas

Multi-line formulas can be saved as a text file by choosing **File** > **Save As** in the Posted Note window. This text file can be reopened at a later time by clicking the Posted Note button in Formula Entry and choosing **File** > **Open**.

6.3 Calculating Statistics with Formula Entry

KaleidaGraph provides 12 commands in the **Statistics** menu to help you analyze data. The statistics can be calculated on individual columns or a specific range of data. The equations used to calculate the statistics are listed in Section D.2.

6.3.1 Individual Column Statistics

Statistics are calculated on individual columns by entering the column number within the statistics function. This function results in a single data value, which can be displayed in a dialog or stored in a column, cell, or memory register. The following table shows some example formulas that calculate statistics on individual columns.

Formula	Result
mean(c12)	Finds the mean of column 12 and displays the result in a dialog.
c4 = stderr(c3)	Determines the standard error for column 3 and places the result in column 4.
cell(0,5) = std(c15)	Finds the standard deviation of column 15 and stores the result in row 0 of column 5.
m2 = cmin(c11)	Determines the minimum value in column 11 and stores the result in memory register 2.

6.3.2 Statistics on a Specific Block of Data

Statistics can also be calculated on a range of data. This is accomplished using the statistics commands and the matrix operator, [ ]. This operator enables you to define a range of rows and columns (around the current row or column index) to use in the calculation. The syntax for the matrix operator is:

[starting row:ending row, starting column:ending column]

It is not necessary to enter starting and ending values if you want to use all of the rows or columns in the calculation. If the starting and ending values are missing, the entire range of rows or columns is used to determine the result.

For example:

[,]	Operate on the entire data window.
[, 1:3]	Operate on all rows in columns 1 through 3.
[0:25,]	Operate on rows 0 through 25 in all columns.

The following table shows some examples of statistics commands using the matrix operator. Each of these formulas display the results in a dialog.

Formula	Result
stderr([,])	Calculates the standard error for the entire data window.
mean([, 1:1])	Finds the mean for all rows in column 1.
std([0:10,])	Calculates the standard deviation for rows 0 through 10 in all columns.
median([0:100, 3:5])	Calculates the median for the data in rows 0 through 100 in columns 3 through 5.
std([–10:10, 0:0])	Finds the standard deviation for a window of data from –10 rows to +10 rows from the highlighted row in column 0.
npts([0:1000, –3:3])	Finds the number of points in rows 0 through 1000 in a window of data from –3 columns to +3 columns from the highlighted column.

6.3.3 Calculating Statistics on a Row-by-Row Basis

To calculate statistics on a row-by-row basis, enter zero for both the starting and ending row. For example, the formula c5 = mean([0:0, 1:4]) calculates the mean for each individual row in columns 1 through 4 and stores the results in column 5.

6.4 Masking Data with Formula Entry

KaleidaGraph supports two functions (**mask** and **filter**) that can be used to mask data via a formula. An **unmask** function is also provided to return the data to normal. These commands are defined in Section 6.9. Some example uses of these commands are listed below.

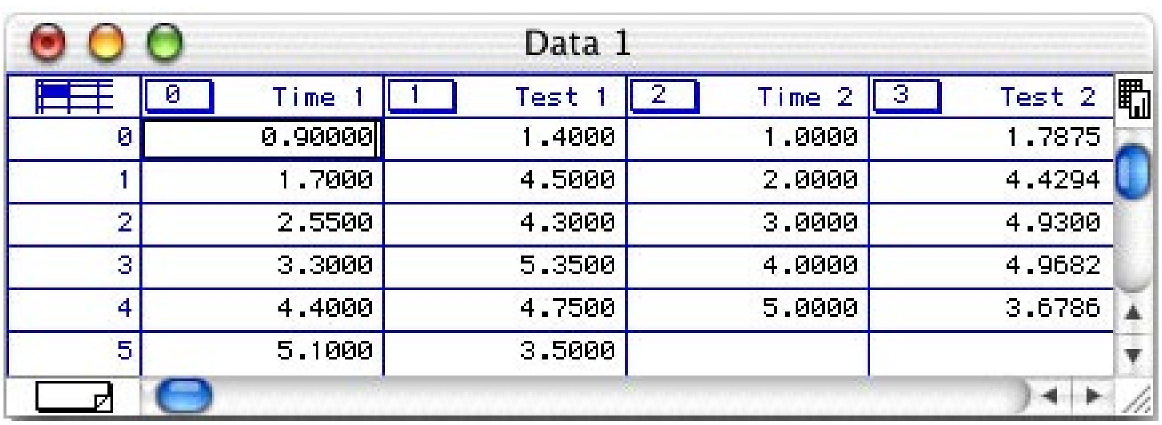
Note: For more information on masking and unmasking data, refer to Section 4.7.

Formula	Result
mask(c0 < 0, c0)	Masks any negative values in c0.
mask(c1 < c2, c2)	Masks any cells in c2 whose value is greater than the corresponding cell in c1.
filter(–1, 1, c0)	Masks all cells in c0 that are less than –1 or greater than 1.
unmask(c0 > 0, c3)	Unmasks any cells in c3 if the corresponding value in c0 is positive.

6.5 Interpolating Data

KaleidaGraph is able to perform linear interpolation through the use of the **table** function in Formula Entry. The table command performs a linear approximation of a number (x) based on the data in two columns. The syntax for this command is **table(x, x col#, y col#)**, such that for a given x value, this function returns an estimate for y.

Figure 6-5 shows an example use of the table function. The first two columns contain X-Y data points. Column 2 contains a new set of X values for which you want to find the corresponding Y values. The Y values in column 3 were found by executing the formula **c3=table(c2, c0, c1)** in Formula Entry.



	0 Time 1	1 Test 1	2 Time 2	3 Test 2
0	0.90000	1.4000	1.0000	1.7875
1	1.7000	4.5000	2.0000	4.4294
2	2.5500	4.3000	3.0000	4.9300
3	3.3000	5.3500	4.0000	4.9682
4	4.4000	4.7500	5.0000	3.6786
5	5.1000	3.5000		

Figure 6-5 Sample data window

6.6 Obtaining Values from a Curve Fit

The **Curve Fits** menu in Formula Entry contains commands that can be used to analyze the values along a fitted curve. These commands enable you to find values at any point on the curve, as well as calculate the residual error between the fitted curve and the original data.

Before you can get values from a curve fit, you must first create a plot and apply a curve fit to the data. Otherwise, an error message is displayed when you try to run the formula.

Note: It is possible to get the curve fit values and residual values into the data window without using Formula Entry. See Section 10.6.3 for more information on exporting the results of a curve fit into the data window.

The following sections show how to get values from a curve fit into the data window. Figure 6-6 shows a plot with a Polynomial curve fit applied (represented by the dashed line) which will be used as an example.

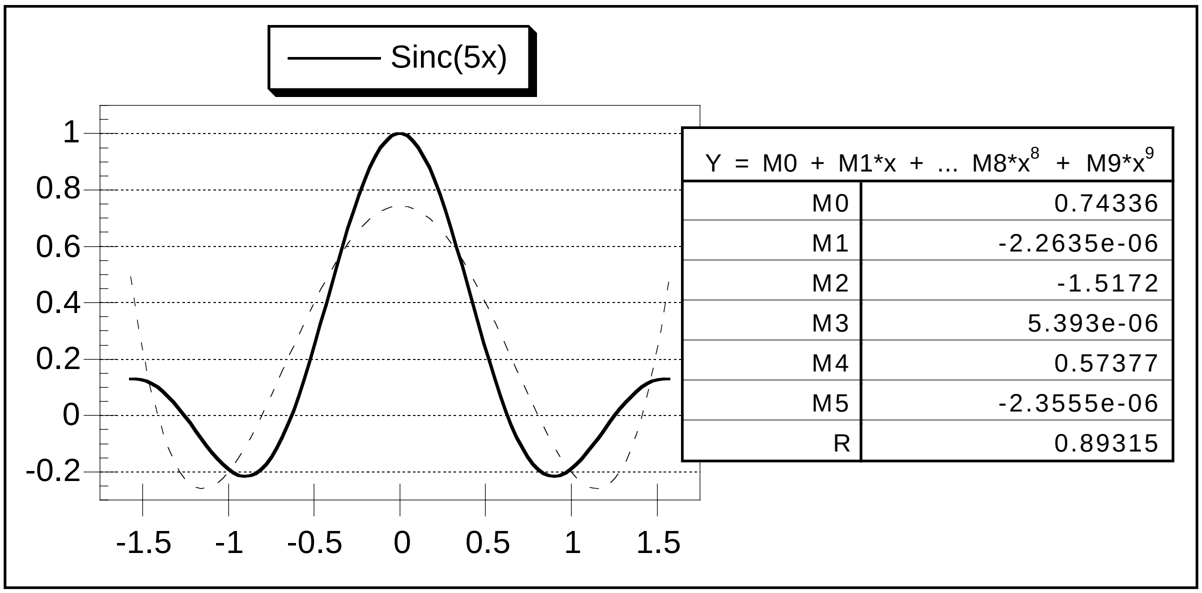


Figure 6-6 Sample plot with Polynomial fit applied

6.6.1 Getting Curve Fit Values into the Data Window

The following steps explain how to use Formula Entry to obtain values from the curve fit at specific X locations. The X positions can be the original column of X values, a new column of values, or a single point along the X axis.

Note: Unless the curve fit is extrapolated to the axis limits (see Section 10.8.1), the results are only accurate if the X values lie within the original range of the data. If the values fall outside the data range, the results are linearly interpolated.

To calculate values from a curve fit:

1. Activate the data window that contains the fitted variables. Figure 6-7 shows a portion of the data used to create the sample plot.

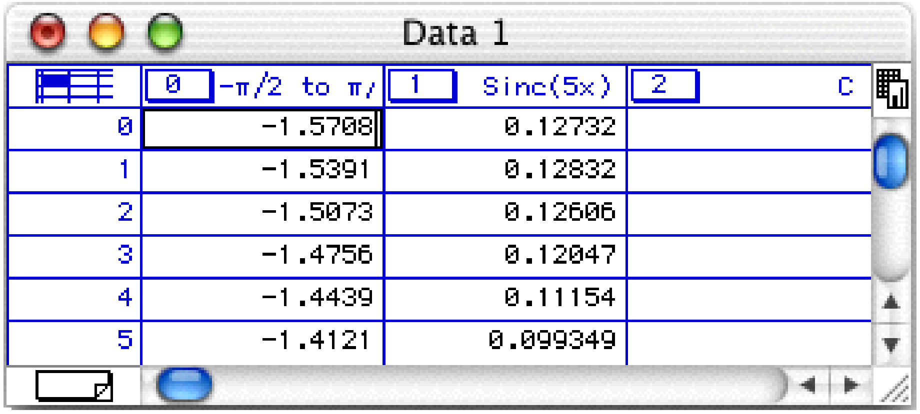


Figure 6-7 Data used to create sample plot

2. Choose **Windows** > **Formula Entry**.

3. Enter the formula into the window. Figure 6-8 shows the formula for this example. The formula should be of the form **z = fit(x,y)**, where:

   z is the destination column.
   x is either a single data value or a column containing X values.
   y is the column that had the curve fit applied.
   fit is replaced by the appropriate command from the **Curve Fits** menu of Formula Entry.

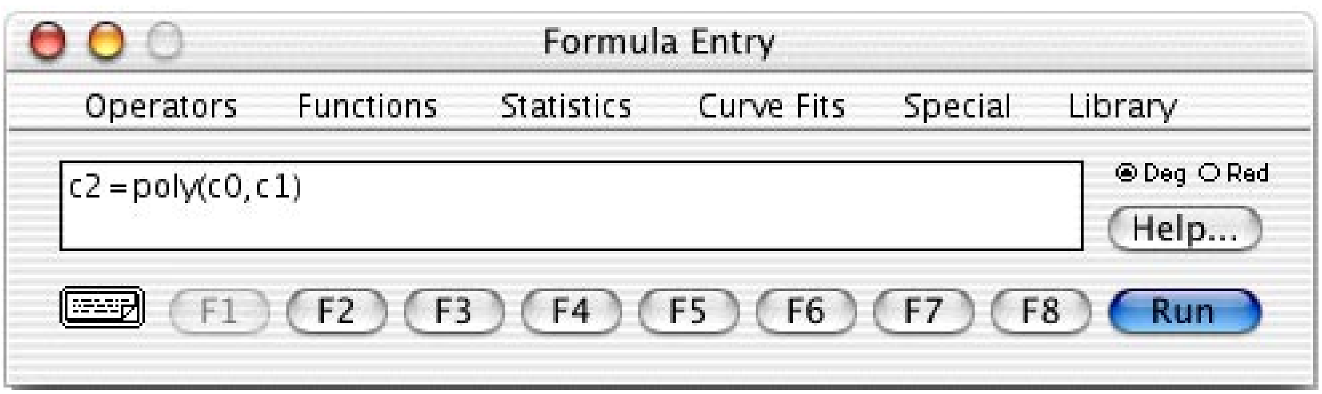
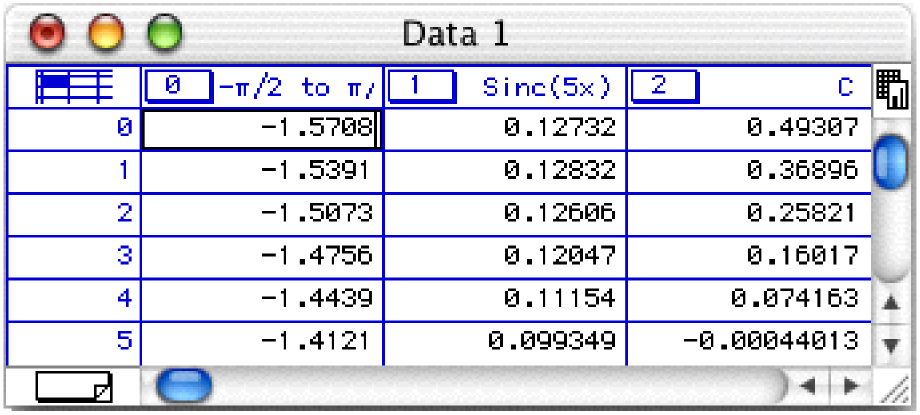


Figure 6-8 Curve fit function entered in Formula Entry

4. Click **Run** to execute the formula. The destination column is filled with the values of the curve fit at the specified X values. Figure 6-9 shows the resulting data window, with the curve fit values in the last column (column 2).



	0 -π/2 to π/	1 Sinc(5x)	2 C
0	-1.5708	0.12732	0.49307
1	-1.5391	0.12832	0.36896
2	-1.5073	0.12606	0.25821
3	-1.4756	0.12047	0.16017
4	-1.4439	0.11154	0.074163
5	-1.4121	0.099349	-0.00044013

Figure 6-9 Data window containing curve fit values

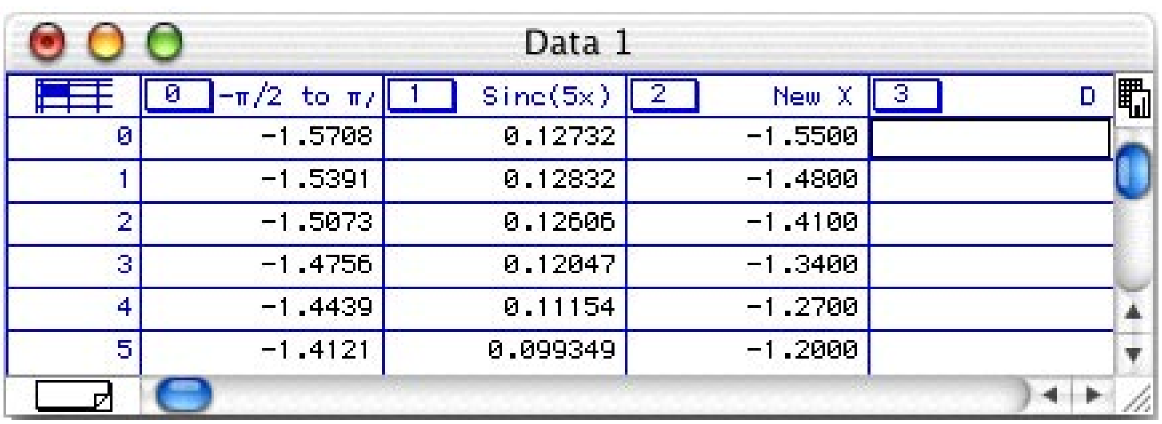
Note: This method uses the curve fit points originally calculated when the fit was applied. You may want to increase the number of calculated points to make the values from the fit more accurate. See Section 10.8.3 for instructions on increasing the number of curve fit points.

6.6.2 Using the Calculated Equation to Find Values

The most accurate method of getting values from a fit is to use the original equation and the parameters calculated by KaleidaGraph. This is not possible when a Smooth, Weighted, Cubic Spline, or Interpolate curve fit is applied to the data, because these curve fits do not result in an equation.

To calculate values using the curve fit's equation:

1. Copy the curve fit coefficients to the Calculator. Refer to Section 10.6.3 if you are unsure of how to do this. If you applied a General curve fit, you can skip this step because the parameters are automatically stored in the Calculator.
2. Activate the data window that contains the fitted variables. Figure 6-10 shows the original data with a column containing new X values (column 2).



	0 -π/2 to π/	1 Sinc(5x)	2 New X	3 D
0	-1.5708	0.12732	-1.5500	
1	-1.5391	0.12832	-1.4800	
2	-1.5073	0.12606	-1.4100	
3	-1.4756	0.12047	-1.3400	
4	-1.4439	0.11154	-1.2700	
5	-1.4121	0.099349	-1.2000	

Figure 6-10 Data window containing new X values

3. Choose **Windows** > **Formula Entry**.
4. Enter your formula into the window. Figure 6-11 shows the formula used to obtain values from a curve fit at designated points, where:

c3 is the destination column for the curve fit values.
c2 is the column containing the new X values.
m0–m5 are the curve fit coefficients previously copied to the Calculator.

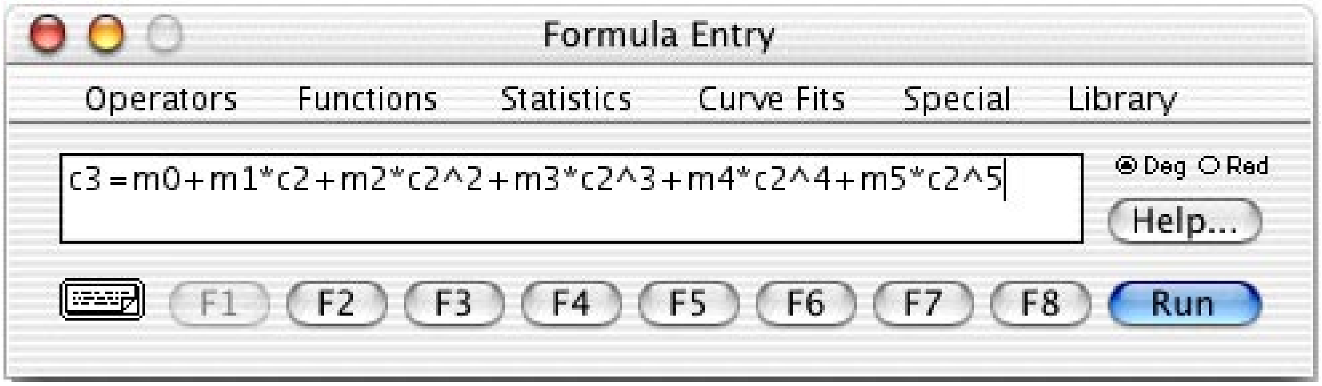


Figure 6-11 Using the curve fit equation to calculate values

5. Click **Run** to execute the formula. The destination column is filled with the values of the curve fit at the specified X values. Figure 6-12 shows the resulting data window, with the curve fit values in the last column (column 3).

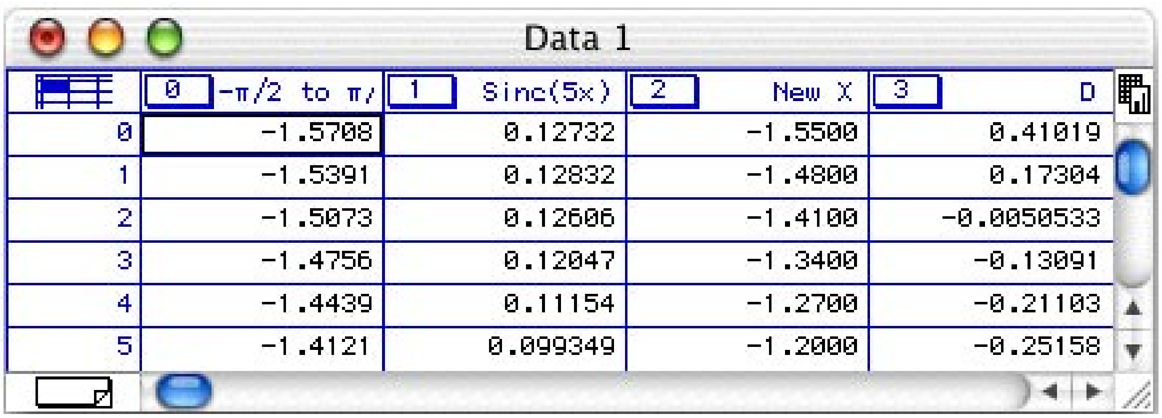


Figure 6-12 Data window containing curve fit values

Note: This method can similarly be used to calculate the X value when the Y value on the fit is known. You need to rearrange the formula so that it solves for X instead of Y.

6.7 Performing Date and Time Calculations

It is possible to perform calculations on columns containing dates and times. Internally, dates are stored as the total number of seconds since Jan. 1, 1904 and times are stored as a relative number of seconds. This allows you to perform calculations on date and time data just as you would with any other kind of data. The easiest way to illustrate this is to use the **Column Format** command (**Data** menu) to change the format of a column containing dates or times to the **Double** format. The column now shows the total number of seconds.

Note: You do not have to change the format of the column before using it in a calculation. This is because KaleidaGraph automatically stores the values as a total number of seconds.

6.8 Using the Library

KaleidaGraph provides a library which can be used to define constants, variables, and functions for use in Formula Entry, the Function plot, and the General curve fit. Using the library, you can create shorthand references to values and functions that you use on a regular basis.

6.8.1 Library Basics

The library is displayed by choosing either **Macros** > **Library** or **Curve Fits** > **General** > **Library**. The current library is displayed in a text editor. Once you are in the editor, you can modify the current library, open a saved library, or create a new library.

The default library provided with KaleidaGraph contains a number of definitions for variables, constants, math functions, and curve fit equations. The definitions are grouped together to make it easier to locate a particular function, but this is not a requirement. You can place the definitions in any order you choose.

One of the main reasons for using the library is to form aliases or shorthand references for functions, variables, and constants. This is particularly useful for defining sections of a formula that exceed the 256 character limit in the Formula Entry, Function plot, and General curve fit windows.

6.8.2 Creating Library Definitions

Any of the definitions in the library can be called from Formula Entry, the Function plot, or the General curve fit. KaleidaGraph searches the current library for any substitutions and makes them before proceeding. Please note that KaleidaGraph substitutes the definition, so you may want to enclose the definition with parentheses to maintain the proper syntax after the substitution takes place.

Use the same syntax shown in the default library to create your own definitions. All of the library definitions are terminated with a semicolon. Anything following the semicolon is treated as a comment, unless the semicolon is preceded by a backslash (\). The purpose of the backslash is to treat the data after the semicolon as a continuation of the function. This is normally used to set up initial conditions when defining curve fit equations.

When creating your shorthand references, it is important to note that the library is not case sensitive. If you had two functions named S1() and s1(), either one could be substituted by KaleidaGraph.

6.8.3 Creating a Custom Library

The library is saved as part of the Macros file. To create a custom library, choose **Macros** > **Library** and save the default library to disk (using the **Save As** command in the editor). Then, choose **File** > **New** to start a new library. Use the same syntax shown for the default library. When completed, save the new library to disk.

KaleidaGraph references whichever library is currently open. To reference a new library, use the **Open** command in the editor to locate and load the library you want to use. If you want to tie a particular library to a macros file, use the **File** > **Export** > **Macros** command to save the macros file. The current library becomes the default library for that particular macros file.

6.9 Function Descriptions and Syntax

This section describes each of the operators and functions available in the Formula Entry menus. Examples are provided for each of the items.

6.9.1 Operators Menu

( ) - The parentheses are used with almost every formula to enclose the variable being used in the calculation. They are also used to set the priority order for calculations.

Example: c3 = (cos(c2) * sin(c1)) + ((2 * tan(c0)) / sqrt(pi))

^ - This operator raises the preceding value to the specified power.

Examples: 5^2, c3 = c1^(1/c0)

* - This operator multiplies two values together.

Examples: 23 * m0, c3 = .9087 * c5

/ - This operator is used for division.

Examples: c4 = c0 / 12.234, c12 = (c8 + c5) / c2

+ - This operator is used for addition.

Examples: c3 = (c2 + c1 + c0) / 3, c12 = 198 + c10

– - This operator is used for subtraction.

Examples: c9 = c8 – c6, c19 = (c12 – c3) / c5

% - This operator performs the **mod** function.

Examples: c9 = c8 % 4, c13 = c12 % c3

! - This operator calculates the factorial of a number.

Examples: 7!, c13 = c10!

< - This operator compares two values to see if one is less than the other.

Examples: mask(c1 < 3, c1), unmask(c2 < –12, c3)

<= - This operator compares two values to see if one is less than or equal to the other.

Examples: mask(c4 <= c5, c6), unmask(c2 <= 23, c12)

> - This operator compares two values to see if one is greater than the other.

Examples: mask(c1>1, c0), unmask(c4 > c3), c5)

>= - This operator compares two values to see if one is greater than or equal to the other.

Examples: mask(c12 >= 12.34, c12), unmask(c11 >= –5, c11)

== - This operator compares two values to see if they are equal to each other.

Examples: mask(c1 == 4, c1), unmask(c3 == c2, c1)

!= - This operator compares two values to see if they are unequal.

Examples: mask(c2 != c4, c2), unmask(c3 != 2, c3)

&& - This operator performs a **logical and** on two Boolean expressions. The result of both expressions must be true (non-zero) for the logical and to be true.

Examples: c3 = (c0>1 && c0<15) ? c0 : 0.5, c1 = (c2>3 && c2<11) ? c2 : 0

|| - This operator performs a **logical or** on two Boolean expressions. The result of one expression must be true (non-zero) for the logical or to be true.

Examples: c1 = (c2>1 || c3>1) ? c1 : c4, c2 = (c1<0 || c1 == 0.5) ? 0 : c1

?: - This operator performs a conditional test on a Boolean expression. This operator can be expressed in one of two ways:

- if (Boolean) expression 1 else expression 2
- (Boolean) ? expression 1 : expression 2

If the result of the Boolean is true (non-zero), return expression 1; if it is false, return expression 2.

Examples: c9 = (c2>.5 || c3>.1) ? c11 : c4, c1 = (c2>1 && c2<10) ? c2 : 0.1

= - This operator is used to place the result of a calculation into a column, cell, or memory location.

Examples: m0 = 12 * m1, c1 = sqrt(c0), c2 = cos(c0), cell(0,5) = cmin(c0)

[ ] - The matrix operator is used to specify a range of data for use in a statistical calculation. The syntax of the operator is: [starting row:ending row, starting column:ending column].

It is not necessary to enter starting and ending values to use all of the rows or columns in the calculation. If the starting and ending values are missing, the entire range of rows or columns is used to determine the result

Examples: cmax([0:23, 3:10]), c18 = median([, 12:14])

6.9.2 Functions Menu

abs(x) - Determines the absolute value of a number.

Examples: abs(–5), abs(m3), c3 = abs(c2)

cell(row#, col#) - Gets or sets the value of a specific cell.

Example: cell(0, 2) = 5.5 * cell(1, 3), cell(10, 13) = sqrt(cell(15, 11))

cos(x) - Determines the cosine of a number. The number is in radians or degrees, based on the setting in the Formula Entry window.

Examples: cos(30), cos(m8), c4 = cos(c2)

diff(x) - Calculates the difference between each value and the one that follows it.

Examples: c5 = diff(c2), c11 = diff(c10)

erf(x) - Calculates the error function of a value. This function has the following limiting values: erf(0) = 0, erf(∞) = 1, and erf(–x) = –erf(x).

Examples: erf(2), c10 = erf(c9)

erfc(x) - Calculates the complementary error function of a value. This function has the following limiting values: erfc(0) = 1, erfc(∞) = 0, and erfc(–x) = 2–erfc(x).

Examples: erfc(.5), c20 = erfc(c19)

exp(x) - Calculates the value of e (2.7182...) raised to a power.

Examples: exp(3), exp(m1), c10 = exp(c2)

fract(x) - Returns the fractional portion of a number.

Examples: fract(2.83), c12 = fract(c9)

index() - Returns the current row number (starting at 0).

Example: c5 = index() * .5, c11 = index() – 64

inorm(x) - Determines the inverse of the normal distribution for a number between zero and 100%.

Examples: inorm(25), inorm(m2), c10 = inorm(c9)

int(x) - Returns the integer portion of a number.

Examples: int(2.83), c12 = int(c9)

invcos(x) - Determines the inverse cosine of a number. The number is in radians or degrees, based on the setting in the Formula Entry window.

Examples: invcos(.5), invcos(m3), c6 = invcos(c5)

invsin(x) - Determines the inverse sine of a number. The number is in radians or degrees, based on the setting in the Formula Entry window.

Examples: invsin(.219), invsin(m0), c15 = invsin(c10)

invtan(x) - Determines the inverse tangent of a number. The number is in radians or degrees, based on the setting in the Formula Entry window.

Examples: invtan(1), invtan(m2), c9 = invtan(c8)

ln(x) - Calculates the natural logarithm (base e) of a number.

Examples: ln(8), ln(m5), c12 = ln(c9)

log(x) - Calculates the common logarithm (base 10) of a number.

Examples: log(20), log(m10), c18 = log(c15)

norm(x) - Determines the normal distribution of a number between zero and 100%.

Examples: norm(23), norm(m5), c7 = norm(c5)

pi - Represents the value of π (3.1415926...).

Example: c10 = pi * c9, c11 = 3 * pi * c9

ran() - Generates a random number between zero and one.

Examples: c15 = ran(), c9 = c8 * ran()

rsum(col#) - Computes the running sum of a column.

Example: c3 = rsum(c2), c10 = rsum(c9)

sin(x) - Determines the sine of a number. The number is in radians or degrees, based on the setting in the Formula Entry window.

Examples: sin(45), sin(m6), c11 = sin(c10)

sqrt(x) - Calculates the square root of a number.

Examples: sqrt(192), sqrt(m9), c6 = sqrt(c5)

table(x, x col#, y col#) - The table command performs a linear approximation of a number (x) based on the data in two columns. The result is that for a given x, this function returns an estimate for y. The order of the columns determines what value is returned. This function has the following constraints:

- You cannot have masked or empty cells in either the x col# or y col#.
- y col# = f(x col#) may only describe a single value function.
- It does not operate on a subset of rows within a column.

Examples: table(2.5, c0, c1), c4 = table(c3, c0, c1)

tan(x) - Determines the tangent of a number. The number is in radians or degrees, based on the setting in the Formula Entry window.

Examples: tan(68), tan(m10), c5 = tan(c4)

6.9.3 Statistics Menu

cmin(x) - Determines the minimum value within the specified data.

Examples: cmin(c5), cmin([, 0:10]), c2 = cmin([,])

cmax(x) - Determines the maximum value within the specified data.

Examples: cmax(c15), cmax([0:23, 3:10]), c13 = cmax([, 0:12])

csum(x) - Calculates the sum of the data values.

Examples: csum(c9), csum([, 0:3]), c12 = csum([0:50, 3:8])

kurtosis(x) - Calculates the kurtosis of the specified data.

Examples: kurtosis(c12), kurtosis([0:5,]), c5 = kurtosis([0:25, 0:1])

mean(x) - Calculates the mean of the data.

Examples: mean(c0), mean([, 2:3]), c11 = mean([,])

median(x) - Calculates the median of the data range.

Examples: median(c3), median([2:10, 0:1]), c17 = median([, 12:14])

npts(x) - Determines the number of points within the specified data range.

Examples: npts(c1), npts([, 0:5]), c10 = npts([,])

rms(x) - Calculates the RMS (root mean square) of the data range.

Examples: rms(c3), rms([, 1:1]), c5 = rms([12:21,])

skew(x) - Calculates the skewness of the data range.

Examples: skew(c15), skew([, 3:10]), c13 = skew([–10:10, 2:2])

std(x) - Calculates the standard deviation of the data range.

Examples: std(c10), std([12:120,]), c3 = std([,])

stderr(x) - Calculates the standard error of the data.

Examples: stderr(c4), stderr([–5:5, 15:19]), c3 = stderr([,])

var(x) - Calculates the variance of the specified data range.

Examples: var(c6), var([0:25,]), c3 = var([,])

6.9.4 Curve Fits Menu

expr(x, col#) - Given an x value and a column that has an Exponential curve fit applied, this function determines the value from the fit at the x point.

Examples: expr(105, c10), c11 = expr(c8,c9)

gen(" ", x, col#) - Given the name of a General curve fit, an x value, and a column that has the fit applied, this function determines the value at the x point. If no name is given, the first General fit that is found in the column is used.

Examples: gen("fit1", 22, c1), c5 = gen(c0, c1), c11 = ("gaussian", c8, c9)

interp(x, col#) - Given an x value and a column that has an Interpolate curve fit applied, this function determines the value from the fit at the x point.

Examples: interp(11.95, c2), c19 = interp(c15, c18)

lin(x, col#) - Given an x value and a column that has a Linear curve fit applied, this function determines the value from the fit at the x point.

Examples: lin(–12.5, c1), c10 = lin(c9, c5)

logr(x, col#) - Given an x value and a column that has a Logarithmic curve fit applied, this function determines the value from the fit at the x point.

Examples: logr(1300, c8), c12 = logr(c3, c4)

poly(x, col#) - Given an x value and a column that has a Polynomial curve fit applied, this function determines the value from the fit at the x point.

Examples: poly(.0034, c15), c14 = poly(c12, c1)

pow(x, col#) - Given an x value and a column that has a Power curve fit applied, this function determines the value from the fit at the x point.

Examples: pow(119, c10), c3 = pow(c0, c2)

smooth(x, col#) - Given an x value and a column that has a Smooth curve fit applied, this function determines the value from the fit at the x point.

Examples: smooth(11.15, c2), c4 = smooth(c2, c3)

spline(x, col#) - Given an x value and a column that has a Cubic Spline curve fit applied, this function determines the value from the fit at the x point.

Examples: spline(–23.098, c17), c4 = spline(c3, c8)

wgt(x, col#) - Given an x value and a column that has a Weighted curve fit applied, this function determines the value from the fit at the x point.

Examples: wgt(15.657, c2), c9 = wgt(c10, c11)

6.9.5 Special Menu

execute("filename") - This function loads a formula or formula script into the Posted Note and executes it. This function can only be used in the Formula Entry window; it cannot be used in the Posted Note window.

Examples: execute("Macintosh HD:Scripts:script1"), execute("C:\KaleidaGraph\Scripts\script1.txt")

macro(" ") - This function executes a macro by name.

Examples: macro("Simplify"), macro("Invert Mask")

mask(logical expression, col#) - Masks the cells in the specified column if the expression is true.

Examples: mask(c4 > 1.25, c6), mask(c9 != c11, c14)

name(" ", col#) - This command uses the text string given to name the specified column.

Examples: name("Y-Error", c2), name("Residuals", c11)

script("filename") - This function loads and runs the specified plot script. If no name is given, the current script is executed.

Examples: script("Scatter Plot Script"), script("Temperature"), script()

unmask(logical expression, col#) - Unmasks the cells in the specified column if the expression is true.

Examples: unmask(c2 == c4, c5), unmask(c2 < 4, c3)

6.9.6 Library Menu

The **Library** menu lists all of the definitions that are present in the current library. If the library is empty, this menu is also empty.

6.10 Formula Scripting

KaleidaGraph supports an internal set of scripting commands in the Posted Note of Formula Entry. Using these commands, you can automate the process of opening and closing data windows, creating plots, and exporting plots to a printer or the Clipboard. Scripts are particularly useful if you repeatedly open the same types of text files, perform the same set of data manipulations, and generate the same types of plots.

6.10.1 Creating a Formula Script

Formula scripts are similar to the multi-line formulas that were discussed in Section 6.2. The main difference is that there are a number of additional commands that are supported by KaleidaGraph. These commands enable you to open data files, create plots, print plots, and export the plots in a variety of formats.

All of the formula script commands are discussed in Section 6.10.2. In addition, the #FORMULA command supports all of the functions that are available in the Formula Entry window, except for the **execute** command

To enter and execute a formula script:

1. Verify that a data window is already open. The script cannot execute unless a data window is available.
2. Choose **Windows** > **Formula Entry**.
3. Click the Posted Note button ( ) to display the text editor.
4. Enter the formula script into the Posted Note.
5. When you are finished entering the script, choose **File** > **Close** to leave the Posted Note. You will return to Formula Entry, which displays the message shown in Figure 6-13.

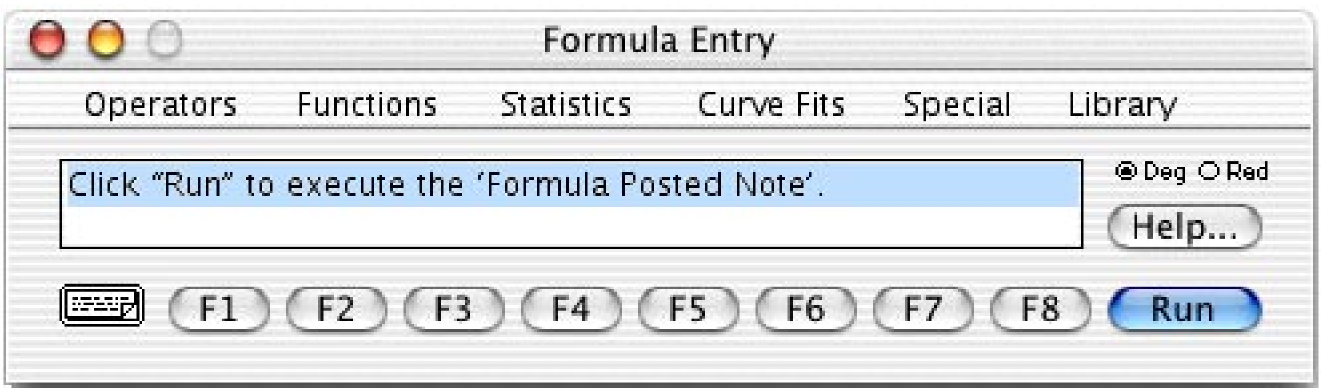


Figure 6-13 Formula Entry after exiting the Posted Note

6. Click **Run** to execute the script in the Posted Note.

You can save the scripts using the **Save As** command from the **File** menu in the Posted Note window. Once the script has been saved, it can be reopened by choosing **File** > **Open** in the Posted Note dialog. Alternately, you can use the **execute** function in Formula Entry to load and execute a saved formula script.

6.10.2 Formula Script Commands

Definitions

BOOLEAN	yes/no/true/false
STRING	"255 character string surrounded by quotes"

The format for specifying a path to a file differs between the Windows and Macintosh versions. Under Windows, the path starts at the hard disk, with subdirectories separated by a backslash (\). On a Macintosh, the path starts at the hard disk, with subdirectories separated by a colon (:).

The equal sign (=) is optional in this syntax. Anywhere the equal sign is used, the command is also valid without it.

The commands are case sensitive. The examples for each command show how it should be entered in the Posted Note window.

Comments can be added between formula script commands with the use of a semicolon. Any lines that begin with a semicolon are treated as comments. Place comment lines before or after individual formula script procedures. A syntax error message is displayed if the comment is included within the formula script command definition.

List of Commands

#CLEAR - This command deletes the selection in the active data window. The cells below the selection shift up to fill the void. You may optionally specify a selection as part of the command options. If no selection is specified, the current selection in the window is used. You may also specify whether to delete column titles in addition to the data. The syntax for this command follows:

```
#CLEAR
[title = BOOLEAN]   [Start_Row   End_Row   Start_Column   End_Column]
#END
```

The numbers should be separated by any white space characters. As a special case, if Start Row has a value of –1, the entire window is selected. The active cell in the data window is moved to the location specified by Start Row, Start Column. The selection addresses for both row and column positions begin at zero and are counted from the upper-left corner of the data window (position 0,0). The default action is to not delete titles.

Example: Delete the data in rows 0–50 of columns 10–20.

```
#CLEAR
title = false 0  50  10  20
#END
```

#CLEARDATA - This command clears the selection in the active data window. You may optionally specify a selection as part of the command options. If no selection is specified, the current selection in the window is used. The syntax for this command follows:

```
#CLEARDATA
[Start_Row   End_Row   Start_Column   End_Column]
#END
```

The numbers should be separated by any white space characters. As a special case, if Start Row has a value of –1, the entire window is selected. The active cell in the data window is moved to the location specified by Start Row, Start Column. The selection addresses for both row and column positions begin at zero and are counted from the upper-left corner of the data window (position 0,0).

Example: Clear the data in rows 50-100 of columns 15–25.

```
#CLEARDATA
50 100 15 25
#END
```

#CLOSE - This command closes the active window, without saving its contents. The syntax for this command follows:

```
#CLOSE
```

#CLOSE/ALL - This command closes all visible windows, without saving their contents. The syntax for this command follows:

```
#CLOSE/ALL
```

#CLOSE/OPT - This command closes the specified plot or data window. You have the option of saving the window before closing it. Only plot and data windows should be taken into account when determining the position number of the window. The syntax for this command follows:

```
#CLOSE/OPT
window = #    (the position of the window from front to back, where the frontmost window is position 1)
save = BOOLEAN
wintype = plot/data    (optional)
#END
```

Example: Save and close the frontmost data window.

```
#CLOSE/OPT
window = 1
save = true
wintype = data
#END
```

#COPY - This command copies the selection in the active data window to the Clipboard. You may optionally specify a selection as part of the command options. If no selection is specified, the current selection in the window is used. You may also specify whether to copy column titles in addition to any data. The syntax for this command follows:

```
#COPY
[title = BOOLEAN]   [Start_Row   End_Row   Start_Column   End_Column]
#END
```

The numbers should be separated by any white space characters. As a special case, if Start Row has a value of –1, the entire window is selected. The active cell in the data window is moved to the location specified by Start Row, Start Column. The selection addresses for both row and column positions begin at zero and are counted from the upper-left corner of the data window (position 0,0). The default action is to not return titles.

Example: Copy the selection in rows 5–15, columns 1–3 (no titles).

```
#COPY
5  15  1  3
#END
```

Example: Copy the selection in rows 0–10, columns 1–2 (with titles).

```
#COPY
title = true 0  10  1  2
#END
```

#CUT - This command cuts the selection in the active data window. The cells below the selection shift up to fill the void. You may optionally specify a selection as part of the command options. If no selection is specified, the current selection in the window is used. You may also specify whether to include the column titles with the data. The syntax for this command follows:

```
#CUT
[title = BOOLEAN]   [Start_Row   End_Row   Start_Column   End_Column]
#END
```

The numbers should be separated by any white space characters. As a special case, if Start Row has a value of –1, the entire window is selected. The active cell in the data window is moved to the location specified by Start Row, Start Column. The selection addresses for both row and column positions begin at zero and are counted from the upper-left corner of the data window (position 0,0). The default action is to not include titles.

Example: Cut the data in rows 100–500 of columns 0–20, without including the titles.

```
#CUT
title = false 100  500  0  20
#END
```

#DATA - This command places tab-separated data in a new data window. The first row should contain titles. The syntax for this command follows:

```
#DATA
(First row contains tab-separated titles.)
(The remaining rows contain tab-separated data.)
#END
```

Example: Use the first row for titles and place the remaining data in a new data window.

```
#DATA
Time	Test#1	Test#2
1.0	38.6	39.7
1.5	41.2	40.9
2.0	42.7	42.3
#END
```

#DATA/NOTITLE - This command places tab-separated data in a new data window. The first row should contain data. The syntax for this command follows:

```
#DATA/NOTITLE
(All rows contain tab-separated data.)
#END
```

Example: Place the data in a new data window.

```
#DATA/NOTITLE
1.0    38.6    39.7
1.5    41.2    40.9
2.0    42.7    42.3
#END
```

#DATAFILE - This command loads the data file named in the string. If the file is a text file, it is loaded using the specified parameters. The syntax for this command follows:

#DATAFILE
file = STRING (This string should be either a simple name or a fully or partially qualified path to the file. The Open dialog is displayed if the path or file name is incorrect, or if "*.*" is used for the string.)
delimiter = tab/space/special/STRING (if STRING is used, only the first character of the string is important)
skip = # (the number of lines to be skipped at the beginning of the file)
read_titles = BOOLEAN
del_number = # (0 -> del == 1, 1 -> del >= 1, 2 -> del >= 2, 3 -> del >= 3)
title_format = STRING (only used when delimiter = special)
data_format = STRING (only used when delimiter = special)
#END

Example: Display the Open dialog to open any data file.

```
#DATAFILE
file = "*.*"
#END
```

Example: Open a comma-delimited text file. The path uses a Windows format.

```
#DATAFILE
file= "C:\kgraph\Data_1.txt"
delimiter = ","
#END
```

Example: Open a text file using the special format. The path uses a Macintosh format.

```
#DATAFILE
file = "Macintosh HD:Data:Test #1"
delimiter = special
skip = 0
read_titles = TRUE
title_format = "3(a)"
data_format = "A f3 2(A v)"
#END
```

#FORMULA - This command executes a list of standard formulas. A semicolon should be placed at the end of each formula. It is valid to omit the #FORMULA and #END statements if you wish. The syntax for this command follows:

#FORMULA
(List of standard formulas.)
#END

Example: Calculate the average of two columns and give the output column a title.

```
#FORMULA
c3 = (c1 + c2) /2;
name("Average", c3);
#END
```

#MERGEFILE - This command loads the data file named in the string, starting at the specified row and column position. The file is loaded using the specified parameters. The syntax for this command follows:

#MERGEFILE
file = STRING   (This string should be either a simple name or a fully or partially qualified path to the file. The Open dialog is displayed if the path or file name is incorrect, or if "*.*" is used for the string.)
delimiter = tab/space/special/STRING   (if STRING is used, only the first character of the string is important)
skip = #   (the number of lines to be skipped at the beginning of the file)
read_titles = BOOLEAN
del_number = #   (0 -> del == 1, 1 -> del >= 1, 2 -> del >= 2, 3 -> del >= 3)
title_format = STRING   (only used when delimiter = special)
data_format = STRING   (only used when delimiter = special)
position = # #   (row number and column number separated by a tab)
#END

Example: Display the Open dialog to merge any data file.

```
#MERGEFILE
file = "*.*"
#END
```

Example: Merge a comma-delimited text file. The path uses a Windows format.

```
#MERGEFILE
file= "C:\kgraph\Data_1.txt"
delimiter = ","
#END
```

#METAFILE - This command exports the active plot or layout window as a Metafile image via the Clipboard. This command is only available under Windows. The syntax for this command follows:

#METAFILE

#METAFILE/OPT - This command exports the active plot or layout window as a Metafile image to a file or the Clipboard, using the specified parameters. It is not possible to export the plot as an OLE object using this command. This command is only available under Windows. The syntax for this command follows:

#METAFILE/OPT
scale = #   (ranging from 1 to 1000)
hires = BOOLEAN
min_whitespace = BOOLEAN
file = STRING
#END

Example: Export the active plot as a Metafile image at 75%.

```
#METAFILE/OPT
scale = 75
hires = TRUE
file = "C:\Images\Image_1.wmf"
#END
```

#OPEN - This command opens the specified file. It can be used to open any file that KaleidaGraph recognizes. The syntax for this command follows:

#OPEN
file = STRING (This string should be either a simple name or a fully or partially qualified path to the file. The Open dialog is displayed if the path or file name is incorrect, or if "*.*" is used for the string.)
#END

Example: Open a saved Style file. The path uses a Macintosh format.

```
#OPEN
file = "Macintosh HD:KaleidaGraph:KG Style"
#END
```

Example: Open a saved Layout file. The path uses a Windows format.

```
#OPEN
file = "C:\KGraph\Layouts\Layout_1.qpl"
#END
```

#PAGE_PRINT - This command prints one or more plot windows using the current settings in the layout window. The syntax for this command follows:

#PAGE_PRINT

#PASTE - This command pastes text or binary data from the Clipboard into the active data window. You may optionally specify a selection as part of the command options. If no selection is specified, the current selection in the window is used. You may also specify whether to paste column titles in addition to any text data. The syntax for this command follows:

#PASTE
[title = BOOLEAN] [Start_Row End_Row Start_Column End_Column]
#END

The numbers should be separated by any white space characters. As a special case, if Start Row has a value of –1, the entire window is selected. The active cell in the data window is moved to the location specified by Start Row, Start Column. The selection addresses for both row and column positions begin at zero and are counted from the upper-left corner of the data window (position 0,0). The default action is to not return titles.

Example: Replace rows 0–10, columns 1–2 with the data on the Clipboard (with titles).

```
#PASTE
title = true 0  10  1  2
#END
```

#PICT - This command exports the active plot or layout window as a PICT image via the Clipboard. The syntax for this command follows:

```
#PICT
```

#PICT/OPT - This command exports the active plot or layout window to a file or the Clipboard, using the specified parameters. The hires_pict parameter is only available for PICT (Macintosh version only) and TIFF files. The post_pict parameter is only available for PICT files on a Macintosh. The syntax for this command follows:

```
#PICT/OPT
type = pict/bmp/gif/jpeg/png/tiff
scale = #    (ranging from 1 to 1000)
hires_pict = BOOLEAN (Same as selecting High Resolution when exporting PICT or TIFF images)
post_pict = BOOLEAN    (Only use in the Macintosh version when type=pict.)
min_whitespace = BOOLEAN
file = STRING
#END
```

Example: Export the active plot as a high resolution PICT file at 75%. The path uses a Macintosh format.

```
#PICT/OPT
type = pict
scale = 75
hires_pict = TRUE
file = "Macintosh HD:PICT Folder:Test file"
#END
```

#PLOT_PRINT - This command prints the active plot window. The syntax for this command follows:

```
#PLOT_PRINT
```

#SAVE - This command saves the specified plot or data window. Only plot and data windows should be taken into account when determining the position number of the window. The syntax for this command follows:

```
#SAVE
window = #    (the position of the window from front to back, where the frontmost window is position 1)
wintype = plot/data    (optional)
file = STRING
#END
```

Example: Save the second plot window from the front. The path uses a Windows format.

```
#SAVE
window = 2
wintype = plot
file = "C:\KGraph\Test Plot.qpc"
#END
```

#SCRIPT - This command can be used to either load a specific script file or create a script from scratch. The y2 options in the script are only used for Double Y and Double XY plots. The x2 options in the script are only used for Double X and Double XY plots. The only way to use a template plot, apply curve fits, or apply error bars is to open a saved plot script file. The syntax for this command follows:

#SCRIPT
file = STRING (Load the script file. This file should be in the same folder as the last script file that was opened or specified by a fully qualified path. The Open dialog is displayed if the path or file name is incorrect, or if "*.*" is used for the string.)
x/y/x2/y2 axis title STRING (These commands are valid for the x, y, x2, and y2 axes.)
x/y/x2/y2 axis limits # # (Limits for dates or times can be entered as a STRING.)
x/y/x2/y2 axis auto
x/y/x2/y2 axis linear/log
auto_print = boolean
print_mode = single/layout
auto_save = boolean
save_mode = with_data/without_data/pict/bitmap/metafile/bmp/gif/jpeg/png/tiff
prefix = STRING (The character limit for the string is 38 (Windows) and 30 for the file name (Macintosh))
plot_type =line/scatter/double_y/double_x/double_xy/hilo/step/probability/xy_probability/histogram/box/percentile/bar/stack_bar/column/ stack_column/polar/pie
begin_group (select variables, plot title, and legend text for the group)
title = STRING (one title per group)
legend = STRING
legend = STRING
x = # (select X column number)
y = # (select Y column number)
y2 = # (if using a Double Y or Double XY plot, select Y2 column number)
x2 = # (if using a Double X or Double XY plot, select X2 column number)
window = # (Specify a data window for plotting. The default is the frontmost data window (window 1). The last window specified is remembered until a new script is encountered or a new window number is specified.)
end_group
(Add as many groups as you need to the script. Each group creates a new plot.)
#END

Example: Display the Open dialog to load and execute a plot script.

```
#SCRIPT
file = "*.*"
#END
```

Example: Create and save a Scatter plot from two data windows.

```
#SCRIPT
auto_save = TRUE
save_mode = with_data
prefix = "1/31_Group"
plot_type = scatter
begin_group
title = "Lot 10938"
x = 0
y = 1
y = 2
window = 2
x = 0
y = 1
y = 2
end_group
#END
```

Example: Create a Line plot from two sets of X-Y data in the same data window.

```
#SCRIPT
plot_type = line
begin_group
x = 0
y = 1
x = 2
y = 3
end_group
#END
```

Example: Create two Double Y plots from a single data window.

```
#SCRIPT
plot_type = double_y
begin_group
x = 0
y = 1
y2 = 2
end_group
begin_group
x = 3
y = 4
y2 = 5
end_group
#END
```

#SELECTION - This command sets the selection in the active data window. The syntax for this command follows:

```
#SELECTION
Start_Row   End_Row   Start_Column   End_Column
#END
```

The numbers should be separated by any white space characters. As a special case, if Start Row has a value of –1, the entire window is selected. The active cell in the data window is moved to the location specified by Start Row, Start Column. The selection addresses for both row and column positions begin at zero and are counted from the upper-left corner of the data window (position 0,0).

Example: In the active data window, select rows 5–15, in columns 1–3.

```
#SELECTION
5  15  1  3
#END
```

Example: Select the entire data window.

```
#SELECTION
–1
#END
```

#SET_DIRECTORY - This command sets the default directory. The syntax for this command follows:

#SET_DIRECTORY
directory = STRING (STRING is a fully or partially qualified path to the desired directory. Note: The path should end in a backslash (Windows) or colon (Macintosh) if the last item is not a file.)
#END

Example: Move up one directory on a Macintosh.

```
#SET_DIRECTORY
directory = "::"
#END
```

Example: Move up one directory under Windows.

```
#SET_DIRECTORY
directory = ".."
#END
```

Example: Change the current directory to the Sample Data folder on a Macintosh.

```
#SET_DIRECTORY
directory = "Macintosh HD:KaleidaGraph:Data:"
#END
```

Example: Set the default directory to be the program directory.

```
#SET_DIRECTORY
directory = ""
#END
```

6.10.3 Example Formula Scripts

The examples in this section follow the same basic format. An overview describes the goal of the script. Then the example script is listed, just as it would appear if it were being entered directly into the Posted Note of the Formula Entry window. Following the script is an explanation of each step in the script.

Note: The **Scripts** folder (**Examples** folder) contains these three example formula scripts. The paths in the scripts have been modified to be correct for your particular platform.

Basic Example

This example script performs the following functions:

- Selects a portion of the data window.
- Executes four Formula Entry functions, two of which generate columns of data.
- Copies the data and the titles to the Clipboard.
- Pastes the information into another part of the data window.

The example script is listed below, followed by a description of how it operates.

```
#SELECTION
0 99 0 1
#END

#FORMULA
c0 = index() + 1;
c1 = log(c0);
name("Series", c0);
name("log(x)", c1);
#END

#COPY
title = true –1
#END

#PASTE
title = true 10 110 2 2
#END
```

The #SELECTION command selects rows 0 through 99 in the first two data columns. The #FORMULA command executes formulas which operate only on the selected data cells. The formulas create a data series from 1 to 100 in the first column, calculate the log of each value in the second column, and name the two columns.

The #COPY command copies the entire data window, including titles, because the starting row is set to –1. Finally, the #PASTE command pastes the information on the Clipboard into the data window, starting at row 10 in the third data column.

Intermediate Example

This example script performs the following functions:

- Imports a tab-delimited text file.
- Executes two Formula Entry functions.
- Creates a Scatter plot of the raw data.
- Exports the plot to the printer.

The example script is listed below, followed by a description of how it operates.

```
#DATAFILE
file = "C:\kgraph\Data\example.txt"
delimiter = tab
skip = 0
read_titles = TRUE
del_number = 1
#END

#FORMULA
c3 = (c1 + c2) /2;
name("Average", c3);
#END

#SCRIPT
auto_print = true
print_mode = single
plot_type = scatter
begin_group
title = "Sample Script Plot"
legend = "Mean of Test Data"
x = 0
y = 3
end_group
#END
```

The #DATAFILE opens the specified text file. The path to the file uses a Windows format. The text file is imported as a tab-delimited file which has a single tab between data values, skipping zero lines, and reading the first line as titles. The #FORMULA command executes two Formula Entry commands which calculate the average of two columns and place a title in the output column.

The #SCRIPT command generates a Scatter plot from the calculated data. The plot title and legend information are set directly via the title and legend options within the script. After the plot is created, it is automatically exported to the printer.

Advanced Example

This example script performs the following functions:

- Sets the program directory as the default directory.
- Opens a data file.
- Makes a selection in the data window.
- Plots the selected data as a Stack Column plot.
- Exports the plot as a TIFF file, scaling the image by 50%.

The example script is listed below, followed by a description of how it operates.

```
#SET_DIRECTORY
directory = ""
#END

#DATAFILE
file = ":Examples:Data:Housing Starts.QDA"
#END

#SELECTION
0 5 0 3
#END

#SCRIPT
y axis title "Range"
y axis limits 0 450
plot_type = stack_column
begin_group
x = 0
y = 1
y = 2
y = 3
end_group
#END

#PICT/OPT
type = tiff
scale = 50
file = "Example.tif"
#END
```

The #SET_DIRECTORY command sets the program directory to be the default path. The #DATAFILE command opens the **Housing Starts** data file. The path to this file uses a Macintosh format and begins in the same directory as the program. Since the path to this file does not start at the root directory, the path begins with a colon (:).

The #SELECTION command selects the first six rows in the data set. The #SCRIPT command creates a Stack Column plot of the selected data. The title and limits for the Y axis are defined within this command. The plot is then exported as a TIFF file with 50% reduction using the #PICT/OPT command. The TIFF file is given the specified name and is saved in the same directory as the program.

Creating Plots

Chapter 7

KaleidaGraph provides 19 different plot types for representing your data. All of the available plots are listed in the **Gallery** menu. This chapter covers:

- The different types of plots that are available in KaleidaGraph.
- Creating a plot.
- Plotting multiple X-Y pairs on a single graph.
- Plotting a function.
- Generating similar plots.

7.1 Types of Plots

7.1.1 Line Plot

Use a Line plot to represent trends in different data sets over a period of time. Each line in the plot represents a separate data column. The data is displayed as a series of X-Y pairs, connected to the next point by a line. The data does not have to be sorted, but it is important to note that the points are connected in the order they appear in the data window.

Line plots give you a great deal of flexibility in how the data is displayed. You can specify the number of markers, line style, and line thickness for each variable. You can also hide the line and display only the markers. All of these characteristics are controlled in the Plot Style dialog.

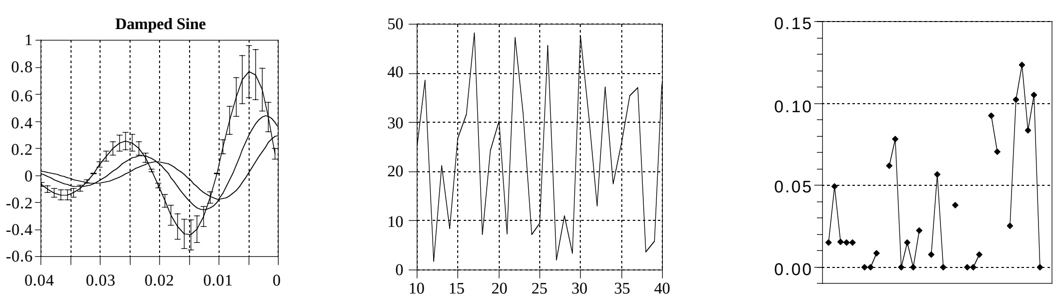


Figure 7-1 Sample Line plots

7.1.2 Scatter Plot

Use a Scatter plot to compare large sets of data. Scatter plots are similar to Line plots, except the points are not connected by lines. Each X-Y pair is represented by a unique marker on the plot. Each column of data is represented by a different marker.

Scatter plots display a marker for each point in the plot. It is not possible to display a partial set of markers or a line between the points. If you want this kind of flexibility, plot the data as a Line plot.

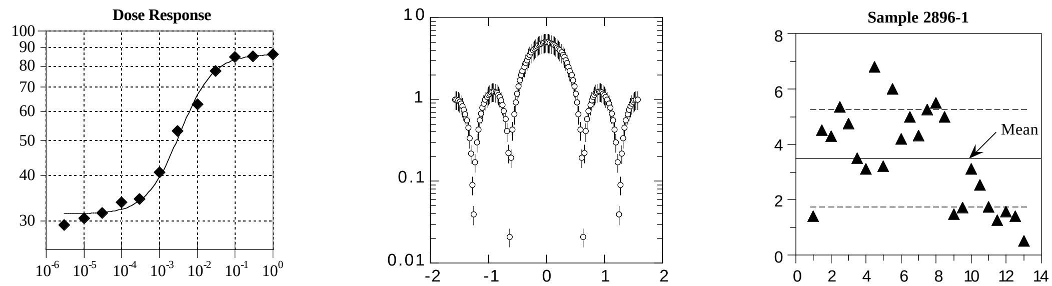


Figure 7-2 Sample Scatter plots

7.1.3 Double Y Plot

Use a Double Y plot to display two groups of data, each with its own vertical scale. Each line in the plot represents a separate data column. The data is displayed as a series of X-Y pairs, connected to the next point by a line. The data does not have to be sorted, but it is important to note that the points are connected in the order they appear in the data window.

The Double Y plot gives you the same amount of control over the display of the data as the Line plot, but with the added capability of having separate scales on the vertical axes. This plot is useful when you are plotting two dependent (Y) groups which share a common independent (X) variable.

7.1.4 Double X Plot

Use a Double X plot to display two groups of data, each with its own horizontal scale. Each line in the plot represents a separate data column. The data is displayed as a series of X-Y pairs, connected to the next point by a line. The data does not have to be sorted, but it is important to note that the points are connected in the order they appear in the data window.

The Double X plot gives you the same amount of control over the display of the data as the Line plot, but with the added capability of having separate scales on the horizontal axes. This plot is useful when you are plotting two groups of data that need to have separate independent (X) axes and share the same dependent (Y) axis.

7.1.5 Double XY Plot

Use a Double XY plot to display two groups of data, each with its own horizontal and vertical scale. Each line in the plot represents a separate data column. The data is displayed as a series of X-Y pairs, connected to the next point by a line. The data does not have to be sorted, but it is important to note that the points are connected in the order they appear in the data window.

The Double XY plot gives you the same amount of control over the display of the data as the Line plot, but with the added capability of having two separate scales on the horizontal and vertical axes. This plot is useful when you are plotting two groups of data that need to have separate independent (X) and dependent (Y) axes

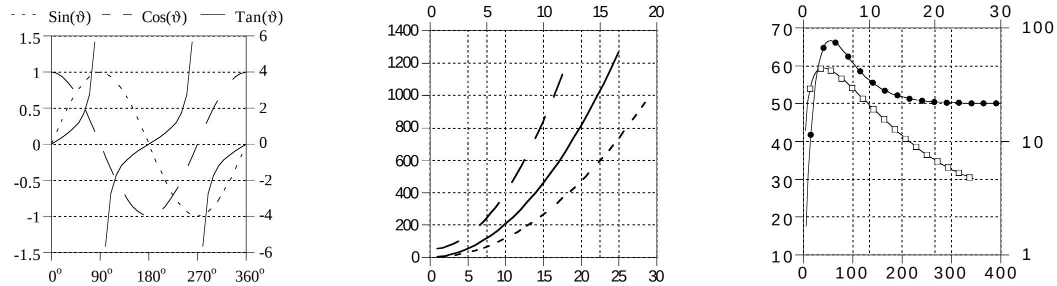


Figure 7-3 Sample Double Y, Double X, and Double XY plots

7.1.6 High/Low Plot

This plot type is designed for many uses. The most common is to create High/Low/Close plots for the financial industry. This plot normally contains a vertical bar, representing the range of values from low to high, and a marker, representing the ending or closing value.

The order of the columns in the data window does not matter. However, this plot appears differently depending on how many Y variables are plotted for a given X.

- One Y variable (Line plot) - A standard Line plot is generated. This is often useful when overlaying a High/Low plot with some additional X-Y data.
- Two Y variables (Spike plot) - A vertical line is drawn between the two data points. The first plotted variable controls the color, line style, width, and marker style.
- Three or more Y variables (High/Low/Close plot) - A vertical line is drawn between the maximum and minimum data points. The first plotted variable controls the color, line style, width, and marker style. All of the data points between the minimum and maximum are drawn as ticks to the right of the vertical line. The size of the tick is controlled by the marker size of the first plotted variable.

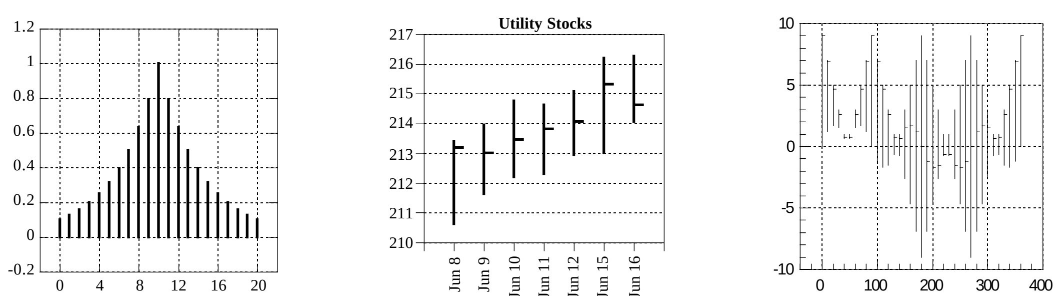


Figure 7-4 Sample High/Low plots

7.1.7 Step Plot

A Step plot is a variation of a Line plot. It can be used to compare data that does not show a trend. Each line in the plot represents a separate data column. The data is plotted as a series of steps. The data does not have to be sorted, but it is important to note that the points are connected in the order they appear in the data window

Note: It is not possible to display markers on a Step plot. The placement of the steps is controlled using the **Step Options** command (**Format** menu).

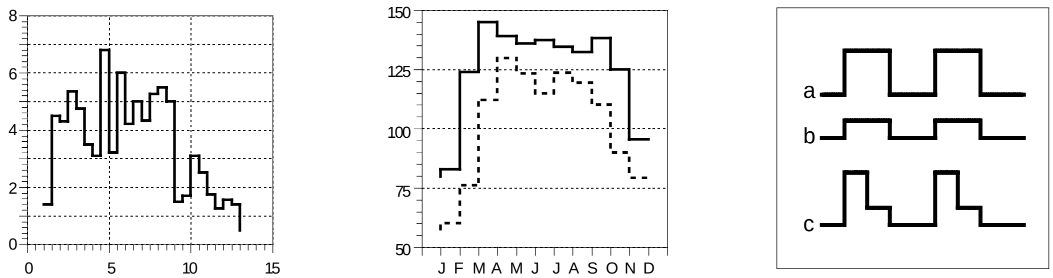


Figure 7-5 Sample Step plots

7.1.8 Probability Plot

Use a Probability plot to display the graph of a variable normalized to either a linear or normal probability distribution. The X axis is scaled in probability (between 0 and 100%) and shows the percentage of the Y variable whose value is less than the data point. The Y axis displays the range of the data variables.

The Probability plot gives rough information about the local density of the data and symmetry. The plotted data points do not coincide, even if there are exact duplicates in the data. It is easy to read information from the plot and it can accommodate a large number of observations. The Probability plot is not a summary but a display of all the data.

Note: The **Probability Options** command (**Format** menu) controls whether a linear or normal distribution is used.

7.1.9 X-Y Probability Plot

The X-Y Probability plot is similar to the Probability plot. The only difference is that in an X-Y Probability plot, you enter the probability associated with each Y value. The data is entered as X-Y pairs, where X is the probability (between 0 and 100%) and Y is the data value.

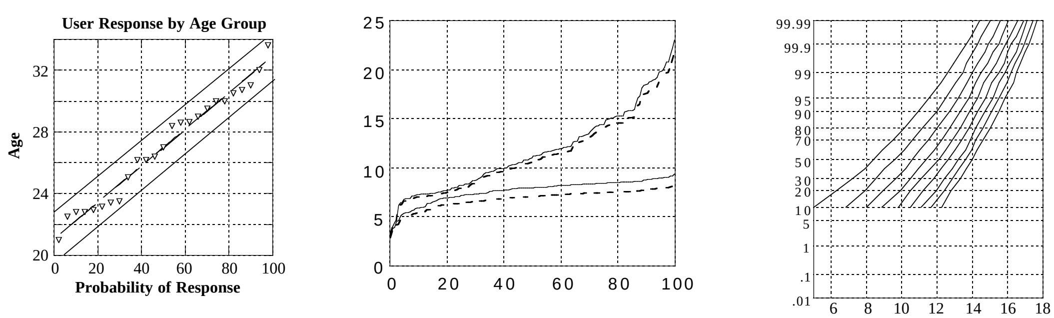


Figure 7-6 Sample Probability and X-Y Probability plots

7.1.10 Histogram Plot

A Histogram calculates the number of data points that fall within a specified bin size. The resulting plot displays the range of the data on the X axis and the number of data points in each bin on the Y axis. If more than one variable is plotted, the Y axis displays the total number of points in each bin.

You can use the **Histogram Options** command (**Format** menu) to specify the bin size, reference value, and number of extra bins that get added to the plot. A maximum of 200 bins may be plotted.

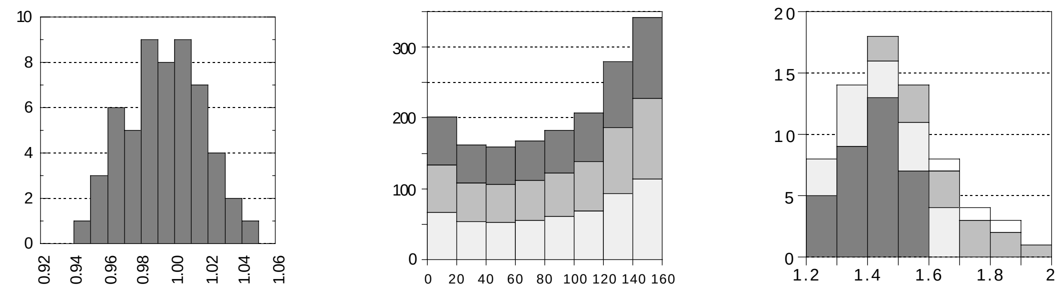


Figure 7-7 Sample Histogram plots

7.1.11 Box Plot

A Box plot enables you to quickly examine a number of variables and extract their major characteristics. While this does not provide detailed information about the data, it does give you a look at its global behavior. A Box plot represents each variable as a separate box. The Y axis displays the range of the data and the X axis displays the names of each variable. See Section D.1 for details on what the boxes represent and how they are calculated.

Up to 20 additional variables may (optionally) be overlaid as markers on each plot. The markers are matched one to one with the plotted variables. The first value in a marker column is matched with the first box, the second value in a marker column is matched with the second box, and so on. When using markers, the boxes do not have a fill pattern so that the markers can be clearly identified.

Note: You may not select more than a total of 20 variables and markers from a single data window. If your needs exceed this limit, place the variables and markers in separate data windows.

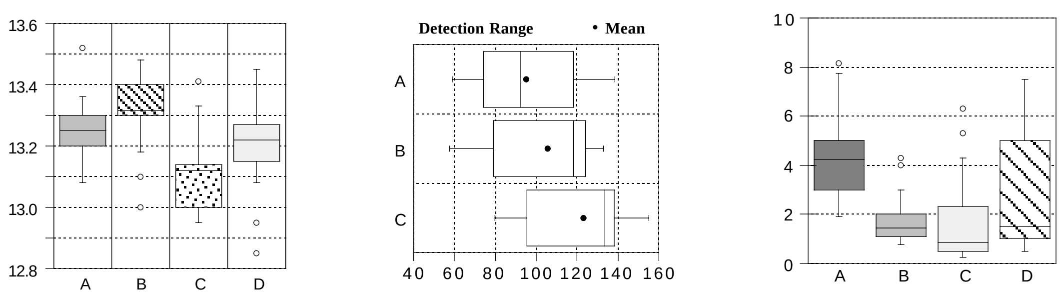


Figure 7-8 Sample Box plots

7.1.12 Percentile Plot

A Percentile plot represents each variable as a separate box. The Y axis displays the range of the data and the X axis displays the names of each variable. Each box encloses 90% of the data.

The bottom and top of each box represent 5% and 95% of the data. Three lines are drawn inside each box. The middle line represents the median value of the data (50%), while the lower and upper dashed lines represent 25% and 75% of the data, respectively. Due to the importance of these lines, the boxes do not have fill patterns

Up to 20 additional variables may (optionally) be overlaid as markers on each plot. The markers are matched one to one with the plotted variables. The first value in a marker column is matched with the first box, the second value in a marker column is matched with the second box, and so on.

Note: It is not possible to display a fill pattern in the boxes. You may not select more than a total of 20 variables and markers from a single data window. If your needs exceed this limit, place the variables and markers in separate data windows.

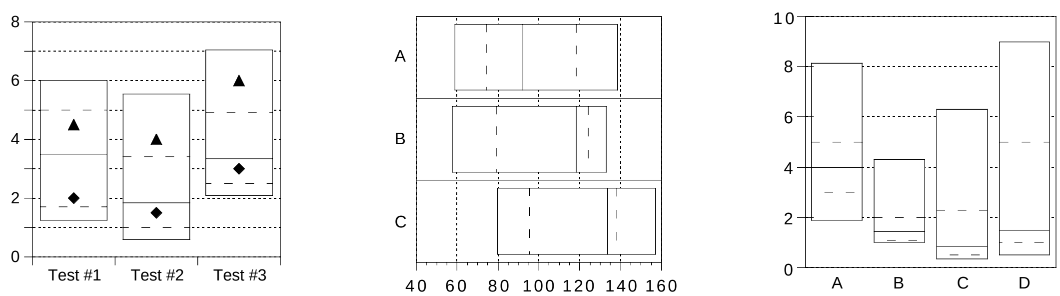


Figure 7-9 Sample Percentile plots

7.1.13 Horizontal Bar Plot

A Horizontal Bar plot is a graph of data selected from one window. The data is displayed as a series of horizontal bars. The length of each bar represents the value of a variable. Multiple values for the same category (row) are displayed as side-by-side bars. The horizontal axis displays the range of the dependent values and the vertical axis displays the categories.

Note: A maximum of one independent (X) variable and 20 dependent (Y) variables may be plotted at once. Customizing Horizontal Bar plots is covered in Section 8.15.

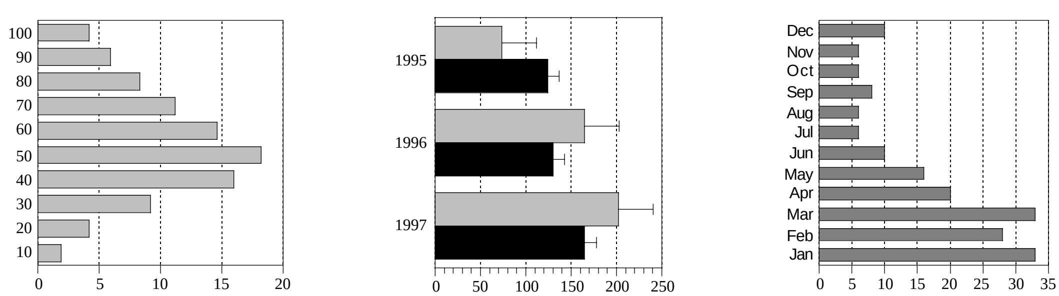


Figure 7-10 Sample Horizontal Bar plots

7.1.14 Stack Bar Plot

A Stack Bar plot is a graph of data selected from one window. As in a Horizontal Bar plot, the data is displayed as a series of horizontal bars. The difference between a Stack Bar plot and a Horizontal Bar plot is that variables belonging to the same category (row) are displayed as bars stacked one on top of another. The length of each bar represents the sum of the data points for a category. The horizontal axis displays the range of sums for a category and the vertical axis displays the categories.

Note: A maximum of one independent (X) variable and 20 dependent (Y) variables may be plotted at once. Customizing Stack Bar plots is covered in Section 8.15.

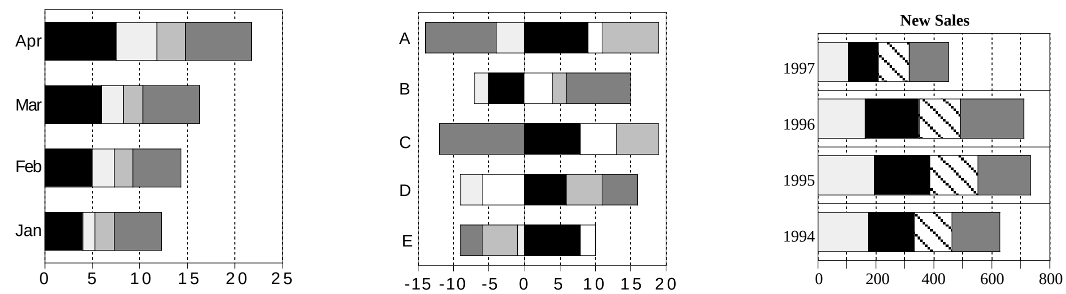


Figure 7-11 Sample Stack Bar plots

7.1.15 Column Plot

A Column plot is a graph of data selected from one window. The data is displayed as a series of vertical bars. The height of each bar represents the value of a variable. Multiple values for the same category (row) are displayed as side-by-side columns. The horizontal axis displays the categories and the vertical axis displays the range of the variables.

Note: A maximum of one independent (X) variable and 20 dependent (Y) variables may be plotted at once. Customizing Column plots is covered in Section 8.15.

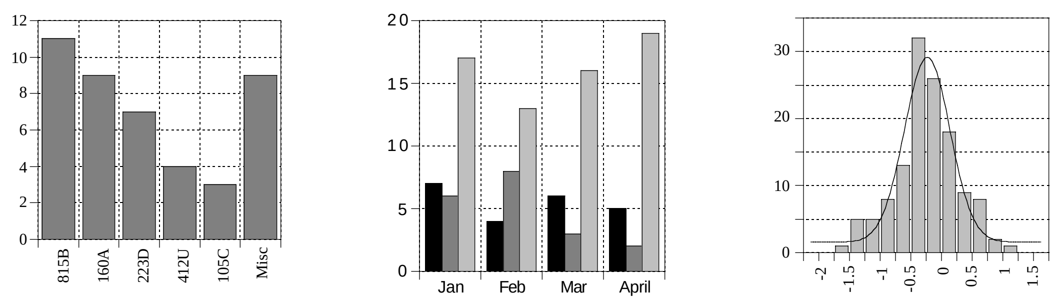


Figure 7-12 Sample Column plots

7.1.16 Stack Column Plot

A Stack Column plot is a graph of data selected from one window. As in a Column plot, the data is displayed as a series of vertical bars. The difference between a Stack Column plot and a Column plot is that variables belonging to the same category (row) are displayed as columns stacked one on top of another. The height of each bar represents the sum of the data points for a category. The horizontal axis displays the categories and the vertical axis displays the range of sums for all variables in that category.

Note: A maximum of one independent (X) variable and 20 dependent (Y) variables may be plotted at once. Customizing Stack Column plots is covered in Section 8.15.

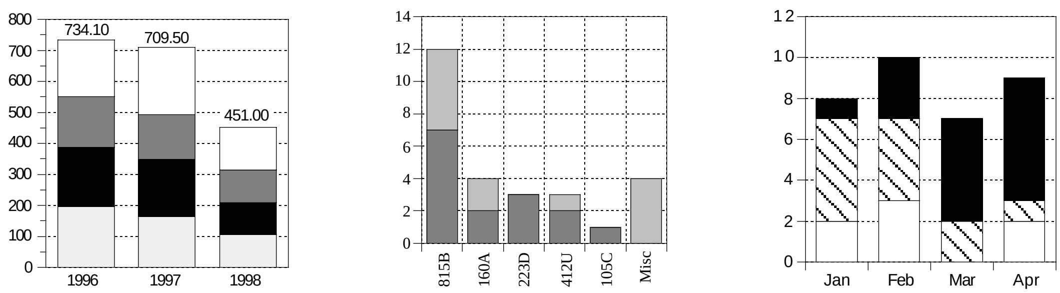


Figure 7-13 Sample Stack Column plots

7.1.17 Polar Plot

In a Polar plot, the data is plotted on the polar coordinate system. Each point in this system is specified by an angle (Ø) in degrees and a radius (R). As an option, an alternate data format can be used, where the data is specified in X-Y format and then transformed to the polar coordinate system. In either case, an independent variable (Ø or X) and one or more dependent variables (R or Y) are selected for plotting.

Using the **Polar Options** command (**Format** menu), you can specify the grid type, data format, reference angle, and direction for Polar plots. Customizing Polar plots is covered in Section 8.16.

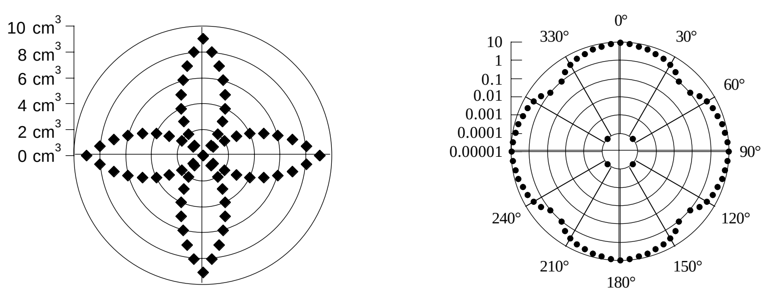


Figure 7-14 Sample Polar plots

7.1.18 Pie Chart

A Pie chart displays the value of up to 20 categories as a percentage of the total area of a circle. Each of the categories is represented as a segment of the circle (or a wedge of the pie). Only two variables may be plotted at one time in a Pie chart. The first is the independent (X) variable or category. This variable contains the name of each category. The second variable (Y) contains the category values, which must be numeric.

Note: If more than 20 categories are selected for plotting, only the first 20 appear on the plot. Customizing Pie charts is covered in Section 8.17. Negative values cannot be plotted in a Pie chart. If the data contains negative values, a dialog gives you the option of plotting the absolute value of the data or canceling the plot.

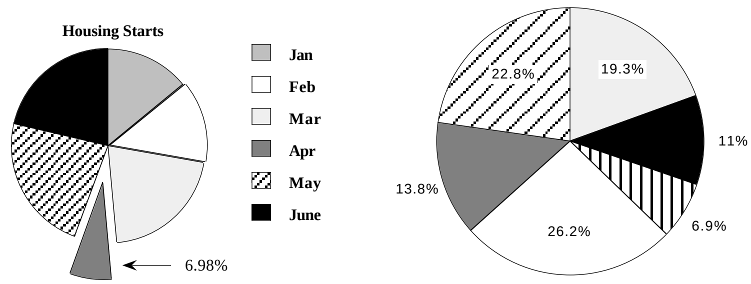


Figure 7-15 Sample Pie charts

7.1.19 Function Plot

The Function plot can be used to display a mathematical function as a plot. Choosing this plot type displays a dialog where you can enter the function, the minimum and maximum X values, and the number of points to plot.

Once the plot is created, it behaves like a Line plot. You have full control over the line style, markers, and all other components of the plot. Section 7.3 provides instructions for creating a Function plot.

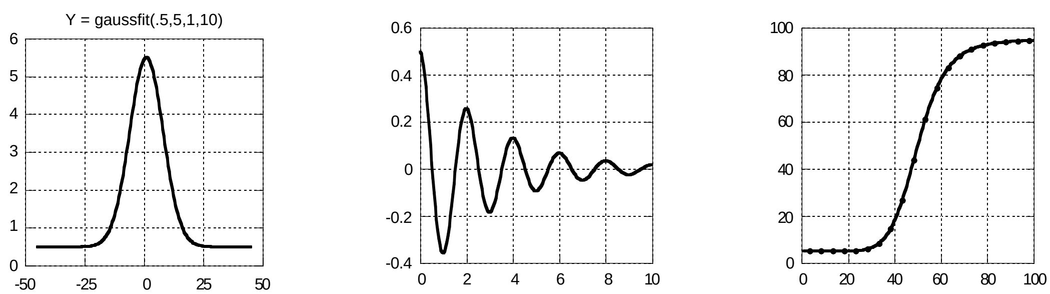


Figure 7-16 Sample Function plots

7.1.20 Text Plot

This plot type allows you to create a plot window whose intended use is text. Text plots are particularly useful when you want to generate a plot that only contains text, lines, boxes, tables, or pictures.

Note: Certain tools in this window are unavailable, because they can only be used when data is plotted.

7.1.21 Template Plot

The Template plot is used to generate the same type of plot, using different data. It allows you to take any plot and use it as a base for plotting new data. This works for both new plots and plots that have been saved. The use of this plot is discussed in Section 7.4.2.

7.2 Creating a Plot

The **Gallery** menu contains the plot types available in KaleidaGraph. You can create a plot using any of the open data windows. If a selection is made in the data you are plotting, only the selected data is used in making the plot.

To create a plot:

1. Open the data to be plotted.
2. Make any selections in the data windows to plot a subset of the data.
3. Choose a plot type from the **Gallery** menu. A dialog similar to Figure 7-17 is displayed.

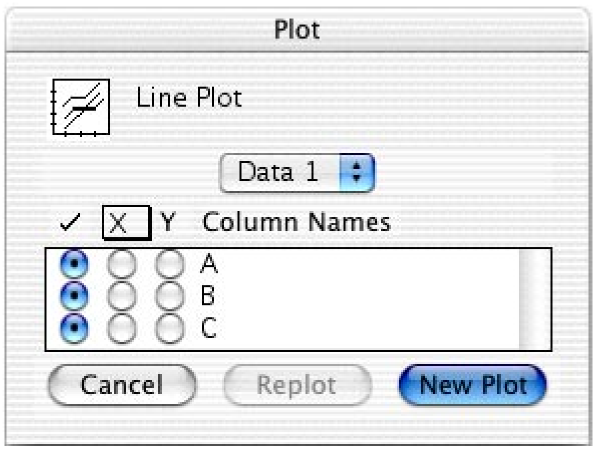


Figure 7-17 Variable Selection dialog

4. Click the appropriate buttons to select the columns to be plotted. You can clear all of the buttons at once by clicking the check mark icon in this dialog.
5. Click **New Plot** to create the plot.

At this point, the plot is linked to the original data window. If another plot is opened, created, or brought to the front, the data for the previous plot becomes embedded in the plot. If the plot is saved, a copy of the data is embedded in the plot and saved as part of the plot file. Once data is embedded in a plot, it can be extracted by choosing **Plot** > **Extract Data** or clicking the Find Data button ( ) in the plot window.

7.2.1 Plotting from Multiple Data Windows

Unless you are creating a Pie chart or one of the plots in the **Bar** submenu, you can plot data from multiple data windows. This is accomplished by making variable selections for each data set to be used.

The Variable Selection dialog contains a pop-up menu listing the names of all open data windows. Using this menu, you can select up to 20 independent and 20 dependent variables to be plotted. If a text column is used as the independent variable, you may not select any other independent variables and you can only plot from that particular data window.

To create a plot from multiple data sets:

1. Open the data to be plotted. To plot subsections of any data sets, select the appropriate rows in each data window.
2. Choose a plot type from the **Gallery** menu. A Variable Selection dialog appears.
3. Make the independent and dependent variable selections for the active data window.
4. Click the data window's name in this dialog to display a pop-up menu containing the name of each data window. A check mark in front of the name indicates that variables are already selected from that data window. If the data window contains a data selection, its name is underlined.

   The dialog in the background of Figure 7-18 displays the variables selected from the first data set and the pop-up menu used to switch between data windows.

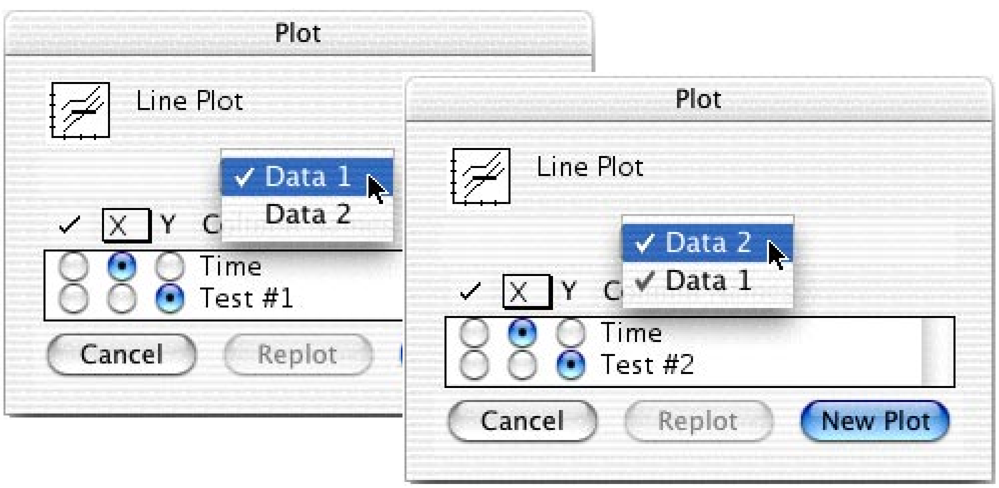


Figure 7-18 Variable Selection dialogs

5. Select the name of a different data window and make the variable selections. Continue doing this until all of the variables to be plotted are selected. The dialog in the foreground of Figure 7-18 shows the selection made from the second data set.
6. Click **New Plot** to create a plot based on the selections.

7.2.2 Plotting Multiple X-Y Pairs from the Same Data Window

The Linear, X-Y Probability, and Polar plots allow you to plot multiple independent variables from the same data window. If the independent variable is a text column, you may only have one independent variable.

For the plot types mentioned above, the Variable Selection dialog contains a pop-up menu listing the independent variables (X–X9). Using this pop-up menu, you can plot up to nine independent and 20 dependent variables from one data window.

To create a plot from one data window, with different X variables:

1. Open the data to be plotted.
2. Choose a plot type from the **Gallery** menu. A Variable Selection dialog appears.
3. Make the first set of X and Y variable selections.
4. Click the X column's shadow box to display a pop-up menu containing the list of X variables (X–X9). If one of the variables already contains a selection, it is preceded by a check mark. The dialog in the background of Figure 7-19 displays the pop-up menu used to select a different independent variable.

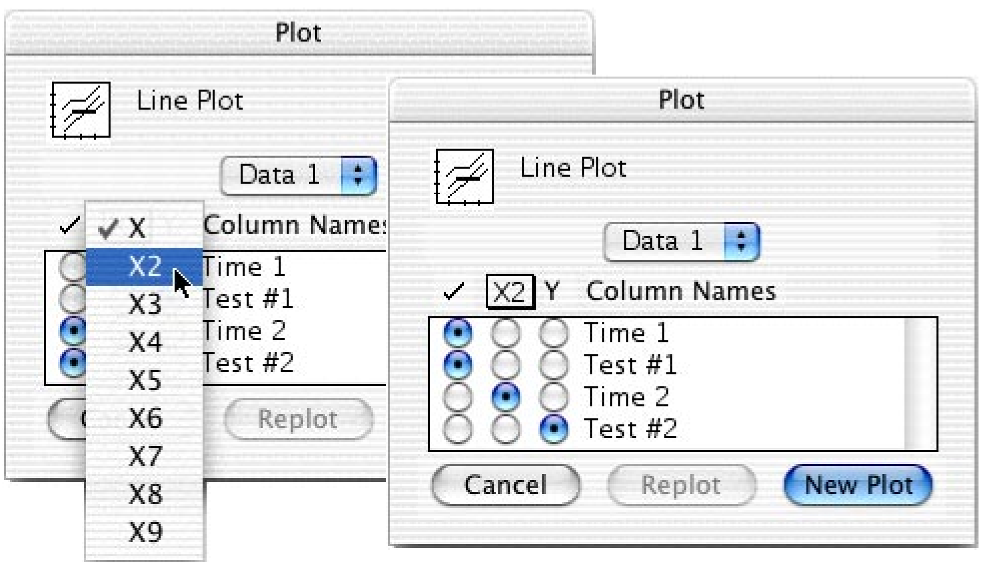


Figure 7-19 Variable Selection dialogs

5. Select a different X variable and choose what is to be plotted against it. Continue doing this until all of the selections are made. The dialog in the foreground of Figure 7-19 shows the selections for the second independent variable.
6. Click **New Plot** to create a plot based on the selections.

7.3 Plotting a Function

KaleidaGraph provides a Function plot that can be used to plot any mathematical function. The function you enter is evaluated between the specified limits and is displayed as a plot.

When defining your function, use **x** for the independent variable. The operators and math functions found in Formula Entry can be used in defining your function. Constants and other shorthand references can be defined in the library. You can also use any of the definitions included in the Curve Fits folder. The table below lists some example definitions:

Example Function Definitions
5 * x^2
gaussfit(1, 5, 1, 10)
m1 + m2 * exp(–m3 * x); m1 = 5; m2 = 1; m3 = 0.5
a * exp(–b * x) * cos(c * x + d); a = 4; b = 0.7; c = 2.2; d = 15

To create a Function plot:

1. Choose **Gallery** > **Function** to display the Function Plot dialog, shown in Figure 7-20.

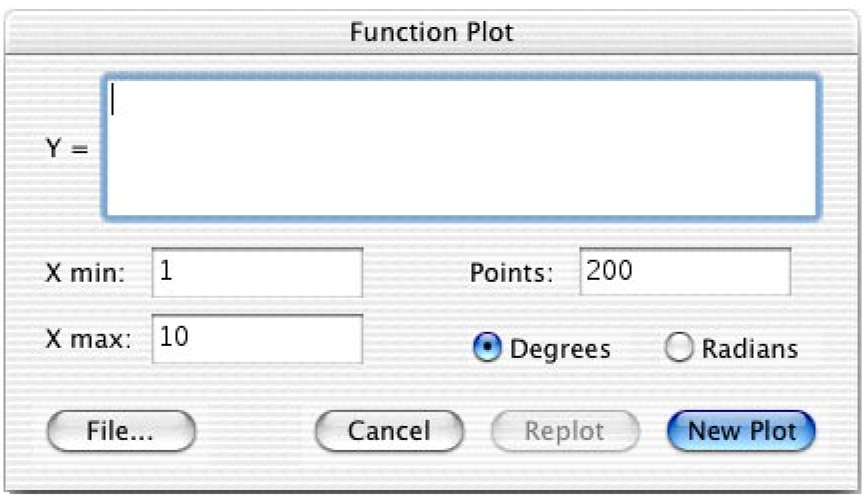


Figure 7-20 Function Plot dialog

2. Enter the function you want to plot. The **File** button can be used to load a saved function definition
3. If necessary, change the values for the minimum, maximum, and number of points.
4. Click **New Plot** to plot the function.

Once the Function plot is created, it behaves like a Line plot. You have full control over the line style, markers, and all other components of the plot. If you want to see the data generated by the function, choose **Plot** > **Extract Data** or click the Find Data button ( ) in the plot window.

7.4 Generating Similar Plots

If you routinely create the same types of plots in KaleidaGraph, it helps to set some defaults or templates so that most of the work can be done automatically. KaleidaGraph provides several features that can be used to generate similar plots.

7.4.1 Using Style Files

One way to generate similar plots is through the use of Style files. KaleidaGraph stores most of your dialog settings and program defaults into a Style file. The Style file also stores layouts for each of the various plot types, in addition to any defaults you may have set for colors and text labels.

Each of the plots in the **Gallery** menu can have its own layout. The layout remembers the default style for the legend, the position of the axis titles, plot title, axes, and legend, and the size of the axes, frame, and legend.

To define the layout for a plot:

1. Open an existing plot or create a plot of the type you want to define.
2. Modify the position and size of the titles, plot, and legend.
3. Choose **Plot** > **Define Layout**. The dialog in Figure 7-21 is displayed, asking if you want to save the layout for this plot type.

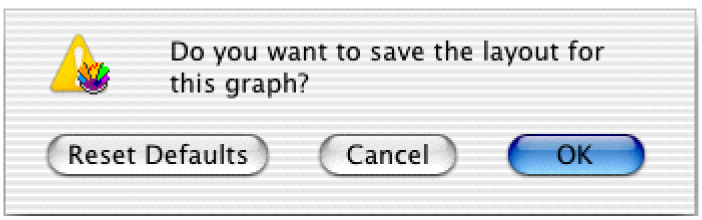


Figure 7-21 Define Layout dialog

4. Click **OK** to save the layout.

Defining the layout of a plot only saves the position of the titles and legend. To change the defaults for text labels, you must use the **Text Options** command (**Format** menu).

You need to save the Style file when quitting for these changes to be remembered the next time you start KaleidaGraph. To create individual Style files for different projects, choose **File** > **Export** > **Style**. This way you can save any number of different Style files which can be opened at a later time using the **Open** command

7.4.2 Using Template Plots

The Template plot is used to generate the same type of plot, using different data. It allows you to take any graph and use it as a base for plotting new data. This works for new graphs and graphs that were saved.

The new data is autoscaled unless the limits are fixed in the Axis Options dialog. The legend reflects the new variables being plotted, but all other text labels and objects remain the same. Any curve fits or error bars in the original plot are applied and recalculated on the new plot.

Once a Template plot has been created, it can be modified like any other plot. If you need the ability to generate multiple plots at once, use the Plot Script feature. This topic is covered in Section 7.4.3.

To use a plot as a template for creating other plots:

1. Open or create the data to be plotted.
2. Open or create the plot you want to use as a base for creating other plots.
3. Choose **Gallery** > **Template** to display a Variable Selection dialog. Figure 7-22 shows the dialog when the original plot is a Line plot.

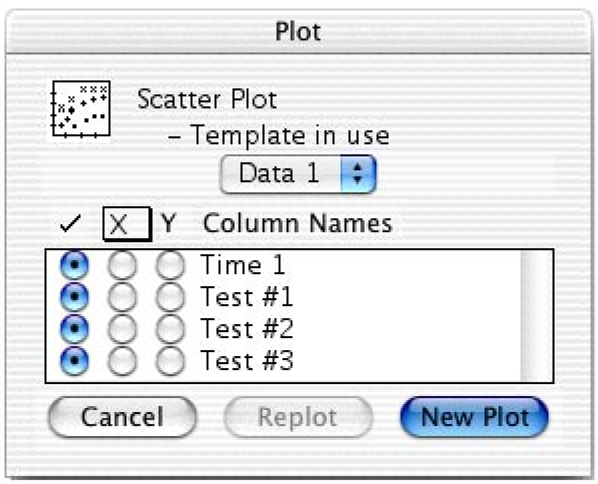


Figure 7-22 Variable Selection dialog

4. Click the appropriate buttons to select the variables to be plotted.
5. Click **New Plot** to generate a plot based on your selections.

Note: It is not necessary to use the Template plot if you are only modifying the original data used to create the plot. Instead, you can extract the data from the plot, modify it, and update the plot. This process is covered in Section 8.13.

7.4.3 Using Plot Scripts

Another way to generate similar plots is the **Plot Script** command (**Windows** menu). This feature enables you to choose any plot type or an existing plot and use it as a template for creating multiple plots at once.

When to Use Scripts

The script feature is intended for the user who has a Template plot and wants to plot different data sets and view the resulting curves. Even if you only need to generate a simple plot, but you generate it all of the time with different data, you will find plot scripts very useful.

The advantage of a script over a Template plot is that a script can generate multiple plots. Also, you can control legend descriptions and plot titles using a script. In a Template plot, the legend descriptions come from the column names of the plotted variables and the plot title remains unchanged.

Creating a Plot Script

The Plot Script command is similar to the Template plot but provides you with more options for creating plots. Using plot scripts, you can create multiple plots at once, automatically print or save the plots, and set the plot title or legend for each plot.

Choosing **Windows** > **Plot Script** displays the dialog in Figure 7-23. This dialog is used to make your selections for the current script. The script selections include: plot type, variable selections, plot titles, legend text, auto print, and auto save options.

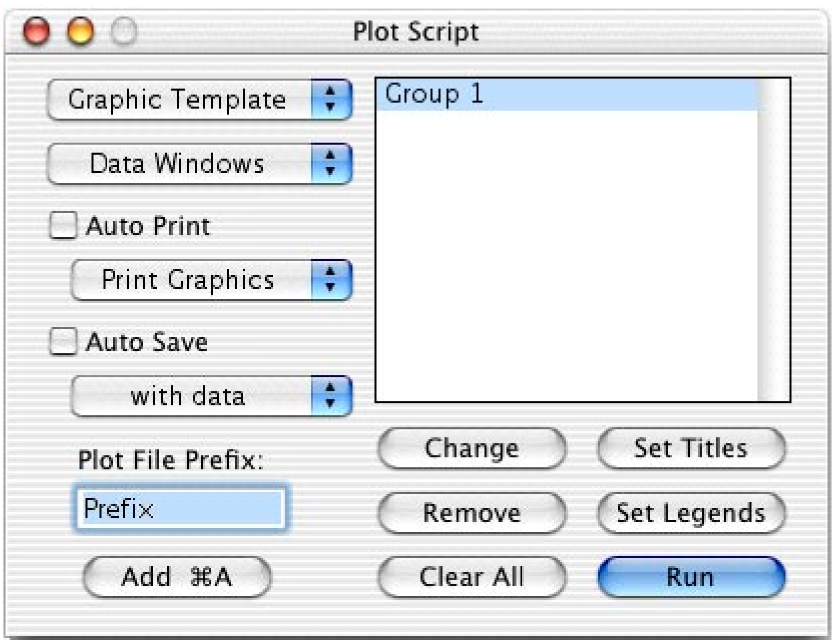


Figure 7-23 Plot Script window

To create a plot script:

1. Choose **Windows** > **Plot Script** to display the Plot Script window.
2. If groups are already listed in this window, click **Clear All** to remove the groups and start a new script.
3. Using the **Graphic Template** pop-up menu, choose either a Gallery plot type or an existing plot to use as a template.

   Note: Only one graphic template can be used in a script.
4. Click **Add**. A Variable Selection dialog similar to Figure 7-24 is displayed.

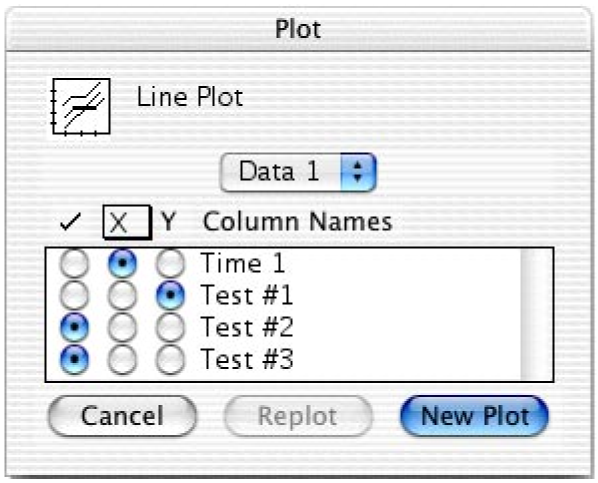


Figure 7-24 Variable Selection dialog

5. Select the variables to be plotted.
6. Click **OK**. Group 1 will be created in the Plot Script window, as shown in Figure 7-25. Continue using the **Add** button to create new groups. Each group generates a separate plot.

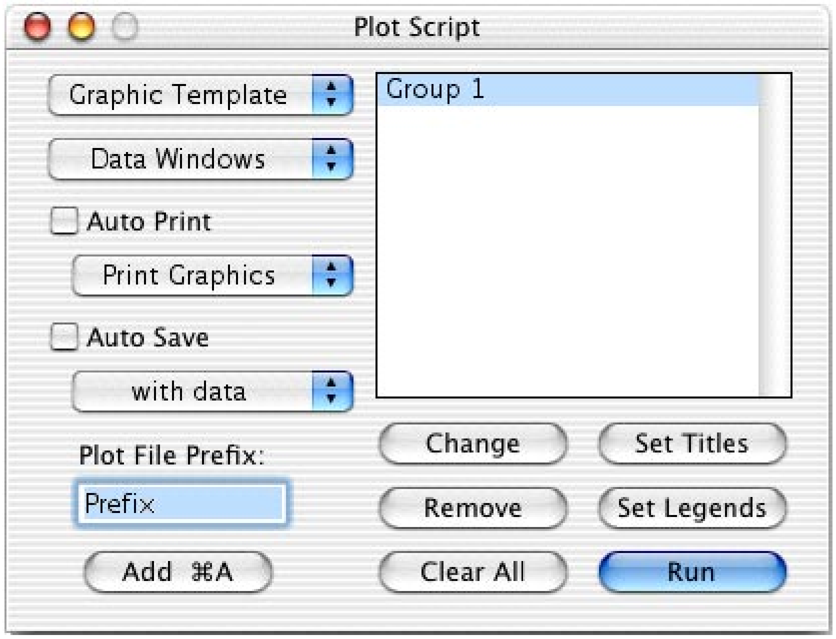


Figure 7-25 Plot Script window

7. When you are finished creating groups, click **Run** to execute the script.

Setting Titles and Legends

By default, the titles are the same as the template and the legend comes from the column names of the plotted variables. You can set the plot titles and legends for each plot being created using the **Set Titles** and **Set Legends** buttons in the Plot Script window. Clicking either of these buttons displays a text editor.

Each line in the editor corresponds to a different plot. Plot titles are entered on separate lines, followed by a return. If you choose to set the titles, you must set them for all of the plots. If a specific plot is not to have a title, create a blank line in the editor by pressing **Enter** (Windows) or **Return** (Macintosh). Each title appears to the right of the corresponding data group in the Plot Script window.

Note: Multi-line titles can be created using the **\r** characters. Any text that follows **\r** will be placed on a separate line.

To set the plot titles for a script:

1. Click **Set Titles**.
2. Enter plot titles as displayed in Figure 7-26.

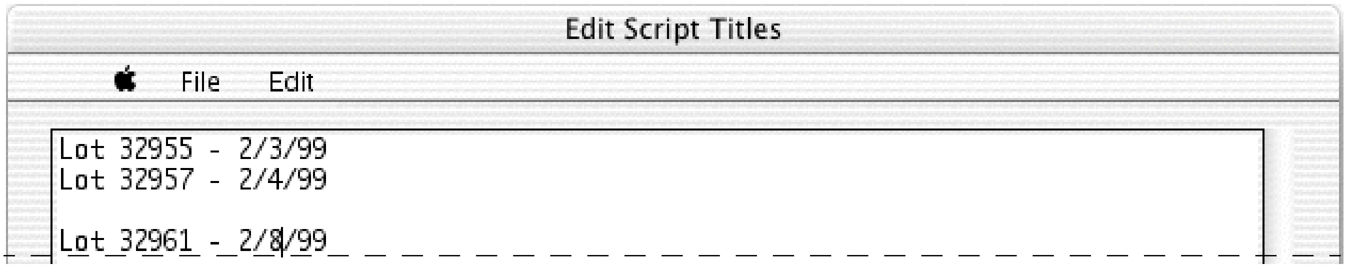


Figure 7-26 Edit Script Titles dialog

3. Choose **File** > **Close**. The titles are listed to the right of the groups, as shown in Figure 7-27.

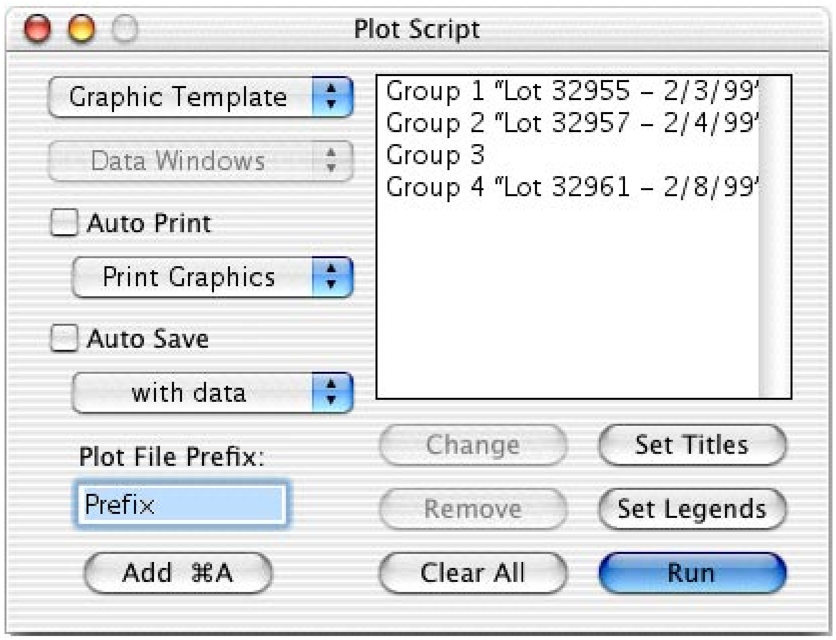


Figure 7-27 Plot Script window after setting titles

The legend information for each plot is also placed on separate lines, followed by a return. To indicate that the labels belong to the same plot, each legend label should be separated with either a comma or a tab. For Double Y or Double XY plots, enter the labels for the Y1 legend followed by the Y2 labels (on the same line). All labels prior to a return belong to the same plot. Only use the **Enter** (Windows) or **Return** (Macintosh) key to separate legend titles associated with different plots.

Note: Multi-line legend labels can be created using the **\r** characters. Any text that follows **\r** will be placed on a separate line.

To set the legends for a script:

1. Click **Set Legends**.
2. Enter legend titles as displayed in Figure 7-28.

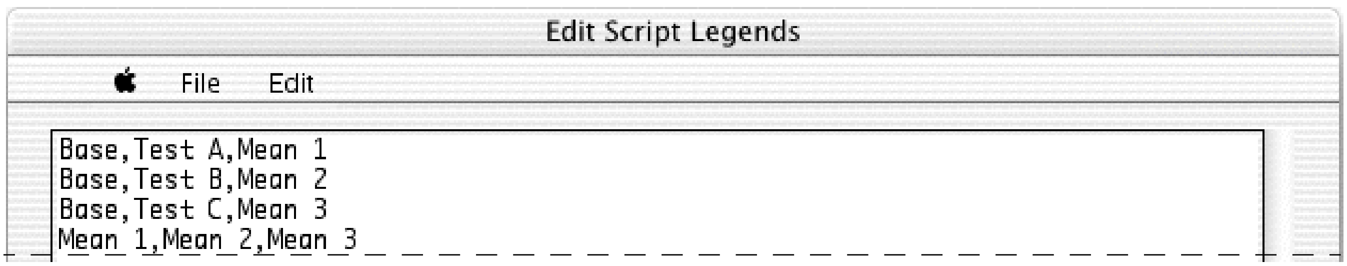


Figure 7-28 Edit Script Legends dialog

3. Choose **File** > **Close**.

Using Pattern Recognition to Create New Groups

The Plot Script window includes a pattern recognition feature. Once you add a third data group, KaleidaGraph tries to identify a pattern in the way you are setting up the groups. If KaleidaGraph's selections are correct, you can replicate the pattern throughout the data window by clicking **Repeat** instead of **OK**. KaleidaGraph continues the selection pattern across all of the columns in your data window, and creates the groups automatically.

Note: The pattern recognition feature only works across the columns of a data window or across the columns of a set of data windows. KaleidaGraph cannot identify a plotting pattern that jumps from one data window to another.

Pointing Groups to Different Data Windows

The **Data Windows** pop-up menu can be used to select a different data window for each individual set of variable selections. This pop-up menu contains up to 20 **X Set** menu items (X–X20). A set is defined as an X variable and its associated Y variables (such as: X, Y1, Y2). The number of **X Set** menu items depends on the number of independent variables selected for the current group.

The sets are listed in the pop-up menu in the order they were created. Selecting one of the sets displays a submenu containing a list of the open data windows. They are listed in the order they are stacked on the screen, from front to back. The data window currently associated with a particular set is preceded by a check mark. Figure 7-29 shows the Plot Script window with the **Data Windows** pop-up menu and the **X Set** submenu.

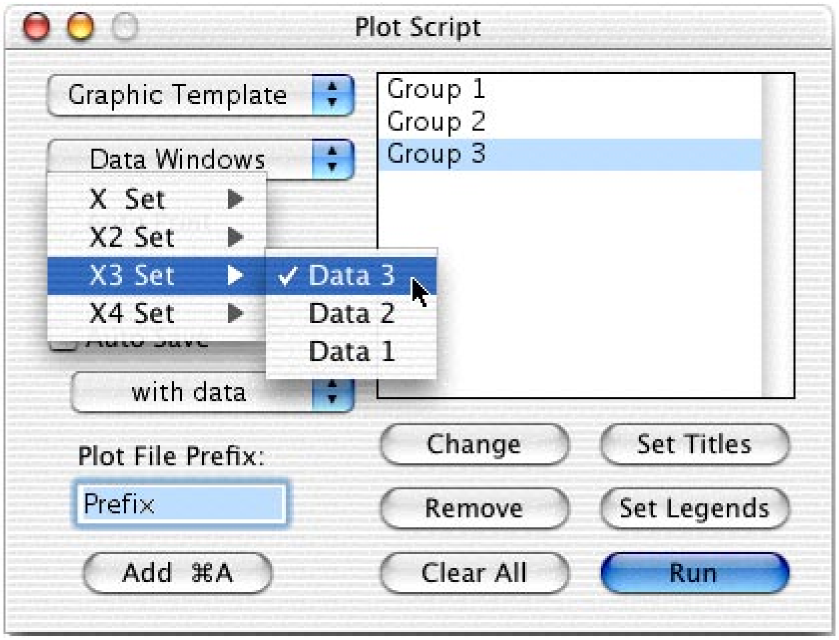


Figure 7-29 Selecting different data windows

To change the data window associated with a particular set:

1. Click the group to be edited.
2. Click the **Data Windows** pop-up menu, select an **X Set**, and choose a different data window. The selected data window is now used for that particular set of variable selections.

Note: The only other way to change the data window referenced by a set is to use the **Change** button after selecting each individual group. This method must be used to select different columns for plotting.

Exporting Plots Automatically

You can automatically export script-generated plots to the printer or to disk using the **Auto Print** and **Auto Save** options.

To automatically print plots:

1. Click the **Auto Print** check box.
2. Choose either **Print Graphics** or **Print Layout** from the pop-up menu immediately below **Auto Print**.
3. Execute the script by clicking **Run**. The plots are automatically printed.

To automatically save plots:

1. Click the **Auto Save** check box.
2. Choose the type of graphic file to be created from the pop-up menu immediately below **Auto Save**
3. Enter a prefix to be used for the saved plot files. By default, the naming convention uses the word **Prefix** followed by an underscore (_) and a number.
4. Click **Run** to execute the script. KaleidaGraph displays a dialog allowing you to specify a location for the saved plots.

Memory Considerations

If KaleidaGraph runs out of memory or you exceed the maximum number of open plot windows while executing a script, the situation is handled based on the current state of the **Automatically dispose of old plots** option in the Preferences dialog (**File** menu).

When this check box is selected, the oldest plot is discarded from memory, without asking for your approval. If this check box is cleared, a dialog asks whether to save or dispose the oldest plot. If you choose not to dispose of a plot to make memory available, the request that caused the out of memory condition is canceled

Saving and Loading Scripts

Once you have created a script, you can save it to disk by choosing either **File** > **Export** > **Script** or **File** > **Save Script**.

If you used an existing plot for the **Graphic Template**, a plot file is created with the plot script document. In the Windows version, the plot has the same name as the script, but with a **.qpc** file name extension. In the Macintosh version, the plot's file name is the first 25 characters of the script's file name, followed by **.plot**. Any future use of the script causes the plot file to open. Once a script and its template have been saved, the plot must remain in the same folder as the script and it cannot be renamed. Otherwise, the script no longer recognizes the plot document.

A default Script file is automatically opened each time KaleidaGraph is started. This way you can save your plot script and have it opened the next time you use the program.

Custom Script files can be opened at any time from within KaleidaGraph by choosing **File** > **Import** > **Script** or **File** > **Open**. The Plot Script window changes to reflect the contents of the new Script file.

7.4.4 Using Formula Scripts

The last feature that allows you to generate similar plots is a formula script. This feature has a command, **#SCRIPT**, which has all of the same options as the Plot Script window. You can create a plot script from scratch or you can use a command to open one that is saved.

The advantage of using a formula script is that you can open and modify data windows as part of the script. You can also execute more than one plot script inside of a formula script.

The disadvantage of using this feature is that you cannot select a specific plot to be used as the template. It must either be a Gallery plot type or a plot that is opened as part of a saved plot script.

For more information on formula scripts, refer to the following sections:

- Section 6.10.1 explains creating and entering formula scripts.
- Section 6.10.2 discusses the syntax for the #SCRIPT command and the rest of the formula script commands.
- Section 6.10.3 contains some example formula scripts.

Working with Plots

Chapter 8

KaleidaGraph provides complete control over the appearance of your plots. This includes the legend, axes, ticks, grid lines, and labels, in addition to how the data is represented on the plot.

This chapter explains how to:

- Edit the legend and legend text.
- Change the axis limits and scaling, as well as removing, exchanging, and linking axes.
- Control the display and number of tick marks.
- Control the display of grid lines.
- Control the display of axis labels.
- Change the marker, fill pattern, line style, line width, and variable color.
- Change the frame, grid, interior, and background colors of a plot.
- Resize the plot.
- Change the data in a plot.
- Customize Bar, Polar, and Pie plots.

8.1 Hiding, Displaying, and Closing Plots

The **Windows** menu contains four commands that can be used to hide and display plot windows. These commands operate on either a single window or all of the plot windows at once.

8.1.1 Changing the Active Window

To make a plot window active, click anywhere in the desired plot window or choose **Windows** > **Show Plot** and select its name.

8.1.2 Hiding Plot Windows

To hide a plot window, choose **Windows** > **Hide Window** and select its name. You can hide all open plots by choosing **Windows** > **Hide All Plots**.

8.1.3 Displaying Plot Windows

You can display a hidden plot window with the **Show Plot** command (**Windows** menu). The names of hidden plot windows appear in italic. Select the name of the desired plot window and it is brought to the front. The **Show All Plots** command can be used to display all of the plot windows at once.

8.1.4 Closing Plot Windows

You can use the **Close All Plots** command (**Windows** menu) to close all of the open plot windows at once. Individual plot windows can be closed by choosing **File** > **Close** or clicking the close box of each window. A dialog may be displayed, asking if you want to save the plot before closing.

Note: If you do not want to save the plot, this dialog can be avoided by pressing **Shift** while either clicking the close box or choosing **File** > **Close/NoSave**.

You can have a maximum of 32 plots open at one time in KaleidaGraph. If you have 32 plots open and try to create another plot, either the oldest plot is automatically disposed or the dialog in Figure 8-1 appears. The Preferences dialog (**File** menu) contains an option that controls the automatic disposal of plots. If **Automatically dispose of old plots** is selected, the oldest plot is discarded. If this check box is cleared, the dialog is displayed.

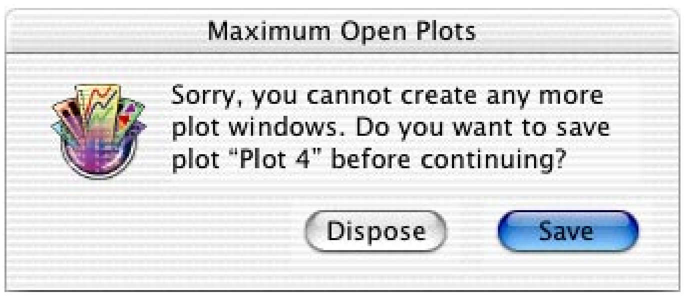


Figure 8-1 Maximum Open Plots dialog

8.2 Changing the View of the Plot Window

KaleidaGraph provides several magnification levels for changing the view of the plot window. To change the magnification of the plot, click the Zoom drop-down menu in the lower-left corner of the plot window and choose the desired zoom level. The available zoom percentages are: 25, 50, 75, 100, 125, 150, 200, 300, and 400%.

8.3 Displaying Rulers and Grids in the Plot Window

To turn the rulers and grids on and off in the plot window:

1. With the plot window active, choose **Plot** > **Ruler & Grids**.
2. Click the appropriate check boxes to display the rulers, grids, or both.
3. Select the proper units that you want to use.
4. Click **OK** to return to the plot window.

8.4 Modifying the Legend

The legend consists of a frame, the plot symbols, and the labels used to identify each variable in the plot. By default, the labels are taken from the column names in the data window. The legend displays the variables in the order they are selected for plotting.

8.4.1 Editing the Legend Frame and Color

Changing Foreground and Background Color

The Color icon on the toolbox consists of two sections: a foreground section (upper left) for setting the foreground color and a background section (lower right) for setting the background color. When you select a fill or pen pattern, the foreground color is applied to the black portion of the pattern and the background color is applied to the white portion of the pattern.

To change the foreground or background color of the legend:

1. Click the legend to select it.
2. Click the foreground or background section of the Color icon. One of the following color palettes appears with the current color choices.

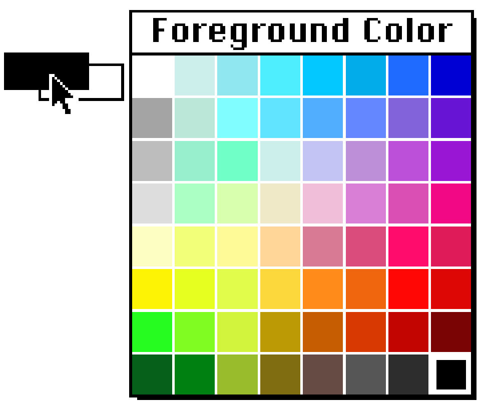


Figure 8-2 Foreground Color palette

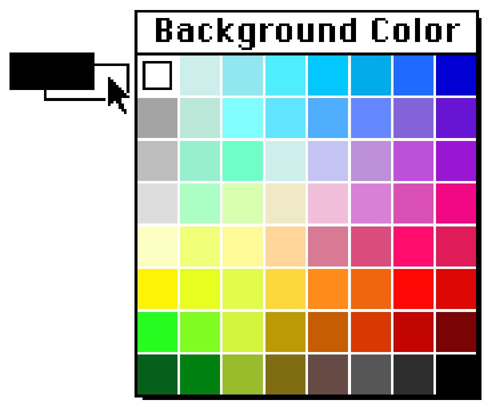


Figure 8-3 Background Color palette

3. Select a different color from the palette.

Changing Fill and Pen Patterns

The Pattern icon on the toolbox consists of two sections: an inner section for setting the fill pattern and an outer section for setting the pen pattern. The fill pattern controls the interior of the legend; the pen pattern controls the legend frame.

You can change the fill or pen pattern by selecting a different pattern from the pop-up menu that appears when you click the appropriate portion of the Pattern icon. Black portions of the pattern represent the foreground color and white portions represent the background color.

To change the fill or pen pattern of the legend:

1. Click the legend to select it.
2. Click the interior or outer portion of the Pattern icon. One of the following pop-up menus appears with the pattern choices.

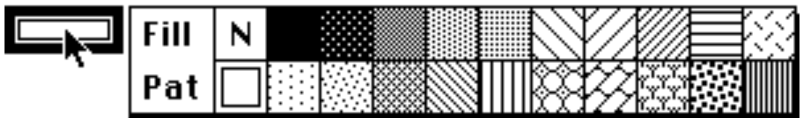


Figure 8-4 Fill Pattern choices

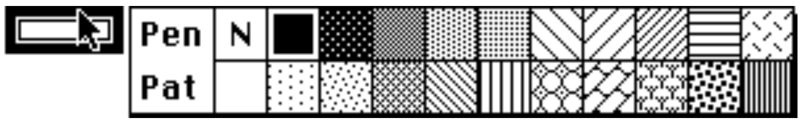


Figure 8-5 Pen Pattern choices

3. Select a different pattern from the pop-up menu.

Changing Line Style and Width

The Style icon on the toolbox controls the line style and width of the legend frame. The left portion of the icon controls the line style; the right portion controls the line width.

To change the line style or width of the legend frame:

1. Click the legend to select it.
2. Click the line style or line width portion of the Style icon. One of the following pop-up menus appears with the available choices.

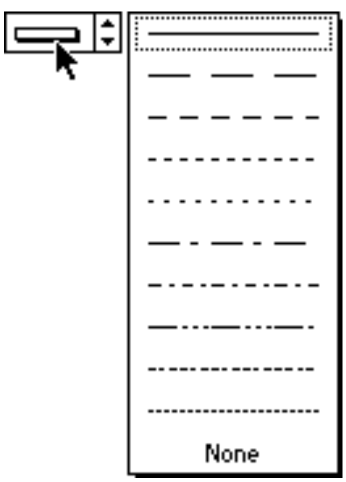


Figure 8-6 Line style pop-up menu

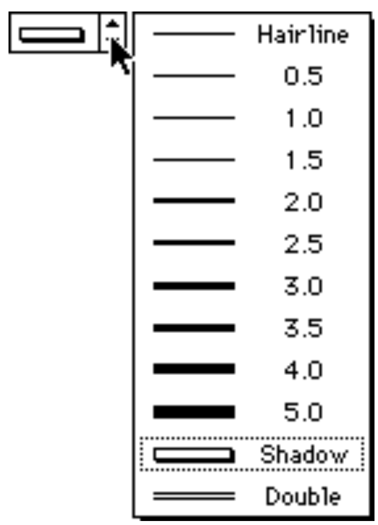


Figure 8-7 Line width pop-up menu

3. Select a different line style or width from the pop-up menu. To completely remove the frame around the legend, choose **None** from the Line Style pop-up menu.

Setting Legend Frame Defaults

To make the current legend style the default, choose **Plot** > **Define Layout**. Choosing this command only remembers the settings for this particular type of plot because each plot type in the **Gallery** menu can have its own layout.

This information is saved as part of the Style file. You must save the Style file for the layout to be remembered the next time you start KaleidaGraph.

8.4.2 Editing Legend Text

Editing Legend Text Attributes as a Group

The attributes of the legend text can be changed as a group by double-clicking any of the labels in the legend with the Selection Arrow. A variation of the Edit String dialog, shown in Figure 8-8, appears which can only be used to modify the attributes of the text labels. This dialog does not allow you to edit any of the text labels. See Section 9.7.4 for more information on the commands available in this dialog.

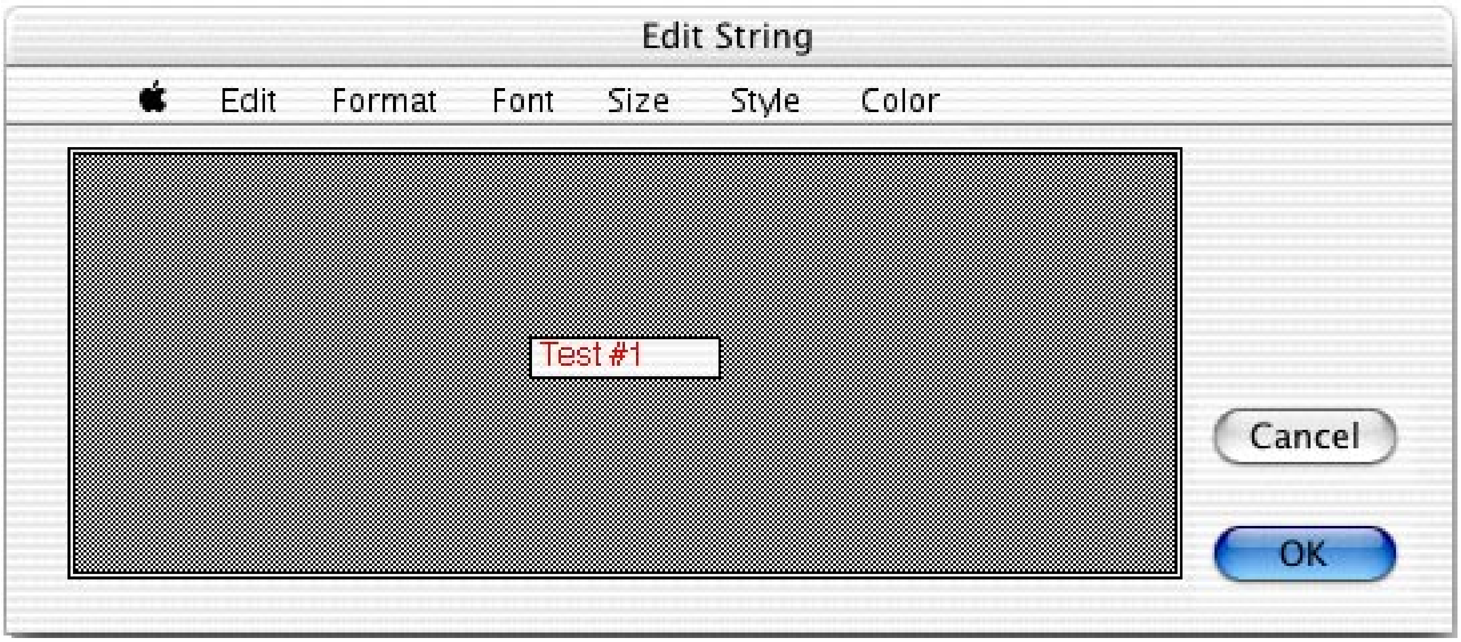


Figure 8-8 Variation of Edit String dialog

Editing Individual Items in the Legend

The labels in the legend can be edited using the Text tool. Double-clicking a text label displays the Edit String dialog. Edit the legend text as needed and click **OK** to return to the plot.

The labels in the legend can be moved to different locations using the Text tool. This is accomplished by dragging a label to a new position with the Text tool. To delete an item in the legend, use the Eraser.

Changing Legend Text Color Independent of Variable Color

By default, the color of the legend text is linked to the variable color set in the Plot Style dialog. To make these different colors, choose **Format** > **Text Options** and clear the **Link Text To Variable Color** check box. The text and variable colors can now be set independently.

8.5 Modifying Axes

This section explains how to control the appearance of the axes. The Axis Options dialog, shown in Figure 8-9, contains almost all of the options related to the display of the axes. Many of these settings are also available directly from the toolbox. Section C.2 explains how to modify the axes using the toolbox.

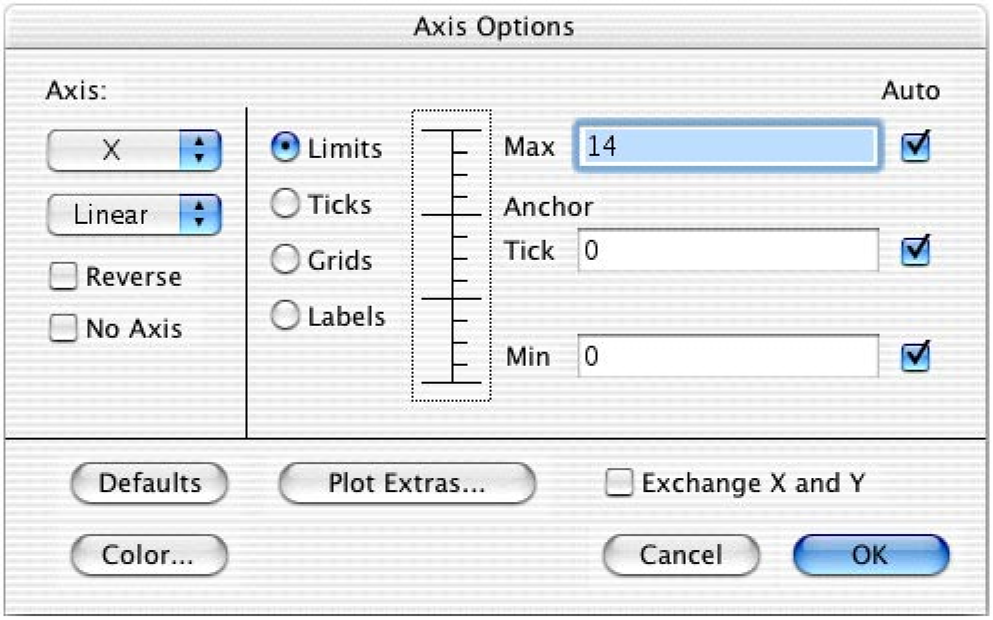


Figure 8-9 Axis Options dialog

8.5.1 Setting the Axis Limits

The axis limits are based on the data used to create the plot. KaleidaGraph automatically calculates the beginning and ending values and the placement of tick marks along the axes.

You can specify your own limits in the Axis Options dialog. The **Min** and **Max** fields determine the lowest and highest values on the selected axis. The **Anchor Tick** field indicates the value from which all major tick marks on a linear axis are calculated. The Anchor Tick field has no affect on axes with logarithmic scales.

To set the axis limits:

1. With the plot window active, choose **Plot** > **Axis Options**.
2. Choose the appropriate axis from the pop-up menu in the upper-left corner of the dialog.
3. Click **Limits** if it is not already selected. The Axis Options dialog changes to display the options available for controlling axis limits.
4. Enter values for **Min**, **Max**, and **Anchor Tick**. To return to the original values calculated by KaleidaGraph, click the **Auto** check boxes.
5. Click **OK** to return to the plot.

8.5.2 Selecting Linear or Log Scaling

The numeric axes in KaleidaGraph can have either linear or logarithmic scaling. When **Linear** scaling is selected, the axis is divided into equal sections. When **Log** scaling is used, each increment of the axis is a power of 10. Partial decades are allowed when using the log scale.

To change the scaling of an axis:

1. With the plot window active, choose **Plot** > **Axis Options**.
2. Choose the appropriate axis from the pop-up menu in the upper-left corner of the dialog.
3. Choose **Linear** or **Log** from the pop-up menu beneath the menu used to select the axis.

   Note: The **Min** value in the Axis Options dialog must be greater than zero to use log scaling. If you try to use non-positive limits, a message appears and the axis scaling reverts to linear.

4. Click **OK** to return to the plot.

8.5.3 Hiding an Axis

It is possible to hide each of the four axes in a plot using the **No Axis** option in the Axis Options dialog.

To hide an axis:

1. With the plot window active, choose **Plot** > **Axis Options**.
2. Choose the appropriate axis from the pop-up menu in the upper-left corner of the dialog.
3. Click the **No Axis** check box.
4. Click **OK** to return to the plot.

8.5.4 Exchanging Axes

The **Exchange X and Y** option in the Axis Options dialog can be used to swap the X and Y axes of a plot. When this check box is selected, the X and X2 axes switch position with the Y and Y2 axes.

Note: Using this option does not change the independent and dependent variables. It only changes their position in the plot. This option has no affect on Double Y, Double X, Double XY, Bar, Polar, and Pie plots.

To exchange the X and Y axes:

1. With the plot window active, choose **Plot** > **Axis Options**.
2. Click the **Exchange X and Y** check box.
3. Click **OK** to return to the plot.

8.5.5 Reversing the Variables on an Axis

You can control whether the variables are plotted in ascending or descending order along an axis. By selecting the **Reverse** check box in the Axis Options dialog, variables are plotted in descending order along the selected axis.

To reverse an axis on the plot:

1. With the plot window active, choose **Plot** > **Axis Options**.
2. Choose the appropriate axis from the pop-up menu in the upper-left corner of the dialog.
3. Click the **Reverse** check box.
4. Click **OK** to return to the plot.

8.5.6 Linking a Second Axis

You can control the display of the tick marks and labels on the X2 and Y2 axes using the **Link To** pop-up menu. This pop-up menu only appears in the Axis Options dialog when the X2 or Y2 axis is selected.

Note: The Link To pop-up menu is not displayed for certain axes in Double Y, Double X, and Double XY plots because they are independent of the main axis.

To link the X2 or Y2 axis:

1. With the plot window active, choose **Plot** > **Axis Options**.
2. Choose the X2 or Y2 axis from the pop-up menu in the upper-left corner of the dialog.
3. From the **Link To** pop-up menu, select how the axis should be linked. The options include:
   - **No Ticks, No Labels** - The X2 or Y2 axis does not have any ticks or labels.
   - **Axis, No Labels** - The X2 or Y2 axis has the same tick marks as the main axis, but does not have any labels.
   - **Axis, Labels** - The X2 or Y2 axis has the same tick marks and labels as the main axis.
   - **Custom Axis, Custom Labels** - Depending on the scaling of the main axis, the X2 or Y2 axis is linked by either a linear or exponential equation. The equation is entered into one of the dialogs shown in Figure 8-10. The tick marks and labels are calculated and displayed on the X2 or Y2 axis.

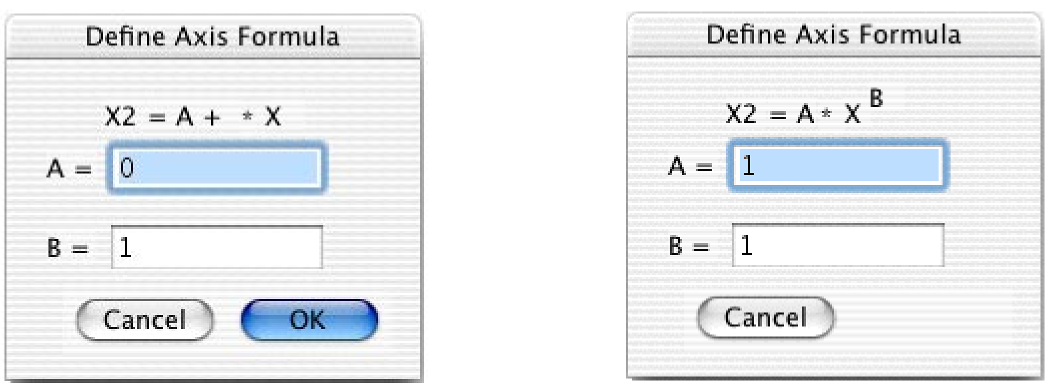


Figure 8-10 Define Axis Formula dialogs

4. Click **OK** to return to the plot.

8.5.7 Changing Axis Thickness

To change the thickness of the axes:

1. Choose **Format** > **Plot Extras**.
2. To increase the thickness, change the **Frame Thickness** to 200, 300, or 400%.

   Note: Depending on the value entered, you may not notice the change on the screen; however, there will be a difference when the plot is printed.

3. Click **OK** to return to the plot.

8.6 Modifying Tick Marks

This section explains how to change the appearance of the tick marks. The Axis Options dialog, shown in Figure 8-11, contains almost all of the options related to the display of the tick marks. Many of these settings are also available directly from the toolbox. Section C.2.4 explains how to modify the tick marks using the toolbox.

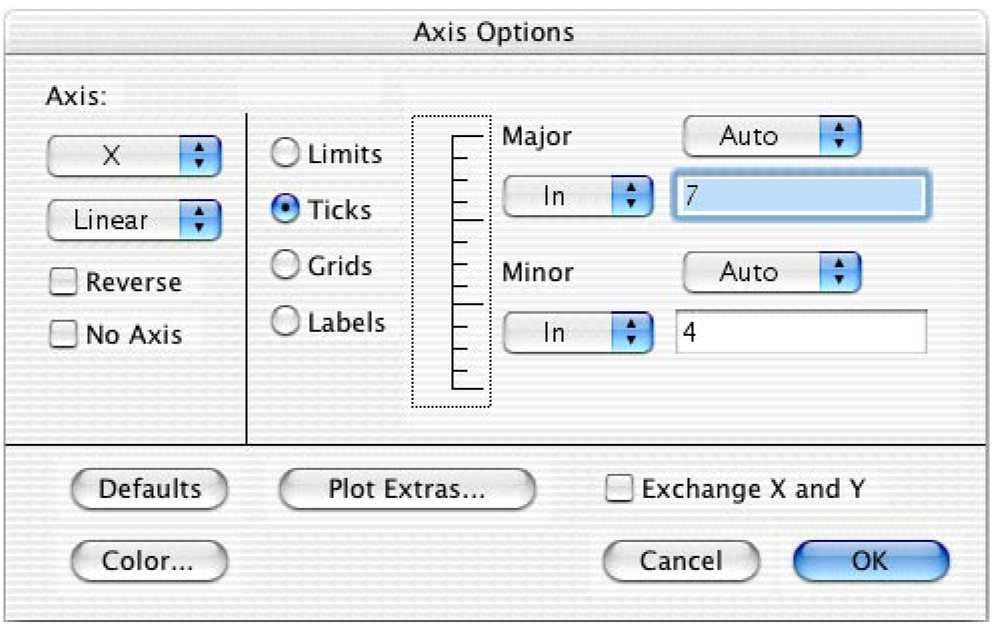


Figure 8-11 Axis Options dialog

8.6.1 Changing the Display of Tick Marks

The major and minor ticks can be displayed inside, outside, or on both sides of the axis. You can also choose not to display them at all.

To control how the major and minor ticks are displayed:

1. With the plot window active, choose **Plot** > **Axis Options**.
2. Choose the appropriate axis from the pop-up menu in the upper-left corner of the dialog.
3. Click **Ticks**. The Axis Options dialog changes to display the options available for controlling major and minor ticks.
4. Choose **In**, **Out**, **Both**, or **None** from the pop-up menus below **Major** and **Minor** to specify how the major and minor ticks should be displayed.
5. Click **OK** to return to the plot.

Figure 8-12 shows how the tick marks appear when you select (in order): **In**, **Out**, **Both**, and **None** for the X and Y axes.

Figure 8-12 Tick mark display

8.6.2 Changing the Number of Tick Marks

When **Linear** scaling is selected for an axis, the number of major tick marks is controlled by the pop-up menu to the right of **Major** in the Axis Options dialog. They can either be calculated automatically or set to a specific number.

To change the number of major ticks:

1. With the plot window active, choose **Plot** > **Axis Options**.
2. Choose the appropriate axis from the pop-up menu in the upper-left corner of the dialog.
3. Click **Ticks**. The Axis Options dialog changes to display the options available for controlling major and minor ticks.
4. Choose **Auto** or **Fixed #** from the pop-up menu to the right of **Major**.
5. If **Fixed #** is selected, enter the desired number of divisions in the field below the pop-up menu.
6. Click **OK** to return to the plot.

The number of minor tick marks is controlled in the same manner as major ticks. You can either specify a fixed number or let KaleidaGraph calculate the number of minor ticks.

8.6.3 Changing the Tick Mark Interval

When **Linear** scaling is selected for an axis, KaleidaGraph automatically determines the number of major tick marks. Using the Axis Options command, it is possible to place the tick marks at specific intervals along an axis.

To change the interval between major tick marks:

1. With the plot window active, choose **Plot** > **Axis Options**.
2. Choose the appropriate axis from the pop-up menu in the upper-left corner of the dialog.
3. Click **Limits** if it is not already selected. The Axis Options dialog changes to display the options available for controlling axis limits.
4. Enter a value in the **Anchor Tick** field. The intervals are calculated from the Anchor Tick value.
5. Click **Ticks**.
6. Choose **Interval** from the pop-up menu to the right of **Major** and enter the desired interval in the field below it. For dates, enter the desired number of days for the interval. For time data, use a D:H:M:S.S format for entering the interval.
7. Click **OK** to return to the plot.

8.6.4 Changing Tick Mark Length or Thickness

To change the length or thickness of the tick marks:

1. Choose **Format** > **Plot Extras**.
2. To change the length, edit the **Inside** and **Outside** values for **Major** or **Minor**.
3. To increase the thickness, change the **Tick Thickness** to 200, 300, or 400%.

   Note: Depending on the value entered, you may not notice the change on the screen; however, there will be a difference when the plot is printed.

4. Click **OK** to return to the plot.

8.7 Modifying Grid Lines

This section explains how to control the appearance of the grid lines. The Axis Options dialog, shown in Figure 8-13, contains most of the options related to the display of the grid lines. Many of these settings are also available directly from the toolbox. Section C.2.5 explains how to modify the grid lines using the toolbox

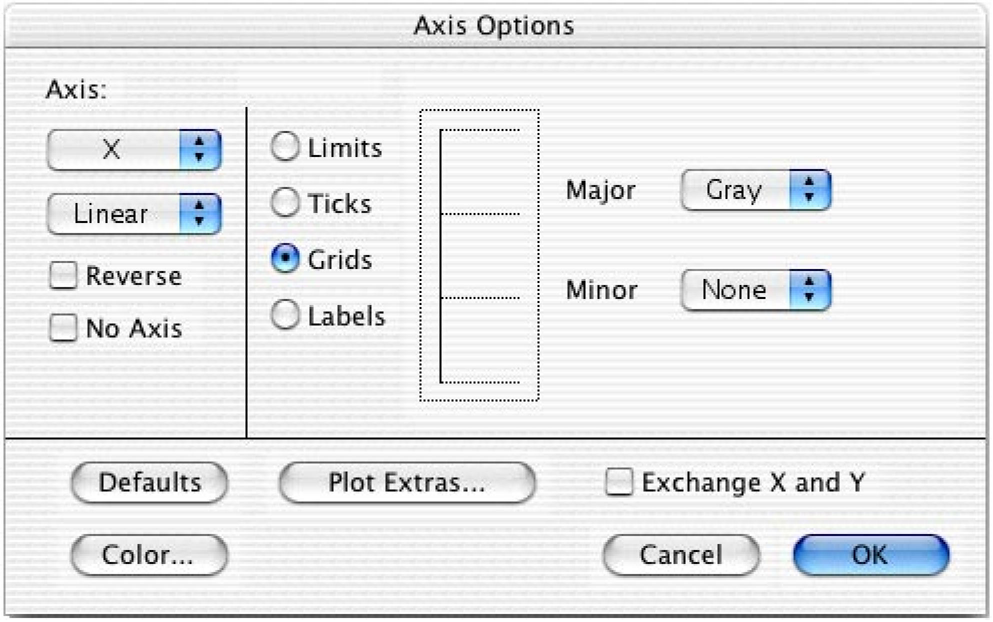


Figure 8-13 Axis Options dialog

8.7.1 Controlling the Display of Grid Lines

Grid lines can be displayed at each of the major and minor tick marks. The lines can be solid, gray (dotted), or not displayed at all. Section 8.10 explains how to change the color of the grid lines.

To control how the grid lines are displayed:

1. With the plot window active, choose **Plot** > **Axis Options**.
2. Choose the appropriate axis from the pop-up menu in the upper-left corner of the dialog.
3. Click **Grids**. The Axis Options dialog changes to display the options available for controlling major and minor grids.
4. Choose **Solid**, **Gray**, or **None** from the pop-up menus to the right of **Major** and **Minor** to specify how the major and minor grids should be displayed.
5. Click **OK** to return to the plot.

8.7.2 Changing Grid Line Thickness

To change the thickness of the grid lines:

1. Choose **Format** > **Plot Extras**.
2. To increase the thickness, change the **Grid Thickness** to 200, 300, or 400%.

   Note: Depending on the value entered, you may not notice the change on the screen; however, there will be a difference when the plot is printed.

3. Click **OK** to return to the plot.

8.8 Modifying Axis Labels

This section explains how to control the frequency and format of the axis labels. The Axis Options dialog, shown in Figure 8-14, contains almost all of the options related to the number and format of the axis labels. Many of these settings are also available directly from the toolbox. Section C.2.6 explains how to modify the axis labels using the toolbox.

Note: For information on changing the text attributes of the labels, see Section 9.7.2.

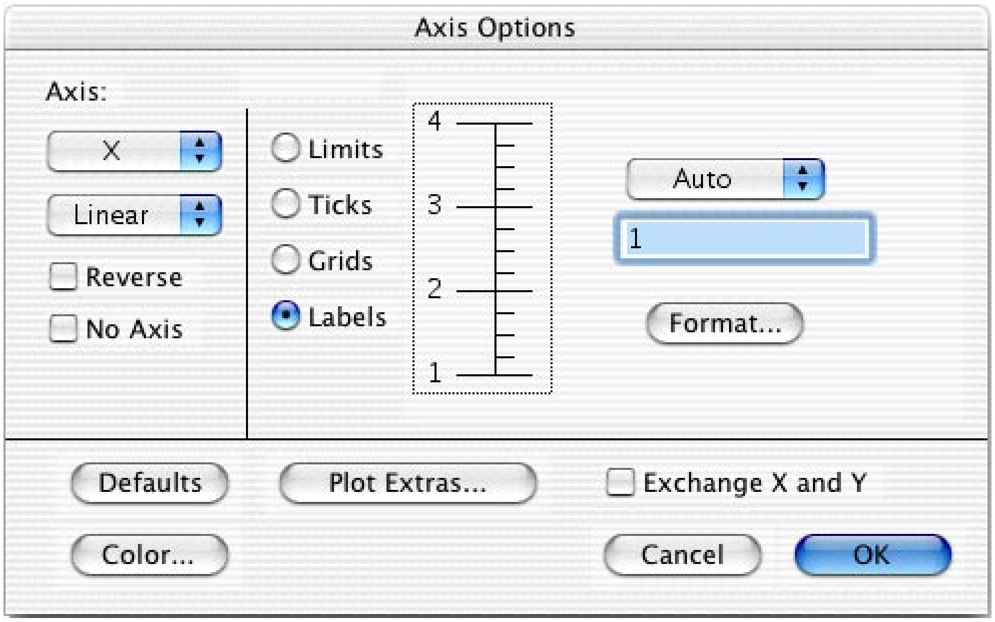


Figure 8-14 Axis Options dialog

8.8.1 Controlling the Frequency of Axis Labels

KaleidaGraph automatically calculates the frequency of axis labels. The frequency depends on the number of major tick marks. You can specify a different frequency or you can choose to remove the axis labels from the plot.

To control the frequency of the axis labels:

1. With the plot window active, choose **Plot** > **Axis Options**.
2. Choose the appropriate axis from the pop-up menu in the upper-left corner of the dialog.
3. Click **Labels**. The Axis Options dialog changes to display the options available for specifying label frequency.
4. Choose **Auto**, **Every Nth**, or **None** from the pop-up menu to specify how often the labels should be displayed. If you select **Every Nth**, enter a number in the field below the menu.
5. Click **OK** to return to the plot.

8.8.2 Formatting Axis Labels

The format of the axis labels can also be controlled using the Axis Options dialog. Clicking the **Format** button (when **Labels** is selected) displays a dialog similar to Figure 8-15. The same dialog can be displayed using any one of four commands from the **Format** menu: **X Label Format**, **Y Label Format**, **Y2 Label Format**, and **X2 Label Format**.

Note: This dialog does not have any affect if the data has a Text format in the data window.

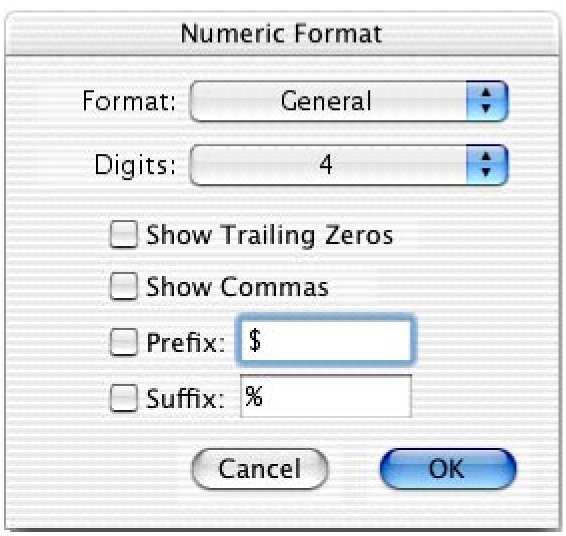


Figure 8-15 Label Format dialog

To change the format of the axis labels:

1. Click **Labels** and then click **Format** in the Axis Options dialog. You can also choose the appropriate **Label Format** command (X, Y, X2, or Y2) from the **Format** menu. The Label Format dialog is displayed using either method.
2. Make your selections in the dialog.
3. Click **OK** to return to the plot window.

8.9 Modifying Lines, Markers, and Fill Patterns

This section explains how to control the display of the data in the plot window. The Plot Style dialog, shown in Figure 8-16, contains most of the options available for the display of the plot variables. Many of these settings are also available directly from the toolbox. Section C.3 explains how to modify the plot style using the toolbox.

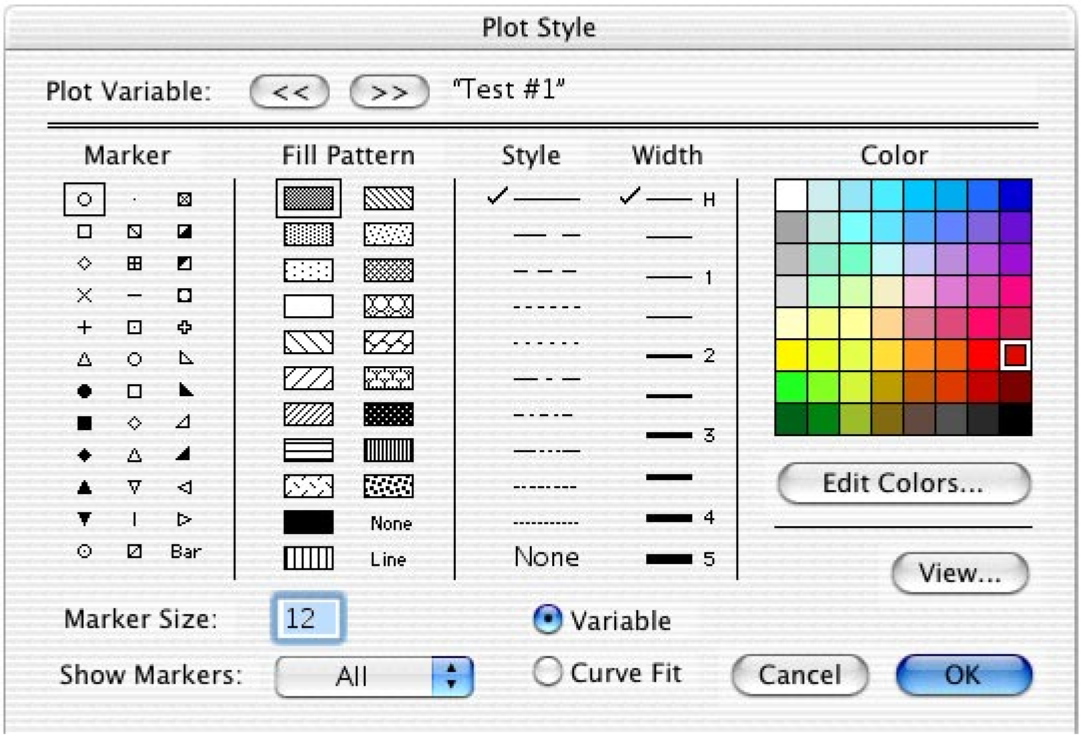


Figure 8-16 Plot Style dialog

8.9.1 Selecting Variables in the Plot Style Dialog

Before variables can be modified, they need to be selected in the Plot Style dialog. Each variable is independently controlled in this dialog.

To select variables for editing in the Plot Style dialog:

1. With the plot window active, choose **Plot** > **Plot Style**.
2. Use the << and >> buttons to locate the desired variable. The name of the current variable appears to the right of these buttons.

   Note: The **Tab** and **Down Arrow** keys can be used to move to the next variable. You can go to the previous variable using **Shift**+**Tab** or the **Up Arrow** key. The **Left Arrow** key takes you to the first variable; the **Right Arrow** key takes you to the last variable.

8.9.2 Changing Marker Type, Size, or Number of Markers

Most of the plots have the ability to display markers on the plot. The exceptions are the Histogram, Stack Bar, Stack Column, and Pie plots. The Plot Style dialog controls the type, size, and number of markers displayed in the plot.

To change the marker type, size, or number of markers:

1. With the plot window active, choose **Plot** > **Plot Style**.
2. Select the variable to be edited. See Section 8.9.1 if you are unsure of how to do this.
3. To change the marker type, click one of the symbols below **Marker**. The first six markers in the left column are transparent; all of the others are opaque. The active marker has a box around it.
4. To change the size of the marker, edit the value in the **Marker Size** field.
5. To change the number of markers displayed, make a selection from the **Show Markers** pop-up menu. You can choose whether all, a fixed number, a percentage, every Nth, or none of the markers are displayed on the plot. The current setting is preceded by a check mark.

   Note: Scatter plots always show all of the markers. Use a Line plot to show fewer markers.
6. Click **OK** to return to the plot.

8.9.3 Selecting Fill Patterns

All of the plots have the ability to have a fill pattern displayed for the variable. For some plots, like the Line plot, this is only possible if **Bar** is selected as the marker type in the Plot Style dialog.

Note: It is not possible to assign a fill pattern to the markers.

To change the fill pattern:

1. With the plot window active, choose **Plot** > **Plot Style**.
2. Select the variable to be edited. See Section 8.9.1 if you are unsure of how to do this.
3. Click one of the patterns below **Fill Pattern**. The active fill pattern has a box around it.
4. Click **OK** to return to the plot.

8.9.4 Changing Appearance of Variable and Curve Fit Lines

Line, Double Y, Double X, Double XY, High/Low, Step, Probability, X-Y Probability, Horizontal Bar, Column, and Function plots can use a line to represent a plotted variable. Curve fits can be applied to all of these plots as well as Scatter plots. The style, width, and color of the variable or curve fit lines can be set using the Plot Style dialog.

Note: Variable and curve fit lines are controlled independently of each other. Changing the variable color also changes the marker color. The associated legend text is also changed if **Link Text To Variable Color** is selected in the Text Options dialog (**Format** menu).

To change the line style, width, or color of variable or curve fit lines:

1. With the plot window active, choose **Plot** > **Plot Style**.
2. Select the variable to be edited. See Section 8.9.1 if you are unsure of how to do this.
3. If changes are being made to the variable line, proceed to the next step. Otherwise, click **Curve Fit** to make changes to the curve fit line.
4. To change the line style, select a different option from the **Style** column. Choose **None** from this list to hide the line. The current style is preceded by a check mark.
5. To change the line width, select a different option from the **Width** column. The current width is preceded by a check mark.

   Note: If you also want to make the markers thicker, follow the instructions in Section 8.9.5.
6. To change the color, select a different color from the palette. The current color has a white square around it.
7. Click **OK** to return to the plot.

8.9.5 Changing Marker and Line Thickness

The Plot Style dialog contains a number of options for setting the thickness of the variable and curve fit lines. However, this setting does not affect the thickness of the markers. The Plot Extras dialog contains a Line Thickness setting that can be used to increase the thickness of markers, variable lines, and curve fit lines.

To change the thickness of the markers, variable lines, and curve fit lines:

1. Choose **Format** > **Plot Extras**.
2. To increase the thickness, change the **Line Thickness** to 200, 300, or 400%.

   Note: Depending on the value entered, you may not notice the change on the screen; however, there will be a difference when the plot is printed.
3. Click **OK** to return to the plot.

8.10 Changing Plot Color

This section explains how to change the frame, grid, interior, and background colors for a plot. These settings are also available directly from the toolbox. Section C.4 explains how to modify the colors using the toolbox

To change the plot colors:

1. Choose **Format** > **Plot Color** or click the **Color** button in the Axis Options dialog. The dialog in Figure 8-17 appears using either method.

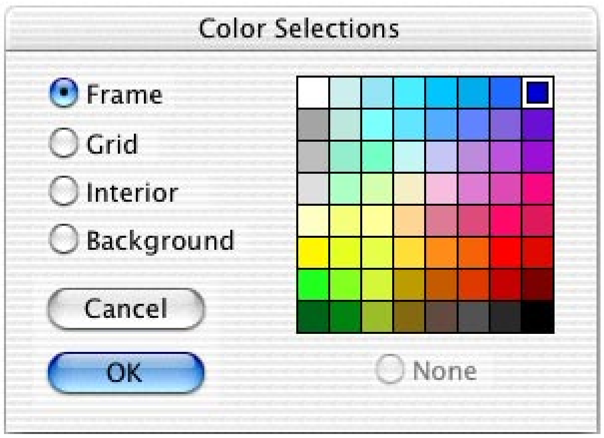


Figure 8-17 Plot Color dialog

2. Click one of the buttons and select a color from the palette for that item. If **None** is selected for the interior or background, that particular portion of the plot is transparent.
3. Click **OK** to return to the plot.

8.11 Resizing the Plot

There are two methods available for changing the size of the plot: the Set Plot Size command and the Selection Arrow. Changing the size of the plot has no affect on the size of the plot window. Once the plot has been resized, you can make these settings the new default by choosing **Plot** > **Define Layout** and clicking **OK**.

8.11.1 Set Plot Size Command

You can use the **Set Plot Size** command (**Plot** menu) to specify the exact dimensions for the axes and frame, as well as the position of the axis origin.

To set the size of the plot:

1. Choose **Plot** > **Set Plot Size**. The dialog in Figure 8-18 appears.

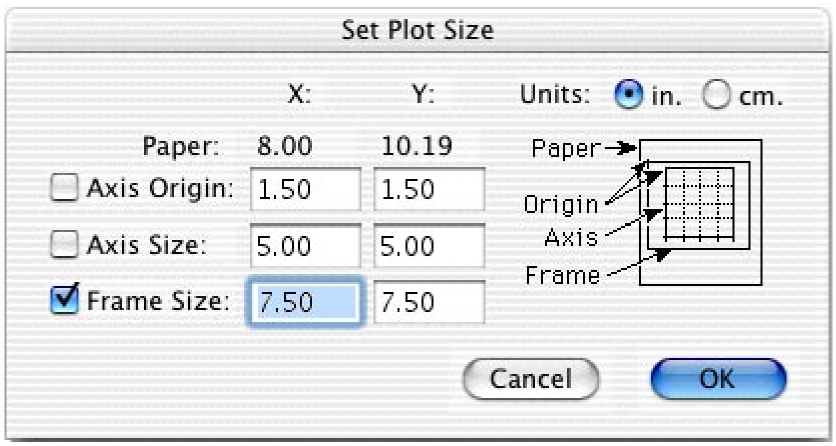


Figure 8-18 Set Plot Size dialog

2. Click **Axis Size**, **Frame Size**, or both and enter the desired X and Y dimensions.
   - If **Axis Size** is selected, the frame is scaled proportionally to the new dimensions.
   - If **Frame Size** is selected, the axis is scaled proportionally to the new dimensions.
   - If both are selected, the axis and frame sizes are set independently. Be careful; this can cause parts of the plot to be placed outside of the plot frame.

     Note: Typing the word **auto** allows a floating dimension, while specifying the size of the other dimension. This allows the plot to remain proportional to its original size.
3. If necessary, click the **Axis Origin** check box and specify where to place the upper-left corner of the axes.
4. Click **OK**. The plot is resized based on the new settings.

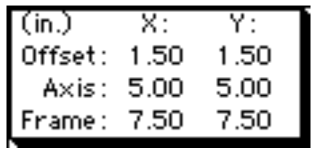
If the values in the **Frame Size** field are larger than the **Paper** dimensions, a dotted line is displayed to represent the printable area of the paper. You can hide these lines by clearing the **Show the plot paper boundary** check box in the Preferences dialog (**File** menu).

8.11.2 Using the Selection Arrow

The Selection Arrow can be used to manually resize the plot or plot frame.

To resize the plot:

1. Click one of the axes with the Selection Arrow. Object handles appear at each corner of the plot.
2. Drag one of the handles to a new position. As you drag, a dialog appears in the upper-left corner of the plot window. This dialog (shown below) contains the current plot dimensions and axis origin (offset).



Note: To resize the plot proportionally about its center, press **Alt** (Windows) or **Option** (Macintosh) when dragging an object handle. By pressing **Shift** as you drag, you can constrain the plot to a square.

3. Release the mouse button when the axes are the desired size.

To resize the plot frame:

1. Locate the lower-right corner of the plot frame. You can use the scroll bars or increase the size of the plot window to find it.
2. Using the Selection Arrow, click the Size box ( ) in the lower-right corner.
3. Drag the box to resize the plot frame. As you drag, a dialog appears in the upper-left corner of the plot window to display the current dimensions.
4. Release the mouse button when the plot frame is the desired size. The size of the plot changes proportionally to the new size of the frame.

8.12 Extracting Data from a Plot

When you save a plot with data, the data windows used to create it are archived with the plot. This enables you to save the graph and data in a single file instead of saving them separately. The data is not saved with the plot if either of the following occurs:

- The data windows used to create the plot are closed before the plot is saved.
- The plot is saved by choosing **File** > **Export** > **Plot w/o Data**.

The next time the plot is opened, you can make any necessary changes to the plot. If you want to modify the data, it needs to be extracted from the plot. This is accomplished by choosing **Plot** > **Extract Data** or clicking the Find Data button ( ) in the plot window. Once the data is extracted, it can be modified, saved to disk, or used to create new plots.

Note: The **Extract Data** command is unavailable if the plot does not have archived data associated with it, the plot was just created, or the data has already been extracted.

To extract a data set:

1. Make a plot window the active window.
2. Choose **Plot** > **Extract Data** or click the Find Data button ( ) in the upper-right corner of the plot window. The data used to create the plot is displayed. Figure 8-19 shows an example of an extracted data set.

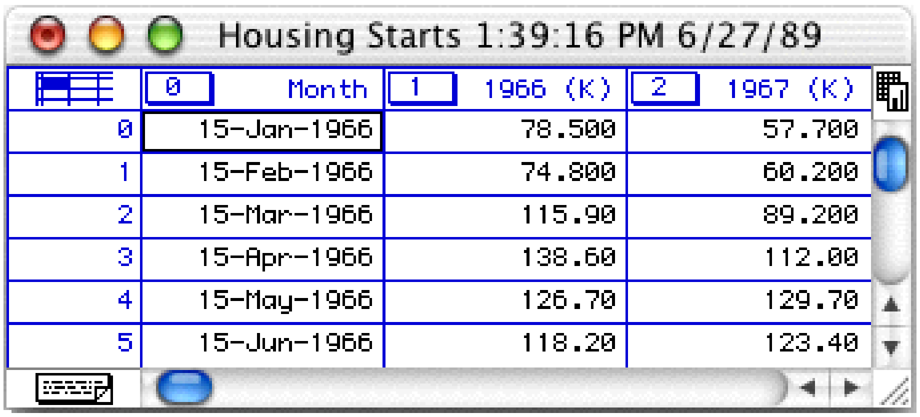


Figure 8-19 Extracted data window

The title of the extracted data window begins with the same name as the original data file. Additionally, a date and time stamp is appended to the name, identifying when the data was archived in the plot.

Note: If you attempt to close an extracted data window, the message in Figure 8-20 appears. If you click **OK**, the plot loses its link with the original data.

Figure 8-20 Warning dialog

8.13 Updating Plotted Data

The **Plot** menu contains two commands that can be used to update the data in a plot: **Auto Link** and **Update Plot**. Both of these commands operate in a similar manner. They cause the current plot to be re-created to reflect any changes in its data. Curve fits and error bars are also recalculated based on any changes in the data

The difference between the two commands is that **Auto Link** revises the plot after each individual change in the data while **Update Plot** only causes an update when you choose the command. Clicking the Update Plot button ( ) in the data window performs the same function as choosing the Update Plot command.

These commands are useful if you have a saved plot and need to add or edit points. You can extract the data from the plot (using either the **Extract Data** command or the Find Data button), modify the data, and use the Auto Link or Update Plot commands to revise the plot.

To update plots using the **Auto Link** command:

1. Open or activate the plot to be updated.
2. Choose **Plot** > **Extract Data** or click the Find Data button ( ) to display the data used to create the plot.
3. Turn on **Auto Link** in the **Plot** menu.
4. Modify the data and move to another cell. After each data value is modified, the plot is redrawn to show the changes.

To update plots using the **Update Plot** command:

1. Open or activate the plot to be updated.
2. Choose **Plot** > **Extract Data** or click the Find Data button ( ) to display the data used to create the plot.
3. Make any changes to the extracted data.
4. Choose **Plot** > **Update Plot** or click the Update Plot button ( ) in the data window to redraw the plot.

8.14 Plotting Additional Data

There may be times when you want to add additional variables to an existing plot. Instead of creating the plot from scratch, you can open a plot and add new data to it.

Note: If you just want to add a few points to an existing curve, refer to Section 8.13. The steps below are for adding a new variable to the plot.

To add additional variables to a plot:

1. Open or activate the plot which will have the new data plotted.
2. Choose **Plot** > **Extract Data** or click the Find Data button ( ) to display the data used to create the plot.
3. Open the new data or add it to an existing data window.
4. Switch back to the plot and choose **Gallery** > **Template**.
5. Select all of the data to be plotted (including the data from the original plot) and click **New Plot**.

A new plot will be created that includes the additional data. Any curve fits and error bars from the original plot will be applied to the new plot.

8.15 Customizing Bar Plots

This section explains some of the special features associated with Bar plots. This includes changing the spacing between bars, displaying value labels, and creating Line/Column plots.

8.15.1 Changing Bar Attributes

The **Plot Extras** command (**Format** menu) contains four options that affect the appearance of the bars. For details on changing the fill pattern in the bars, see Section 8.9.3.

To change the attributes of the bars:

1. Choose **Format** > **Plot Extras**. The dialog in Figure 8-21 appears.

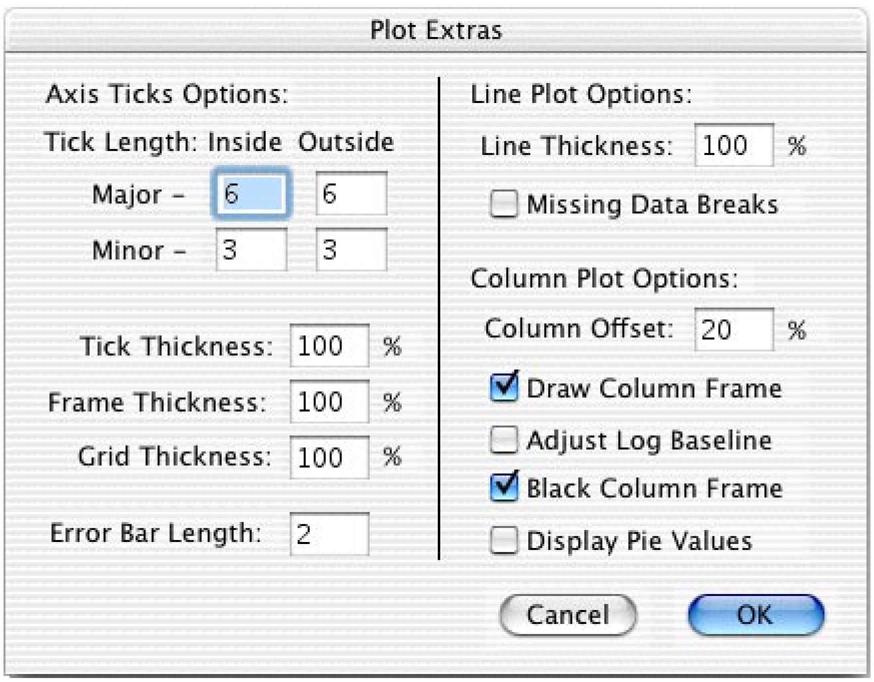


Figure 8-21 Plot Extras dialog

2. To change the spacing between the bars, edit the **Column Offset** percentage. Entering a value of 0% removes all space between the bars; a value of 100% displays the bars as spikes.
3. To remove the frame surrounding each bar, clear the **Draw Column Frame** check box.
4. To make the frame the same color as the fill pattern, clear the **Black Column Frame** check box.
5. To use a baseline of 1.0 for log scales, select the Adjust Log Baseline check box. Otherwise, the minimum axis limit is used for the baseline.
6. Click **OK** to return to the plot.

8.15.2 Adding Value Labels

Labels that display the value of each bar can be added to Bar plots. For Horizontal Bar and Column plots, the label is placed at the end of each bar. For Stack Bar and Stack Column plots, labels are placed on each section of the bar.

To add value labels to a Bar plot:

1. Select the plot which will have the value labels added.
2. Turn on **Add Values** in the **Plot** menu. The actual values of each bar are added to the plot.

Figure 8-22 shows a Bar plot that contains value labels. These labels can be moved individually with the Text tool or as a group with the Selection Arrow. The **Value Label Format** command (**Format** menu) can be used to change the numeric format of the labels.

Note: Any time the plot is redrawn, the value labels are placed back at their original position. If you are displaying only a few of the labels, use the Text tool to add them to the plot. You can place them exactly where you want and you do not have to worry about the label moving when the plot is redrawn

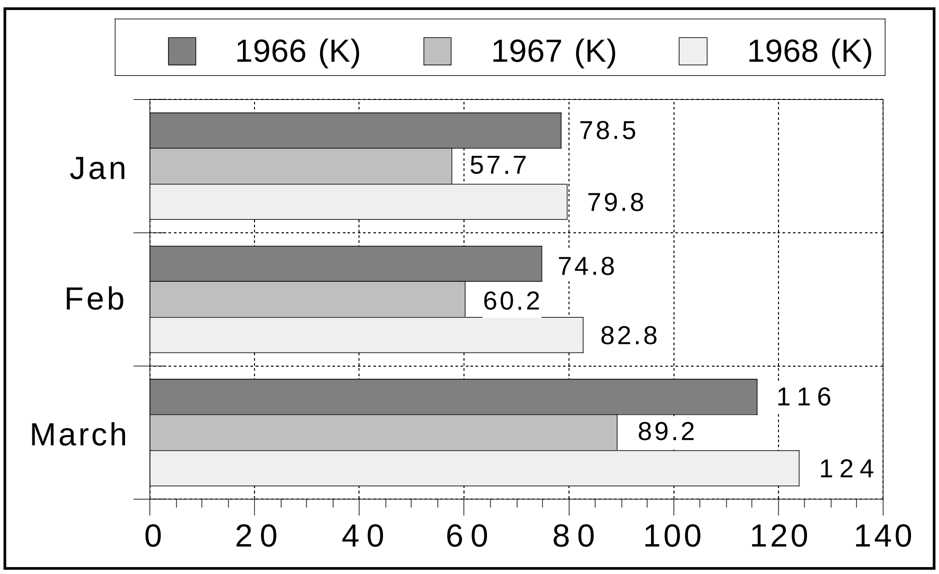


Figure 8-22 Bar plot with value labels

To change the text attributes of the value labels:

1. Double-click one of the text labels with the Selection Arrow. The Edit String dialog appears.
2. Make any changes to the label attributes. The options in this dialog are covered in Section 9.7.4. Setting default attributes is covered in Section 9.7.5.
3. Click **OK** to update the value labels.

8.15.3 Adding Lines to Bar and Column Plots

Horizontal Bar and Column plots can have lines added to create Line/Column plots.

To make a Line/Column plot:

1. Create a Horizontal Bar or Column plot, including the variable to be displayed as a line. Figure 8-23 shows a sample Column plot with two variables plotted.
2. Choose **Plot** > **Plot Style**. Use the << and >> buttons to locate the variable to be displayed as a line
3. Select a Fill Pattern of **Line**, as well as the line style, line width, and number of markers to be displayed.
4. Click **OK** and the plot updates to display a Line/Column plot. Figure 8-24 shows the sample plot after displaying the second variable as a line.

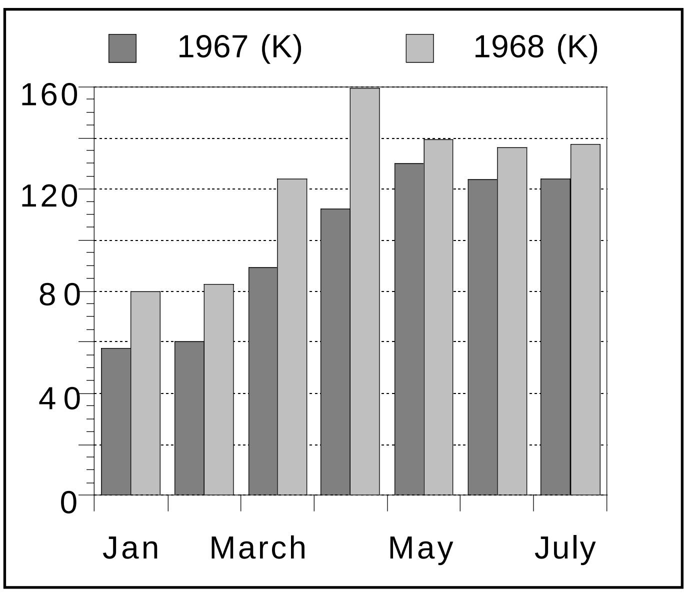


Figure 8-23 Sample Column plot

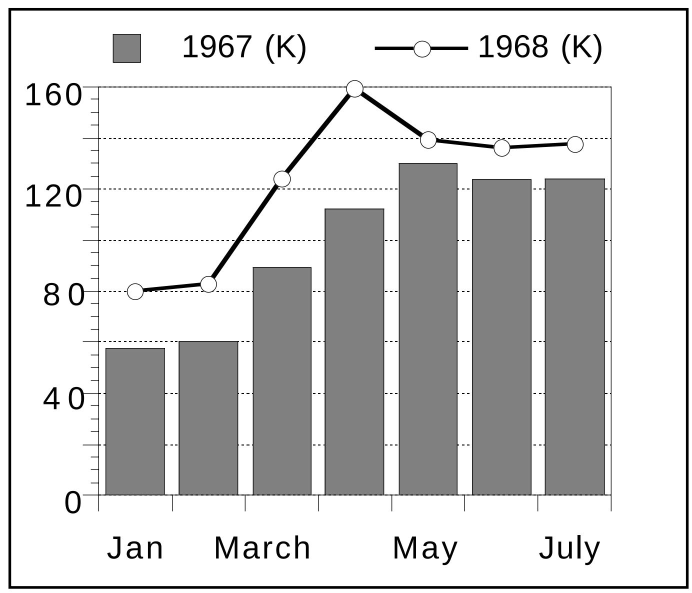


Figure 8-24 Plot displaying variable as a line

8.16 Customizing Polar Plots

There are two commands that can be used to change the appearance of Polar plots: **Axis Options** (**Plot** menu) and **Polar Options** (**Format** menu). Axis Options controls the display of tick marks, grid lines, and axis labels while Polar Options controls the grid type, reference angle, and direction.

8.16.1 Changing the Display of Tick Marks, Grid Lines, and Axis Labels

The angular (Ø or X) and the radial (R or Y) axes can be modified using the Axis Options command. Choosing **Plot** > **Axis Options** displays the dialog in Figure 8-25. The options for each of the axes are discussed in the following paragraphs.

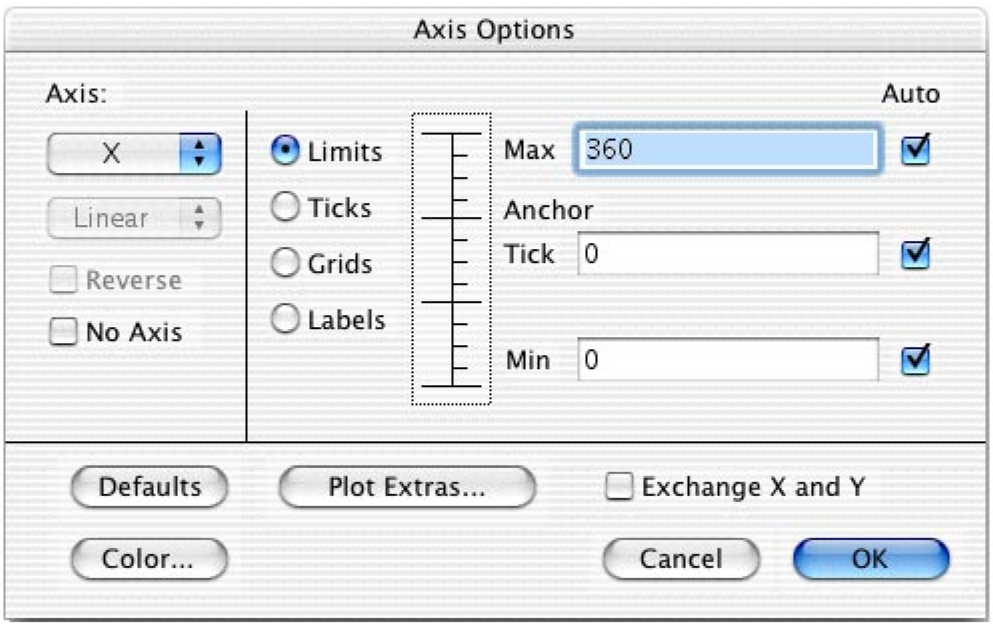


Figure 8-25 Axis Options dialog

The angular axis is controlled by the X axis settings in the Axis Options dialog. You can use the options for **Ticks**, **Grids**, and **Labels** to modify this axis. You can also use the **No Axis** option to remove the angular axis

The radial axis is controlled by the Y axis settings in the Axis Options dialog. You can use the options for **Limits**, **Ticks**, **Grids**, and **Labels** to modify the axis. You can also choose between **Linear** and **Log** scaling, or remove the axis with the **No Axis** option.

8.16.2 Setting the Grid Type, Reference Angle, and Direction

Polar plots have several unique options that control their appearance. These options include the grid type, data format, reference angle, and direction. All of these options are controlled using the **Polar Options** command (**Format** menu).

To change the Polar plot options:

1. Choose **Format** > **Polar Options**. The dialog in Figure 8-26 is displayed.

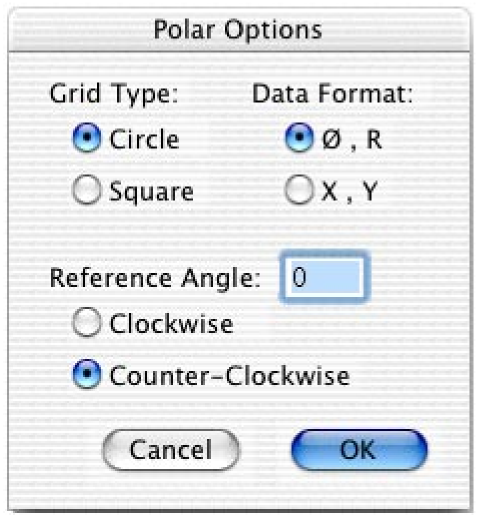


Figure 8-26 Polar Options dialog

2. Make any changes to the settings in this dialog.
3. Click **OK** to return to the plot.

Figure 8-27 shows a Polar plot displayed in a **Clockwise** direction, with a circular grid and a 45^{o} **Reference Angle**. Figure 8-28 shows a Polar plot displayed on a square grid.

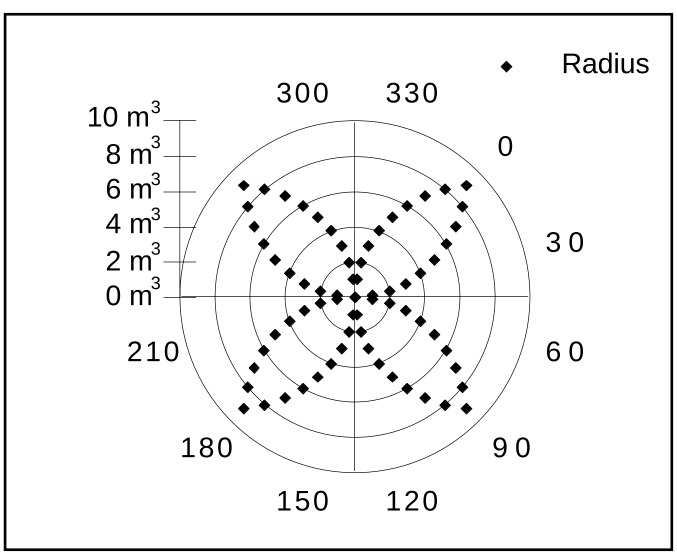


Figure 8-27 Polar plot on circular grid

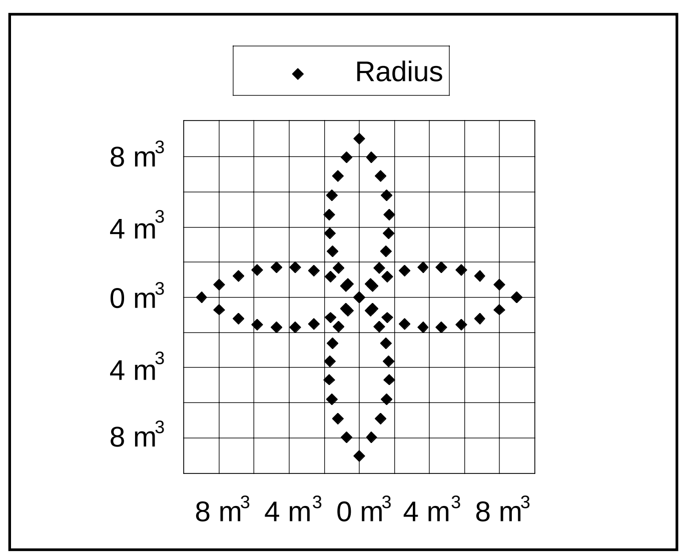


Figure 8-28 Polar plot on square grid

8.17 Customizing Pie Charts

This section explains some of the special features associated with Pie charts. This includes exploding a segment of the chart, removing the frame, and displaying value labels.

8.17.1 Exploding a Pie Chart

A common method of emphasizing one segment of a Pie chart is to drag that segment away from the chart. This is called an exploded Pie chart.

To move a pie segment:

1. Using the Selection Arrow, click twice on the segment to be moved. The first click selects the entire pie. The second click selects the individual segment.
2. Drag the segment away from the rest of the pie. Do not worry about what direction you drag, the segment only moves along its radius. Figure 8-29 shows an example of an exploded Pie chart.

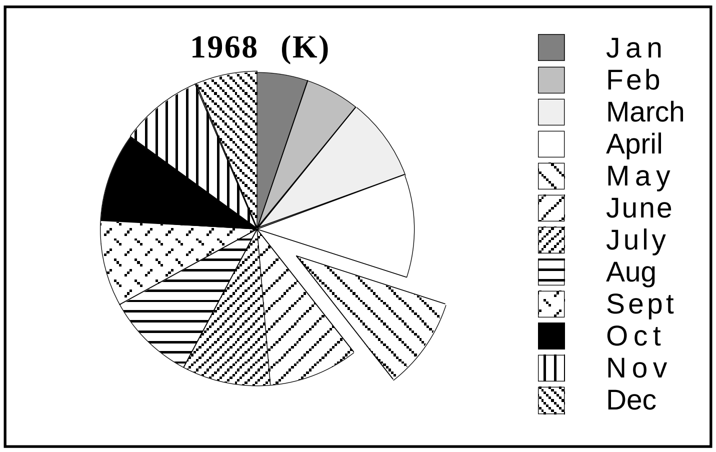
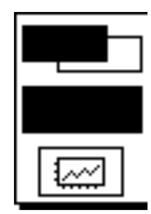


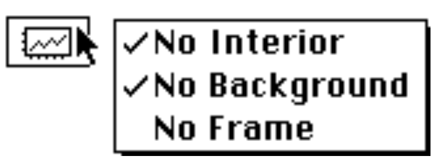
Figure 8-29 Exploded Pie chart

8.17.2 Removing the Frame Surrounding each Slice

There are two methods available to remove the frame that surrounds each piece of the pie. The first is to set the **Frame** color to **None** in the Plot Color dialog (**Format** menu). The second method is to make the change directly from the toolbox.

When the entire Pie is selected, the toolbox displays the icons shown to the right. Click the last icon to display the pop-up menu shown below. Turn on **No Frame** in the menu to remove the frame.





8.17.3 Adding Value Labels

Labels that display the percentage of each slice relative to the entire pie can be added to Pie charts.

To add value labels to a Pie chart:

1. Select the Pie chart which will have the value labels added.
2. Turn on **Add Values** in the **Plot** menu. The percentage values of each pie slice are added to the plot. To display the actual values, select the **Display Pie Values** check box in the Plot Extras dialog (**Format** menu).

Figure 8-30 shows a Pie chart that contains value labels. These labels can be moved individually with the Text tool or as a group with the Selection Arrow. The **Value Label Format** command (**Format** menu) can be used to change the numeric format of the labels.

Note: Any time the plot is redrawn, the value labels are placed back at their original position. If you are displaying only a few of the labels, use the Text tool to add them to the plot. You can place them exactly where you want and do not have to worry about the label moving when the plot is redrawn.

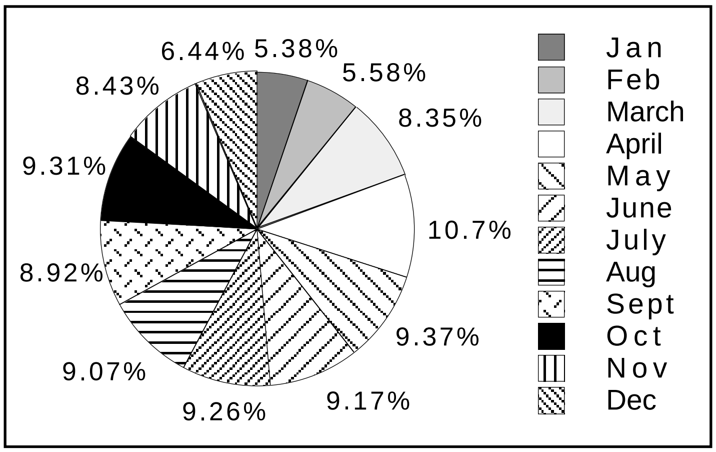


Figure 8-30 Pie chart with value labels

To change the attributes of the value labels:

1. Double-click one of the text labels with the Selection Arrow. The Edit String dialog appears.
2. Make any changes to the label attributes. The options in this dialog are covered in Section 9.7.4. Setting default attributes is covered in Section 9.7.5.
3. Click **OK** to update the value labels.

Working with Plot Tools and Objects

Chapter 9

This chapter covers:

- Using the tools and icons displayed on the toolbox.
- Selecting, moving, resizing, and deleting objects.
- Aligning objects.
- Grouping and ungrouping objects.
- Sending objects up, down, to the front, and to the back.
- Creating and editing text labels.
- Creating and editing lines, arrows, rectangles, ovals, polygons, arcs, and Bezier curves.

9.1 The KaleidaGraph Toolbox

The toolbox, shown in Figure 9-1, is displayed whenever a plot or layout window is active. It contains all of the plot tools, as well as a number of icons that can be used to quickly make changes in the plot or layout window. When switching between plot and layout windows, the toolbox remembers the last tool selected for each window.

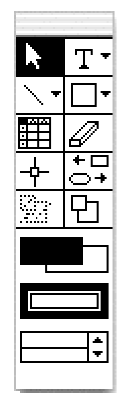


Figure 9-1 Toolbox

9.1.1 Plot Tools

Selection Arrow 

The Selection Arrow is used to select tools, commands, and objects in the plot or layout window. Sections 9.6.1–9.6.3 discuss editing objects with the Selection Arrow. Resizing the plot with this tool is covered in Section 8.11.2.

The Selection Arrow is the default tool in the toolbox. It is automatically selected whenever a new plot is created. Also, most tool selections revert to the Selection Arrow after being used. This tool is used to select, move, and resize objects in the plot or layout window. Once an object is selected, you can copy, duplicate, or delete it.

Text Tool

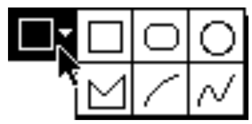
This tool is used to add and modify plot labels. Clicking this tool and holding down the mouse button displays a pop-up menu that can be used to edit selected text labels. Using the Text tool is covered in Section 9.7.

Line/Arrow Tool

This tool is used to draw lines or arrows. Clicking this tool and holding down the mouse button displays a pop-up menu that contains the lines and arrows that can be drawn. Creating lines and arrows is discussed in Section 9.8.1.

Object Tool

Clicking the object tool (to the right of the Line/Arrow tool) displays the pop-up menu shown below. This menu contains six drawing tools that can be used to create various shapes. Creating and editing shapes is covered in Section 9.8.



Box Tool

This tool is used to draw square and rectangular boxes. Creating boxes is discussed in Section 9.8.2.

Rounded Rectangle Tool

This tool is used to draw squares and rectangles with rounded corners. Creating rectangles with round corners is discussed in Section 9.8.2.

Oval Tool

This tool is used to draw ovals and circles. Creating ovals and circles is discussed in Section 9.8.2.

Polygon Tool

This tool is used to draw polygons (multi-sided objects). Creating polygons is discussed in Section 9.8.3.

Arc Tool

This tool is used to create arcs. Creating arcs is discussed in Section 9.8.2.

Bezier Curve Tool

This tool is used to draw Bezier curves. Creating Bezier curves is covered in Section 9.8.4.

Table Tool

This tool is used to create tables of data. The use of this tool is covered in Section 9.2.

Eraser Tool

The Eraser is used to delete labels and objects from the plot or layout window. Grouped items, such as the legend and axis labels, can also be removed using this tool. The Eraser cannot remove any markers or lines that represent plotted data.

Identify Tool

This tool is used to display the coordinates of points on a plot. The use of this tool is discussed in Section 9.3

Align Tool

This tool is used to align a group of selected objects. Aligning objects is covered in Section 9.6.9.

Data Selection Tool

This tool is used to select a portion of the data on the plot for further plotting and analysis. The use of this tool is discussed in Section 9.4.

Zoom Selection Tool

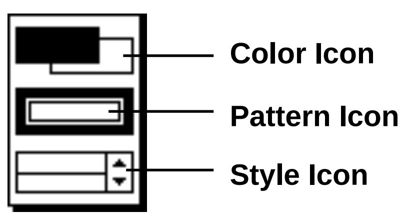
This tool is used to quickly change the axis limits. The use of this tool is covered in Section 9.5.

9.1.2 Toolbox Icons

The bottom portion of the toolbox contains icons that vary according to what is currently selected. Clicking any of the icons displays a pop-up menu that can be used to quickly edit an item’s attributes, without having to choose a command or display a dialog.

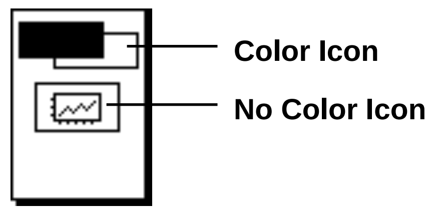
When a Shape or the Legend Is Selected

The Color icon controls the foreground and background color of the selected object. The Pattern icon controls the fill and pen patterns. The Style icon sets the line style and width for selected objects. For more information on using these icons, see Section 9.9.



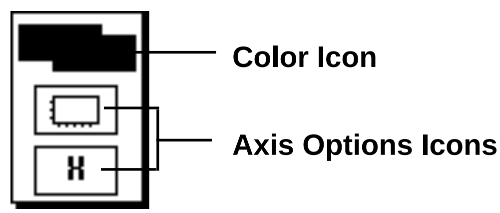
When the Plot Frame Is Selected

The Color icon controls the interior and background colors for the plot. The Transparent icon allows you to leave the interior or background transparent. For more information on using these icons, see Section C.4.



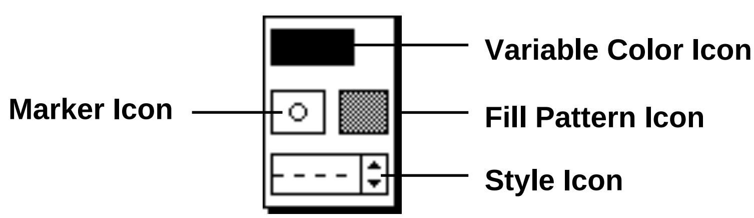
When an Axis Is Selected

The Color icon determines the frame and grid colors. The first of the two Axis Options icons allows you to hide the selected axis or exchange the X and Y axes. The second Axis Options icon controls axis scaling, tick mark display, grid line display, and the format of the axis labels. If the X2 or Y2 axis is selected, you can also link the axes. The interior of the icon displays which axis is currently selected. The Color icon is covered in Section C.4; the Axis Options icons are covered in Section C.2.



When a Marker Is Selected in the Legend

The Variable Color icon controls the color of the variable. The two icons in the middle control the variable markers and fill patterns. The Marker icon determines the marker type, size, and number of markers displayed. The Fill Pattern icon controls the fill pattern. The Style icon determines the variable's line style and width. For more information on using these icons, see Section C.3.



9.2 Using the Table Tool

The Table tool is used to create tables that have up to 16 columns and 64 rows. Clicking in the plot or layout window after selecting this tool displays the dialog in Figure 9-2. Use this dialog to create and edit tables of data within your plots.

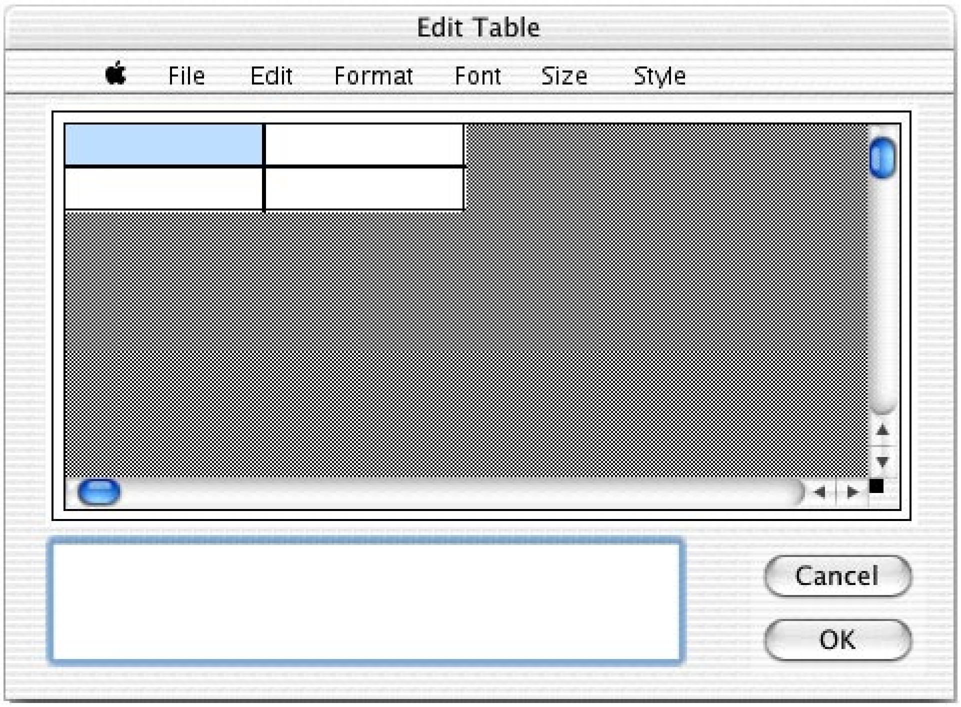


Figure 9-2 Edit Table dialog

9.2.1 Creating and Editing Tables

Tables can be added to any plot or layout window in KaleidaGraph. Tables can be created by either using the Table tool or by pasting text from the Clipboard. Tables are also created to display the results of General and Polynomial curve fits when **Display Equation** is turned on in the **Plot** menu.

To create a table:

1. Select the Table tool from the toolbox.
2. Click anywhere in the plot or layout window to display the Edit Table dialog.
3. Customize the appearance of the table by adding rows, adding columns, or applying style changes, using the commands covered in Section 9.2.2. Then, type data into the cells.
4. Click **OK** to return to the plot or layout window. Figure 9-3 shows a sample table created with the Table tool.

Statistics for the Example #1 data set			
	Test #1	Test #2	Test #3
Minimum	0.5	0.55000001	2.05
Maximum	6.8000002	6.1999998	7.6999998
Sum	87.43	59.07	96.57
Points	25	25	25
Mean	3.4972	2.3628	3.8628
Median	3.5	1.84	3.3400002
RMS	3.9020287	2.8699568	4.192261
Std Deviation	1.7664171	1.6626498	1.6626498
Variance	3.1202294	2.7644043	2.7644043
Std Error	0.35328342	0.33252996	0.33252996
Skewness	-0.013367556	0.84262208	0.84262208
Kurtosis	-1.1877315	-0.402795	-0.40279496

Figure 9-3 Sample table

To edit a table:

1. Double-click the table with the Selection Arrow. The Edit Table dialog appears, displaying the contents of the table.
2. Make the necessary changes to the table, using the commands covered in Section 9.2.2.
3. Click **OK** to return to the plot or layout window.

9.2.2 Table Tool Commands

File Menu

- **New** - This command deletes any existing cells and creates a new table.
- **Open** - This command loads a text file into the table. A tab in the text forces the data into the next cell. A return in the text forces the data into the next row of the table.
- **Set Defaults** - This command saves the current table settings as the default. For information on setting the defaults, see Section 9.2.3.
- **Close** - This command closes the dialog. Clicking **OK** performs the same function.

Edit Menu

- **Cut** - This command deletes the contents of the selected cells and places them on the Clipboard.
- **Copy** - This command copies the selection to the Clipboard, replacing the previous contents, if any
- **Paste** - This command places the Clipboard's contents into the selected cell positions. If necessary, this command adds cells to the table. A tab in the text forces the data following it into the next cell. A return in the text causes the data following it into the next row of the table.
- **Clear** - This command deletes the contents of the selected cells. This command operates similar to **Cut**, except that the contents of the Clipboard are not disturbed.

- **Copy Table** - This command copies the table to the Clipboard as an image, for inclusion in another program.
- **Copy Style** - This command copies the style of the current selection. This allows you to quickly apply a particular style to sections of the table. If necessary, this command adds cells to the table when the style is pasted.
- **Select All** - This command selects the entire contents of the table.
- **Insert Rows** - This command inserts a row above the current selection.
- **Insert Columns** - This command inserts a column to the left of the current selection.
- **Delete Rows** - This command deletes any selected rows. The entire row is deleted using this command.
- **Delete Columns** - This command deletes any selected columns. The entire column is deleted using this command.

Format Menu

- **Left, Right, or Center Justification** - These commands control the alignment of the cell's contents
- **Cell Horizontal Line** - This command sets the line width and color of the line on the bottom of any selected cells.
- **Cell Vertical Line** - This command sets the line width and color of the line on the right side of any selected cells.

  Note: If **None** is selected for the line width, a vertical line is not displayed between adjacent cells. The cells are treated as a single unit. Type the string in the leftmost cell and choose the justification. The contents appear across all cells in the unit. This technique is used to generate labels that span multiple cells.
- **Cell Background** - This command sets a background pattern and color within selected cells. If you want a solid color, choose **100% Fill**.
- **Frame Style** - This command sets the style of the table's frame.
- **Frame Color** - This command sets the color of the table's frame.

- **Options** - This command displays the dialog in Figure 9-4.

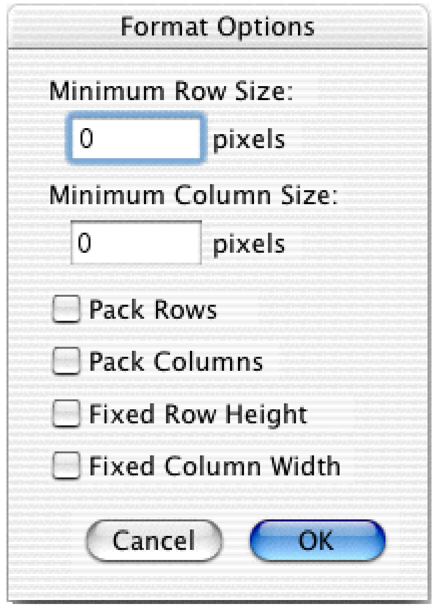


Figure 9-4 Format Options dialog

- **Minimum Row Size**, **Minimum Column Size** - The values entered in these fields determine the minimum row height and column width (one pixel equals 1/72 inch).
- **Pack Rows**, **Pack Columns** - Packing minimizes the white space within cells (horizontally with **Pack Rows** and vertically with **Pack Columns**).
- **Fixed Row Height**, **Fixed Column Width** - These options control how much white space exists within cells. When these check boxes are selected, the row or column sizes are the same throughout the table.

Font Menu

The **Font** menu contains all of the fonts that are currently installed. Choosing a different font from this menu changes the font for the entire cell, even if only a portion of the cell is selected. If you need to use a second font in the cell, use the **Alt. Font** command in the **Style** menu.

Size Menu

This menu contains the sizes that are available for displaying fonts. You can specify a size that is not listed using the **Other** command in this menu. The font size is applied to the entire cell, even if only a portion of the cell is selected.

Style Menu

The **Style** menu is divided into three sections. The commands in the first section control the style of the entire cell, even if only a portion of the cell is selected. The color of the text is controlled using the command in the second section. The commands in the third section allow you to display selected characters in an alternate font, as subscripts, or as superscripts. The **Normal**, **Superscript**, **Subscript**, and **Alt. Font** commands are the only commands in the Edit Table dialog that have an affect on selected portions of a cell.

9.2.3 Setting Defaults for Tables

The Edit Table dialog initially displays four cells. In addition to being part of the table, they are used to set the default layout for generating new tables. They control the layout as follows:

- cell 1 (upper left): This cell controls itself and does not affect the defaults for the rest of the table.
- cell 2 (upper right): This cell controls the default settings for the top row of the table (excluding cell 1).
- cell 3 (lower left): This cell controls the default settings for column one of the table (excluding cell 1).
- cell 4 (lower right): This cell controls the default settings for the body of the table (everything except row one and column one).

Change the attributes (font, size, style, justification, and so on) in each of the four cells. Then, choose **File** > **Set Defaults** to remember the settings. Your default settings are saved as part of the Style file.

9.3 Using the Identify Tool

Use the Identify tool to find the coordinates of any point in the plot window. It can also be used to place the coordinates directly on the plot or the Clipboard.

To use the Identify tool:

1. Select the Identify tool from the toolbox.
2. Click anywhere in the plot. The coordinates are displayed in the upper-left corner of the plot window.
3. To leave the coordinates on the plot, press **Alt** (Windows) or **Option** (Macintosh) when releasing the mouse button. A tag is placed at the current location. The appearance of the X and Y coordinate labels is based on the current settings in the X and Y Label Format dialogs (**Format** menu), with the exception of adding one extra digit of accuracy.
4. To place the coordinates on the Clipboard, press **Shift** when releasing the mouse button. The first time you do this, the Clipboard is cleared and the first set of coordinates is copied. Any additional coordinates are appended to the information already stored on the Clipboard.

9.4 Using the Data Selection Tool

Use the Data Selection tool to select a portion of the displayed data for further plotting and analysis. This is accomplished by enclosing a region of the plot in a polygon.

This tool is particularly useful for eliminating outliers from a plot during a regression analysis. After the Data Selection tool is used, all curve fits are automatically recalculated based on the remaining data. This allows arbitrary data points to be removed from a plot in a very selective, graphical manner.

To enclose the data points in a polygon:

1. Select the Data Selection tool from the toolbox.
2. Position the crosshair at the point where you want to begin forming line segments. Then, click the mouse button to begin. A small square marks the starting point.
3. Move the mouse to the point where you want the first segment of the polygon to end.

   Note: Pressing **Alt** (Windows) or **Option** (Macintosh) when creating the polygon removes the points that lie within it. This feature enables you to select the points you want to remove instead of the points you want to keep.
4. Move the mouse and click again to create the second segment.
5. Continue moving and clicking the mouse until the desired data is enclosed in a polygon. You can complete the polygon by clicking the hollow square, double-clicking the mouse, pressing **Enter** (Windows) or **Return** (Macintosh), or clicking when the crosshair is positioned over the toolbox. Figure 9-5 shows a sample plot that has a region of data selected.

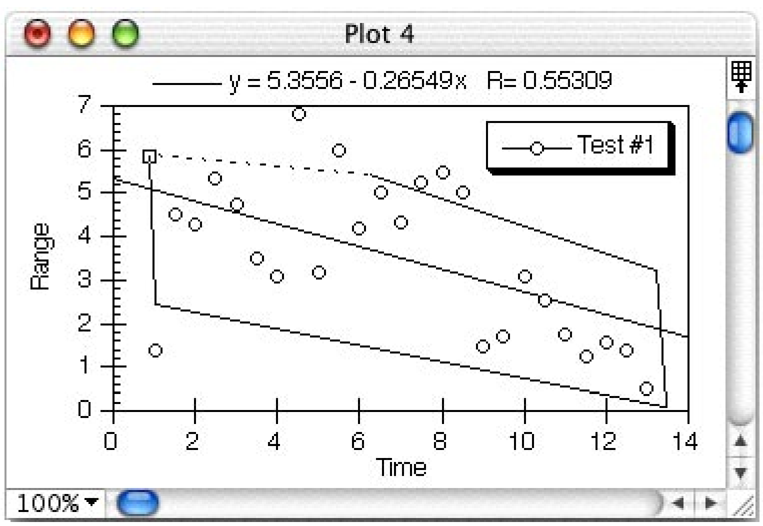


Figure 9-5 Using the Data Selection tool

The plot is then redrawn and only the points that lie within the polygon (or outside when using the **Alt** or **Option** key) are replotted. The points that are removed become masked in the data window. Figure 9-6 shows the plot after being redrawn to only include the points within the polygon.

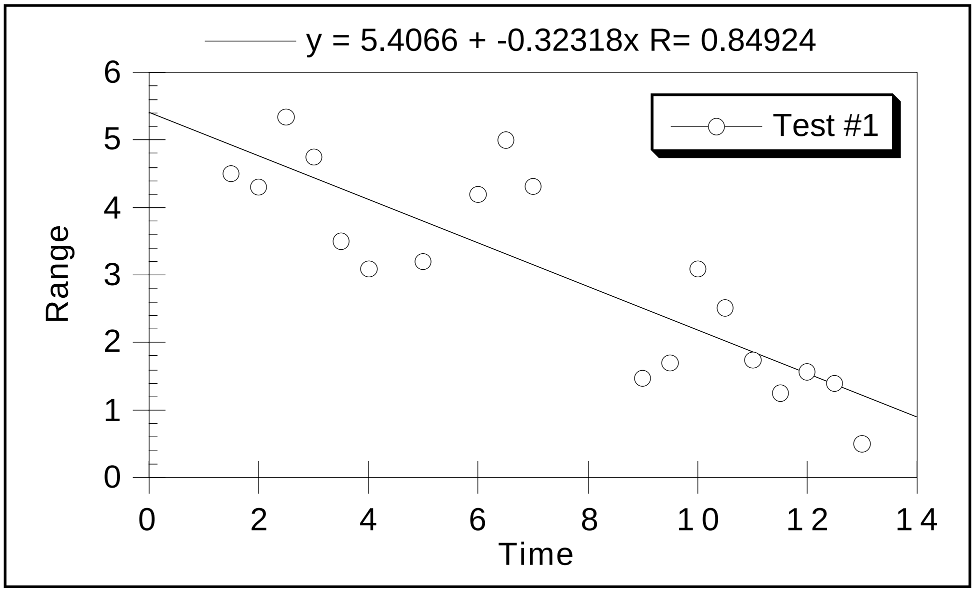


Figure 9-6 Resulting plot after using the Data Selection tool

Note: Any polygon bounding a selection region can have a maximum of 100 bounding points. To return to the original plot and unmask the data, double-click the Data Selection tool.

9.5 Using the Zoom Selection Tool

Use the Zoom Selection tool to quickly change the axis limits. This is accomplished by creating a zoom box, which defines the new limits for the plot. This tool produces an effect similar to specifying axis limits in the Axis Options dialog (**Plot** menu).

To use the Zoom Selection tool:

1. Select the Zoom Selection tool from the toolbox.
2. Drag over a region of the graph. The upper-left and lower-right corners of the box define the new limits for the plot. For Double Y, Double X, and Double XY plots, you can rescale the X2 and Y2 axes by pressing **Ctrl** (Windows) or ⌘ (Macintosh) when creating the zoom box. Figure 9-7 shows a sample plot with new limits being selected.

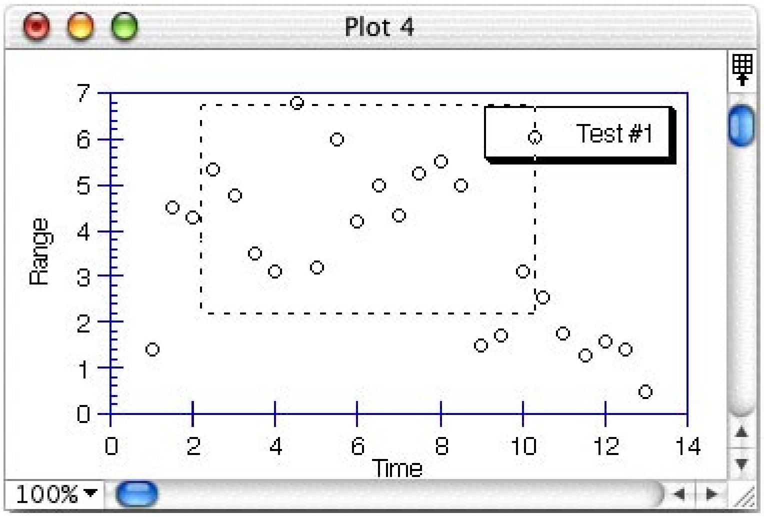


Figure 9-7 Using the Zoom Selection tool

3. Release the mouse button when the box extends to the desired limits. The plot is redrawn to the new set of limits. Figure 9-8 shows the plot after being redrawn using the new axis limits.

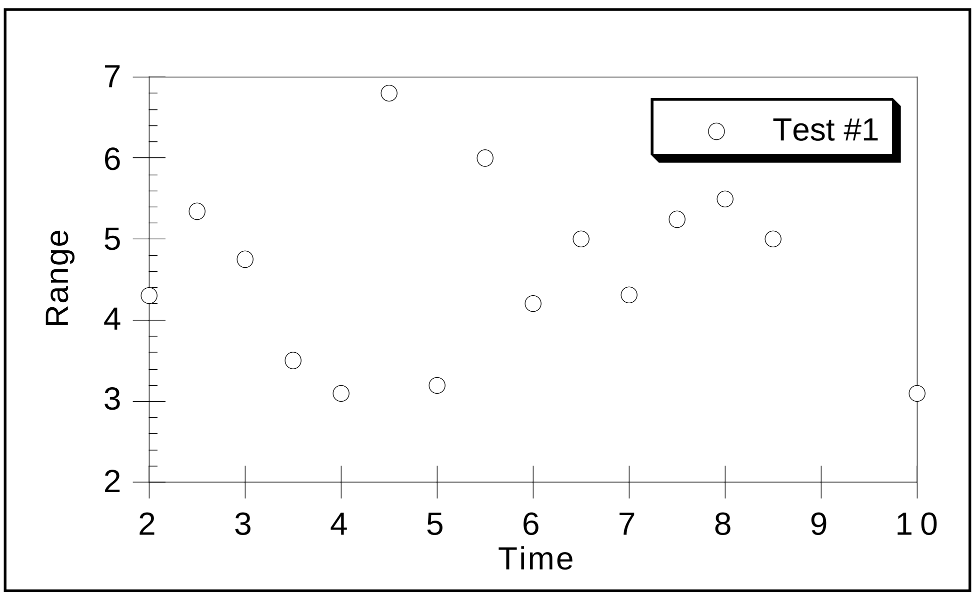


Figure 9-8 Resulting plot after using the Zoom Selection tool

Note: Double-click the Zoom Selection tool to return to the original plot.

When the plot is zoomed, you can adjust the position by pressing **Alt** (Windows) or **Option** (Macintosh) and clicking on an area of the plot. For Double Y, Double X, and Double XY plots, you can adjust the X2 and Y2 axes by pressing **Ctrl**+**Alt** (Windows) or ⌘+**Option** (Macintosh) and clicking. The point you click is placed in the center of the redrawn plot.

9.6 Working with Objects

9.6.1 Selecting and Deselecting Objects

To modify an object, it must first be selected. To select an object, position the pointer over the object and click. Object handles appear around the object.

There are three methods for selecting multiple objects:

- Press **Shift** and use the Selection Arrow to click the objects one at a time. Figure 9-9 shows an example of selecting objects one at a time.
- Use the Selection Arrow to drag a dotted box around the objects to be selected. Figure 9-10 shows objects being selected in this manner.
- Choose **Edit** > **Select All** to select all of the objects.

Note: When multiple objects are selected, changes affect each of the selected objects. For example, if you change the fill pattern, it changes in all of the selected objects.

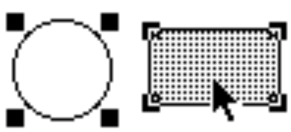


Figure 9-9 Clicking objects individually

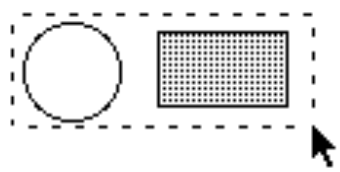


Figure 9-10 Dragging the Selection Arrow

To deselect objects:

- To deselect all selected objects, click a blank portion of the window or press **Esc**.
- To deselect objects one at a time, press **Shift** while clicking the object. The object you click is deselected. All other objects remain selected.

9.6.2 Moving Objects

After objects are created in the plot or layout window, it may be necessary to reposition them. The plot, legend, and any text labels can also be moved using the method described below.

To move objects:

1. Select the object to be moved.
2. Position the pointer over the selected object and drag it to a new position. As you drag, an outline of the object is displayed. If you drag to the edge of the window, the window scrolls.
3. Release the mouse button when the object is in the desired location. Figure 9-11 shows an example of an object being moved.

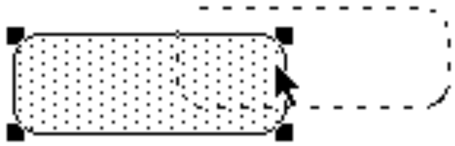


Figure 9-11 Moving an object

Note: You can also move selected objects vertically or horizontally using the arrow keys. If the arrow keys are used, the objects move one pixel in the specified direction (10 pixels if you press **Shift** at the same time).

9.6.3 Resizing Objects

Any object can be resized in the plot or layout window. This includes the plot axes, legend, and any shapes created with the drawing tools.

Note: The legend cannot be resized if it only contains a single item.

To resize an object:

1. Select the object to be resized.
2. Drag any one of the handles until the object reaches the desired size. Figure 9-12 shows an object being resized.

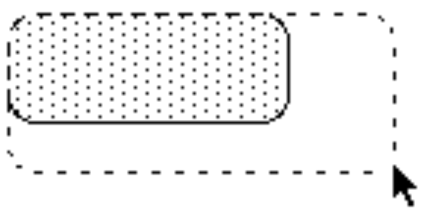


Figure 9-12 Resizing an object

Note: To resize the object proportionally about its center, press **Alt** (Windows) or **Option** (Macintosh) when dragging the handle. By pressing **Shift** as you drag, you can constrain the shape to a square or circle.

9.6.4 Duplicating Objects

To duplicate an object:

1. Select the object to be duplicated.
2. Choose **Edit** > **Duplicate**. A copy of the object is placed slightly below and to the right of the original. Its handles are visible to indicate that it is selected. An example of a duplicated object appears in Figure 9-13.

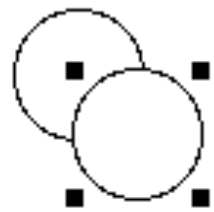


Figure 9-13 Duplicating objects

Note: You can also duplicate an object by pressing **Alt** (Windows) or **Option** (Macintosh) and dragging the object.

9.6.5 Copying Objects

To copy one or more objects:

1. Select the objects to be copied.
2. Choose **Edit** > **Copy Selection**. The objects are copied to the Clipboard, replacing the previous contents, if any.

   Note: If you copy anything associated with the original plot (legend, plot title, axis title, and so on), the entire plot is copied to the Clipboard.

9.6.6 Pasting Objects

To paste objects from the Clipboard:

1. Open the plot or layout window which will receive the object on the Clipboard.
2. Choose **Edit** > **Paste**. The object is placed in the active window.

9.6.7 Deleting Objects

To delete an object in the plot or layout window:

1. Select the object to be deleted.
2. Press **Backspace** (Windows) or **Delete** (Macintosh), or choose **Cut** or **Clear** from the **Edit** menu. **Cut** places the object on the Clipboard; the other methods remove the object from the window.

Note: You can also use the Eraser tool to delete individual objects.

9.6.8 Grouping and Ungrouping Objects

Grouping allows you to combine several objects into a single object. This is useful if you want to move several objects and maintain their relative spacing or if you want to apply the same changes to multiple objects.

To group objects:

1. Select the objects to be grouped. Object handles appear around each object that is selected, as shown in Figure 9-14.
2. Choose **Object** > **Group**. Each object's handles disappear and are replaced by a single set of object handles around the grouped object, as shown in Figure 9-15.

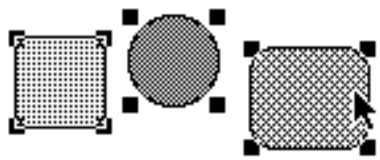


Figure 9-14 Before grouping

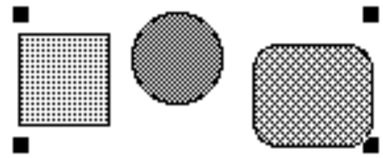


Figure 9-15 After grouping

Once you have grouped objects together, you can no longer select or modify the objects individually. The **UnGroup** command can be used to separate a grouped object into its original components.

To ungroup an object:

1. Select the object to be ungrouped.
2. Choose **Object** > **UnGroup**. Object handles appear for each individual object.

9.6.9 Aligning Objects

KaleidaGraph has two ways to align objects: the Align tool from the toolbox and the **Align** command (**Object** menu). Selecting either displays the pop-up menu in Figure 9-16. The icons in this menu indicate the type of alignment applied to the objects.

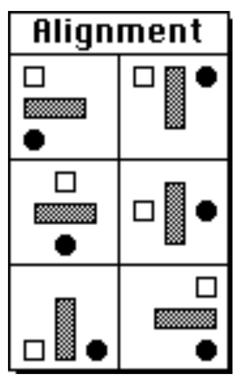


Figure 9-16 Alignment pop-up menu

To align objects:

1. Select the objects to be aligned. Figure 9-17 shows some sample objects that are selected, along with a reference line.
2. Choose **Object** > **Align** or select the Align tool from the toolbox to display the alignment options.
3. Select one of the icons from the menu. The objects are aligned based on your selection. Figure 9-18 shows the objects after aligning them by their left edges.

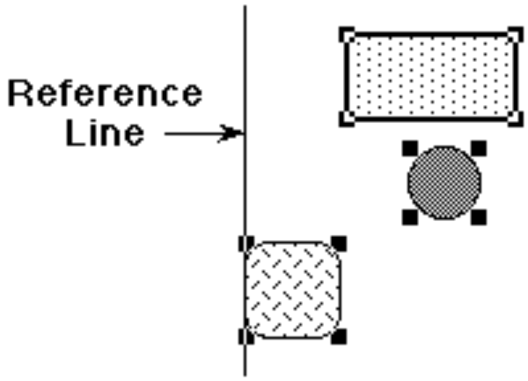


Figure 9-17 Objects before being aligned

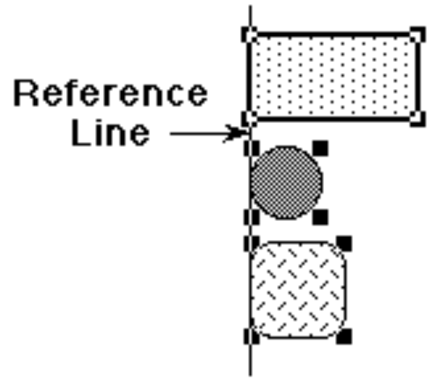


Figure 9-18 Objects after being aligned

9.6.10 Sending Objects Up or Down

Each time you create an object in the plot or layout window, it is placed on top of all other objects in the object stacking order. This order is important because objects near the bottom of the stack may be hidden or difficult to modify.

There are four commands in the **Object** menu that enable you to change the ordering of objects. These commands are: **Up**, **Bring To Front**, **Down**, and **Send To Back**. These commands are useful when objects overlap in a plot or layout.

To change the ordering of an object:

1. Select the object to be modified.
2. Choose the appropriate command from the **Object** menu. The selected object is repositioned in the window.
   - **Up** - Moves the selected object forward one level in the object order.
   - **Bring To Front** - Moves the selected object in front of all other objects.
   - **Down** - Moves the selected object backward one level in the object order.
   - **Send To Back** - Moves the selected object behind all other objects.

Figure 9-19 shows five stacked objects with the middle object (the oval) selected. The four figures below it show how the original objects appear after choosing each of the four commands.

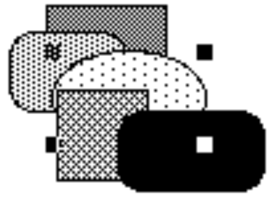
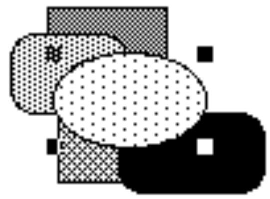
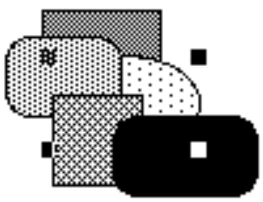


Figure 9-19 Original set of objects

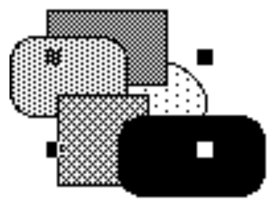
After choosing **Up**



After choosing **Bring To Front**



After choosing **Down**



After choosing **Send To Back**

9.7 Creating and Editing Text Labels

The Text tool is used to create and edit text labels. Clicking within the plot or layout window after selecting this tool displays the dialog in Figure 9-20.

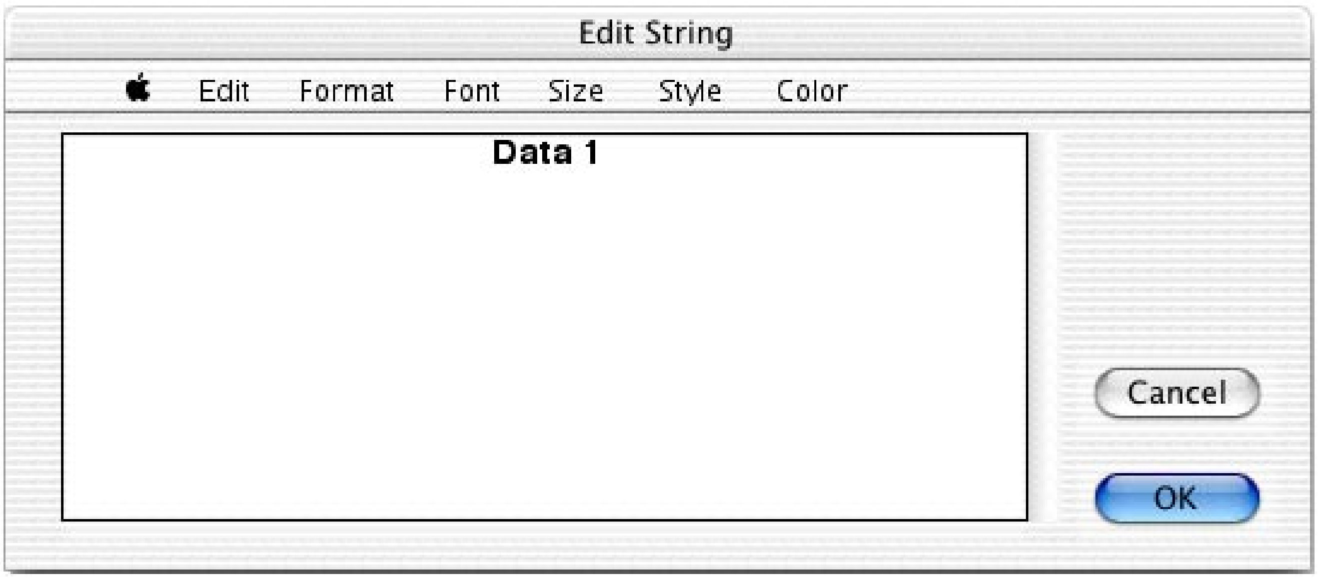


Figure 9-20 Edit String dialog

9.7.1 Creating a Text Label

Text labels can be created anywhere in the plot or layout window. The initial position of the label is not important because once it is created, you can use the Selection Arrow to reposition it.

To create a text label in the plot or layout window:

1. Select the Text tool from the toolbox.
2. Position the text cursor, , anywhere in the window and click once. The Edit String dialog appears
3. Type text and modify the attributes using the menus in the dialog. This dialog supports fully-stylized text, so you can use multiple fonts, styles, and colors in a text label. A maximum of 2000 characters can be entered for a single text label. Figure 9-21 shows the Edit String dialog containing a sample text string.

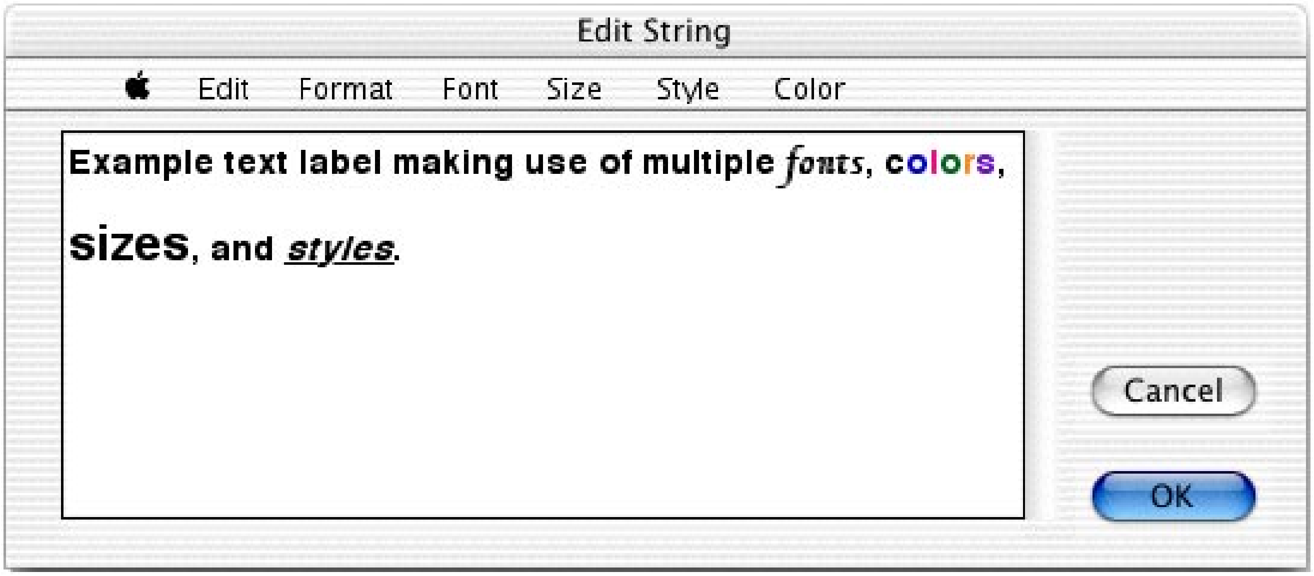


Figure 9-21 Sample string in the Edit String dialog

4. Click **OK** to return to the plot or layout window.

9.7.2 Editing Plot Labels

Legend Text

The legend text can be edited using the Text tool. Double-clicking a text label displays the Edit String dialog. Change the legend text as needed and click **OK** to return to the plot.

It is also possible to move items in the legend to different locations using the Text tool. This is accomplished by dragging a label to a new position with the Text tool.

Plot and Axis Titles

Plot and axis titles can be edited by double-clicking them with either the Text tool or the Selection Arrow. The Edit String dialog appears, containing the current text string. Change the text as needed and click **OK** to return to the plot.

Value and Axis Labels

Value and axis labels are unique because they can be edited individually or as a group.

To edit value or axis labels individually:

1. Select the Text tool from the toolbox.
2. Double-click the label to be modified. The label appears in the Edit String dialog.
3. Change the label text and formatting.
4. Click **OK** to return to the plot.

Note: Be careful when editing value and axis labels in this manner. If the plot is redrawn, the labels revert to their original state.

To edit value or axis labels as a group:

1. Select the Selection Arrow from the toolbox.
2. Double-click one of the labels in the group. The label appears in the Edit String dialog, but the label text cannot be edited because the labels are selected as a group.
3. Make the necessary formatting changes to the label.
4. Click **OK** to return to the plot. The entire group is redrawn to reflect the changes.

Individual Text Labels

Any text label created with the Text tool can be edited by double-clicking it with either the Text tool or the Selection Arrow. The Edit String dialog appears, containing the current text string. Change the text as needed and click **OK** to return to the plot.

9.7.3 Adding Subscripts or Superscripts to a Text Label

To add subscripts or superscripts to a label:

1. Using the Text tool, double-click an existing label or click within the plot or layout window to create a new label.
2. Select the text to be converted.
3. Choose **Subscript** or **Superscript** from the **Style** menu.
4. Click **OK**.

Note: The Edit String dialog does not display characters as subscripts or superscripts. Instead, KaleidaGraph inserts either **\d** (subscript) or **\u** (superscript) before the characters and **\n** (normal) after the characters. These extra characters are only displayed in the Edit String dialog. As soon as you return to the plot, the subscripts or superscripts are displayed correctly.

The Text Options dialog (**Format** menu) contains four settings (**Superscript Offset**, **Subscript Offset**, **Sub/Super Size**, and **Only use built-in sizes**) that affect the appearance of subscripts and superscripts. These settings can be modified to control the offset and size of these characters.

9.7.4 Text Tool Commands

Edit Menu

- **Cut** - This command removes the selection and places it on the Clipboard.
- **Copy** - This command copies the selection to the Clipboard, replacing the previous contents, if any.
- **Paste** - This command places the contents of the Clipboard at the insertion point.
- **Clear** - This command deletes the selection. This command operates similar to **Cut**, except that the contents of the Clipboard are not disturbed.
- **Select All** - This command selects all of the text in the dialog.

Format Menu

- **Left, Right, or Center Justification -** The justification of text labels can be set to either **Left**, **Right**, or **Center**. This setting determines the label placement in relation to where you clicked in the plot or layout window with the Text tool.
- **0, 90, 180, and 270^{o} Rotation -** These commands set the orientation of the text label to one of four positions, moving counter-clockwise from horizontal. You will not see any changes to the text until you return to the plot or layout window.
- **Erase Background -** When this command is turned on, anything behind the text label is erased.

Font Menu

The **Font** menu contains all of the fonts that are currently installed. It is possible to use multiple fonts in the same text label by choosing a different font for selected portions of the label.

Size Menu

This menu contains the sizes that are available for displaying fonts. You can specify a size that is not listed using the **Other** command in this menu. Multiple font sizes can be used in the same text label by choosing a different font size for selected portions of the label.

Style Menu

This menu lists the different styles that can be applied to the text. It also includes commands for displaying characters as subscripts or superscripts.

Color Menu

This menu displays the current color choices available for the text label. Multiple colors can be used in the same text label by choosing a different color for selected portions of the label.

9.7.5 Setting Defaults for Text Labels

You can set defaults for a number of different text labels in the Text Options dialog (**Format** menu). The attributes that you select are saved as part of the Style file. This enables you to set your preferences and have KaleidaGraph remember them the next time you start the program.

To set defaults for text labels:

1. Choose **Format** > **Text Options** to display the dialog in Figure 9-22.

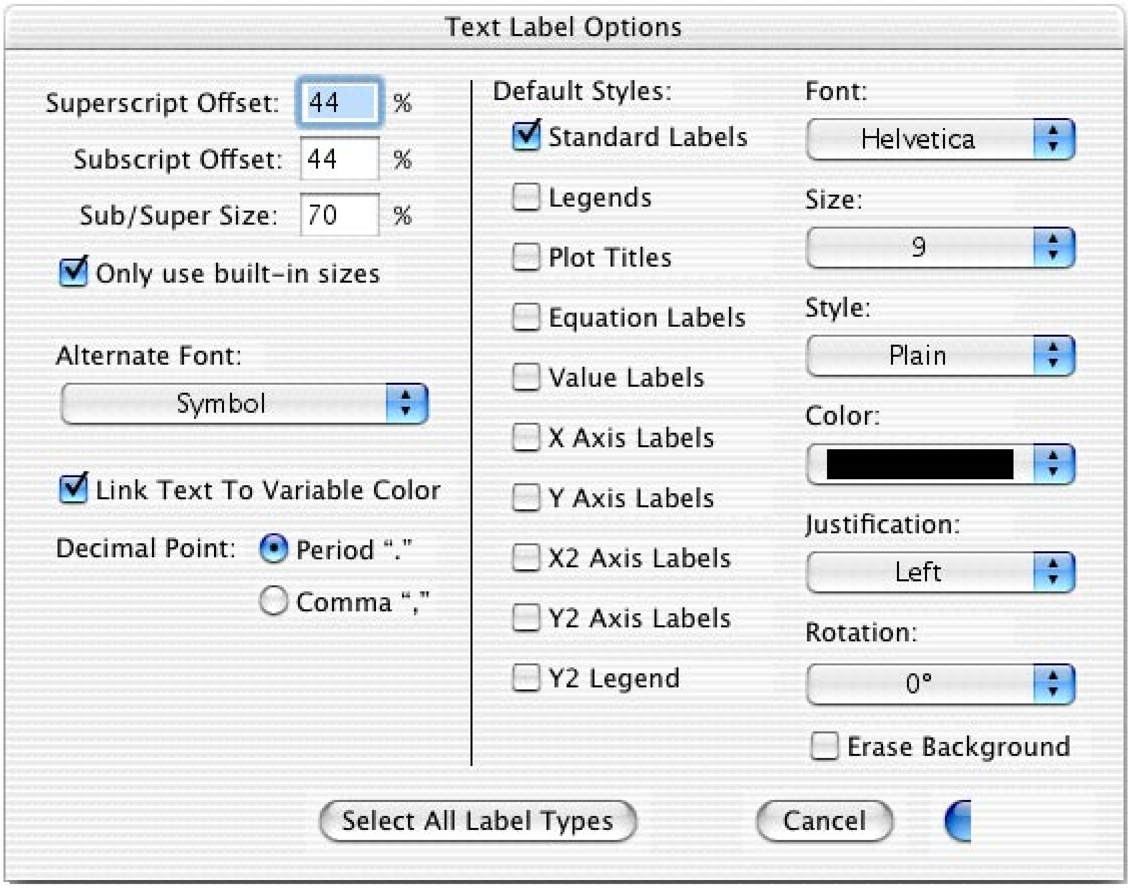


Figure 9-22 Text Options dialog

2. Click the labels for which you want to set new defaults. The different text labels are described below

   - **Standard Labels** - Axis titles and any text created with the Text tool.
   - **Legends** - Any text that appears in the legend.
   - **Plot Titles** - The text representing the plot's title.
   - **Equation Labels** - Text that appears when displaying the equation calculated by a curve fit.
   - **Value Labels** - Text that appears when **Add Values** is turned on or text error bars are applied.
   - **X**, **Y**, **X2**, or **Y2 Axis Labels** - The labels placed at major tick marks to display their value.
   - **Y2 Legend** - Text that appears in the Y2 legend of a Double Y or Double XY plot.

3. Select your new defaults from the pop-up menus.
4. Click **OK** to return to the plot window. Any labels that had their attributes changed are updated.
5. To make these settings permanent, make sure you save changes to the Style when quitting KaleidaGraph.

9.8 Creating and Editing Shapes

You can create lines, arrows, rectangles, rounded rectangles, ovals, polygons, arcs, and Bezier curves in the plot or layout window. Once a shape is created, you can use the icons in the toolbox to change the object's attributes. This topic is covered in Section 9.9.

9.8.1 Lines and Arrows

To create a line or arrow:

1. Click the Line tool on the toolbox.
2. Click the Line tool again and hold down the mouse button to display the pop-up menu in Figure 9-23. Choose the first item in this menu to draw a line or select the type of arrow you want to create.
3. Position the crosshair at the point where you want to begin drawing the line. When creating an arrow, the arrowhead is placed at this position.
4. Drag to draw the line. By pressing **Shift** as you drag, you can constrain the line to rotate in 45^{o} increments.
5. Release the mouse button when the line is the desired length. The line appears with object handles to indicate that it is selected.

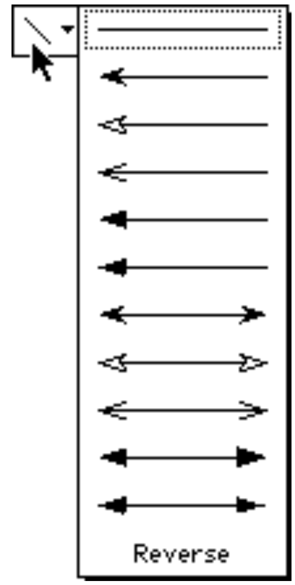


Figure 9-23 Line tool menu

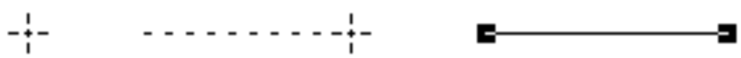


Figure 9-24 Drawing a line

To edit a line or arrow:

1. Select the line or arrow with the Selection Arrow. Object handles appear at the ends of the line.
2. Position the pointer over one of the handles.
3. Drag the handle to adjust the length or angle of the line.
4. You can change the arrowhead type by selecting a different arrow from the pop-up menu. You can switch the ends of the arrow by choosing **Reverse** from the menu.
5. When you are finished, click away from the line to deselect it.

To edit an arrowhead:

1. Double-click the arrow (or click once and choose **Object** > **Edit Object**) to display the dialog in Figure 9-25.

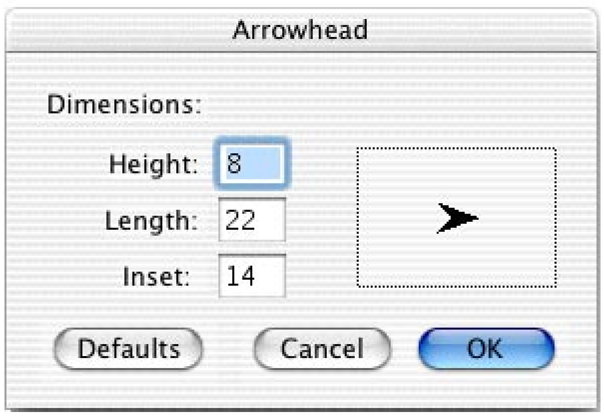


Figure 9-25 Arrow Head dialog

2. Edit the values in the dialog until the arrowhead in the preview window looks the way you want it.
3. Click **OK** to update the arrow.

Note: If you want to modify the default set of arrowheads, use the **Edit Arrow Heads** command (**Object** menu).

9.8.2 Rectangles, Rounded Rectangles, Ovals, and Arcs

To create a rectangle, rounded rectangle, oval, or arc:

1. Select the Box, Rounded Rectangle, Oval, or Arc tool from the toolbox.
2. Position the crosshair at the point where you want to begin drawing.
3. Drag to draw the shape. Depending on the object being created, you can constrain its shape to a square, circle, or quarter-circle by pressing **Shift** as you drag.
4. Release the mouse button when the dotted outline of the object is the desired size. The shape appears with object handles to indicate that it is selected.

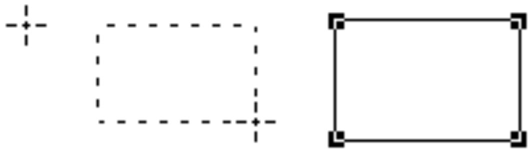


Figure 9-26 Drawing a rectangle

Figure 9-27 Drawing a square

To edit a rectangle, rounded rectangle, oval, or arc:

1. Select the shape with the Selection Arrow. Object handles appear at the corners of the area occupied by the shape.
2. Drag any one of the handles until the object reaches the desired size.
3. When you are finished, click away from the shape to deselect it.

9.8.3 Polygons

The Polygon tool is used to create polygons. Polygons consist of multiple line segments connected at the segment end points. They can either be closed (the first and last segments connect) or open (the first and last segments do not connect).

To create a polygon:

1. Select the Polygon tool from the toolbox.
2. Position the crosshair at the point where you want to begin drawing. Then, click the mouse button to begin drawing. A small square marks the starting point.
3. Move the mouse to where you want the first segment of the polygon to end. By pressing **Shift** as you click, you can constrain the segment to rotate in 45^{o} increments.
4. Move the mouse and click again to create the second segment.
5. Continue moving and clicking the mouse until the desired number of segments are created. You can close the polygon by clicking the hollow square. To leave the polygon open, double-click the mouse, press **Enter** (Windows) or **Return** (Macintosh), or position the crosshair over the toolbox and click

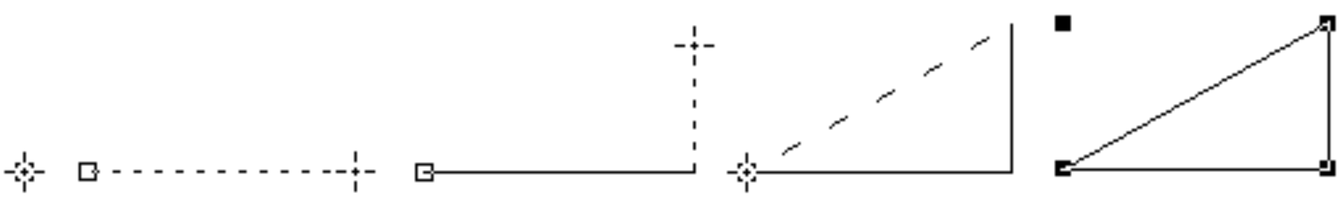


Figure 9-28 Creating a polygon

To edit the segment points of the polygon:

1. Double-click the polygon (or click once and choose **Object** > **Edit Object**). The four object handles disappear and are replaced by handles at each segment point of the polygon.
2. Drag any one of the segment points until the polygon reaches the desired size.
3. When you are finished editing, click outside of the polygon to deselect it.

9.8.4 Bezier Curves

The Bezier curve tool can be used to draw complex objects. Bezier curves are irregularly shaped, multi-sided objects that can have curved line segments. They can either be closed (the first and last segments connect) or open (the first and last segments do not connect).

Each segment in a Bezier curve is defined by four points: two anchor points and two direction points. The anchor points determine where the segments begin and end. The direction points, along with the tangent lines that connect them to the anchor points, determine the size and positioning of the curve segments.

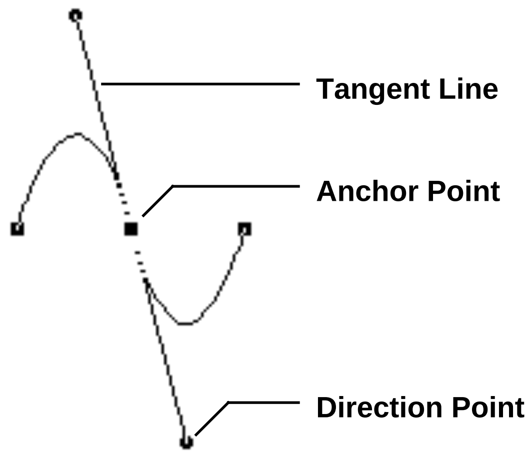


Figure 9-29 Bezier curve definition

Creating a Bezier Curve

To create a Bezier curve:

1. Select the Bezier curve tool from the toolbox.
2. Position the crosshair at the point where you want to begin drawing and click the mouse.
3. Move the mouse until the crosshair is where you want the first segment to end.
4. Click and hold the mouse button down. If you release the button, a straight line appears between the points. If you hold down the button and drag, a tangent line appears that allows you to drag the curve into the position you want.
5. Release the mouse button when the segment is the desired shape. The tangent line disappears and handles appear at the anchor points.
6. Continue creating more segments and setting the angle and shape of the curve with the tangent lines. The segments are automatically connected.
7. You can close the curve by clicking the hollow square. To leave the curve open, double-click the mouse, press **Enter** (Windows) or **Return** (Macintosh), or position the crosshair over the toolbox and click.

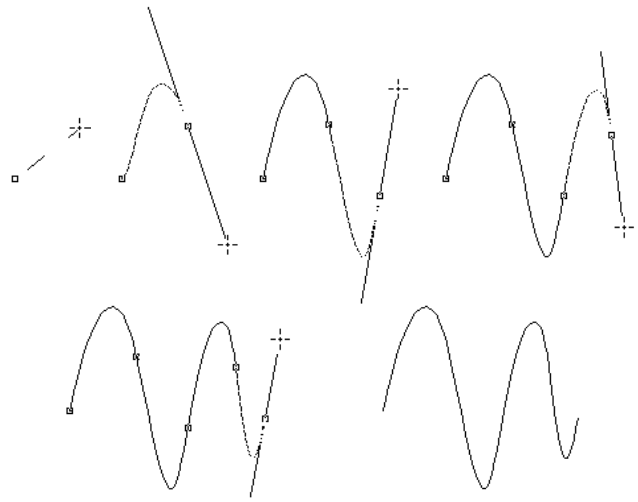


Figure 9-30 Creating a Bezier curve

Editing a Bezier Curve

You can change the position of each anchor point and the shape of each curve. If a segment is straight, there is no tangent line and you can only change its position. Segments can be deleted by selecting an anchor point and pressing **Backspace** (Windows) or **Delete** (Macintosh).

To edit a Bezier curve:

1. Double-click the curve (or click once and choose **Object** > **Edit Object**). The four object handles disappear and are replaced by handles at each anchor point of the curve.
2. Position the arrow over each anchor point you want to move and drag until the point is in the desired position.
3. Release the mouse button. The tangent line is displayed with round handles at each direction point. You can drag these handles to control the size and shape of the curves.
4. Select either of the direction points and drag to edit the shape of the curve. The curve is reshaped in the direction you move the handle.
5. Continue moving anchor points and reshaping curves until the curve is the desired shape.
6. When you are finished editing, click outside of the curve to deselect it.

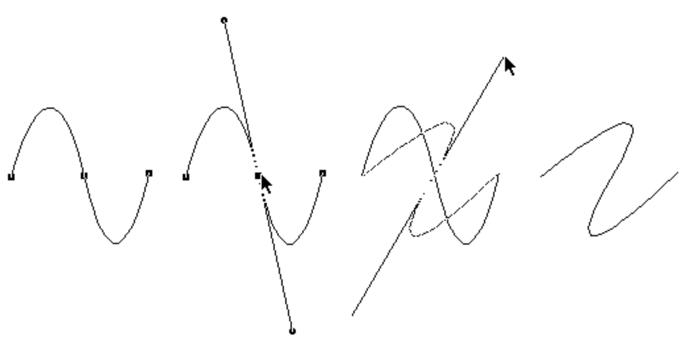


Figure 9-31 Editing a Bezier curve

9.9 Adding Colors and Patterns to Shapes

The area beneath the tools on the toolbox contains several icons that are used to set and modify the attributes of an object. These attributes include: foreground color, background color, fill pattern, pen pattern, line style, and line width. Using these icons, you can control the appearance of any object in the plot or layout window

9.9.1 Setting Default Attributes for Shapes

Attributes for objects drawn in the plot or layout window are controlled using the icons shown in Figure 9-32. If no objects are selected in the window, you can set default attributes. These attributes are applied to any new shapes created in KaleidaGraph.

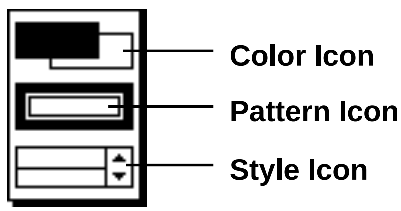


Figure 9-32 Object attribute icons

To set the default attributes:

1. Verify that no items are selected in the plot or layout window.
2. Make selections for the foreground color, background color, fill pattern, pen pattern, line style, and line width. The following sections explain how to make selections for each.

The attributes that you select are saved as part of the Style file. This enables you to set your preferences and have KaleidaGraph remember them the next time you start the program.

9.9.2 Changing Foreground and Background Colors

The Color icon consists of two sections: a foreground section (upper left) for setting the foreground color and a background section (lower right) for setting the background color. When you select a fill or pen pattern, the foreground color is applied to the black portion of the pattern and the background color is applied to the white portion of the pattern.

Note: To edit the colors that appear in the color palettes, use the **Edit Colors** command (**Object** menu).

To change the foreground or background color of an object:

1. Select the object to be edited.
2. Click the foreground or background section of the Color icon. One of the following color palettes appears with the current color choices.

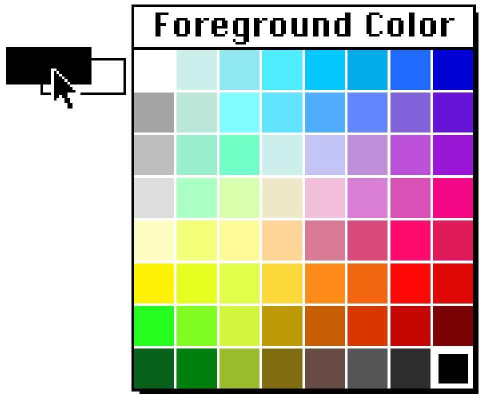


Figure 9-33 Foreground Color palette

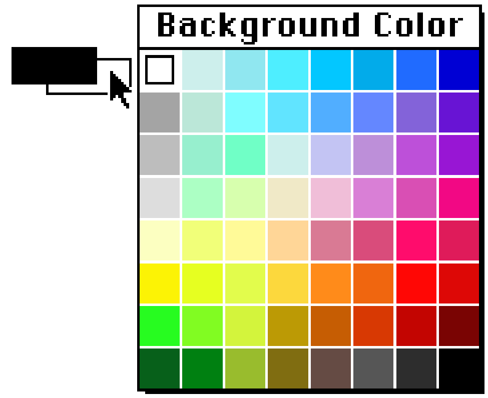


Figure 9-34 Background Color palette

3. Select a different color from the palette.

9.9.3 Changing Fill and Pen Patterns

The fill pattern is used to represent the interior of an object. The pen pattern is used to represent object lines. You can change the fill or pen pattern by selecting a different pattern from the pop-up menu that appears when you click the appropriate portion of the Pattern icon. Black portions of the pattern represent the foreground color and white portions represent the background color.

To change the fill or pen pattern of an object:

1. Select the object to be edited.
2. Click the interior or outer portion of the Pattern icon. One of the following pop-up menus appears with the pattern choices.

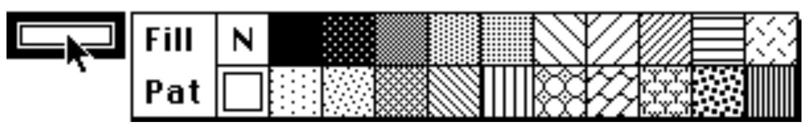


Figure 9-35 Fill pattern choices

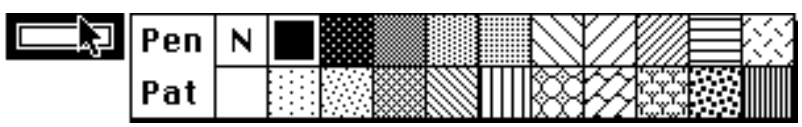


Figure 9-36 Pen pattern choices

3. Select a different pattern from the pop-up menu.

9.9.4 Changing Line Styles and Widths

The Style icon controls the line styles and widths used for objects. The left portion of the icon controls the line style; the right portion controls the line width.

To change the line style or line width of an object:

1. Select the object to be edited.
2. Click the line style or line width section of the Style icon. One of the following pop-up menus appears with the available choices.

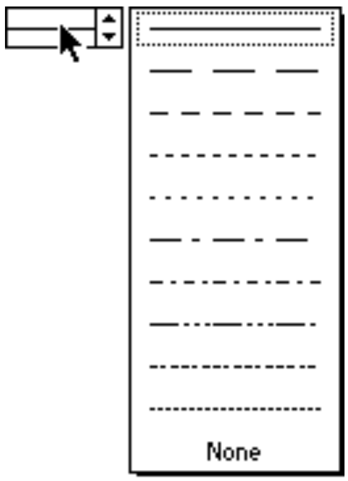


Figure 9-37 Line style menu

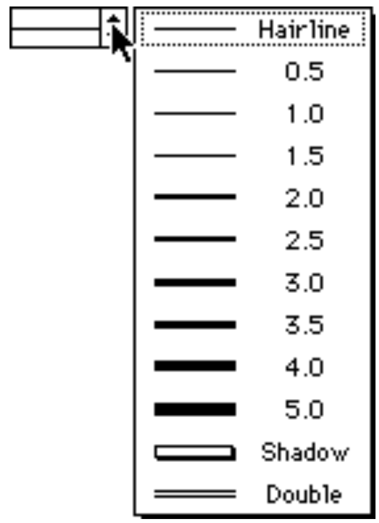


Figure 9-38 Line width menu

3. Select a different line style or thickness from the pop-up menu.

Working with Curve Fits and Error Bars

Chapter 10

This chapter covers:

- The types of curve fits available in KaleidaGraph.
- Applying a Least Squares curve fit.
- Applying a smoothing curve fit.
- Defining and applying a General (user-defined) curve fit.
- Displaying the results of a curve fit.
- Removing a curve fit from a plot.
- Extrapolating curve fits to the axis limits.
- Adding error bars to a plot.

10.1 Types of Curve Fits

This section provides a brief description and lists the data restrictions for each of the curve fits supported by KaleidaGraph. Refer to Section D.5 for reference information for each of the curve fits.

10.1.1 General

This is the most powerful curve fit in KaleidaGraph. It takes a user-defined equation and uses it to fit a set of plotted data. Any function of x or m0, where either x or m0 is the independent variable, can be defined using up to nine parameters (m1–m9).

Examples:

m1 * m0
m1 + m2 * m0
m1 * exp(–m2 * m0)

When a General curve fit is selected, a curve, defined by the equation in the Curve Fit Definition dialog, is fitted to the data using the Levenberg-Marquardt algorithm. There are no data restrictions associated with this curve fit.

Section 2.5.3 provides a guided tour of defining and applying a General curve fit. For more information on the General curve fit, refer to Sections 10.4–10.5. These sections include details about equation structure, initial guesses, and weighting data.

10.1.2 Linear

This function fits a straight line through your data, of the form y = m0 + m1 * x. There are no data restrictions associated with this curve fit.

Note: On a semi-log or log-log plot, this fit appears as a curved line.

10.1.3 Polynomial

This function fits a curve through your data, of the form $y = m0 + m1 * x + m2 * x^2 + m3 * x^3 +...+ m9 * x^9$. The more complex the curvature of the data, the higher the polynomial order required to fit it. There are no data restrictions associated with this curve fit.

10.1.4 Exponential

This function fits a curve through your data, of the form $y = m0 * e^{m1 * x}$. It is generally used to fit data that increases or decreases at a high rate. This curve fit cannot fit negative data or data equal to zero.

Note: On a semi-log plot with a linear X axis, the curve appears as a straight line.

10.1.5 Logarithmic

This function fits a curve through your data, of the form y = m0 + m1 * log(x). A logarithmic curve fit is generally used with data that spans decades (10^0, 10^1, 10^2, and so on). This curve fit cannot be used to fit negative data or data equal to zero.

Note: On a semi-log plot with a linear Y axis, the curve appears as a straight line.

10.1.6 Power

This function fits a curve through your data, of the form $y = m0 * x^{m1}$. This curve fit cannot be used to fit negative data or data equal to zero.

Note: On a log-log plot, the curve appears as a straight line.

10.1.7 Smooth

This function fits a smoothed curve to the data. KaleidaGraph starts by applying a Stineman function to the data. The output of this function then has a geometric weight applied to the current point and ±10% of the data range, to arrive at the smoothed curve.

The only data restriction associated with this curve fit is that the independent (X) variable must be in ascending order in the data window. There is no single expression that represents this curve; therefore, there are no parameters to view or copy.

10.1.8 Weighted

This function fits a curve to the data, using the locally weighted Least Squared error method. The result of this curve fit is to plot a best fit smooth curve through the center of the data. This is an extremely robust fitting technique. Unlike the standard Least Squared error method, this technique is nearly insensitive to outliers. However, this does not come for free. The Weighted curve fit is computationally complex and can take a very long time for a large data set.

The only data restriction associated with this curve fit is that the independent (X) variable must be in ascending order in the data window. There is no single expression that represents this curve; therefore, there are no parameters to view or copy.

10.1.9 Cubic Spline

Moving left to right along the curve in increasing X, every group of four consecutive data points defines a cubic polynomial equation. A series of these cubic polynomials connected together define a cubic spline curve. Using this curve fit is similar to using a French curve to connect all of the data points.

The only data restriction associated with this curve fit is that the independent (X) variable must be in ascending order in the data window. There is no single expression that represents this curve; therefore, there are no parameters to view or copy.

10.1.10 Interpolate

This function fits a curve that passes through the data points and matches the slopes at those points. The advantage of this curve fit is that it does not produce wild results near an abrupt change of slope, as can happen with the Cubic Spline curve fit.

The only data restriction associated with this curve fit is that the independent (X) variable must be in ascending order in the data window. There is no single expression that represents this curve; therefore, there are no parameters to view or copy.

10.2 Applying a Least Squares Fit

Least Squares is a method of curve fitting that has been popular for a long time. Least Squares minimizes the square of the error between the original data and the values predicted by the equation. While this technique may not be the most statistically robust method of fitting a function to a data set, it has the advantage of being relatively simple (in terms of required computing power) and of being well understood.

The major weakness of the Least Squared method is its sensitivity to outliers in the data. If a data point is widely different from the majority of the data, it can skew the results of the regression. For this reason, the data should always be examined for reasonableness before fitting. The Data Selection tool provides a simple method of removing outliers from a plot. The use of this tool is covered in Section 9.4.

The **Curve Fit** menu contains several curve fits that use the Least Squares method. These curve fits are: Linear, Polynomial, Exponential, Logarithmic, and Power. For descriptions and any data restrictions associated with these curve fits, see Sections 10.1.2–10.1.6.

The following steps explain how to apply a Least Squares fit, using the Polynomial curve fit as an example. The procedure is basically the same for applying the other Least Square fits. Section 2.2.4 in the guided tour includes an example of applying a Linear curve fit.

To apply a Polynomial curve fit:

1. Open the plot window which will have the curve fit applied. Figure 10-1 shows a sample plot.

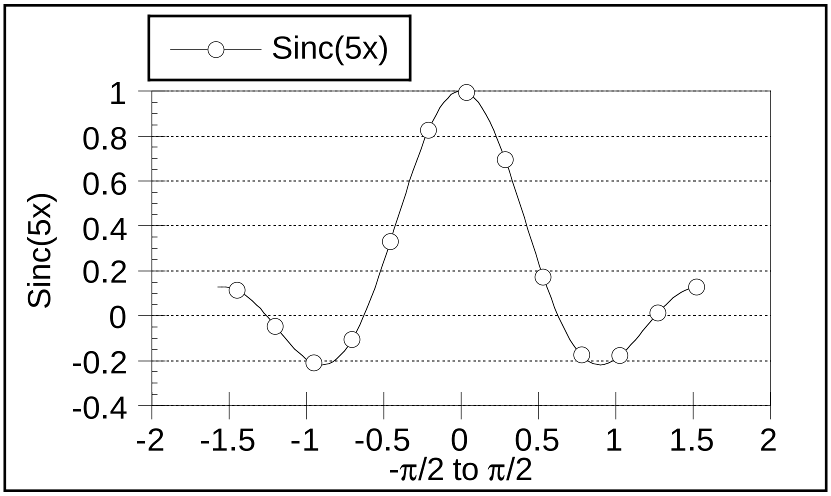


Figure 10-1 Sample plot

2. Choose **Curve Fit** > **Polynomial**. A Curve Fit Selections dialog similar to Figure 10-2 appears. All dependent variables are listed under **Column Names**.

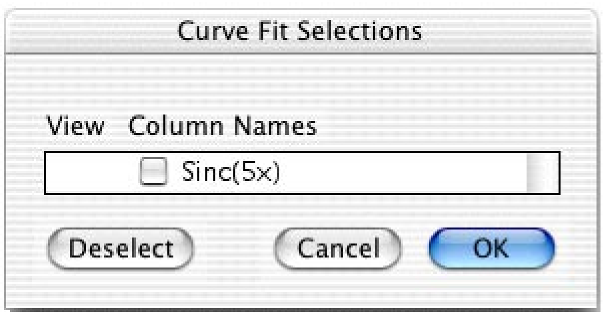


Figure 10-2 Curve Fit Selections dialog

3. Click the check box for each variable to be fit. A keyboard shortcut (**Ctrl**+**A** under Windows or ⌘+**A** on a Macintosh) is available to select all of the variables at once. Each time a variable is selected, a second dialog appears (Figure 10-3). Use this dialog to specify the order of the polynomial to be fit. The maximum value that can be selected is nine.

   Note: This extra dialog is only displayed when using the Polynomial fit.

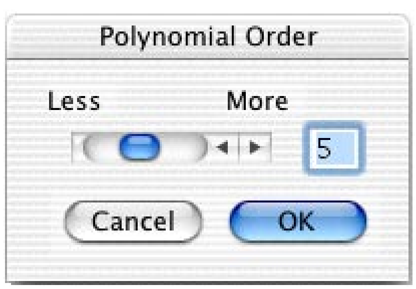


Figure 10-3 Polynomial Order dialog

4. Click **OK** to exit each dialog and apply the curve fit.

   After the curve fit is applied, you can display its equation or extrapolate the fit to the axis limits. These are a few of the features covered in Sections 10.6–10.8. Figure 10-4 shows the sample plot after applying a Polynomial curve fit and displaying the equation.

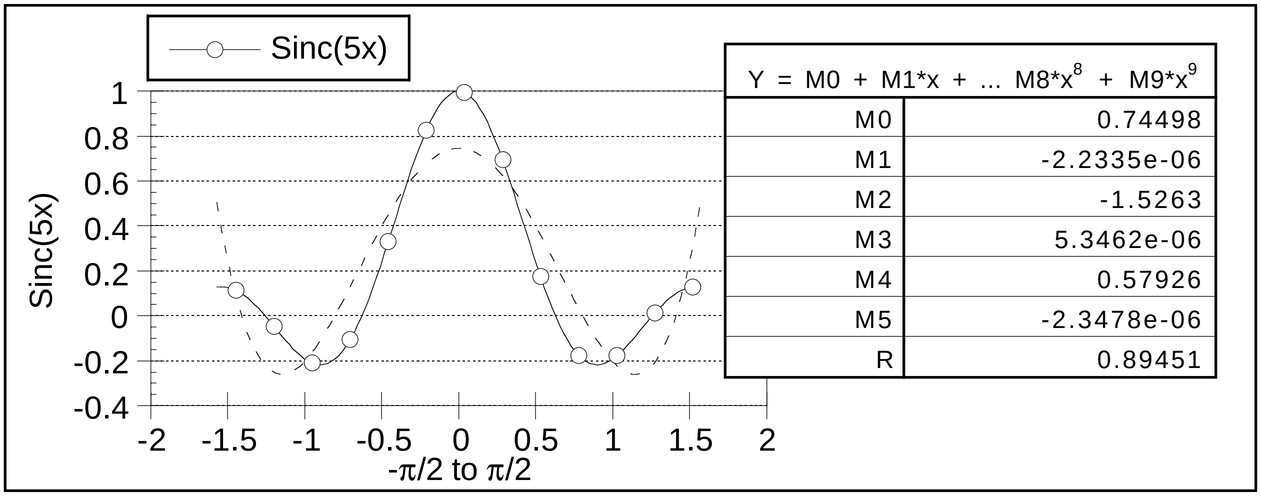


Figure 10-4 Sample plot with Polynomial curve fit applied

10.3 Applying a Smoothing Fit

The **Curve Fit** menu contains four commands designed to fit a smooth curve to the data. These curve fits are: Smooth, Weighted, Cubic Spline, and Interpolate. For descriptions and any data restrictions associated with these curve fits, see Sections 10.1.7–10.1.10.

The following steps explain how to apply a smoothing fit, using the Cubic Spline curve fit as an example. The procedure is basically the same for applying the other smoothing fits.

To apply a Cubic Spline curve fit:

1. Open the plot window which will have the curve fit applied. Figure 10-5 shows a sample plot.

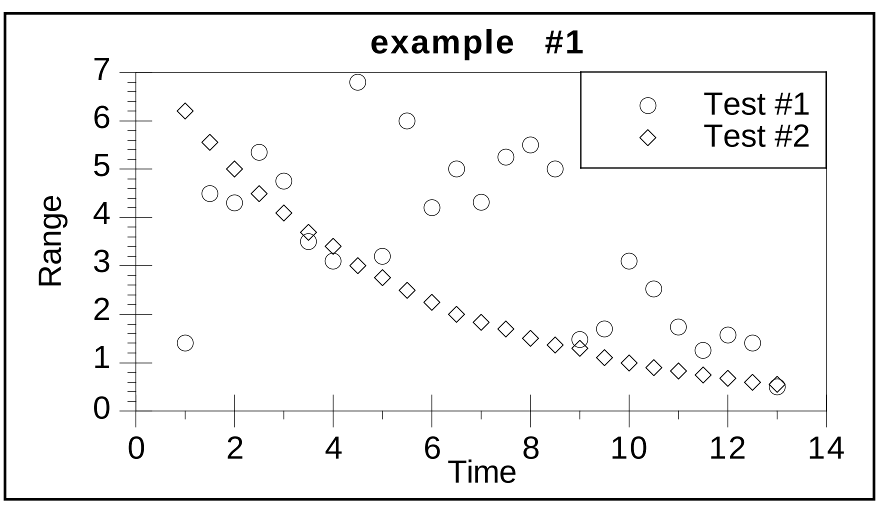


Figure 10-5 Sample plot

2. Choose **Curve Fit** > **Cubic Spline**. A Curve Fit Selections dialog similar to Figure 10-6 appears. All dependent variables are listed under **Column Names**.

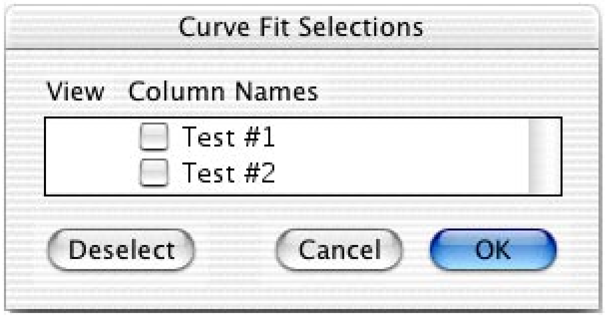


Figure 10-6 Curve Fit Selections dialog

3. Click the check box for each variable to be fit. A keyboard shortcut (**Ctrl**+**A** under Windows or ⌘+**A** on a Macintosh) is available to select all of the variables at once.

Note: When applying a Weighted fit, a second dialog appears each time a variable is selected. Use this dialog to specify the smoothing factor.

4. Click **OK** to apply the curve fit.

After the curve fit is applied, you can increase the number of curve fit points or hide the original data to only display the curve fit. These are a few of the features covered in Sections 10.6–10.8. Figure 10-7 shows the sample plot after applying a Cubic Spline curve fit.

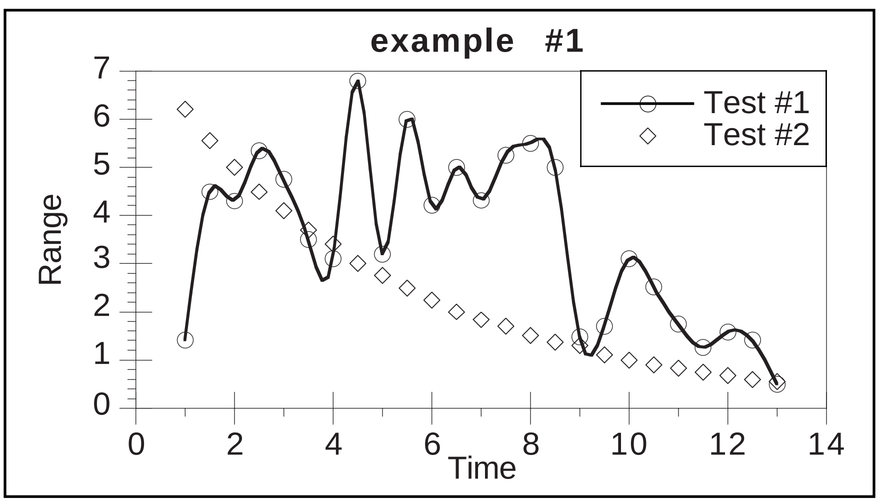


Figure 10-7 Sample plot with Cubic Spline curve fit applied

10.4 General Curve Fit Basics

Nonlinear curve fitting is accommodated in KaleidaGraph through the General curve fit function. The General curve fit can be used to define your own equation and apply a fit to the data, using the Levenberg-Marquardt algorithm. This is the most powerful curve fit in KaleidaGraph because you can specify virtually any equation to be fitted to the data.

This section explains the structure of the curve fit equation, setting initial conditions for the parameters, the Curve Fit Definition dialog, and the use of the commands in the **General** submenu.

10.4.1 Curve Fit Definition

The curve fit definition is the equation that is fitted to the data. Curve fit definitions are composed of the same items as formulas in Formula Entry: operators, parameters, functions, and constants, in addition to an independent variable. When entering the curve fit definition, follow these guidelines:

- The independent variable in an equation is represented by m0.
- The parameters being calculated by the fit are represented by m1–m9. Parameters should be entered wherever you have an unknown quantity and you want KaleidaGraph to calculate the best value.
- Use an asterisk (*) for multiplication and a caret (^) for raising quantities to a power.
- KaleidaGraph does not distinguish between upper and lower case letters (m0 and M0 are the same)
- Constants and other shorthand references can be defined in the library for use in a curve fit definition. Some definitions in the default library are used to substitute x for m0, and a, b, c, and d for m1, m2, m3, and m4, respectively. Section 10.5.7 provides an example of defining a curve fit equation in the library.
- To include a column number as part of the curve fit definition, the **table** function must be used. Section 10.5.5 shows how to use the table function in a General curve fit definition. This method also enables you to fit equations containing multiple independent variables.

The following table lists some equations and how they appear as curve fit definitions in KaleidaGraph.

Original Equations	Curve Fit Definitions
a * x + b	m1 * m0 + m2
a + b * x^c	m1 + m2 * m0^m3
a + b * exp(–c * x)	m1 + m2 * exp(–m3 * m0)
a * exp(–b * x) * cos(c * x + d)	m1 * exp(–m2 * m0) * cos(m3 * m0 + m4)

10.4.2 Initial Conditions

Even though KaleidaGraph calculates values for unknown parameters, you must supply an initial value for each parameter. The algorithm used to calculate the final results requires a reasonable guess to point it in the right direction. KaleidaGraph improves upon these values during each iteration of the fit.

Note: Since the fit starts with initial guesses and improves upon them with each iteration, the final values may be different for a separate set of initial guesses.

Do not get carried away trying to find good guesses; the initial conditions are just that, guesses. If you are not sure what the guesses should be, keep the initial conditions non-zero, and at the correct order of magnitude that you might expect (be it 0.001, 1, or 1000).

The initial conditions are entered immediately following the curve fit definition. Place a semicolon at the end of the definition and begin entering the initial guesses, separating each with a semicolon. The table below shows the definitions from the previous section with some sample initial guesses.

Curve Fit Definitions with Initial Guesses
m1 * m0 + m2; m1 = 0.5; m2 = 23
m1 + m2 * m0^m3; m1 = –2; m2 = 3; m3 = 2
m1 + m2 * exp(–m3 * m0); m1 = 5; m2 = 1; m3 = 0.5
m1 * exp(–m2 * m0) * cos(m3 * m0 + m4); m1 = 4; m2 = 0.7; m3 = 2.2; m4 = 15

10.4.3 Curve Fit Definition Dialog

The Curve Fit Definition dialog is used to enter the equation, initial guesses, and allowable error to be used in the curve fit. You can also choose to specify partial derivatives or to weight the data.

This dialog can be displayed using either of the following methods:

- Choose a curve fit from the **General** submenu (**Curve Fit** menu). In the dialog that is displayed, click **Define**.
- Choose **Curve Fit** > **Edit General**. In the dialog that is displayed, select the name of a curve fit and click **Edit**.

The various features of the Curve Fit Definition dialog are discussed below.

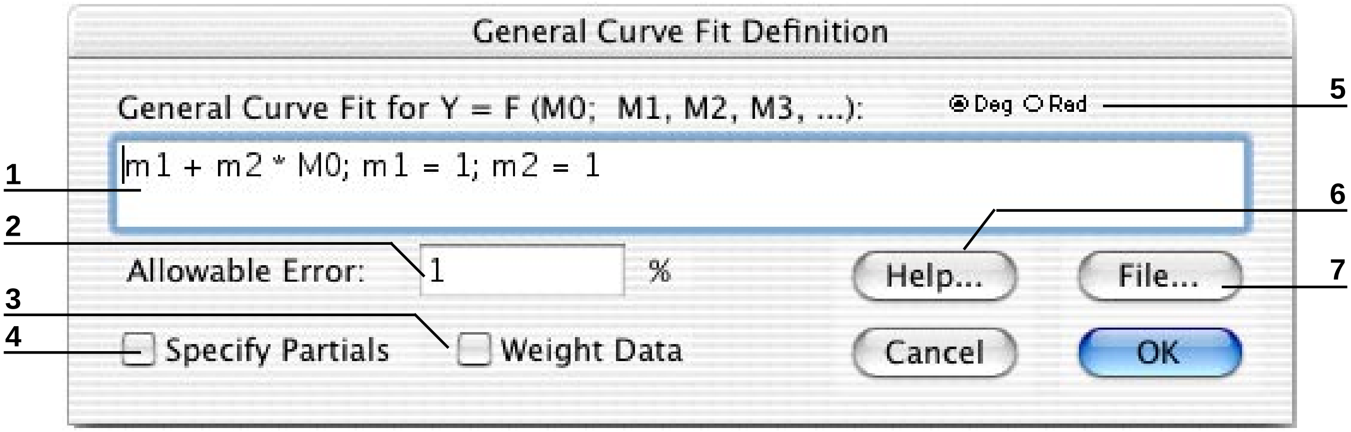


Figure 10-8 Curve Fit Definition dialog

1. **Text Field**
   Curve fit definitions and initial guesses are entered in this area of the dialog.

2. **Allowable Error:**
   The value entered in this field helps KaleidaGraph determine when to stop iterating. Iterations stop if either of the following occurs:

   - Chi Square does not change for a certain number of iterations.
   - The percent change in the normalized Chi Square is less than the Allowable Error.

3. **Weight Data**
   When selected, you can specify a data column that contains weights for the variable you are fitting. These weights should represent the individual errors of the data values. Each weight is used internally as: 1/(weight^2). The smaller the error value, the larger the internal weight. If this check box is not selected, a weight value of 1.0 is used for all data points.

4. **Specify Partials**
   The algorithm that KaleidaGraph uses in calculating each iteration of the General curve fit requires evaluating the partial derivative of the function with respect to each parameter. If **Specify Partials** is not selected, KaleidaGraph numerically approximates the derivative. If this option is selected, the dialog expands, as shown in Figure 10-9.

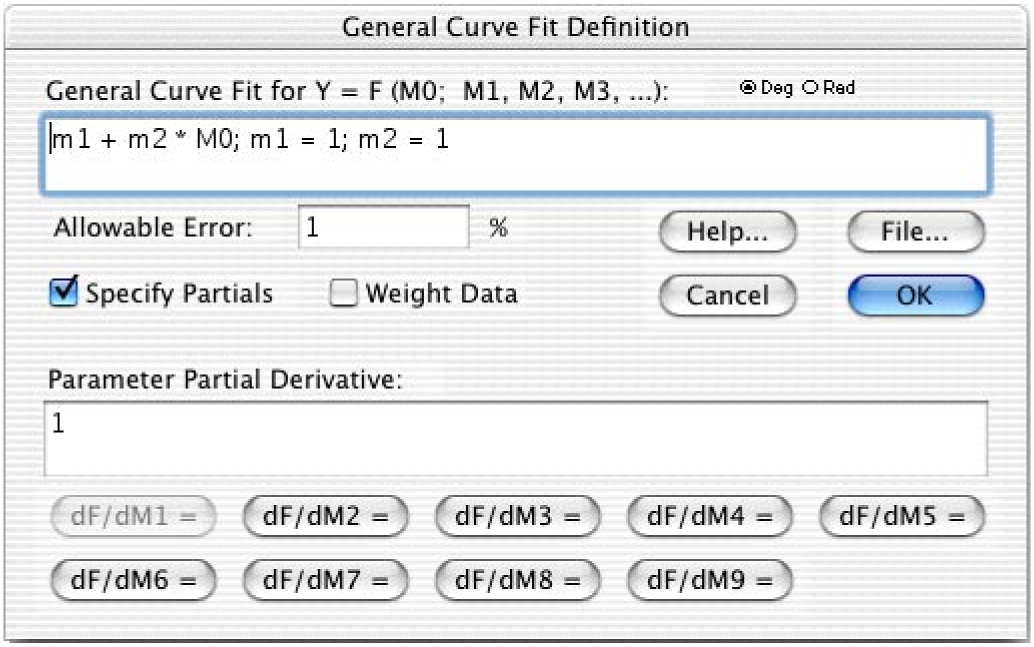


Figure 10-9 Expanded Curve Fit Definition dialog

The nine buttons at the bottom of the dialog allow each partial derivative to be entered in the text field above the buttons. The biggest advantage of specifying the partial derivatives is accuracy. The curve fit algorithm uses these partial derivatives to direct itself where to move after each iteration to find the best solution. In general, sharp deviations in the surface may not be accurately approximated, so the actual partial derivatives are preferred.

You can choose to specify only some of the partial derivatives, but not all of them. It is better to let KaleidaGraph approximate the partial instead of specifying the wrong derivative. If a partial derivative is missing, KaleidaGraph numerically approximates that partial derivative.

5. **Degrees/Radians Buttons**
   These buttons determine whether the results of trigonometric functions are in degrees or radians.

6. **Help...**
   Click this button to display the Help dialog.

7. **File...**
   Clicking this button displays a text editor that can be used to create, open, or save a curve fit definition. The definition appears on the first line and any partial derivatives appear on the lines below it.

10.4.4 Managing the Curve Fit List

KaleidaGraph maintains a master list of curve fits in the **General** submenu. This submenu can store up to 16 user-defined equations that can be applied whenever a new plot is created. The following sections explain how to edit and save changes to the curve fits listed in the General submenu.

Note: The only way to edit the master list is to create a new plot and use the **Edit General** command. The master list is not affected if you do either of the following:

- Use a saved plot (or any plot that had its data archived).
- Choose a curve fit from the **General** submenu and click **Define** to edit the definition.

Adding a Curve Fit to the Master List

To add a curve fit to the master curve fit list:

1. Create a new plot.
2. Choose **Curve Fit** > **General** > **Edit General**.
3. Select a fit in the current list.
4. Click **Add** and a **New Fit** appears below the selected fit in the list. The name of the fit can be changed by selecting it and editing the name in the text field.
5. Click **Edit** to enter the definition in the Curve Fit Definition dialog.
6. Click **OK** when you are finished. Now you can use this fit in the active plot or any new plots that are created. Remember to save changes to the Macros file to make the changes permanent.

Rearranging the Master Curve Fit List

To rearrange the curve fit list:

1. Create a new plot.
2. Choose **Curve Fit** > **General** > **Edit General**.
3. Select the curve fit to be repositioned.
4. Click **Copy** to copy the fit to the trash bin.
5. Click **Remove** to delete the name from the listing.
6. Select a curve fit in the current list.
7. Select the name of the curve fit that was copied to the trash bin.
8. Click **Add** and the curve fit appears below the selected fit in the list.

Note: When rearranging curve fits, do not leave the Edit General dialog until you are completely finished. As soon as you click **OK** or **Cancel** in this dialog, anything in the trash bin is deleted. The only permanent trash bin item is **New Fit**.

Removing a Curve Fit from the Master List

To remove a curve fit from the master list:

1. Create a new plot.
2. Choose **Curve Fit** > **General** > **Edit General**.
3. Select the curve fit to be deleted.
4. Click **Remove** to delete the curve fit from the listing.
5. Click **OK**.

Adding Curve Fits from the Master List to Saved Plots

When a saved plot is opened, the **General** submenu displays the curve fits that were available when the plot was originally created, regardless of what is currently in the master list. To add curve fits from the master curve fit list to a saved plot:

1. Open a saved plot.
2. Choose **Curve Fit** > **General** > **Edit General**. The curve fits stored in the master list appear below the dotted line in the trash bin.
3. Select the name of the curve fit to be added from the trash bin listing.
4. Select a fit in the current list.
5. Click **Add** and the curve fit appears below the selected fit in the list.
6. Click **OK** when you are finished. Now you can use this fit in the saved plot.

Saving Changes to the Curve Fit Menu

The curve fits in the **General** submenu are saved, by default, in a file called **KGMacros** (Windows) or **KG Macros** (Macintosh). This file is loaded each time you start KaleidaGraph.

To save any changes made to the **Curve Fit** menu, it is necessary to save the changes to the default Macros file using one of the following methods:

- Choose **File** > **Preferences** and select **Prompt** from the **Macros** pop-up menu. When exiting KaleidaGraph, a dialog asks whether or not to overwrite the Macros file that was opened at launch.
- Choose **File** > **Preferences** and select **Always** from the **Macros** pop-up menu. When exiting KaleidaGraph, the Macros file that was opened at launch is automatically overwritten.
- Choose **File** > **Export** > **Macros** and overwrite the default Macros file.

Note: Only changes made to a new plot using the **Edit General** command are saved as part of the macros file. Any changes made to a saved plot or by choosing a curve fit from the **General** submenu and clicking **Define** apply solely to that particular plot.

10.5 Using the General Curve Fit

This section explains how to define and apply a General curve fit equation. Other topics include weighting data, limiting parameters, and defining equations in the library. If you are new to using the General curve fit, refer to Section 10.4 which covers some basic concepts, including equation structure, initial conditions, the Curve Fit Definition dialog, and managing the curve fit list in the **General** submenu. You may also want to take the guided tour in Section 2.5.3 to define and apply a General curve fit.

10.5.1 Using the Predefined Curve Fit Definitions

KaleidaGraph includes over 100 curve fit definitions that are divided into several categories (Polynomial, Logarithmic, Sigmoidal, and so on). These curve fit definitions are provided in the following formats for use in KaleidaGraph:

- Each curve fit definition is stored as a text file in the **Curve Fits** folder.
- Each category of definitions is stored as a Macros file that can be imported to load the definitions into the **General** submenu.
- Each category of definitions is stored as a text file for use in the Library.

Importing Definitions from a Text File

The **Curve Fits** folder contains each curve fit definition as a text file. This allows you to import specific curve fit definitions for use in KaleidaGraph.

To load a text file containing a curve fit definition (and have it available for all future plots):

1. Create a new plot.
2. Choose **Curve Fit** > **General** > **Edit General**. The dialog in Figure 10-10 appears.

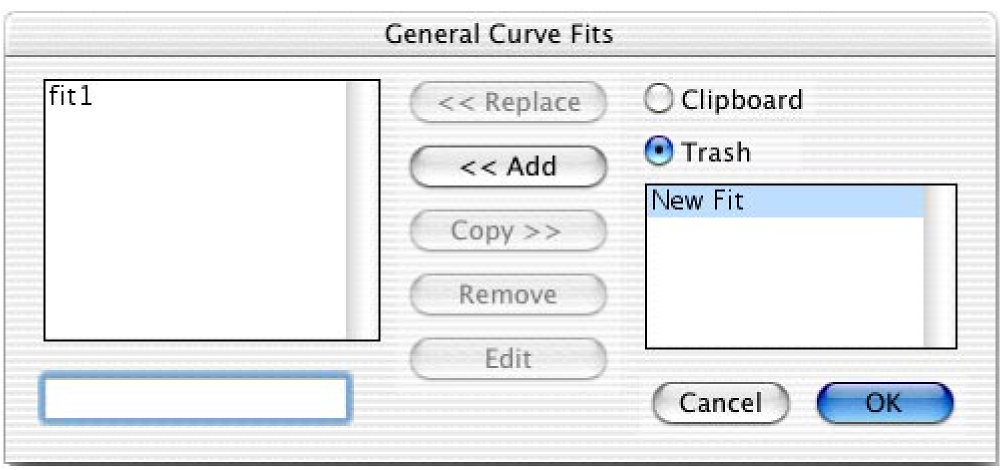


Figure 10-10 Edit General dialog

3. Select a fit in the current list.
4. Click **Add** and a **New Fit** appears below the selected fit in the list. The name of the fit can be changed by selecting it and editing the name in the text field.
5. Click **Edit** to display the Curve Fit Definition dialog.

6. Click **File** in the Curve Fit Definition dialog. A text editor appears.
7. Choose **File** > **Open**.
8. Locate and open a curve fit definition from the **Curve Fits** folder. The definition and initial guesses for the parameters will be displayed in the editor.
9. Choose **File** > **Close** to display the curve fit equation in the Curve Fit Definition dialog. Make sure you change the initial guesses, because all of the initial guesses are set equal to 1.0 by default.
10. Click **OK** until you return to the plot window. Now you can use this fit in the active plot or any new plots that are created. Remember to save changes to the Macros file to make the changes permanent

Importing Definitions from a Macros File

The **Macros** folder contains a Macros file for each category of curve fit definitions. Importing one of these Macros files loads the curve fit definitions into the **General** submenu.

Note: If a plot is already open, you will not see the imported curve fit definitions until a new plot is created

To load the definitions from a Macros file:

1. Choose **File** > **Open**.
2. Locate and open the Macros file of interest from the **Macros** folder.
3. Create a new plot. The curve fit definitions stored in the Macros file will be listed under the **General** submenu.
4. You can now apply one of these fits to your data. Make sure you change the initial guesses, because all of the initial guesses are set equal to 1.0 by default.

Importing Definitions into the Library

The **Library** folder contains text files for each category of curve fit definitions. These text files can be loaded into the library for use in the General curve fit.

To load a text file into the library:

1. Choose **Macros** > **Library** or **Curve Fit** > **General** > **Library**.
2. Choose **File** > **Open**.
3. Locate and open the curve fit category of interest from the Library folder.
4. Browse through the library and make note of the shorthand reference and number of unknowns for the curve fits that you will be using.
5. Choose **File** > **Close**.

You can now use these library definitions to apply a General curve fit. When entering the definition in the Curve Fit Definition dialog, you can enter the shorthand reference defined in the library with the initial guesses, for example **gaussfit(1,5,1,10)**. KaleidaGraph will perform the necessary substitutions before applying the fit.

10.5.2 Entering a Custom Curve Fit Definition

If you need to use a curve fit equation that is not included with the predefined equations, you can easily enter a custom equation and use it in KaleidaGraph.

Note: Custom curve fit equations can also be defined in the library. Refer to Section 10.5.7 for more information on this topic.

To create a custom curve fit definition (and have it available for all future plots):

1. Create a new plot.
2. Choose **Curve Fit** > **General** > **Edit General**.
3. Select a fit in the current list.
4. Click **Add** and a **New Fit** appears below the selected fit in the list. The name of the fit can be changed by selecting it and editing the name in the text field.
5. Click **Edit** to display the Curve Fit Definition dialog.
6. Enter the curve fit equation in the Curve Fit Definition dialog. Make sure you include the initial guesses for each of the unknown parameters. Figure 10-11 shows an example curve fit equation and its initial guesses.

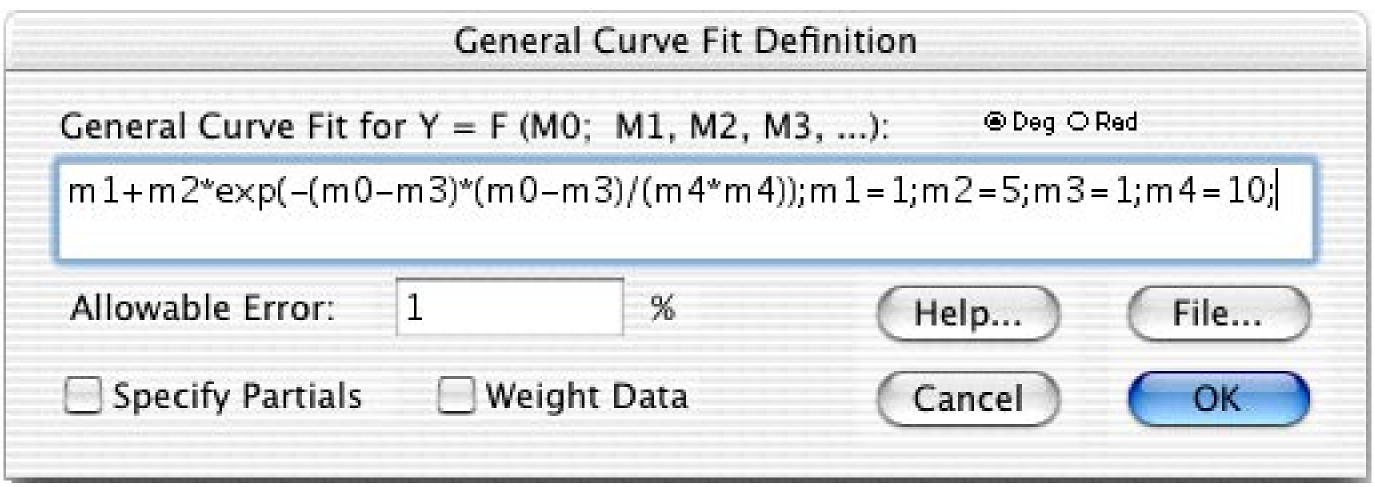


Figure 10-11 Example curve fit equation

7. Click **OK** until you return to the plot window. Now you can use this fit in the active plot or any new plots that are created. Remember to save changes to the Macros file to make the changes permanent

10.5.3 Applying a General Curve Fit

To apply a General curve fit:

1. Open the plot window which will have the curve fit applied. Figure 10-12 shows a sample plot.

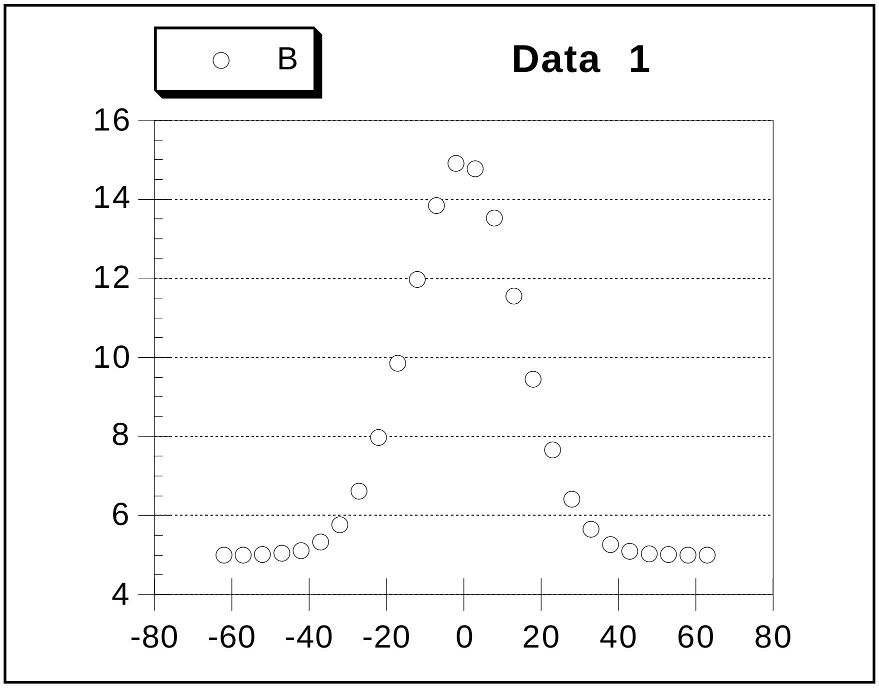


Figure 10-12 Sample plot

2. Choose a curve fit from the **General** submenu (**Curve Fit** menu). A Curve Fit Selections dialog similar to Figure 10-13 appears. All dependent variables are listed under **Column Names**.

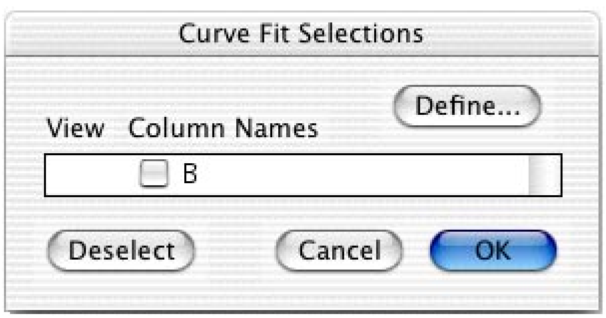


Figure 10-13 Curve Fit Selections dialog

3. If the equation has not been entered, click **Define** to enter the equation or shorthand reference to be applied to the data. Figure 10-14 shows the Curve Fit Definition dialog containing a curve fit definition. When you are finished, click **OK**.

   Note: For information on including this fit in your master curve fit list, refer to Section 10.4.4.

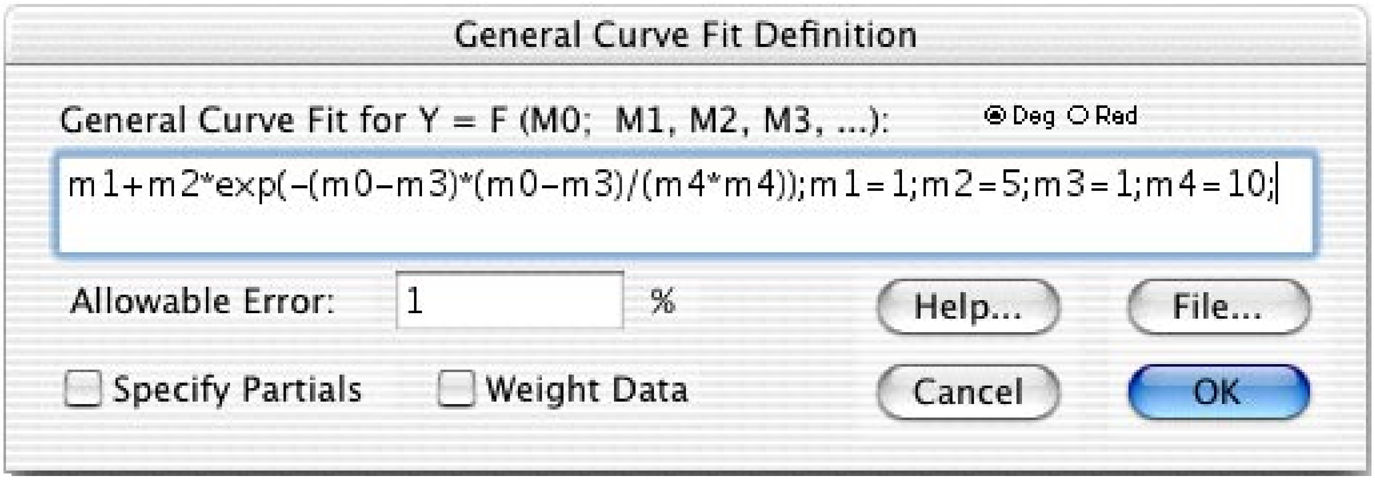


Figure 10-14 Curve Fit Definition dialog

4. Click the check box of each variable to be fit. Typing **Ctrl**+**A** (Windows) or ⌘+**A** (Macintosh) selects all of the variables at once.

5. Click **OK** to apply the curve fit. A status dialog will show the progress of the curve fit. It is updated after each iteration.

Note: Typing **Ctrl**+**Period** (Windows) or ⌘+**Period** (Macintosh) cancels a General curve fit. The current state of the fit is calculated and the resulting curve is drawn.

When the curve fitting process is complete, the curve is drawn on the plot. Once the curve fit is applied, you can display its equation or extrapolate the fit to the axis limits. These are a few of the features covered in Sections 10.6–10.8. Figure 10-15 shows the sample plot after applying a General curve fit and displaying the equation.

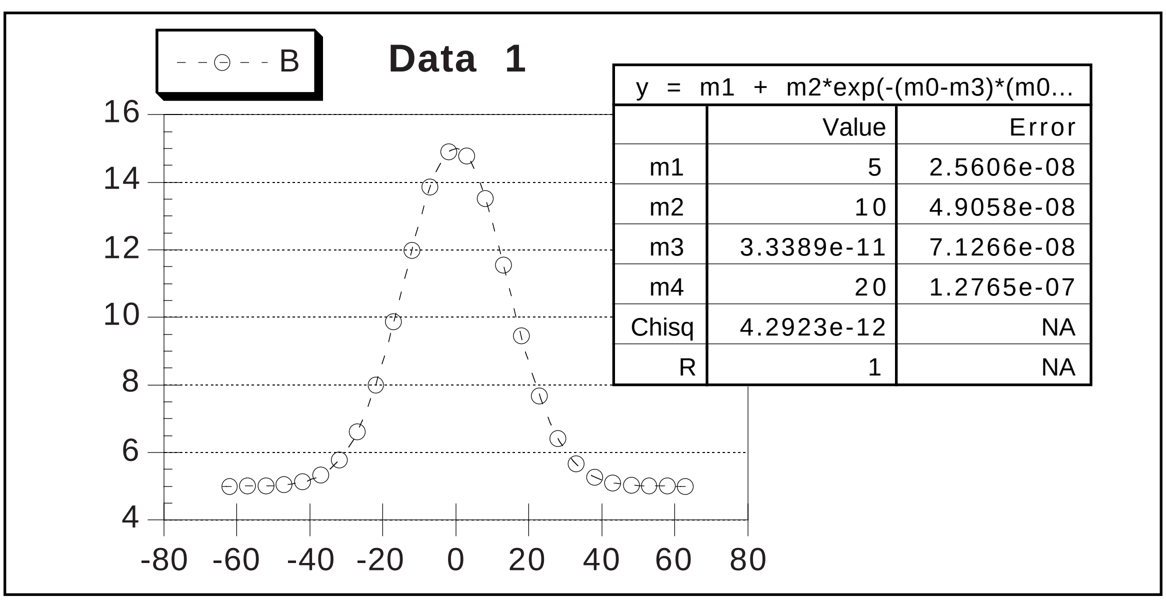


Figure 10-15 Sample plot with General curve fit applied

10.5.4 Weighting Data

The General curve fit provides an option to weight the data. When **Weight Data** is selected in the Curve Fit Definition dialog, the data is weighted during the curve fit. These weights are taken from a specified column in the data window. The weights should represent the individual error values of the fitted data.

Each weight is used internally as: 1/(weight^2). Specifying a weight less than 1.0 for a data point increases the internal weight of that point and can make the total error much larger. Weight values larger than 1.0 reduce the internal weights. When **Weight Data** is not selected, a weight value of 1.0 is used for all data points.

When the Weight Data option is selected and you choose a variable to fit, the dialog in Figure 10-16 appears. Use this dialog to select the column containing the weights for your data.

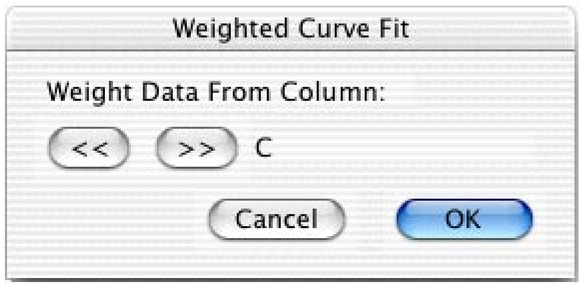


Figure 10-16 Selecting column of weights

10.5.5 Fitting Equations with Multiple Independent Variables

Through the use of the **table** function, KaleidaGraph can fit equations containing multiple independent variables. For example, to fit an equation like **y = a*x + b*x2 +c**, you would enter the following curve fit definition: **a*x + b*table(m0, x col, x2 col) + c**. You need to replace **x col** and **x2 col** with the appropriate column numbers from the data window (for example, table(m0, c0, c2)). The result of using the table function is that for each value of x, the corresponding value from the x2 column is evaluated as part of the curve fit.

10.5.6 Setting Limits for Curve Fit Parameters

Limits on curve fit parameters can be set in one of two ways. The first method is to use conditional testing on the parameter. The second method is to use the **limit** function from the default library.

Suppose that a General curve fit is being defined to fit a straight line to a set of data. The equation is of the form: **a+b*x**. In this example, you want to limit the value of b so that it is not a negative number.

In the Curve Fit Definition dialog, enter either of the following definitions:

```
a + (b=(b<0) ? 0 : b) *x; a=1; b=1     or     a + (b=limit(0, 1000, b))*x; a=1; b=1
```

In the first definition, a conditional test is performed to see if b is less than zero. If true, b is set to zero; if false, b is set equal to itself.

In the second definition, the limit function is used. The limit function requires that a lower and upper boundary be set for the parameter. In this case, zero and 1000 are used to bound the extremes of b.

Whenever a parameter is bound in a General curve fit definition, you must specify partial derivatives. Otherwise, a **Singular Coefficient Matrix error** may occur if the parameter being limited gets pegged at a bounding point for multiple iterations.

10.5.7 Defining Curve Fits in the Library

This section explains how to define two equations in the library and call them from a General curve fit. For information on the syntax used, see Section 6.8.

Creating a Basic General Curve Fit Definition

This example defines the equation **(a*x)/(b+a*x)** in the library.

To add the definition to the library:

1. Choose **Macros** > **Library** or **Curve Fit** > **General** > **Library** to open the library. The text editor is displayed containing the contents of the library.
2. Scroll to the end of the library to include the definition with the other curve fit definitions already present. The definition can be placed anywhere in the library, but it makes it easier to find if all of the definitions are grouped together.
3. Position the cursor on a new line after the last definition and type the following:

```
sample1(a0,b0) = (a*x)/(b+a*x)\; a=a0\; b=b0;
```

4. Once the definition is entered, choose **File** > **Close** to leave the library. The definition is now available for use in the Curve Fit Definition dialog.

Figure 10-17 shows how to call the library definition from within the Curve Fit Definition dialog. Because of the two commands (a=a0 and b=b0) that follow the main definition in the library, initial guesses can be entered directly into the curve fit definition.

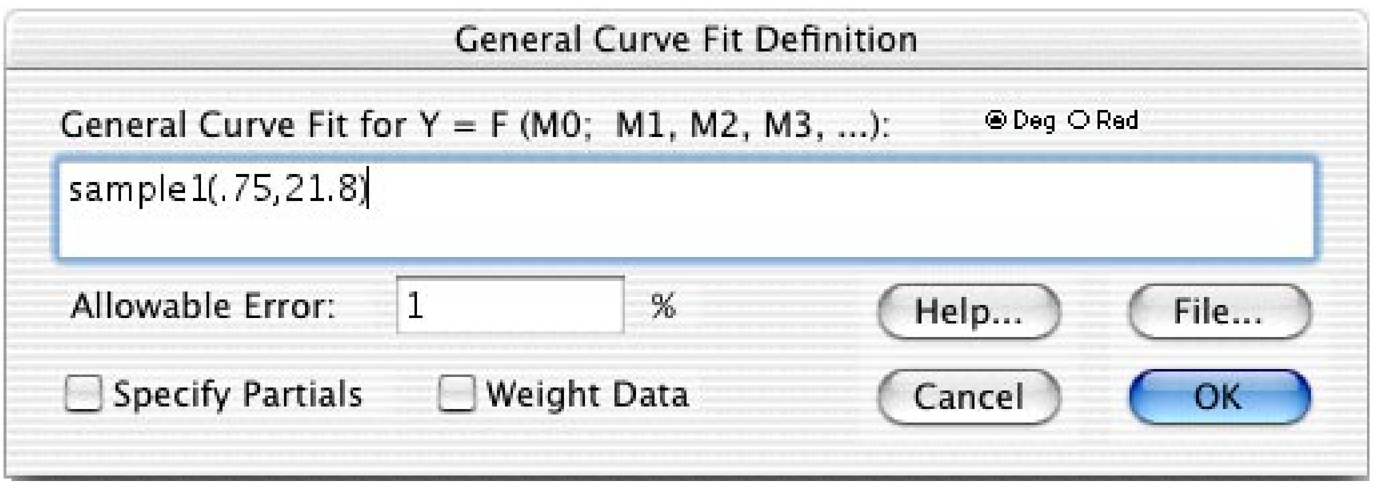


Figure 10-17 Curve Fit Definition dialog

Creating a Complex General Curve Fit Definition

Now that a basic curve fit definition has been defined, let's create a more complex curve fit definition. This example creates a definition for the following equation:

a*exp(–(x–b)^2/(c*c)) + d*exp(–(x–f)^2/(g*g)) + h*exp(–(x–i)^2/(j*j))

To add this definition to the library:

1. Open the library again so that the definition can be added. The following commands should be added to the existing Variable Definitions. These commands are necessary to define a formula that uses all of the parameters available (m1–m9).

```
f=m5;
g=m6;
h=m7;
i=m8;
j=m9;
```

2. Add the following commands to the existing Curve Fit Definitions.

```
form1() = a*exp(–(x–b)^2/(c*c));
;
form2() = d*exp(–(x–f)^2/(g*g));
;
form3() = h*exp(–(x–i)^2/(j*j));
;
sample2(a0, b0, c0, d0, f0, g0, h0, i0, j0) = form1() + form2() + form3()\;
a=a0\; b=b0\; c=c0\; d=d0\; f=f0\; g=g0\; h=h0\; i=i0\; j=j0;
```

Notice the original equation has been split into three smaller parts and all three parts are tied together into one formula. This makes it possible to use curve fit definitions that are longer than the 256 character limit in the Curve Fit Definition dialog.

Figure 10-18 shows how to call the library definition from within the Curve Fit Definition dialog. The name of the definition is entered, followed by all of the initial guesses.

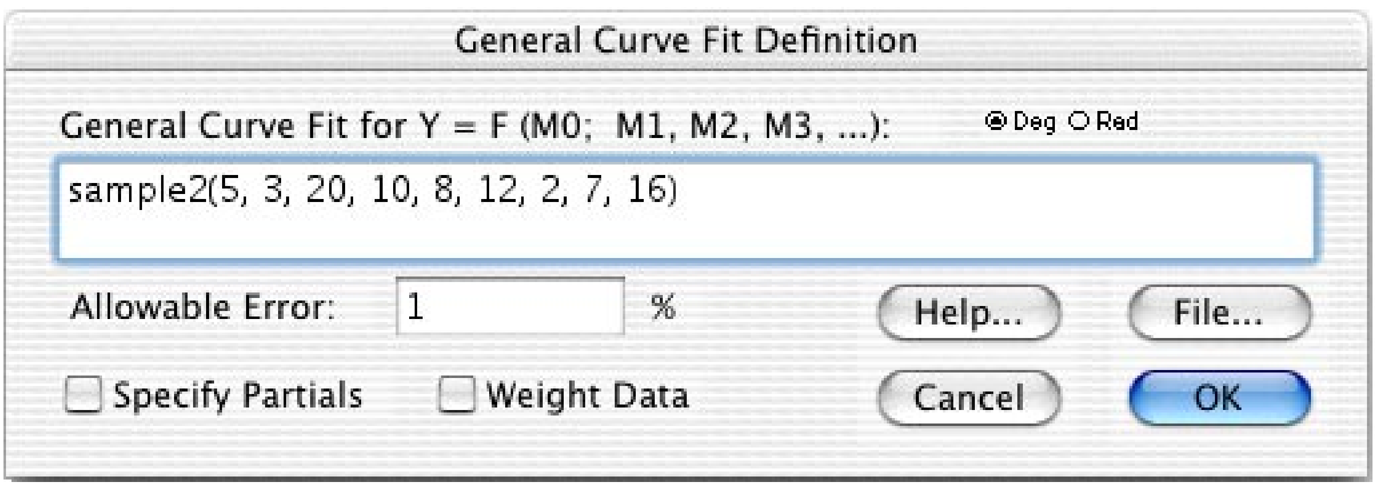


Figure 10-18 Curve Fit Definition dialog

10.5.8 The General Curve Fit and the Macro Calculator

Calculations for the General curve fit are performed in the Macro Calculator. After performing a General curve fit, the memory registers in the Macro Calculator store information about the fit. The memory registers and their contents are listed below:

m0	R value
m1–m9	The values of the parameters.
m22	Chi Square value
m23	% error term
m24	Chi Square of the last iteration.
m25	Number of iterations completed.
m26	Number of parameters in the fit.

Note: Almost all memory registers, alpha registers, and program space are overwritten during the execution of a General curve fit.

10.6 Working with Curve Fit Results

This section explains how to view the results of a curve fit, interpret the values that are reported by KaleidaGraph, and export the curve fit results for further analysis.

10.6.1 Viewing Curve Fit Results

KaleidaGraph provides two methods to view the results of a curve fit. You can either display the equation on the plot or view the coefficients in a dialog.

Displaying the Curve Fit Equation

The curve fit equation can be displayed by turning on **Display Equation** in the **Plot** menu. The results from any General, Linear, Polynomial, Exponential, Logarithmic, or Power curve fits are displayed in the plot window. Not only is the resulting equation displayed, but if any updates occur to the curve fit, the equation updates automatically.

Depending on the type of curve fit, the equation is displayed in a table or as a text label. The General and Polynomial curve fits display equations in a table because the equations for these fits can be much longer than the others. The appearance of the equation can be edited by double-clicking it with the Selection Arrow.

The numeric format of the equation can be controlled using the **Equation Label Format** command (**Format** menu). Some of the options available include number format, number of significant digits, and trailing zeros

Viewing Coefficients

The other method of displaying curve fit results is to view the curve fit's coefficients in a dialog.

To view the coefficients from the curve fit:

1. Reselect the curve fit from the **Curve Fit** menu. A Curve Fit Selections dialog similar to Figure 10-19 appears with a drop-down arrow under **View** for each variable that has a curve fit applied.

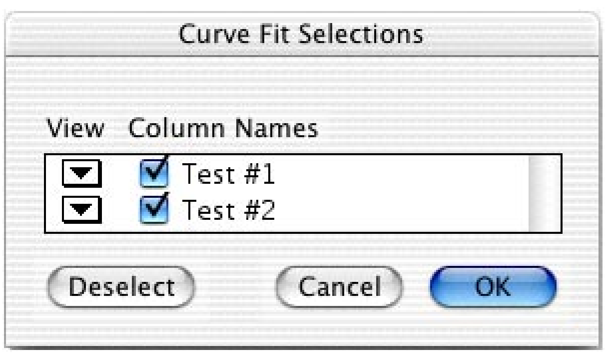


Figure 10-19 Curve Fit Selections dialog

2. Click one of the drop-down arrows and choose **View Results** from the pop-up menu that appears. A dialog similar to Figure 10-20 will be displayed containing the coefficients from the curve fit.

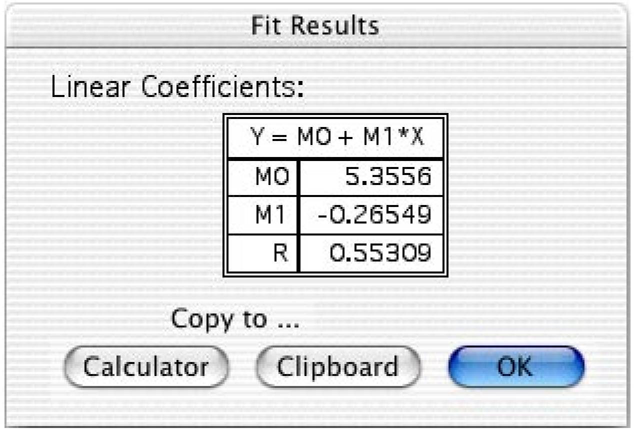


Figure 10-20 Viewing coefficients

3. To copy the coefficients to the Clipboard, click **Clipboard**. This enables you to paste the results into another window or program.
4. To copy the coefficients to the Calculator, click **Calculator**. This enables you to use the results in a Formula Entry function.
5. Click **OK** until you return to the plot window.

10.6.2 Interpreting Curve Fit Results

This section explains some of the values that are calculated when using the General, Linear, Polynomial, Exponential, Logarithmic, and Power curve fits. The correlation coefficient is calculated for each of these fits, but only the General curve fit displays the parameter errors and Chi Square value.

Correlation Coefficients

A correlation coefficient (R or R^2) is displayed with the curve fit's equation and coefficients. The correlation coefficient indicates how well the calculated curve fits the original data.

This value ranges from zero to one. The closer to one, the better the fit. The equation used to calculate the correlation coefficient is listed in Section D.6.

You can choose to display the Linear Correlation Coefficient (R) or the Coefficient of Determination (R^2) as part of the results. The **Display R^2 instead of R** check box in the Curve Fit Options dialog (**Format** menu) controls which is displayed.

Parameter Errors

The General curve fit displays a column of errors to the right of the parameter values. The errors represent the standard error values of the parameters. These values should be read as: parameter value ± error.

Chi Square Value

The General curve fit reports a Chi Square value as part of the results. This value is the sum of the squared error between the original data and the calculated curve fit. In general, the lower the Chi Square value, the better the fit. The equation used to calculate Chi Square is listed in Section D.6.

10.6.3 Exporting Curve Fit Results

Once a curve fit has been applied, the results of the fit can be exported for further analysis. The export options include copying the parameter values to the Clipboard or the Calculator, copying the values of the curve fit line into the data window, and copying the residual values into the data window.

Copying the Parameters to the Clipboard or Calculator

To copy the parameter values to the Clipboard or Calculator:

1. Reselect the curve fit from the **Curve Fit** menu. A Curve Fit Selections dialog similar to Figure 10-21 appears with a drop-down arrow under **View** for each variable that has a curve fit applied.

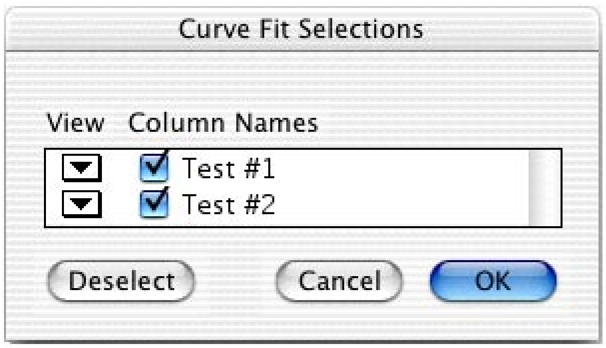


Figure 10-21 Curve Fit Selections dialog

2. To copy the coefficients to the Clipboard, click one of the drop-down arrows and choose **Copy Parameters to Clipboard** from the pop-up menu that appears. This enables you to paste the results into another window or program.

   Note: If you paste the results into the plot window, the coefficients in the table do not change if the curve fit is updated. For this reason, we recommend that you use the **Display Equation** command instead.

3. To copy the coefficients to the Calculator, click one of the drop-down arrows and choose **Copy Parameters to Calculator** from the pop-up menu that appears. This enables you to use the results in a Formula Entry function.
4. Click **OK** to return to the plot window.

Note: It is also possible to copy the parameters by choosing **View Results** from the pop-up menu and clicking the **Clipboard** or **Calculator** button.

Copying the Curve Fit Values to the Data Window

To copy the values of the curve fit line to the data window:

1. Reselect the curve fit from the **Curve Fit** menu. A Curve Fit Selections dialog similar to Figure 10-21 appears with a drop-down arrow under **View** for each variable that has a curve fit applied.
2. Click one of the drop-down arrows and choose **Copy Curve Fit to Data Window** from the pop-up menu that appears.
3. Click **OK** to return to the plot window.

KaleidaGraph will append the results after any existing data in your data window. The first column will be a series of X values. Except for the Weighted curve fit which uses the original X values, the number of X values will be equal to the number of curve fit points specified in the Curve Fit Options dialog (**Format** menu). The second column will contain the values from the curve fit at each of these locations.

Note: If you want to use the original X values or supply your own X values, you need to use Formula Entry to calculate the curve fit values. Section 6.6 provides instructions for using the curve fit functions in Formula Entry to obtain values from the curve fit.

Copying the Residuals to the Data Window

To copy the residual values to the data window:

1. Reselect the curve fit from the **Curve Fit** menu. A Curve Fit Selections dialog similar to Figure 10-21 appears with a drop-down arrow under **View** for each variable that has a curve fit applied.
2. Click one of the drop-down arrows and choose **Copy Residuals to Data Window** from the pop-up menu that appears.
3. Click **OK** to return to the plot window.

KaleidaGraph will append the residual values after any existing data in the data window. These values represent the difference between the original Y values and the values calculated by the curve fit.

10.7 Removing a Curve Fit

Curve fits can be removed from a plot in the same way that they are added. After applying a curve fit, a check mark appears next to that type of fit in the **Curve Fit** menu to indicate that a curve fit is applied.

To remove a curve fit from the plot:

1. From the **Curve Fit** menu, choose the type of curve fit to be removed. A dialog similar to Figure 10-22 is displayed containing the list of variables. The check box of each fitted variable is selected.

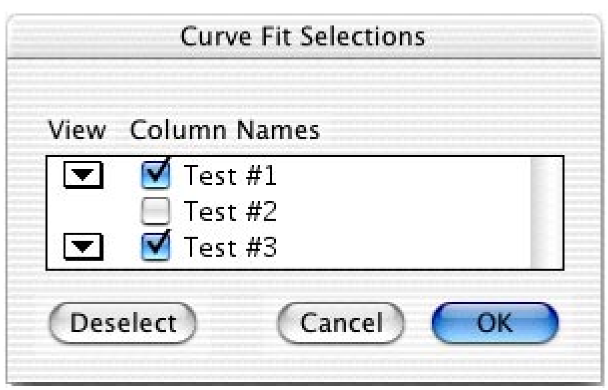


Figure 10-22 Curve Fit Selections dialog

2. Clear any check boxes for the fits to be removed. Clicking **Deselect** clears all of the check boxes at once.
3. Click **OK** to return to the plot and remove the curve fits and any associated equations.

10.8 Other Curve Fit Features

The following sections show how to use some of the options available for curve fits. Some of these options include extrapolating to the axis limits and increasing the number of curve fit points.

The line style, width, and color of the curve fit are controlled using the **Plot Style** command (**Plot** menu). The method for changing these settings is described in Section 8.9.4.

10.8.1 Extrapolating to the Axis Limits

Curve fits are normally displayed between the minimum and maximum data values plotted on the independent axis. However, you can choose to extrapolate the curve fits to the axis limits. The extrapolated portion of the fit is based on the calculated equation.

Note: It is not possible to extrapolate the Smoothing fits (Smooth, Weighted, Cubic Spline, and Interpolate) because they cannot be represented by a single equation.

To extrapolate a curve fit:

1. Choose **Format** > **Curve Fit Options**. The dialog in Figure 10-23 appears.

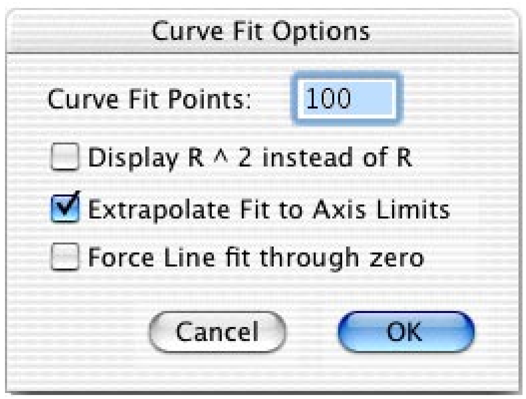


Figure 10-23 Curve Fit Options dialog

2. Select the **Extrapolate Fit to Axis Limits** check box.
3. Click **OK** to return to the plot. After recalculating the curve fits, the plot is redrawn to extend the curve fits to the axis limits.

10.8.2 Forcing a Linear Fit Through the Origin

By default, the slope and intercept are calculated when applying a Linear curve fit. KaleidaGraph provides an option that forces the curve fit to pass through the origin (0,0), so that only the slope of the line is calculated

To force a Linear curve fit through the origin:

1. Choose **Format** > **Curve Fit Options**.
2. Select the **Force Line fit through zero** check box.
3. Click **OK** to return to the plot. Any Linear curve fits are recalculated with a Y-intercept of zero.

10.8.3 Increasing the Number of Curve Fit Points

Any time a curve fit is applied (except for the Weighted fit), KaleidaGraph calculates a predetermined number of points along the fit. These points are used to draw the curve fit and calculate values from the fit. You may find it useful to increase the number of curve fit points when using Formula Entry to place values from the fit into the data window. This increases the accuracy of these values.

To increase the number of curve fit points:

1. Choose **Format** > **Curve Fit Options**.
2. Enter a different value in the **Curve Fit Points** field. A good starting point is to enter the number of points that were plotted. You may want to make this value even higher if there are a number of peaks and valleys in the data.
3. Click **OK** to return to the plot window and recalculate the fits.

10.8.4 Displaying Only the Curve Fit

There may be times when you are only interested in displaying the curve fit on the plot without the original data points. This can be accomplished using the Plot Style dialog, provided the plot is not a Scatter plot.

To display only the curve fit:

1. With the plot window active, choose **Plot** > **Plot Style**. The Plot Style dialog appears.
2. Use the << and >> buttons to locate the variable to be hidden. The name of the current variable appears to the right of these buttons.
3. Choose **None** from the **Show Markers** pop-up menu and **None** for the Line Style. This hides the markers and line used to represent the original data.
4. Click **OK** to return to the plot. The variable is hidden and only the curve fit is displayed.

10.9 Working with Error Bars

Error bars illustrate the amount of error for the plotted data. Error bars normally appear as lines extending from the data points, with caps on either end to represent the upper and lower values of the spread. KaleidaGraph supports the use of error bars in all of the plots in the **Linear** submenu, as well as Probability, X-Y Probability, Horizontal Bar, Column, and Function plots.

10.9.1 Adding Error Bars

To add error bars to a graph:

1. Select the plot which will have the error bars added.
2. Choose **Plot** > **Error Bars**. A dialog similar to Figure 10-24 appears. Each plotted variable is listed, with two check boxes preceding it.

   Note: If you are adding error bars to a Probability, Horizontal Bar, or Column plot, the **X Err** check boxes are unavailable.

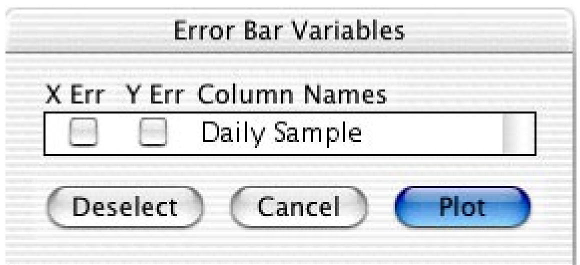


Figure 10-24 Error Bar Variables dialog

3. Click one of the check boxes for the variable to which you want to add error bars. A dialog similar to Figure 10-25 appears so that you can specify the options for this particular set of error bars.

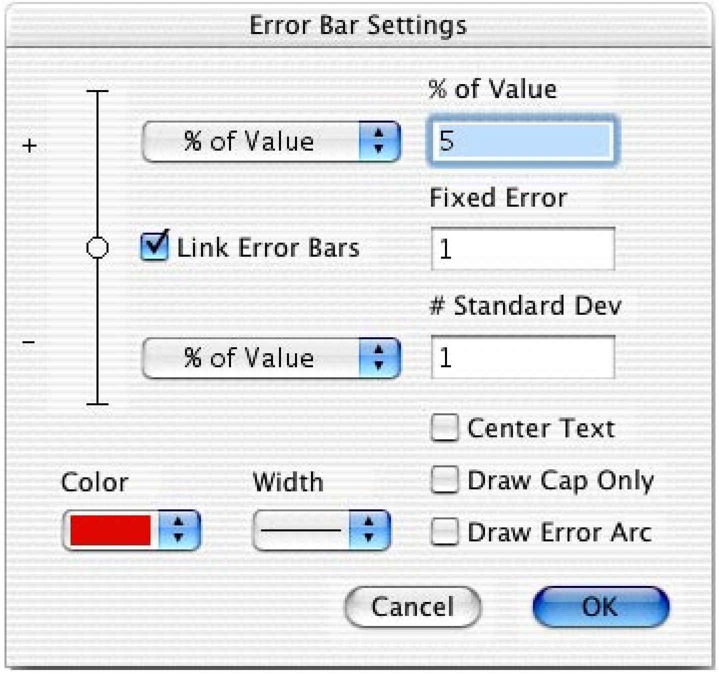


Figure 10-25 Error Bar Settings dialog

4. Make your selections, including the color and thickness for the error bars, and click **OK**.
5. Repeat steps 3 and 4 until error bars have been selected for all of the desired variables.
6. Click **Plot** to add error bars to the variables in the plot. The plot is redrawn and the error bars are displayed. Figure 10-26 shows a sample plot containing error bars.

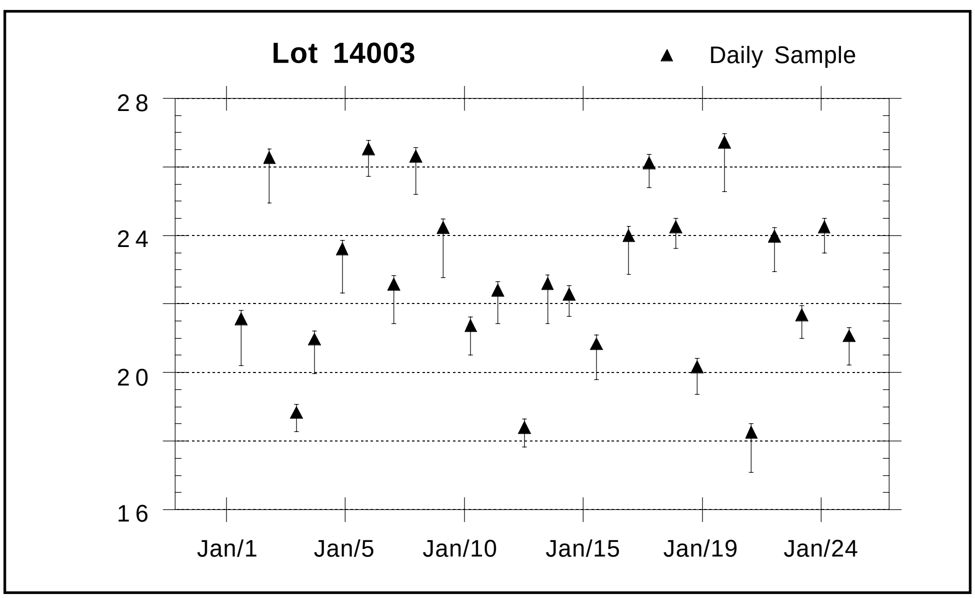


Figure 10-26 Sample plot with error bars

10.9.2 Plotting Error Values from a Data Column

If you do not want to use one of the available error types, you can supply your own set of error values and plot them as error bars.

To plot error values from a data column:

1. In the data window, create a column containing the error values.
2. Select the plot which will have the error bars added.
3. Choose **Plot** > **Error Bars** to display the Error Bar Variables dialog.
4. Click one of the check boxes to display the Error Bar Settings dialog.
5. Choose **Data Column** for the error type and select the name of the data column containing the error values.
6. Click **OK**. Then, click **Plot** to add the error bars to the plot.

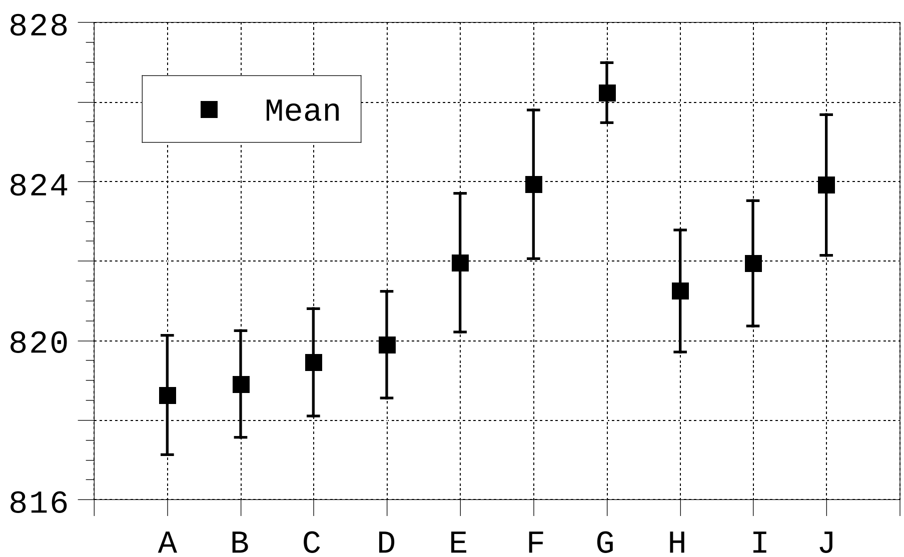


Figure 10-27 Plot with error bars from a data column

10.9.3 Adding Text Error Bars

Error bars can also be used to annotate specific markers or columns in the plot with text labels. When a Text data column is used for the error type, the data in the column is displayed as text labels at the corresponding points on the plot. Only data points that have errors associated with them have text labels displayed.

The direction of the labels is the same as the error type selected (horizontal for X Err or vertical for Y Err). If the **Center Text** check box is selected, the labels are centered on the markers. Otherwise, they are justified and placed on the side of the marker which has the error selected. The text labels can be moved individually with the Text tool or as a group with the Selection Arrow.

Note: Text error bars use the same default attributes as value labels. These labels can also include subscripts or superscripts. For more information on adding these features to text labels, see Section 9.7.3.

To add text error bars to a plot:

1. In the data window, place the data to be displayed as labels into a Text column. Figure 10-28 shows a sample data set containing a Text column with error values.

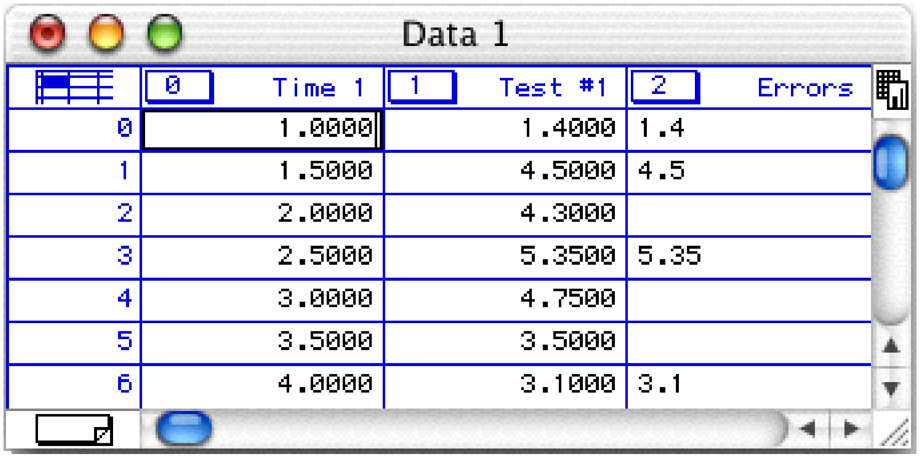


Figure 10-28 Sample data set with text errors

2. Select the plot which will have the error bars added.
3. Choose **Plot** > **Error Bars** to display the Error Bar Variables dialog.
4. Click one of the check boxes to display the Error Bar Settings dialog, shown in Figure 10-29.

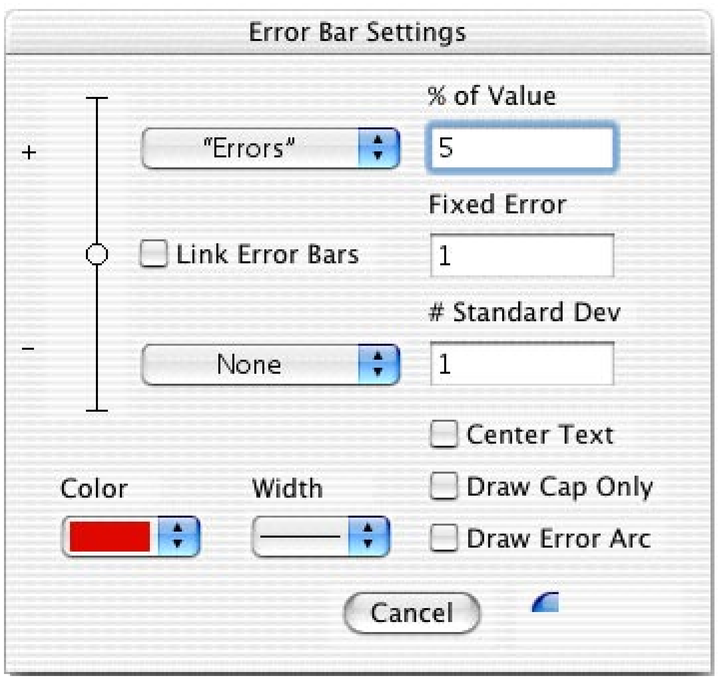


Figure 10-29 Error Bar Settings dialog

5. Clear the **Link Error Bars** check box.
6. Choose **None** for one of the portions of the bar. For the other, choose **Data Column** and select the text column containing the labels (in this case **Errors**).
7. To center the labels on the marker, select the **Center Text** check box. Otherwise, the labels are justified.

   Note: If the **Link Error Bars** check box is selected and the **Center Text** check box is cleared, text labels are placed on both sides of the marker.

8. Click **OK**. Then, click **Plot** to add the labels to the plot. Figure 10-30 shows a sample plot with text error bars added.

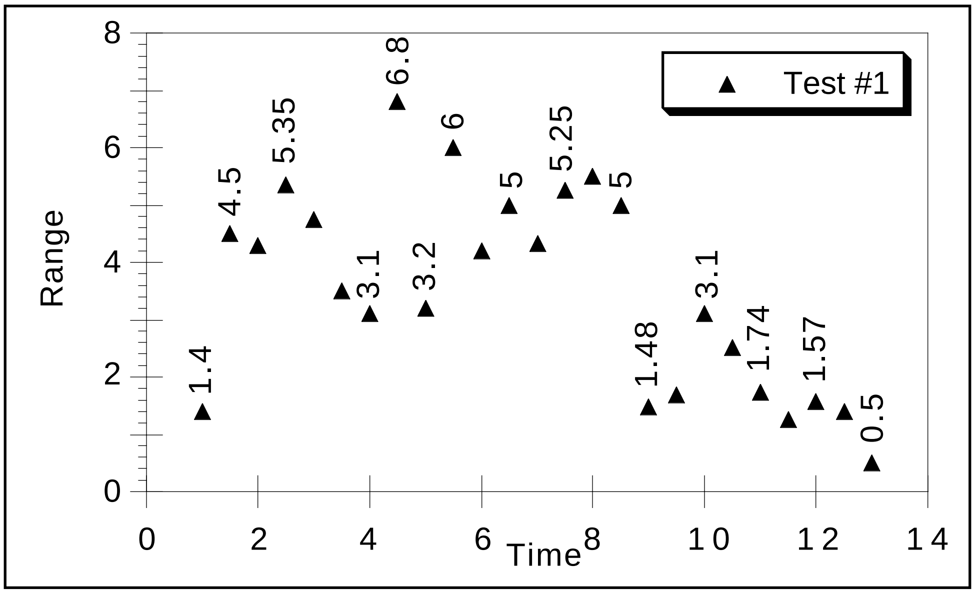


Figure 10-30 Plot with text error bars added

10.9.4 Creating Single-Sided Error Bars

Single-sided error bars are useful when you only want to display the bar on one side of the data point. You can choose a single error type and decide whether to display the positive or negative portion of the bar.

A special case of these error bars results when the errors are taken from a data column. Here the direction of the error bar is controlled by the sign of the data and whether positive or negative error types have been selected. For positive data, bars are displayed in the same direction as the error type. For negative data, the bars are displayed in the opposite direction.

To create single-sided error bars:

1. Select the plot which will have the error bars added.
2. Choose **Plot** > **Error Bars** to display the Error Bar Variables dialog.
3. Click one of the check boxes to display the Error Bar Settings dialog.
4. Clear the **Link Error Bars** check box.
5. Choose **None** for either the positive or negative error type.
6. Choose the type of error to be displayed for the other portion of the bar.
7. Click **OK**. Then, click **Plot** to add the error bars to the plot. Figure 10-31 shows a sample plot with single-sided error bars added.

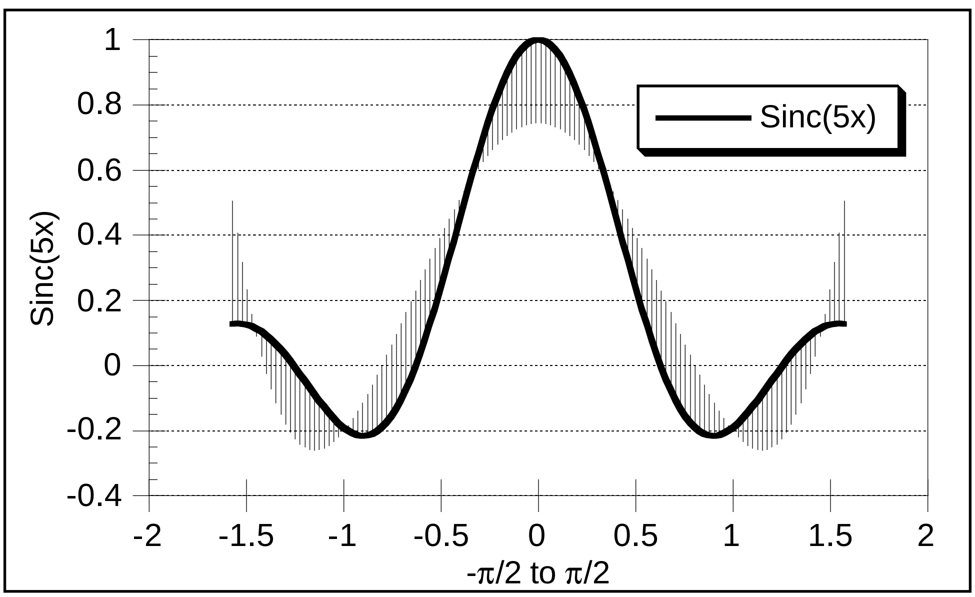


Figure 10-31 Plot with single-sided error bars

10.9.5 Adding Error Arcs

Error arcs can be displayed instead of error bars. This is done by selecting the **Draw Error Arc** check box in the Error Bar Settings dialog. An arc is displayed when one of the error types is set to **None**. If both positive and negative errors are selected for X Err and Y Err, an ellipse is displayed.

All four quadrants of the ellipse can be controlled independently. The vertical distance is determined by the Y Err and the horizontal distance is determined by the X Err. This gives you a number of possibilities for displaying error arcs and ellipses.

To add error arcs:

1. Select the plot which will have the error arcs added.
2. Choose **Plot** > **Error Bars** to display the Error Bar Variables dialog.
3. Click the **X Err** check box for one of the variables to display the Error Bar Settings dialog.
4. Select the **Draw Error Arc** check box.
5. Choose the positive and negative error types. Then, click **OK**. Figure 10-33 shows the dialog containing selections for the X error.
6. Click the **Y Err** check box for that same variable to define the vertical dimension of the arc. Repeat steps 4 and 5 to set up all of the errors of interest. Figure 10-32 shows the Error Bar Settings dialog containing selections for the Y error.

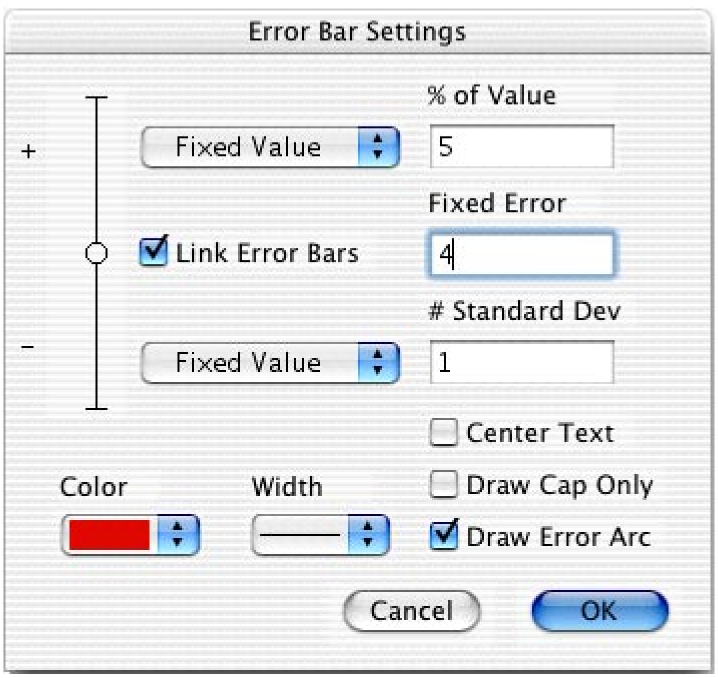


Figure 10-32 X error selections

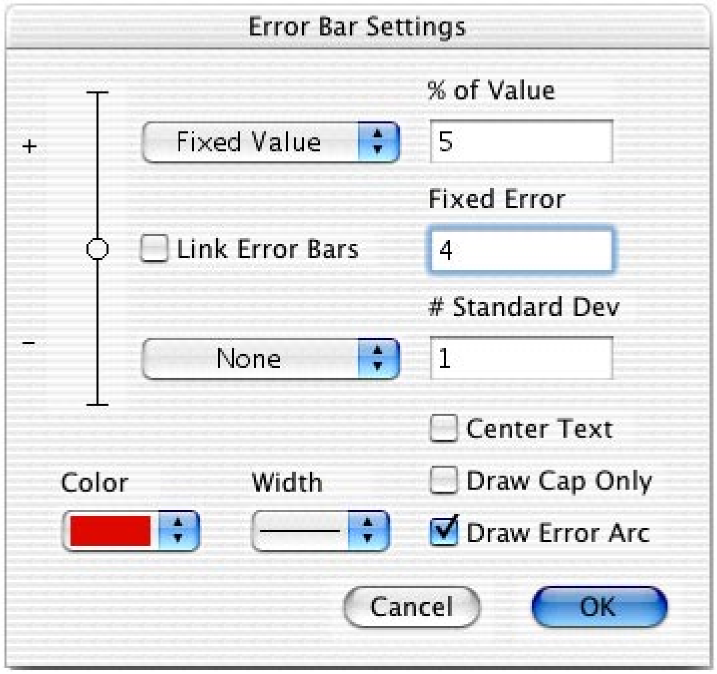


Figure 10-33 Y error selections

7. Click **Plot** to add the error arcs to the plot. Figure 10-34 shows a sample plot with error arcs added, using the settings for the X and Y Error shown above.

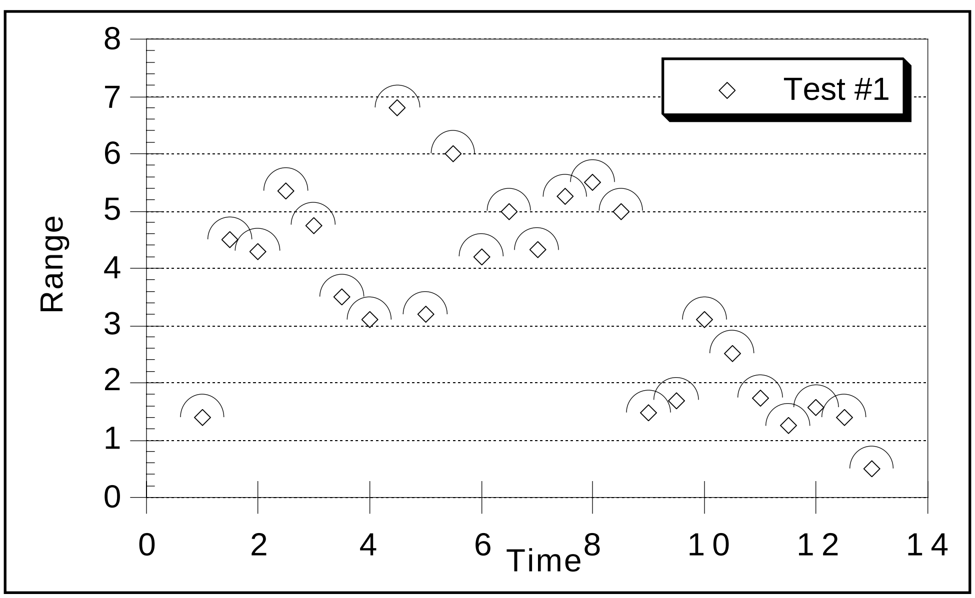


Figure 10-34 Plot with error arcs added

10.9.6 Displaying Only the Cap

You have the option to remove the line that connects the data point with the error cap. This is accomplished by selecting the **Draw Cap Only** check box in the Error Bar Settings dialog.

This option can help remove a number of unnecessary lines from the plot and make it easier to see the data. This feature is useful when the **Standard Deviation** error type is selected. Because all of the error bars are drawn from the mean, the lines connecting the point and the cap are not needed. By removing the lines and adjusting the length of the caps, you can have a dashed or solid error line across the entire plot. See Section 10.9.7 for controlling the cap length.

To display only the error cap:

1. Select the plot which will have the error bars added.
2. Choose **Plot** > **Error Bars** to display the Error Bar Variables dialog.
3. Click one of the check boxes to display the Error Bar Settings dialog.
4. Select the **Draw Cap Only** check box.
5. Choose the positive and negative error types.
6. Click **OK**. Then, click **Plot** to add error bars to the plot. Figure 10-35 shows a sample plot with only the caps displayed and a Standard Deviation error type.

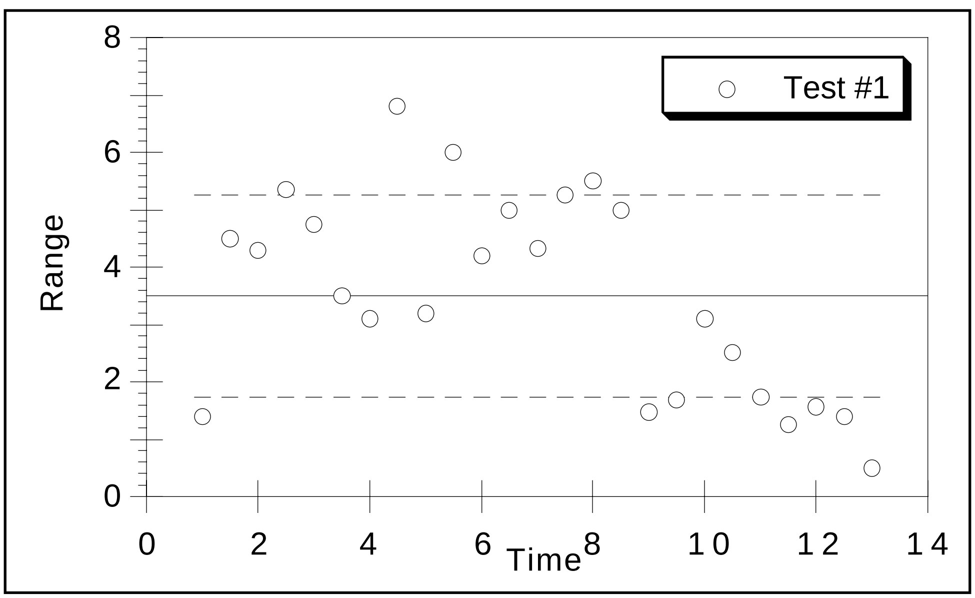


Figure 10-35 Plot displaying only the error caps

10.9.7 Changing the Cap Length

To change the cap length of error bars:

1. Choose **Format** > **Plot Extras**. The dialog in Figure 10-36 is displayed.

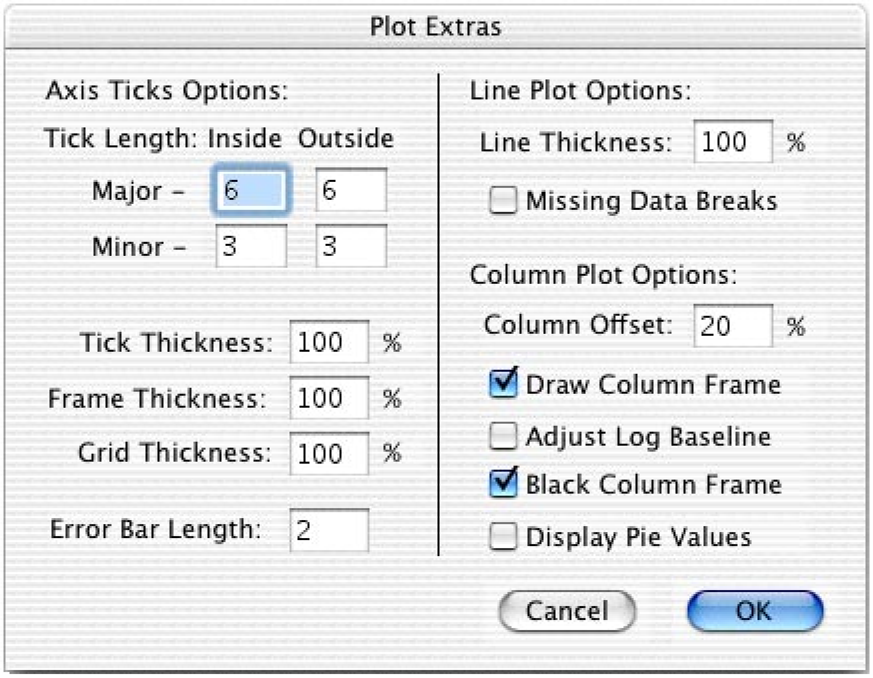


Figure 10-36 Plot Extras dialog

2. Edit the value in the **Error Bar Length** field. A cap is not displayed if zero is entered in this field.
3. Click **OK** to return to the plot.

10.9.8 Removing Error Bars

You can easily remove error bars from individual variables or from all the variables at once.

To remove error bars:

1. Choose **Plot** > **Error Bars**. A dialog similar to Figure 10-37 is displayed.

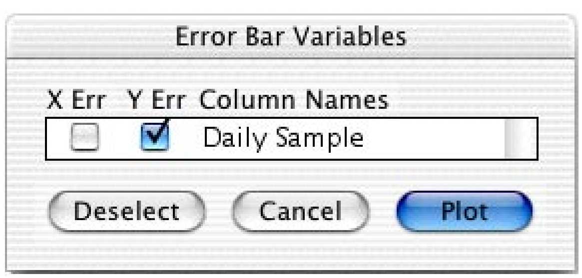


Figure 10-37 Error Bar Variables dialog

2. Clear the check boxes of the error bars to be removed. Clicking **Deselect** clears all of the check boxes at once.
3. Click **Plot** to return to the plot window. The plot is redrawn without any error bars.

Importing and Exporting Graphics

Chapter 11

This chapter explains how to:

- Import graphics files from other programs into the plot window.
- Print plots.
- Save plots.
- Copy plots to the Clipboard.
- Export plots for use in another program.
- Use Object Linking and Embedding for exporting plots in the Windows version.

11.1 Importing Graphics

KaleidaGraph is able to import a variety of graphic files created in other programs. Choosing **File** > **Import** displays a submenu which contains the file formats supported by KaleidaGraph.

When you import a graphic file, the picture is placed in the active plot or layout window. You cannot edit any parts of the imported picture. However, you can move and resize the image, and modify its frame.

Note: A plot or layout window must be the frontmost window on the screen. Otherwise, the graphics options are unavailable in the **Import** submenu.

To import a graphic file:

1. Make the desired plot or layout window the active window.
2. From the **File** menu, choose **Open** or the appropriate file type from the **Import** submenu.
3. Locate and open the file to be imported. The imported image appears as an object in the plot or layout window, with four object handles around it.

11.2 Pasting Graphics

You can paste graphics cut or copied from other programs into the plot and layout windows. When you paste a graphic, it is placed in the active plot or layout window. You cannot edit any parts of the graphic. However, you can move and resize the image, and modify its frame.

To paste a graphic from the Clipboard:

1. Make the desired plot or layout window the active window.
2. Choose **Edit** > **Paste**. The graphic appears as an object in the plot or layout window, with four object handles around it.

11.3 Printing Plots

KaleidaGraph supports laser printers, plotters, ink jet printers, and any other device that has the appropriate printer driver installed.

To export a plot to a printer:

1. Make sure the plot window to be printed is active.
2. To change page options, such as page orientation or paper size, use the **Print Setup** (Windows) or **Page Setup** (Macintosh) command from the **File** menu.
3. Choose **File** > **Print Graphics**.
4. Review your printing options and click **Print**.

Note: The options in the Print Setup, Page Setup, and Print dialogs vary depending on the printer and operating system.

11.4 Saving Plots

KaleidaGraph can save the plots with or without the data used to create the plot. By default, when you save a plot, a copy of the data is saved as part of the plot file. This allows you to open a saved plot and make changes to the plot and the data associated with it (see Section 8.13).

The advantage of saving the plot without the data is that it saves space on the hard disk, especially if you create plots from large data sets. The drawback is that you cannot make any changes that require the plot to be rebuilt from the original data. This includes resizing axes, scaling to a new set of limits, editing markers, and modifying error bars or curve fits.

Note: If the data window used to create a plot is closed before saving the plot, it is saved without the data.

To save a plot with its data:

1. Make sure the plot to be saved is active.
2. Choose **Save Graph**, **Save Graph As**, or **Export** > **Plot w/ Data** from the **File** menu.
3. Name the file and select where it should be stored. Then, click **Save** to export the file to disk.

To save a plot without the data:

1. Make sure the plot to be saved is active.
2. Choose **File** > **Export** > **Plot w/o Data**.
3. Name the file and select where it should be stored. Then, click **Save** to export the file to disk.

11.5 Copying Plots to the Clipboard

Using the **Copy Graph** command (**Edit** menu), plots can be copied to the Clipboard for inclusion in another program. The Copy Graph command is available when a plot window is active and does not contain a selection. This command copies the entire plot to the Clipboard. You can then paste the image into another plot window or into a different program.

To copy a plot to the Clipboard:

1. Make sure the plot window to be exported is active.
2. Choose **Edit** > **Copy Graph**. The following dialog is displayed.

Note: If the **Copy Selection** command is displayed in the **Edit** menu, something is currently selected in the plot window. Do not use this command because it causes the image to have lower resolution.

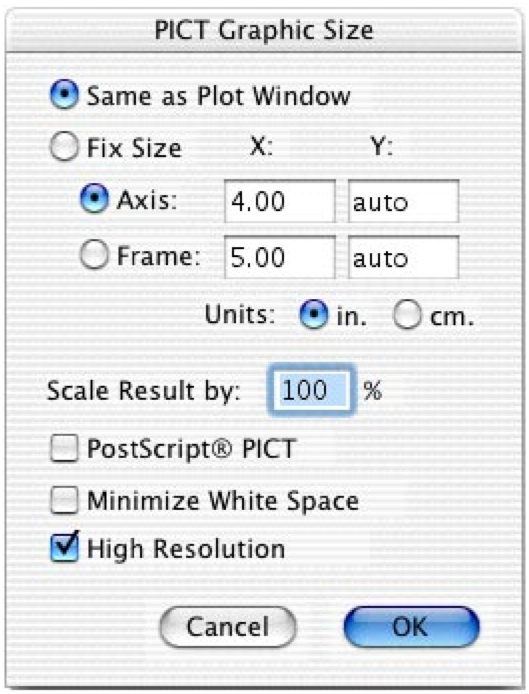


Figure 11-1 Copy Graph dialog

3. Make your selections in the dialog and click **OK**.
4. The entire plot is now on the Clipboard in a Metafile (Windows) or PICT (Macintosh) format and can be inserted into another program using the **Paste** or **Paste Special** command.

11.6 Exporting Graphics

KaleidaGraph is able to save plots in a variety of different formats for inclusion in another program. If one of these files is loaded back into KaleidaGraph, it is treated as a picture. You cannot edit any parts of the picture. However, you can move and resize the image within the plot or layout window.

To export a plot for use in another program:

1. Make sure the plot window to be exported is active.
2. Choose **File** > **Export** and select the desired file format. A dialog similar to Figure 11-2 is displayed to set the size and scaling of the exported file.

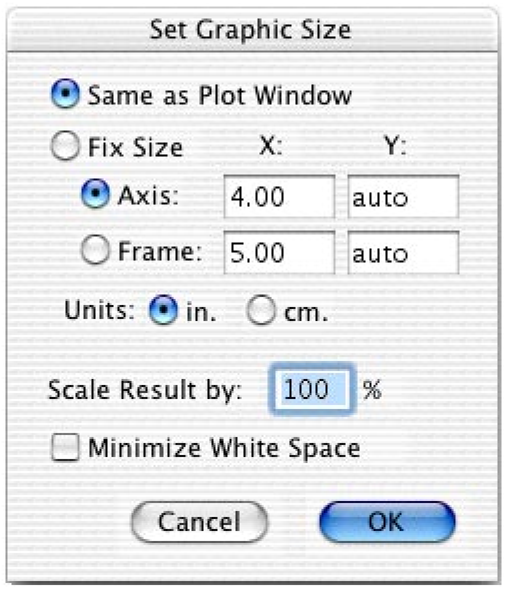


Figure 11-2 Set Graphic Size dialog

3. Click **OK** after making your selections.
4. Name the file and select where it should be stored. Then, click **Save** to export the file to disk.

11.7 Exporting EPS Files

Although KaleidaGraph does not directly have the ability to export plots in an EPS format, it is possible to generate EPS files using PostScript printer drivers. The steps to do this vary between the Windows and Macintosh platforms.

11.7.1 Windows Version

The following steps explain how to generate an EPS file under Windows 95 and 98. Under Windows NT and Windows 2000, an Adobe PostScript printer driver must be selected in the Print dialog. Other drivers may not allow you to select EPS for the Output Format.

1. Choose **Print Graphics** or **Print Layout** from the **File** menu.
2. Select the **Print to File** check box.
3. Choose a PostScript driver and click **Properties**.
4. Click the **PostScript** tab and choose **EPS** for the **Output Format**.
5. Click **OK** until the Print To File dialog is displayed.
6. Enter a file name and change the file name extension from .prn to .eps. Once you click **OK**, an EPS file is created on your hard disk that can be imported into another program. You cannot preview the image in the destination program, but it will print correctly.

11.7.2 Macintosh Version

The steps below are valid for OS 9 and earlier. Under OS X, the export options from the printer driver are limited to PostScript and PDF. Depending on the application you are using, you may be able to import one of these types of files into your document. If not, you can export the image as a high resolution TIFF or PICT file

To export a plot or layout as an EPS file:

1. Make sure v8.0 or later of the Apple LaserWriter driver is selected in the Chooser.
2. Choose **Print Graphics** or **Print Layout** from the **File** menu.
3. The remaining steps depend on the version of the printer driver.
   - LaserWriter drivers 8.4 and newer:
     - Using the pop-up menus in the Print dialog, change the **Destination** to **File**, change **General** to **Save as File**, and change the **Format** to **EPS Mac Enhanced Preview**.
     - Click **Save**. Once you enter a file name and click **Save**, an EPS file is created on your hard disk.
   - LaserWriter drivers 8.0–8.3:
     - Change the **Destination** from **Printer** to **File** and click **Save**.
     - Change the **Format** from **PostScript Job** to **EPS Mac Enhanced Preview**. Once you enter a file name and click **Save**, an EPS file is created on your hard disk.

Working with the Layout Window

Chapter 12

The layout window can be used to place multiple plots on a single page. It can also be used to add text labels, graphics, or tables to a single plot or to multiple plots on a page. The layout window replaces the **Show Page** feature that was available in earlier versions of KaleidaGraph.

This chapter explains how to:

- Display the layout window.
- Place plots into the layout window.
- Change the size and position of plots in the layout.
- Arrange the plots in the layout window.
- Overlay multiple plots.
- Add a background to the layout.
- Save layout files.
- Export the contents of the layout window.

12.1 Displaying the Layout Window

The layout window can be displayed using the **Show Layout** (**Windows** menu) and **New Layout** (**File** menu) commands. The Show Layout command lists any layouts that have been opened or created since the program was started. If a default layout has been saved (in the KG Layout settings file), this submenu will always contain an item called **KG Layout**.

The **New Layout** command creates an empty layout window. If any changes were made to the current layout, a dialog asks if you want to save the changes before displaying the new layout window.

When the layout window is active, a **Layout** menu is added to the menu bar. You can use the commands in this menu to place plots in the layout, arrange the plots, and add a background or frame to the layout.

12.2 Placing Plots in the Layout Window

You can place up to 32 different plots in the layout window. Plots are added to the layout window using the **Select Plot** command (**Layout** menu). Choosing this command displays a submenu that lists the plots that are currently open, from newest to oldest. When you choose a plot from the list, it is placed in the layout window

The name of the plot appears in the lower-left corner of the plot, as shown in Figure 12-2. This is for identification purposes only; it is not included when you print or export the layout.

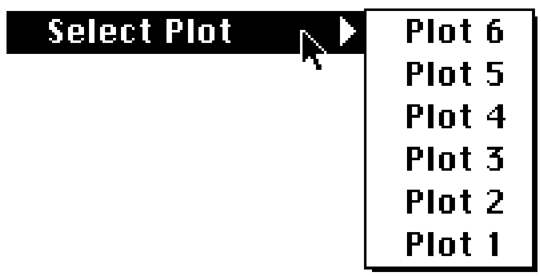


Figure 12-1 Select Plot submenu

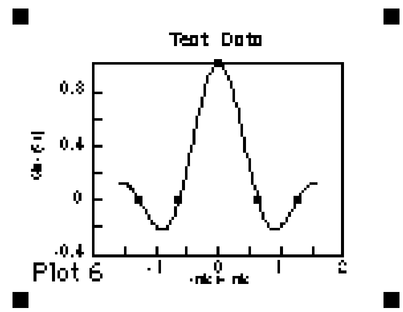


Figure 12-2 Imported plot

As plots are added to the layout window, KaleidaGraph maintains a link between the cell in the layout and the position of the plot in the **Select Plot** submenu. If a particular cell contains the first plot in the list, opening or creating another plot will cause the cell to display this new plot.

Any time you open, create, or close a plot, it is possible for the plots in the layout to change positions because the location of the plots in the **Select Plot** submenu has changed. For this reason, it is best if you open or create all of your plots before arranging them in the layout window.

It is also possible to use empty cells, shown in Figure 12-3, to organize the layout. These can be added to the layout window by choosing **Layout** > **New Position**. The empty cells can be replaced at a later time by clicking the cell and choosing a plot from the **Select Plot** submenu.

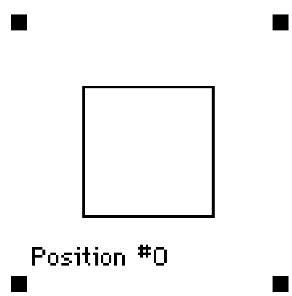


Figure 12-3 Empty cell position

Note: If you open several plots in the Macintosh version but find that some of them are not listed in the **Show Layout** submenu, you need to allocate more memory to the KaleidaGraph program.

12.3 Setting Plot Size and Position

There are two methods available for setting the size and position of plots in the layout window. The first method is to manually drag the plot to resize it or move it to a new position. The second method is to double-click the plot to specify the size and position via the Plot Size dialog.

12.3.1 Manually Resizing the Plot

The plot size and position can be set directly in the layout window by clicking a plot and dragging. When resizing a plot, the **Axis** and **Frame** buttons in the Plot Size dialog determine whether to resize relative to the axis or the frame.

When a plot is selected, object handles appear in one of two places. When **Axis** is selected in the Plot Size dialog, object handles are displayed at the end of each axis. When **Frame** is selected, object handles are placed at the corners of the plot frame. Figure 12-4 shows a plot with **Axis** selected; Figure 12-5 shows a plot with **Frame** selected (in the Plot Size dialog).

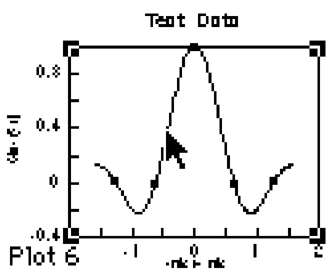


Figure 12-4 Plot with Axis selected

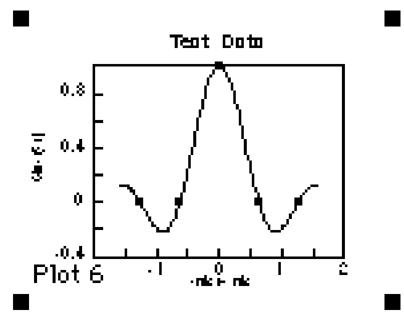
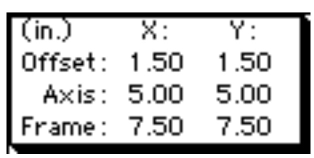


Figure 12-5 Plot with Frame selected

To manually resize the plot, drag one of the object handles. As you drag, a dialog appears in the upper-left corner of the layout window. This dialog (shown below) contains the current plot dimensions and axis origin (offset).



The aspect ratio of the plot is maintained as you resize the plot. By pressing **Shift** as you drag, it is possible to resize the plot without maintaining the aspect ratio.

12.3.2 Using the Plot Size Dialog

The size and position of the plots in the layout can also be controlled using the Plot Size dialog. This dialog, shown in Figure 12-6, is displayed whenever you double-click a plot in the layout window. The settings in this dialog determine the size and location of the plot, as well as whether or not the aspect ratio of the plot is maintained.

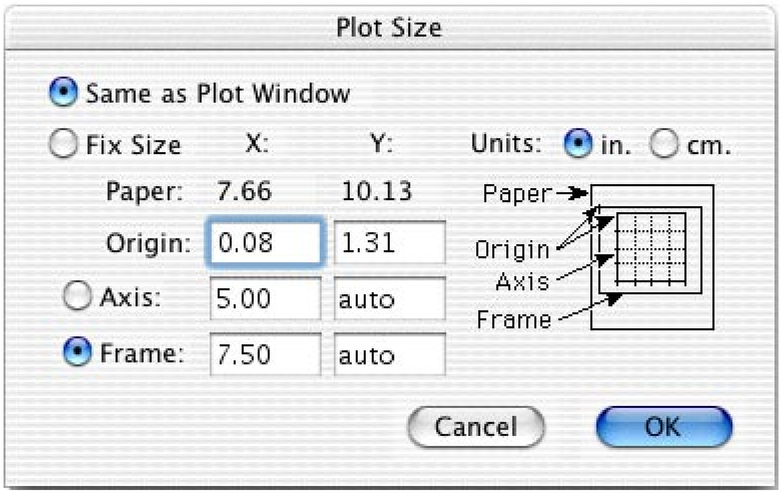


Figure 12-6 Plot Size dialog

The settings in the Plot Size dialog are discussed below:

- **Same as Plot Window -** This setting forces the plot in the layout window to be the same size as the original plot. The location of the plot is controlled using the **Origin** settings.
- **Fix Size -** This setting lets you specify the exact size and placement of the plot in the layout.
  - The **Paper** values, listed for X and Y, are the horizontal and vertical dimensions of the printable space on a sheet of paper.
  - The **Origin** specifies the distance, from the upper-left corner of the print area to the upper-left corner of either the axis or the frame of the plot (depending on which button is selected).
  - The **Axis** settings, when selected, indicate the length of the axes. If the Axis values are changed, the Frame values change proportionately.
  - The **Frame** settings, when selected, indicate the length of the frame. If the Frame values are changed, the Axis values change proportionately.

    Note: Typing the word **auto** allows a floating dimension, while you specify the size of the other dimension. This maintains the plot's aspect ratio. When both X and Y values are entered, without using **auto** for one of them, the aspect ratio is distorted, resulting in ellipses rather than circles for the markers.

  - The **Units** being used must be identified as either inches or centimeters.

12.4 Arranging Plots

The **Arrange Layout** command (**Layout** menu) provides a fast and easy way to take the plots and organize them in the layout window. The plots are arranged in the order they were added to the layout window, starting at the top of the page and working down from left to right. When you choose **Arrange Layout**, the dialog in Figure 12-7 is displayed.

Note: This command can only be used to arrange plots in the layout window. It does not have any affect on any other objects in the layout window.

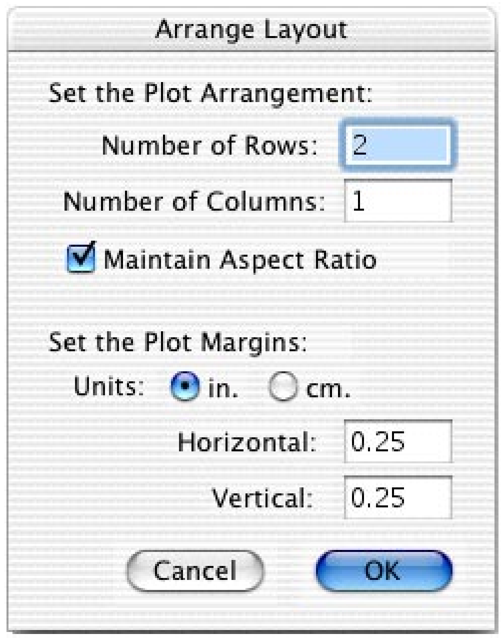


Figure 12-7 Arrange Layout dialog

To arrange the plots in the layout window:

1. Place all of the plots in the layout window.
2. Choose **Layout** > **Arrange Layout**.
3. Enter the desired number of rows and columns into the Arrange Layout dialog. You can also specify the size of the margin around each of the positions in the layout.
4. Click **OK**. The layout window is divided equally, based on the number of rows and columns entered. The plots are automatically resized and placed into the positions.

Figure 12-8 shows a layout window containing five plots in a random pattern. Figure 12-9 shows the same layout window after dividing the window into three rows and two columns using the **Arrange Layout** command.

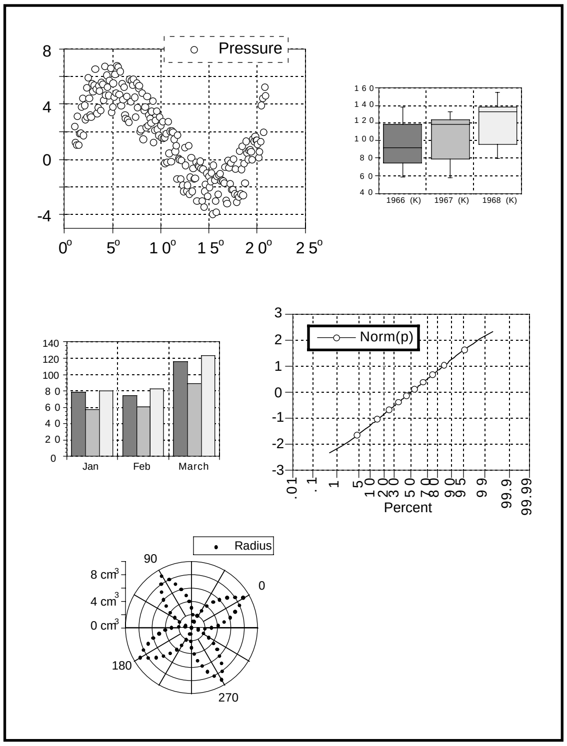


Figure 12-8 Before choosing Arrange Layout

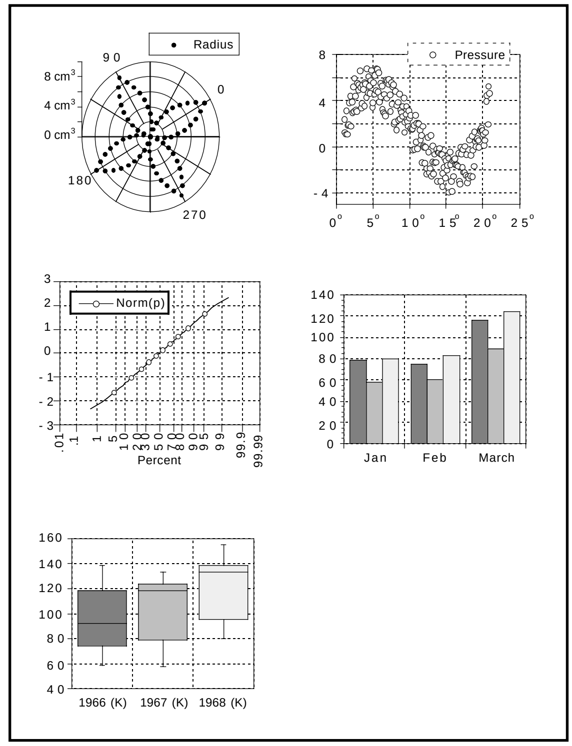


Figure 12-9 After choosing Arrange Layout

12.5 Overlaying Plots

You can use the layout window to position two or more plots on top of each other. These plots must have the same frame and axis dimensions. This is useful if you need to display a single plot with more than the maximum number of variables allowed. By creating separate plots, you can overlay them to get the appearance of a single plot.

To overlay two or more plots:

1. Open or create all of the plots that you need to overlay. The plots must be identical in size for them to overlay properly. To ensure that the plots are of equal size, either choose **Plot** > **Define Layout** after creating the first plot or use the first plot as a template for creating any remaining plots. You can also use the **Set Plot Size** command (**Plot** menu) for each plot to make sure the origin, axis, and frame settings are identical.
2. Remove the text labels from all but one of the plots. The following two plots are used in this example. Figure 12-10 shows a normal plot with all of its labels intact. Figure 12-11 shows a similar plot with everything but the legend deleted.

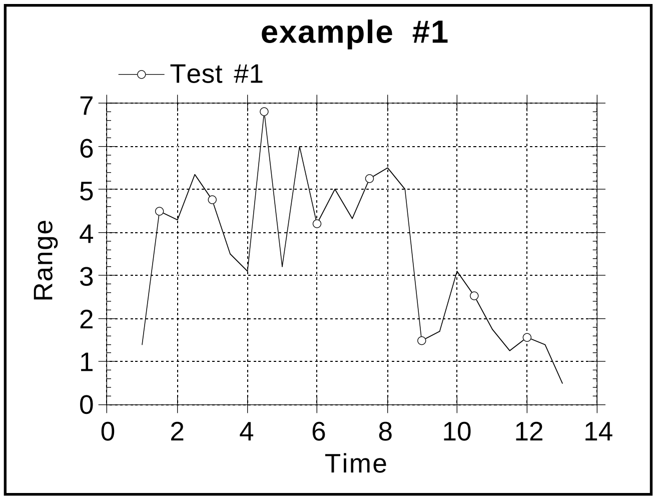


Figure 12-10 Sample plot

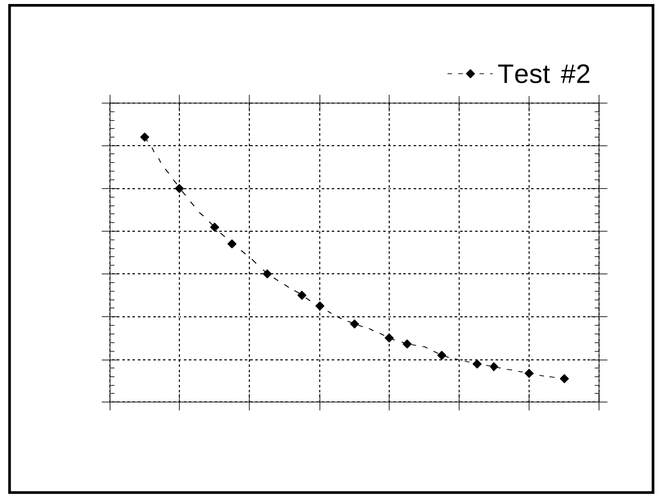


Figure 12-11 Second plot with labels removed

3. Switch to the layout window.
4. Use the **Select Plot** command (**Layout** menu) to import the plots into the layout window.
5. Provided that all of the plots are identical in size, you can use the Align tool to overlay the plots. This is done by selecting the plots and using the Align tool to align either the edges or centers in both the horizontal and vertical directions. Figure 12-12 shows the result of overlaying the two sample plots
6. The layout window can then be printed, copied to the Clipboard, or saved in a variety of formats. You can also save this layout for future use.

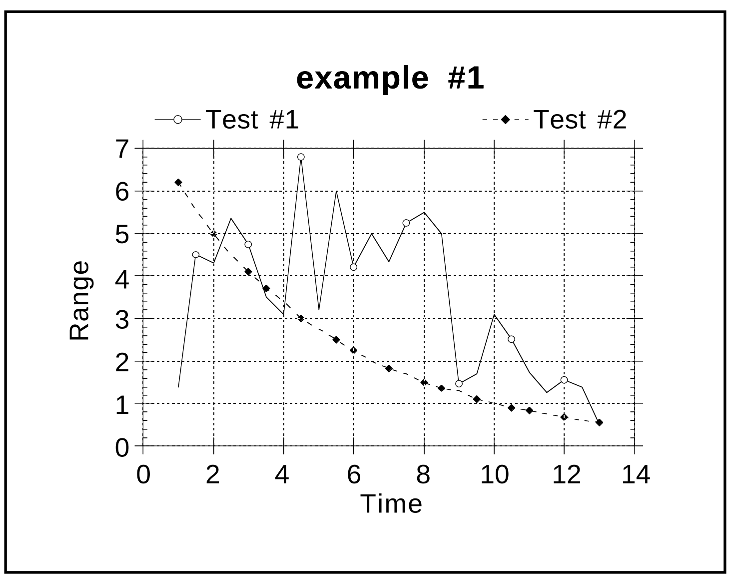


Figure 12-12 Result of overlaying two plots

12.6 Adding a Background or Border

A background or border can be added to the layout window or to the plots in the layout window. The following sections explain how to add backgrounds and borders in the layout window.

12.6.1 Adding to the Layout Window

A background can be added to the layout window using the **Set Background** command (**Layout** menu). This allows you to select a fill pattern for any part of the page not covered by a plot. It also lets you have a frame around the entire layout.

Note: Text labels may or may not have the background displayed behind the text. The **Erase Background** command (in the **Format** menu of the Edit String dialog) controls what happens. When this command is turned on, the background is erased.

To add a background or border to the layout:

1. Choose **Layout** > **Set Background**. The dialog in Figure 12-13 appears.

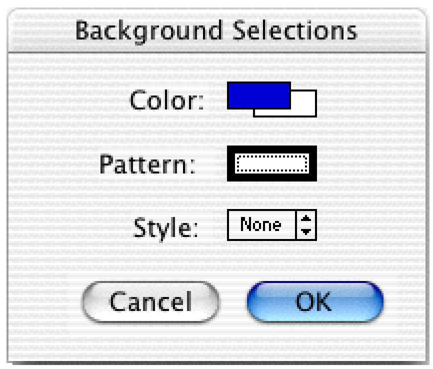


Figure 12-13 Set Background dialog

2. Make your selections by clicking the appropriate section of each icon. The background is controlled by the Color icon and the Fill Pattern portion of the Pattern icon. The border is controlled by the Style icon and the Pen Pattern portion of the Pattern icon. Section 9.9 explains the use of these icons.
3. Click **OK** to apply the background or border to the layout window. Figure 12-14 shows a layout with a background applied. The text label at the top has **Erase Background** turned on and the text labels below each plot do not.

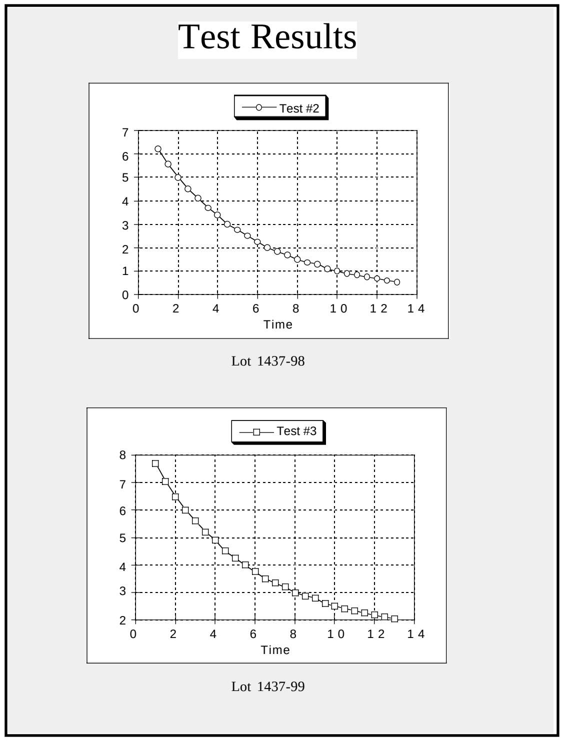


Figure 12-14 Sample layout with background

12.6.2 Adding to a Plot in the Layout

It is possible for each plot in the layout window to have its own background and border. This is accomplished by selecting the plot in the layout and using the Color, Pattern, and Style icons from the bottom of the toolbox

The background is controlled by the Color icon and the Fill Pattern portion of the Pattern icon. If a background color is specified in the original plot window, these settings do not have any affect on the appearance of the plot. The border is controlled by the Style icon and the Pen Pattern portion of the Pattern icon.

Note: Any background applied to the layout is also applied to any plots which have **None** selected for the Fill Pattern and do not have a background color selected in the original plot window.

12.7 Exporting the Layout

The contents of the layout window can be exported by copying the layout to the Clipboard, exporting the layout to a file, or printing the layout.

12.7.1 Copying Layouts

To export a layout to the Clipboard:

1. Make sure the layout window to be exported is active.
2. Choose **Edit** > **Copy Layout**. A dialog is displayed to set the size and scaling of the exported image

   Note: If the **Copy Selection** command is displayed in the **Edit** menu, something is currently selected in the layout window. Do not use this command because it causes the image to have lower resolution.

3. Make your selections in the dialog and click **OK**.
4. The entire layout is now on the Clipboard in a Metafile (Windows) or PICT (Macintosh) format and can be inserted into another program using the **Paste** or **Paste Special** command.

12.7.2 Exporting the Layout to a File

KaleidaGraph is able to save layouts in a variety of different formats for inclusion in another program. If one of these files is loaded into KaleidaGraph, it is treated as a picture. You cannot edit any parts of the picture. However, you can move and resize the image within the plot or layout window.

Note: For information on saving the layout in an EPS format, refer to Section 11.7.

To export a layout for use in another application:

1. Make sure the layout window to be exported is active.
2. Choose **File** > **Export** and select the desired file format. A dialog is displayed to set the size and scaling of the exported file.
3. Click **OK** after making your selections.
4. Name the file and select where it should be stored. Then, click **Save** to export the file to disk.

12.7.3 Printing Layouts

To print a layout:

1. Make sure the layout window to be printed is active.
2. To change page options, such as page orientation or paper size, use the **Print Setup** (Windows) or **Page Setup** (Macintosh) command from the **File** menu.
3. Choose **File** > **Print Layout**.
4. Review your printing options and click **Print**.

Note: The options in the Print Setup, Page Setup, and Print dialogs vary depending on the printer and operating system.

12.8 Opening and Saving Layout Files

Once you have created a layout, you can save the organization of the layout for future use. The layout file stores the following information: size and position of the cells, the contents and location of any text labels or objects, and the background settings.

The plots are not saved as part of the layout file; however, the cells are assigned position numbers based on the order of the plots in the **Select Plot** submenu. For example, if a cell contains the second plot listed in the **Select Plot** submenu, the next time you open this layout file, the second plot in the **Select Plot** submenu will automatically be loaded into this cell.

Note: To see the position numbers for each of the cells, open the layout file before opening or creating any plots. Any cells that show Position #0 are empty cells and will not automatically have plots loaded into them.

To save a layout file:

1. With the desired layout as the active window, choose **Save Layout**, **Save Layout As**, or **Export** > **Layout** from the **File** menu.
2. Give the file an appropriate name and click **Save**.

To open a saved layout file:

1. Open or create the plots you want to display. There is a precedence for loading plots into the layout window, as discussed earlier in this section.
2. Use the **Open** or **Import** > **Layout** command (**File** menu) to open the layout file you saved previously. The plots are automatically positioned according to the layout. The background and any text or graphic elements are also displayed in the layout window.

12.9 Other Layout Window Features

12.9.1 Changing the View of the Layout Window

KaleidaGraph provides several magnification levels for changing the view of the layout window. To change the magnification of the layout, click the Zoom drop-down menu in the lower-left corner of the layout window and choose the desired zoom level. The available zoom percentages are: 25, 50, 75, 100, 125, 150, 200, 300, and 400%.

12.9.2 Displaying Rulers and Grids in the Layout Window

To turn the rulers and grids on and off in the layout window:

1. With the layout window active, choose **Plot** > **Ruler & Grids**.
2. Click the appropriate check boxes to display the rulers, grids, or both.
3. Select the proper units that you want to use.
4. Click **OK** to return to the layout window.

12.9.3 Switching Between Layouts

KaleidaGraph provides a **Show Layout** command in the **Windows** menu which can be used to switch between the default KG Layout and any other layouts that have been opened or created since the program was started. The active layout is listed first in the submenu. Older layouts are listed below the dotted line.

12.9.4 Deleting Items from the Layout

You can delete any unwanted items from the layout window in one of two ways. You can either select the item and press **Backspace** (Windows) or **Delete** (Macintosh), or select the Eraser from the toolbox and click the unwanted item. These methods work for plots or any other objects you want to delete.

12.9.5 Refreshing the Layout

The layout window utilizes the **Auto Refresh** command from the **Plot** menu. When this command is turned on, the plots are updated whenever a change is made in the layout window. When Auto Refresh is turned off, the plots are not updated and are replaced by placeholders in the layout. The plots are not redrawn until the Auto Refresh command is turned on again.

Turning this command off allows you to make several changes in the layout window without having to wait for the window to update after each change. This is particularly useful when arranging several plots on a page or when working with large data sets.

Command Reference

Appendix A

This chapter provides an overview of the commands available in KaleidaGraph. For more detailed information, including the options in the various dialogs, refer to the online help.

A.1 File Menu

New

This command creates a blank data window for entering data. By default, new data windows are created 10 columns by 100 rows in size and are named **Data 1**, **Data 2**, and so on. To change any of the attributes of the data window, use the **Column Format** command (**Data** menu).

Open

This command opens any KaleidaGraph file that has been previously saved. You can also open text and Microsoft Excel files. If a plot or layout window is active, this command can import a variety of graphics into the window.

Merge

You can use this command to combine data from another file with the active data window. This command opens the data file and places it at the current row and column position. Files may be either KaleidaGraph data files, Microsoft Excel files, or text files.

Note: If you exceed the 1000 column limit while merging files, only the number of columns needed to reach the limit is added.

Close

This command closes the active window. Clicking the close box performs the same function.

Note: If you want to close the window without saving any changes, press **Shift** while either clicking the close box or choosing **File** > **Close/NoSave**.

Update & Return

Note: This command is only available in the Windows version of KaleidaGraph.

This command is displayed when editing an embedded OLE object. Choosing this command updates the embedded plot and returns to the source document. If KaleidaGraph was open prior to editing the OLE object or if other files were opened before choosing this command, the source document is not automatically displayed.

Save

This command saves the active data, plot, layout, or plot script window. Depending on which window is active, a **Save Data**, **Save Graph**, **Save Layout**, or **Save Script** command is displayed.

- **Save Data** - This command saves the data as a KaleidaGraph data file. If text format is desired, use the **Export** command.
- **Save Graph** - This command saves the plot with its data, provided the data window is still open. If you want to save the plot using a different format, use the **Export** command.
- **Save Layout** - This command saves the cell positions and background, as well as any text or graphic objects in the active layout window. This command does not save the plots as part of the layout file. Only the positions of the cells are saved.
- **Save Script** - This command saves the active plot script. If a plot is used as the graphic template, it is saved at the same time as the plot script file. In the Windows version, the plot has the same name as the script, but with a **.qpc** file name extension. In the Macintosh version, the plot's file name contains the first 25 characters of the script's file name, followed by **.plot**.

Update

Note: This command is only available in the Windows version of KaleidaGraph.

This command is displayed when editing an embedded OLE object. Choosing this command sends the updated object to the source document.

Save As

This command saves a copy of the current data, plot, layout, or plot script window. Depending on which window is active, a **Save Data As**, **Save Graph As**, **Save Layout As**, or **Save Script As** command is displayed.

- **Save Data As** - This command saves a copy of the data as a KaleidaGraph data file. If text format is desired, use the **Export** command.
- **Save Graph As** - This command saves a copy of the plot with its data, provided the data window is still open. If you want to save the plot using a different format, use the **Export** command.
- **Save Layout As** - This command saves a copy of the cell positions and background, as well as any text or graphic objects in the active layout window. This command does not save the plots as part of the layout file. Only the positions of the cells are saved.
- **Save Script As** - This command saves a copy of the active plot script. If a plot is used as the graphic template, it is saved at the same time as the plot script file. In the Windows version, the plot has the same name as the script, but with a **.qpc** file name extension. In the Macintosh version, the plot's file name contains the first 25 characters of the script's file name, followed by **.plot**.

Import

You can use this command to open a variety of files from within KaleidaGraph. Use the submenu to select the type of document to be imported. The Open dialog is displayed, no matter which type of file you specify. Only files that are the same type as what you choose from the submenu are displayed.

Export

You can use this command to save a variety of files from within KaleidaGraph. Use the submenu to select the type of document to be saved. The Save dialog is displayed, no matter which type of file you specify.

Preferences

You can use this command to control how various files are handled when you quit the program. This command also controls some other general settings for the program.

Note: The settings for this dialog are saved as part of the Style file. If you choose **Never** for the Style, you must manually overwrite the default Style file by choosing **File** > **Export** > **Style**.

Print Setup/Page Setup

This command controls printing options such as paper size and page orientation. The options available in this dialog vary depending on which printer is selected and what operating system you are using. Under Windows, the **Print Setup** command is displayed; on a Macintosh, the **Page Setup** command is displayed.

New Layout

You can use this command to create new layouts from scratch. Choosing this command displays an empty layout window. For more information on using the layout window, see Chapter 12.

Print

This command prints the contents of the active data, plot, or layout window. If the Macro Calculator or Formula Entry window is active, you can print the active macro in the Calculator. Depending on which window is active, a **Print Data**, **Print Graphics**, **Print Layout**, or **Print Program** command is displayed.

- **Print Data** - This command prints the entire data set, if a selection does not exist. A maximum of 15 columns may be printed on a single page.
- **Print Graphics** - This command prints the contents of the active plot window.
- **Print Layout** - This command prints the contents of the active layout window.
- **Print Bitmap** - This command is only available in the Macintosh version of KaleidaGraph. This command prints a bitmapped version of the graph. To print a bitmap, press **Shift** while choosing the **Print Bitmap** command from the **File** menu.
- **Print Program** - This command prints the active program in the Macro Calculator.

Recent Files

This command displays a submenu that lists the last eight data or plot files that have been opened or saved in KaleidaGraph. You can open a listed file by choosing its file name from the submenu.

Exit/Quit

This command closes the KaleidaGraph program and returns to the desktop. Under Windows, the **Exit** command is displayed; on a Macintosh, the **Quit** command is displayed.

KaleidaGraph saves any changes to the default Style, Macros, Script, and Layout files, depending on what is specified in the Preferences dialog. If **Prompt** is selected for any of the items in the Preferences dialog and changes have occurred to those particular items, a dialog is displayed.

A.2 Edit Menu

Undo/Redo

The **Undo** command causes the data, plot, or layout window to revert to its state prior to issuing the last command or operation. If a command or operation cannot be undone, this command is unavailable and is named **Can't Undo**. If it is possible to redo an operation, the **Undo** command changes to **Redo**.

Cut

In the data window, the **Cut** command removes a data selection and places it on the Clipboard. In the plot and layout windows, the **Cut** command removes objects, equations, and text labels that have been added to the original plot. If you cut any part of the original plot, including the labels, titles, or legend, the entire plot is placed on the Clipboard.

- **Cut w/Titles** - If **Shift** is pressed, **Cut** is replaced by **Cut w/Titles**. This command includes the column titles when you cut a data selection.

Copy

If the data window is active, the **Copy** command places a copy of the data selection on the Clipboard. If a plot or layout window is active, the **Copy Graph**, **Copy Selection**, or **Copy Layout** command is displayed.

- **Copy w/Titles** - If **Shift** is pressed, **Copy** is replaced by **Copy w/Titles**. This command includes the column titles when copying a data selection.
- **Copy Graph** - This command is available when the plot window is active and nothing is selected. This command copies the contents of the plot window to the Clipboard as a Metafile (Windows) or PICT (Macintosh) image. You can then paste the plot into another program.
- **Copy Selection** - This command is displayed when any object is selected in the plot or layout window. This command copies the selected object to the Clipboard. If you copy any part of the original plot, including the labels, titles, or legend, a copy of the entire plot is placed on the Clipboard.
- **Copy Layout** - This command is available when a layout window is active and nothing is selected. This command copies the contents of the layout window to the Clipboard as a Metafile (Windows) or PICT (Macintosh) image.

Paste

This command places the contents of the Clipboard into a data, plot, or layout window. When data is pasted into a data window, the data in the first row determines the column's format. When data is pasted into a plot or layout window, the data is placed in a table.

- **Paste w/Titles** - If **Shift** is pressed, **Paste** is replaced by **Paste w/Titles**. This command includes the column titles when you paste a selection from the Clipboard into a data window.

Note: There is no difference between the **Paste** and **Paste w/ Titles** commands when selections are cut or copied from KaleidaGraph's data windows (using the **Cut**, **Copy**, **Cut w/Titles**, or **Copy w/ Titles** command). If the data on the Clipboard is not from a KaleidaGraph data window, **Paste** and **Paste w/ Titles** operate as described above.

Clear

In the data window, choose **Clear** to delete all selected data cells. The cells below the selection shift up to fill the void.

- **Clear w/Titles** - If **Shift** is pressed, **Clear** is replaced by **Clear w/Titles**. This command deletes the column titles along with the data.

In a plot or layout window, **Clear** removes the selected item from the window. This command does not affect the contents of the Clipboard. You can also use the **Backspace** (Windows) or **Delete** key (Macintosh) to perform the same function.

Binary Mode/Text Mode

This command controls whether the **Cut** and **Copy** commands use binary or text format to copy large amounts of data to the Clipboard. When copying 10000 cells or less, both a binary and text representation of the data are copied to the Clipboard. This occurs regardless of the Binary Mode/Text Mode setting.

Binary Mode is much faster and allows you to quickly move large amounts of data between data windows. **Text Mode** should be used if you need to copy more than 10000 cells and paste them into another program.

The active mode is listed in the **Edit** menu and has a check mark in front of it. To switch between modes, choose the mode that is currently active.

Select All

This command selects all objects in the current window. If a data window is active, **Select All** makes a data selection that includes all cells containing data. Any empty cells embedded in the data are also included.

Duplicate

This command makes a copy of selected text or objects in a plot or layout window. The copy is placed slightly below and to the right of the original. If you duplicate any part of the original plot, including the labels, titles, or legend, the entire plot is duplicated.

Clear Data

This command clears the contents of an entire data selection. Only the data is removed; the cells are not deleted as they are with the **Clear** command. You can also use the **Backspace** (Windows) or **Delete** key (Macintosh) to perform the same function.

Paste Link

Note: This command is only available in the Windows version of KaleidaGraph.

This command pastes the contents of the Clipboard in the data window as a linked DDE object. The data pasted into KaleidaGraph maintains a reference to the source document so that when the original file changes, the changes are reflected in the KaleidaGraph data file.

Create Link

Note: This command is only available in the Windows version of KaleidaGraph.

This command creates a DDE link to data in a source document. The linked data maintains a reference to the source document so that when the original file changes, the changes are reflected in the KaleidaGraph data file

Link Options

Note: This command is only available in the Windows version of KaleidaGraph.

This command controls whether the linked data is updated automatically or manually. By default, new links are set to update automatically. This command can also be used to cancel a DDE link.

OLE Options

Note: This command is only available in the Windows version of KaleidaGraph.

This command is only displayed when editing an embedded OLE object. This command controls some of the settings (size, scaling, and resolution) of the embedded object.

Hide Borders/Show Borders

Note: This command is only available in the Windows version of KaleidaGraph.

The **Hide Borders** command hides the borders surrounding linked data under Windows. Once you hide the borders, the command changes to **Show Borders**. Use the **Show Borders** command to redisplay the borders

Edit Text

This command displays KaleidaGraph's text editing facility. The text editor is useful for editing the contents of the Clipboard and other small editing tasks.

A.3 Gallery Menu

The **Gallery** menu lists the types of plots that can be created in KaleidaGraph. For more information on a specific plot type, see Section 7.1 or refer to the online help.

A.4 Plot Menu

Axis Options

This command controls the limits, ticks, grids, and labels for the horizontal and vertical axes in the active plot

Plot Style

This command controls the markers, fill patterns, line styles, and colors which are used in various plots to represent the plotted variables.

Auto Link

When this command is turned on, the plot automatically updates whenever a value is changed in the data window. After changing a data value, moving to another cell causes the update to occur. All curve fits and error bars are automatically recalculated. When **Auto Link** is turned off, the plot is not updated.

Update Plot

This command forces an immediate regeneration of the plot from the original data set. Clicking the Update Plot button ( ) in the data window performs the same function.

Auto Refresh

When this command is turned on, the plot or layout window is redrawn whenever a change is made. When **Auto Refresh** is turned off, the plot or layout window is not redrawn until this command is turned on again.

Display Legend

When this command is turned on, the legend is displayed in the plot window. The legend can be removed by turning off this command or using the **Backspace** (Windows) or **Delete** key (Macintosh) when the legend is selected (which also turns off the **Display Legend** command). The Eraser tool removes the legend permanently, without turning off this command.

Display Title

When this command is turned on, the plot title is displayed in the plot window. The plot title can be removed by turning off this command or using the **Backspace** (Windows) or **Delete** key (Macintosh) when the title is selected (which also turns off the **Display Title** command). The Eraser tool removes the plot title permanently, without turning off this command.

Display Equation

When this command is turned on, the results from curve fits that have closed form solutions (equations) are displayed in the plot window. This includes the General, Linear, Polynomial, Exponential, Logarithmic, and Power curve fits. To change the precision of the results, use the **Equation Label Format** command (**Format** menu).

Error Bars

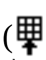
You can use this command to add horizontal and vertical error bars to all of the Linear plots, as well as Probability, X-Y Probability, Horizontal Bar, and Column plots. Error bars illustrate the amount of error for the plotted data. They normally appear as lines extending from the data points, with caps on either end to represent the upper and lower values of the spread.

Add Values

When this command is turned on, the numerical values of bars or columns, or the percentage value of pie slices are automatically placed on the plot. This command affects Pie, Horizontal Bar, Stack Bar, Column, and Stack Column plots. The values can be removed by turning off this command or using the **Backspace** (Windows) or **Delete** key (Macintosh) when the labels are selected as a group. The Eraser tool can be used to remove individual labels.

The values may be moved as a group with the Selection Arrow, or individually with the Text tool. To change the format of the values displayed, choose **Format** > **Value Label Format**.

Extract Data

This command displays the data used to create a saved plot. The Find Data button () in the plot window can also be used to extract this data. The **Extract Data** command is unavailable if the data was not saved with the plot or if it was already extracted.

Define Layout

This command is used to save the basic characteristics of the current plot as a mini-template. Any new plots of that type have the same layout as the original. The layout saves the position of the legend, axes, and titles, the default style for the legend, and the size of the axes, frame, and legend.

Each plot type has its own layout. All layouts are saved in the Style file.

Set Plot Size

You can use this command to specify the exact size and position of the axes or frame for the active plot.

Ruler & Grids

You can use this command to display a ruler and a grid in the plot and layout windows.

A.5 Data Menu

Overwrite Mode

When this command is selected, cells are automatically selected when you use the mouse or keyboard to move to a cell. Typing data into the current cell replaces any existing data.

Insert Mode

When this command is selected, clicking a cell does not automatically select the cell. Instead, a cursor is placed in the cell to edit the current value. To select a cell in Insert Mode, you must double-click the active cell.

Append Columns

This command appends the specified number of columns to the data window. If the current cell is in the last column of the data window, pressing **Right Arrow** automatically adds a new column.

Insert Column

This command inserts an empty column to the left of the column currently selected.

Delete Column

This command removes selected columns from the data window. You can remove a single column or a group of columns using this command.

Insert Row

This command inserts an empty row above the row currently selected. A row can be inserted in a single column or a range of columns.

Delete Row

This command removes selected rows from the data window. You can remove an entire row or a data selection from the data window.

Add Rows

This command adds the specified number of rows to the data window. Rows are added in multiples of 100. If the active cell is in the last row of the data window, pressing the **Enter** (Windows), **Return** (Macintosh), or **Down Arrow** key adds 100 rows to the data window.

Compress Rows

This command compresses a data window to its minimum size by deleting all empty rows that follow the last physical row of data. Rows are deleted in multiples of 100. A data window will never have less than 100 rows

Go To Cell

This command is used to view an exact cell location in the data window. Choosing **Go To Cell** displays a dialog to enter the row and column numbers of the desired cell. The data window automatically scrolls to make that cell position visible.

Column Format

This command is used to edit column titles and change the format of any column in the data window.

Posted Note

This command displays the Posted Note of the active data window. The same result can be obtained by clicking the Posted Note button ( ) in the data window. The Posted Note can be used to store information about the active data set.

A.6 Functions Menu

Ascending Sort

This command sorts numeric or text data in ascending order (from low value to high value). You can sort a column, a range of columns, or any data selection.

Descending Sort

This command sorts numeric or text data in descending order (from high value to low value). You can sort a column, a range of columns, or any data selection.

Create Series

This command fills a selection in a data window with an arithmetic, geometric, or time series of data. This command does not generate a series in a Text column.

Transpose

This command converts columns to rows and rows to columns. For example, if a selection contains four columns and 128 rows, it contains 128 columns and four rows after transposing the data.

Bin Data

This command provides a Histogram-like binning feature. The Bin Data dialog calculates the number of data points within each bin, based on the number of bins and the specified data range. The results can be exported in any one of three formats for plotting or for annotating a plot.

Statistics

This command displays the statistics for the frontmost data window. The Statistics dialog provides a split-screen display so that any two columns within the same data window can be compared.

Student t

This command performs a t-test on selected data. KaleidaGraph provides four types of tests: the single sample t-test, the paired t-test, the unpaired t-test with equal variance, and the unpaired t-test with unequal variance.

ANOVA

This command performs an analysis of variance (ANOVA) for the active data window. KaleidaGraph provides four types of tests: one way ANOVA, two way ANOVA, one way repeated measures ANOVA, and two way repeated measures ANOVA. In addition, four post hoc tests can be used to determine which treatments have an effect or which groups are different.

Wilcoxon

This command can be used to perform three nonparametric tests for the active data window: the Wilcoxon signed rank test, the Wilcoxon matched pairs test, and the Wilcoxon-Mann-Whitney test (also known as the Mann-Whitney test).

Kruskal-Wallis

This command performs a Kruskal-Wallis test to see if three or more groups are affected by a single factor.

Friedman

This command performs a Friedman test to see if a single group of individuals is affected by a series of three or more treatments.

Mask

This command masks one or more selected cells. Masked data is not plotted or used in curve fits. In the data window, masked data is shaded (with a stipple pattern) to make it stand out from unmasked data.

Unmask

This command unmasks any masked data in the selected region of the data window.

A.7 Macros Menu

The **Macros** menu lists three commands followed by a list of the macros stored in your Macros file. This section contains an overview of the macros provided with KaleidaGraph. For more information about a specific macro, refer to the online help.

Show Macros

This command can be used to add new macros to the menu, remove macros already present, and rearrange the macros. A maximum of 100 macros can be stored in this menu at once.

Note: The macros stored in the **Macros** menu are saved, by default, in a file called **KGMacros** (Windows) or **KG Macros** (Macintosh). This file is loaded each time you start KaleidaGraph. To open a custom file, use the **Open** or **Import** commands in the **File** menu.

Edit Macros

You can use this command to edit macros currently listed in the **Macros** menu.

Library

This command displays the contents of the current library. The library is a place where you can enter aliases or shorthand references for functions, variables, and constants. For more information on using the library, refer to Section 6.8.

Default Macros

This section describes the default macros that are supplied with KaleidaGraph. If these macros are not listed in the **Macros** menu, see the troubleshooting section in the online help for instructions on adding them to the menu.

- **Filter** - This macro eliminates outliers from a varying curve. It computes a moving-average curve with a user-defined window size as the reference curve for comparison against the data.
- **Smooth** - This macro computes a moving average of a curve. The resulting column of data is the average of the **n**-points surrounding the current data point. In this macro, **n** must be odd.
- **Simplify** - This macro masks any consecutive data points that do not differ from the last unmasked value by a certain error. Notice that this is the opposite of what the **Filter** macro does.
- **Invert Mask** - This macro **masks** unmasked data and **unmasks** masked data in a single data column.
- **Integrate - Area** - This macro finds the area under a curve, given a lower and upper limit and X-Y data points describing the curve. The area is found by calculating the sum of the trapezoids formed by the data points and, if necessary, interpolating the points forming the lower and upper limits.
- **Integral - Curve** - This macro finds the incremental area under a curve, given the X-Y data points describing the curve. The macro then generates a new curve. The area is the sum of the trapezoids formed by the data points and the reference value.
- **Derivative** - This macro finds the incremental slope of a curve, given the X-Y data points describing the curve. The macro then generates a new curve. The slope is calculated using the two-point slope formula.
- **Series** - This macro creates the series: x_o, $2*x_o$, $3*x_o$, ... (by default, x_o=1).
- **Unit Series** - This macro creates the series: x_o, $1+x_o$, $2+x_o$, ... (by default, x_o=1).
- **π Series** - This macro creates the series: $-\pi/2$, ..., $-\varepsilon$, $+\varepsilon$, ..., $+\pi/2$ (where $\varepsilon=\pi/$(# of selected rows)).
- **Sinc (5x)** - This macro creates a column containing sin(5x)/(5x).
- **abs(x)** - This macro takes the absolute value of the selected column and replaces the column in place.
- **Random#** - This macro generates a series of random numbers between a lower and upper limit. The resulting series of numbers is stored in column 0.

Other Macros

The following macros are not in the default Macros file, but are stored in the **Macro Source** folder as text files. You may load these files into the Macro Calculator using either the **File** > **Import** > **Program Text** command or the **Open** command in the program editor of the Macro Calculator.

- **View General** - This macro displays the results of the previous General curve fit.
- **View Covariances** - This macro displays the covariances of the previous General curve fit.
- **Paste Covariances** - This macro places the covariance/correlation matrix of the previous General curve fit in the selected region of the data window. A group of cells must be selected before executing this macro.

A.8 Format Menu

Plot Extras

This command controls the length and thickness of several plot elements. You can also set some of the options associated with Line and Column plots.

Plot Color

This command controls the frame, grid, interior, and background colors of the plot. If **None** is selected for the interior or background color, that portion of the plot is transparent.

X Label Format

This command controls the format of the labels on the X axis.

Y Label Format

This command controls the format of the labels on the Y axis.

Y2 Label Format

This command controls the format of the labels on the Y2 axis.

X2 Label Format

This command controls the format of the labels on the X2 axis.

Value Label Format

This command controls the format of any value labels on the plot. Value labels are displayed when you use the **Add Values** command (**Plot** menu) or add text error bars.

Equation Label Format

This command controls the format and precision of any curve fit equations on the plot.

Text Options

This command controls several options associated with text labels, including the default attributes used for the various labels in the plot window.

Polar Options

This command controls the grid type, data format, reference angle, and direction for Polar plots.

Curve Fit Options

This command controls several options associated with curve fits, including whether or not curve fits are drawn to the axis limits.

Probability Options

This command controls the type of distribution used in Probability plots. You can choose between a linear or normal distribution.

Histogram Options

This command controls the bin size, reference value, and number of extra bins in a Histogram plot.

Step Options

This command controls the placement of the steps when using a Step plot.

A.9 Object Menu

Align

You can use this command to align selected objects in the plot or layout window. When using one of the edge options, objects are aligned by their border centers, regardless of the thickness.

Up

This command moves one or more selected objects forward one level in the object order of a plot or layout window, changing the draw order of the objects.

Bring To Front

This command moves one or more selected objects in front of all other objects in a plot or layout window, changing the draw order of the objects.

Down

This command moves one or more selected objects backward one level in the object order of a plot or layout window, changing the draw order of the objects.

Send To Back

This command moves one or more selected objects behind all other objects in the plot or layout window, changing the draw order of the objects.

Group

This command combines two or more objects into a single object.

UnGroup

This command separates a previously grouped object into its original components.

Edit Object

You can use this command to edit text, arrows, polygons, Bezier curves, tables, and plot axes. If a different type of object is selected when you choose this command, a beep sounds to let you know that the selected item cannot be edited.

Edit Colors

You can use this command to edit the colors on the palette, save a custom color file, load a saved color file, or print the current palette. You can also reset the palette to its original defaults using this command.

Edit Arrow Heads

You can use this command to customize the height, length, and inset for each arrow style. You can also reset the arrowheads to their default settings using this command.

A.10 Curve Fit Menu

The Curve Fit menu lists the types of curve fits that can be applied within KaleidaGraph. For more information on a specific type of fit, refer to Section 10.1 or the online help. The following two commands are associated with the General curve fit. See Section 10.5 for more information on the use of these commands.

Edit General

This command can be used to add new fits to the submenu, remove fits already present, and rearrange fits. A maximum of 16 curve fits can be stored in this submenu.

Library

This command displays the contents of the current library. One of the main reasons for using the library is to form aliases or shorthand references for functions, variables, and constants. This is particularly useful for defining sections of a formula that exceed the 256 character limit in the Curve Fit Definition dialog for the General curve fit. Section 10.5.7 provides some examples for defining curve fit equations in the library and calling the definitions from the Curve Fit Definition dialog.

Note: Do not use the library to open text files containing saved curve fit definitions. These should be opened from within the Curve Fit Definition dialog.

A.11 Layout Menu

New Position

This command places an empty cell in the layout window. To place a plot into an empty cell position, click the cell and select a plot from the **Select Plot** submenu.

Select Plot

You can use this command to place plots into the layout window. If you want to replace a plot or empty cell in the layout, click the item to be replaced and choose a plot from the **Select Plot** submenu.

Arrange Layout

You can use this command to arrange the plots in the layout window by specifying the number of rows and columns you want to have. The layout window is divided into equal sections, with each plot occupying one of the divisions. The plots are arranged in the order they were added to the layout window, starting at the top of the page and working down from left to right.

Set Background

You can use this command to set a background and border for the layout window.

A.12 Windows Menu

Hide Window

This command can be used to hide an open window. Hidden data and plot windows can be redisplayed using the **Show Data** and **Show Plot** commands.

Show Data

This command can be used to bring any data window to the front.

Show Plot

This command can be used to bring any plot window to the front.

Show Layout

This command brings the selected layout window to the front. Each time you open or create a layout, it is added to the **Show Layout** submenu.

If a default layout has been saved (in the KG Layout settings file), this submenu will always contain an item called **KG Layout**. It is not possible to permanently add layouts to this submenu. They are only remembered until you quit the program.

Clipboard

This command displays the Clipboard window, which contains the last item that was cut or copied. The contents of the Clipboard remain intact until another selection is cut or copied. The contents also remain when switching between programs. This allows you to easily transfer selections between programs.

Macro Calculator

This command displays the Macro Calculator. The calculator combines the RPN (Reverse Polish Notation) programming language with support for 1000 program steps to give you a number of ways to manipulate and transform data. Refer to the online help or the Macro Calculator online manual (on the CD-ROM) for more information on the Macro Calculator.

Formula Entry

This command displays the Formula Entry window. See Chapter 6 for information on using the Formula Entry window.

Plot Script

This command can be used to create a number of plots at once that all have the same characteristics. See Section 7.4.3 for information on using the Plot Script command.

Show All Plots

This command displays all open plot windows at once.

Hide All Plots

This command hides all open plot windows at once.

Close All Plots

This command closes all open plot windows at once. A dialog appears if any of the plots contain unsaved changes.

Close All Data

This command closes all open data windows at once. A dialog appears if any of the data windows contain unsaved changes.

Menu Shortcuts

This command can be used to add or change keyboard shortcuts for commands in KaleidaGraph. These shortcuts are saved as part of the Macros file.

Rename

This command can be used to edit the name of the data, plot, or layout window, without having to save it to disk.

KaleidaGraph's Settings Files

Appendix B

KaleidaGraph maintains four settings files to save the modifications made to the program. These files are opened each time you start KaleidaGraph, provided they are in the correct location and have the correct file name. If a settings file has a different name or is located in a different directory, the file is skipped.

B.1 Style Files

Style files contain the settings for most of KaleidaGraph's dialogs. Any changes made to the original defaults, such as text attributes, plot layout, and dialog settings, are saved in the Style file.

A default Style file is automatically opened each time KaleidaGraph is started. This way you can save the settings and have them opened the next time you use the program. The name and location of this file differs between the Windows and Macintosh versions of KaleidaGraph.

- **Windows:** The default settings file is named **KGStyle**. This file must be in the same directory as KaleidaGraph for it to be opened.
- **Macintosh:** The default settings file is named **KG Style**. This file must be in the KaleidaGraph Preferences folder or in the same folder as KaleidaGraph for it to be opened. The KaleidaGraph Preferences folder is located in the Preferences folder (in the users Library folder under OS X or in the System Folder for OS 9 and earlier).

Custom Style files can be created by choosing **File** > **Export** > **Style**. These files can be opened by choosing **Open** or **Import** > **Style** from the **File** menu. All of the settings change to reflect the contents of the new Style file.

B.2 Macros Files

Macros files contain the macros in the **Macros** menu, the contents of the library, any formulas assigned to the function buttons in Formula Entry, the settings in the Text File Input Format dialog, the keyboard shortcuts, and the curve fits in the **General** submenu.

A default Macros file is automatically opened each time KaleidaGraph is started. This way you can save the settings and have them opened the next time you use the program. The name and location of this file differs between the Windows and Macintosh versions of KaleidaGraph.

- **Windows:** The default settings file is named **KGMacros**. This file must be in the same directory as KaleidaGraph for it to be opened.

- **Macintosh:** The default settings file is named **KG Macros**. This file must be in the KaleidaGraph Preferences folder or in the same folder as KaleidaGraph for it to be opened. The KaleidaGraph Preferences folder is located in the Preferences folder (in the users Library folder under OS X or in the System Folder for OS 9 and earlier).

Custom Macros files can be created by choosing **File** > **Export** > **Macros**. These files can be opened by choosing **Open**, **Import** > **New Macros**, or **Import** > **Append Macros** from the **File** menu. All of the affected menus and options change to reflect the contents of the new Macros file.

B.3 Layout Files

Layout files contain the cell positions and background, as well as any text or graphic objects in the active layout window. These files do not save the plots as part of the layout file. Only the positions of the cells are saved.

A default Layout file is automatically opened each time KaleidaGraph is started. This way you can save a layout and have it opened the next time you use the program. You can access the default layout by choosing **Windows** > **Show Layout** > **KG Layout**. The name and location of this file differs between the Windows and Macintosh versions of KaleidaGraph.

- **Windows:** The default settings file is named **KGLayout**. This file must be in the same directory as KaleidaGraph for it to be opened.
- **Macintosh:** The default settings file is named **KG Layout**. This file must be in the KaleidaGraph Preferences folder or in the same folder as KaleidaGraph for it to be opened. The KaleidaGraph Preferences folder is located in the Preferences folder (in the users Library folder under OS X or in the System Folder for OS 9 and earlier).

Custom Layout files can be created by choosing **Save Layout** or **Export** > **Layout** from the **File** menu when a layout window is active. These files can be opened by choosing **Open** or **Import** > **Layout** from the **File** menu. The layout opens into a new window, displaying any items that were saved as part of the layout.

B.4 Script Files

Script files contain the settings and group selections from the Plot Script window.

If one has been created, a default Script file is automatically opened each time KaleidaGraph is started. This way you can save the plot script and have it opened the next time you use the program. The name and location of this file differs between the Windows and Macintosh versions of KaleidaGraph.

- **Windows:** The default settings file is named **KGScript**. This file must be in the same directory as KaleidaGraph for it to be opened.
- **Macintosh:** The default settings file is named **KG Script**. This file must be in the KaleidaGraph Preferences folder or in the same folder as KaleidaGraph for it to be opened. The KaleidaGraph Preferences folder is located in the Preferences folder (in the users Library folder under OS X or in the System Folder for OS 9 and earlier).

Custom Script files can be created by choosing **Save Script** or **Export** > **Script** from the **File** menu when the Plot Script window is active. These files can be opened by choosing **Open** or **Import** > **Script** from the **File** menu. The Plot Script window changes to reflect the contents of the new Script file.

Toolbox Shortcuts

Appendix C

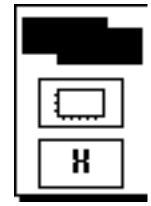
This chapter lists the shortcuts available for selecting plot tools and making changes to plot elements directly from the toolbox. The online help contains information about the other kinds of shortcuts available in KaleidaGraph.

C.1 Shortcuts for Selecting Plot Tools

The table below lists the keyboard shortcuts available for selecting the various plot tools.

To:	Press:
Select the Selection Arrow	Spacebar
Select the Text tool	T
Select the Line/Arrow tool	L
Select the Box tool	1
Select the Rounded-Rectangle tool	2
Select the Oval tool	3
Select the Polygon tool	4
Select the Arc tool	5
Select the Bezier curve tool	6
Activate whatever Object tool is selected in the pop-up menu	O
Select the Table tool	A
Select the Eraser tool	E
Select the Identify tool	I
Select the Data Selection tool	S
Select the Zoom Selection tool	Z
Activate the last tool selected	Ctrl (Windows) or ⌘ (Macintosh)

C.2 Changing Axis Options from the Toolbox



Before using the toolbox to change the axis options, an axis must be selected by clicking twice on the desired axis. The bottom portion of the toolbox will display the icons shown to the right.

C.2.1 Exchanging and Removing Axes

To exchange or remove axes:

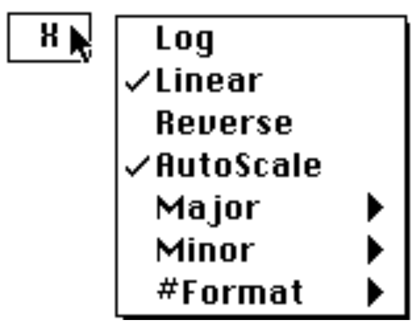
1. Select the desired axis.
2. Click the first Axis Options icon to display the pop-up menu shown below.

3. Turn on **Swap X & Y** to exchange the X and Y axes or **No Axis** to remove the selected axis and its tick marks.

C.2.2 Axis Scaling, Reversing, and Auto Limits

To change scaling, reverse an axis, or return to auto limits:

1. Select the desired axis.
2. Click the second Axis Options icon to display the pop-up menu shown below.

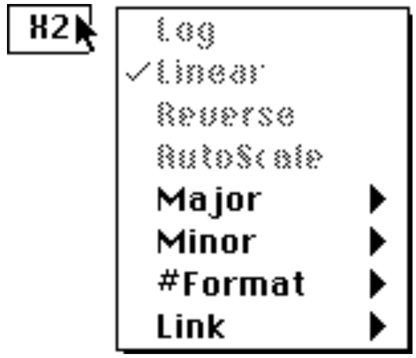


3. Choose **Log** or **Linear** to change the axis scaling, turn on **Reverse** to reverse the axis, or turn on **AutoScale** to return to the limits originally calculated by KaleidaGraph. The rest of the options are discussed in Sections C.2.4–C.2.6.

C.2.3 Linking Axes

To link the X2 or Y2 axes:

1. Select the X2 or Y2 axis.
2. Click the second of the two Axis Options icons to display the pop-up menu shown below.



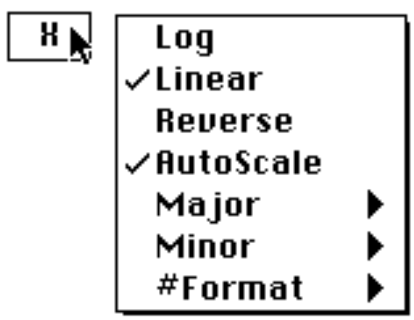
3. Choose **Link** to display the submenu shown below.

4. Choose the type of link to be used for the selected axis.

C.2.4 Tick Marks

To change the display of tick marks or let KaleidaGraph calculate the number of ticks:

1. Select the appropriate axis.
2. Click the second of the two Axis Options icons to display the pop-up menu shown below.



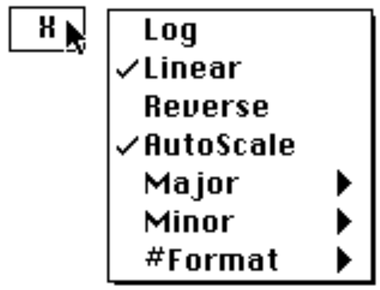
3. Choose **Major** or **Minor** to display a submenu that contains the tick mark options. The submenus for Major and Minor are identical.

4. Use the **Inside** and **Outside** commands to change how the ticks are displayed. Turn on **Auto** to have KaleidaGraph calculate the number of tick marks displayed on the selected axis.

C.2.5 Grid Lines

To change how the grid lines are displayed:

1. Select the appropriate axis.
2. Click the second of the two Axis Options icons to display the pop-up menu shown below.



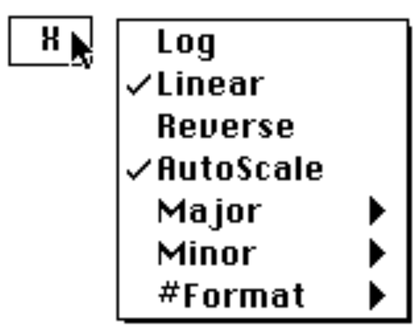
3. Choose **Major** or **Minor** to display a submenu that contains the grid line options. The submenus for Major and Minor are identical.

4. Turn on **Grid** to display a gray (dotted) line or **Solid** to display a solid line. If both items are turned off, grid lines are not displayed.

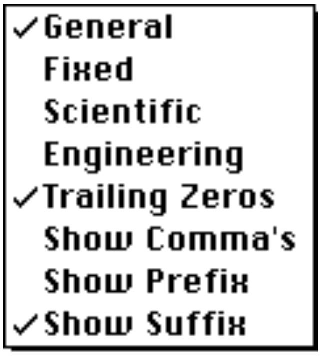
C.2.6 Axis Labels

To change the display of the axis labels:

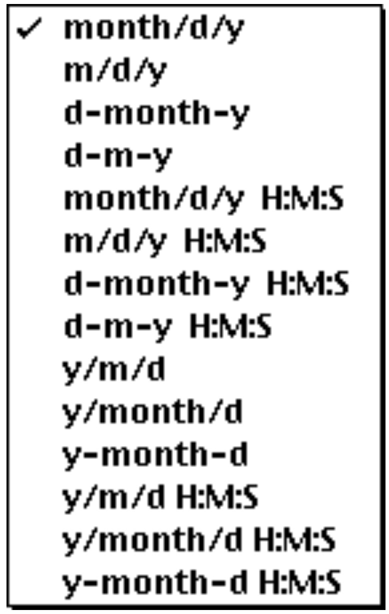
1. Select the desired axis.
2. Click the second of the two Axis Options icons to display the pop-up menu shown below.



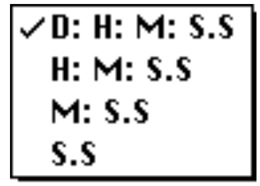
3. Choose **# Format** to display a submenu that contains the display options. The options in the submenu depend on the format of the data plotted along the selected axis. The following three figures show the submenus that appear for Numeric (Float, Double, or Integer), Date, and Time formats.



Numeric formats



Date formats



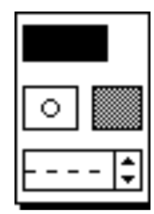
Time formats

Using these options, you can change the format of the axis labels as well as display trailing zeros, commas, a prefix, or a suffix. Changes are based on the current settings in the appropriate Label Format dialog (**Format** menu).

4. Make the desired changes to the submenu.

C.3 Changing Plot Style from the Toolbox

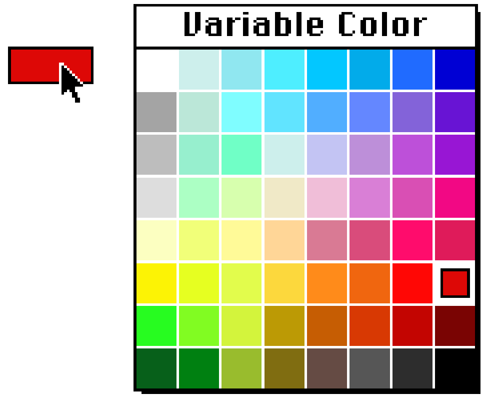
Before using the toolbox to change the variable options, a plot symbol must be selected in the legend by clicking twice on the symbol. The bottom portion of the toolbox will display the Plot Style icons shown to the right.



C.3.1 Variable Color

To change the variable color from the toolbox:

1. Select a plot symbol in the legend.
2. Click the Variable Color icon to display a color palette containing the current color choices.

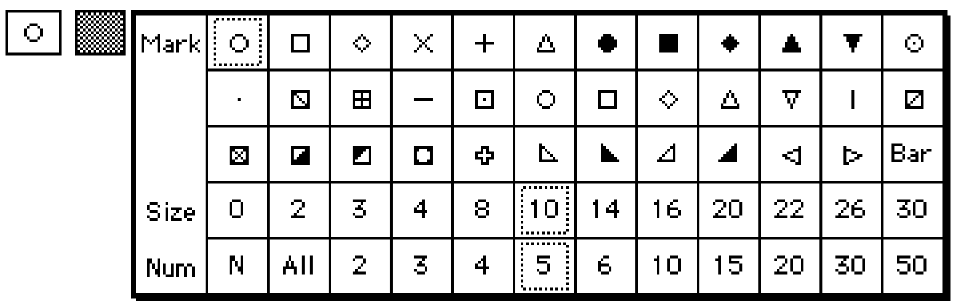


3. Select a different color from the palette.

C.3.2 Marker Type, Size, or Number of Markers

To change the marker type, size, or number of markers displayed:

1. Select a plot symbol in the legend.
2. Click the Marker icon to display the pop-up menu shown below.

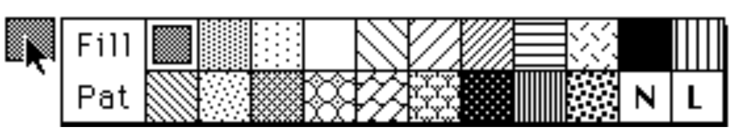


3. Select a different marker type, size, or number from the pop-up menu.

C.3.3 Fill Pattern

To change the fill pattern of the variable:

1. Select a plot symbol in the legend.
2. Click the Fill Pattern icon to display the pop-up menu shown below.

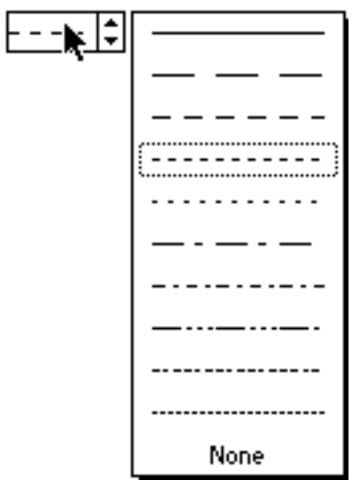


3. Select a different fill pattern from the pop-up menu.

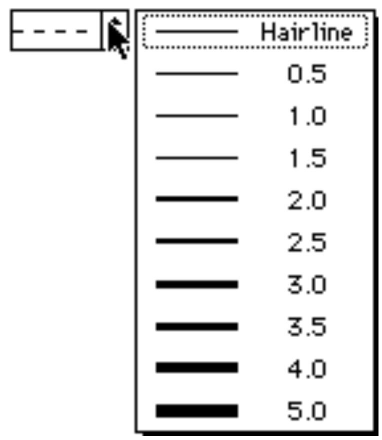
C.3.4 Variable Line Style or Width

To change the variable's line style or width:

1. Select a plot symbol in the legend.
2. To change the line style, click the left portion of the Style icon and select a different line style. The active line style is enclosed in a dotted box.
3. To change the line width, click the right portion of the Style icon and select a different line width. The active line width is enclosed in a dotted box.



Line Style pop-up menu



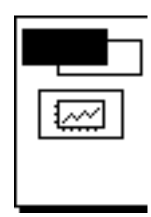
Line Width pop-up menu

C.4 Changing Plot Color from the Toolbox

Before using the toolbox to change the plot colors, either the plot frame or an axis must be selected. If the frame is selected, you can edit the interior and background colors. If an axis is selected, you can change the frame and grid colors.

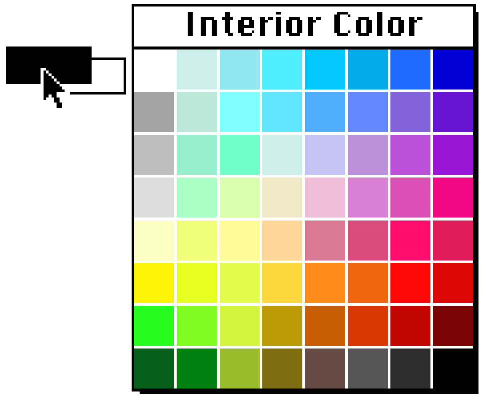
C.4.1 Interior and Background Colors

To change the interior or background colors from the toolbox:



1. Click any of the axes once to select the plot's frame. The bottom portion of the toolbox will display the icons shown to the right.

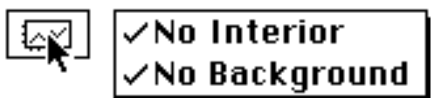
   The Color icon controls the interior and background color. It consists of two sections: an interior color icon (upper left) for setting the plot's interior color, and a background icon (lower right) for setting the background color.
2. Click one of the icons in the palette to display a color palette containing the color choices.



3. Select a different color from the palette.

To make the interior or background of the plot transparent:

1. Click any of the axes once to select the plot's frame. The bottom portion of the toolbox will display two icons.
2. Click the bottom icon to display the following pop-up menu.



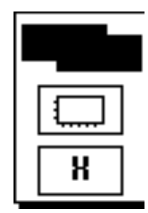
3. Turn on either of the commands in this pop-up menu to make that portion of the plot transparent.

C.4.2 Frame and Grid Colors

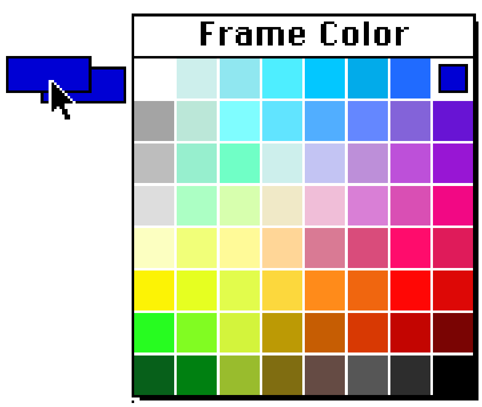
To change the frame or grid colors from the toolbox:

1. Select one of the plot's axes. The bottom portion of the toolbox will display the icons shown to the right.

   The first icon controls the frame and grid color. It consists of two sections: a frame color icon (upper left) for setting the plot's frame color, and a grid icon (lower right) for setting the grid color.



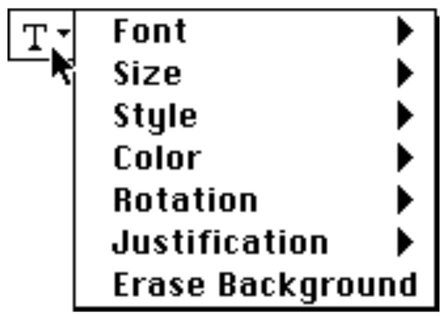
2. Click one of the icons in the palette to display a color palette containing the color choices.



3. Select a different color from the palette.

C.5 Changing Text Attributes from the Toolbox

Holding down the mouse button after clicking the Text tool displays the pop-up menu shown below. You can use this pop-up menu to change the attributes of selected text objects directly from the toolbox. If a selection is not present in the plot window, the default attributes can be set for any text labels created with the Text tool



To change the attributes of a text label directly from the toolbox:

1. Select the label(s) to be modified.
    - Individual labels are selected by clicking the label once with either the Text tool or Selection Arrow.
    - Legend text is selected by clicking the label once with the Text tool.
    - Axis or value labels are selected as a group by clicking any of the labels in the group with the Selection Arrow. They may also be selected individually by clicking with the Text tool.
    - Multiple text labels can be selected by pressing **Shift** and clicking each label with the Selection Arrow.
    - All of the text labels can be selected at once by choosing **Edit** > **Select All**. Everything in the plot is selected, but changes made using the Text tool only affect text objects.
2. Click the Text tool and hold down the mouse button to display the pop-up menu.
3. Choose any of the commands in the pop-up menu. Choosing any of the commands except **Erase Background** displays a submenu that contains the options for that item.
4. Continue making selections from the pop-up menu until all of the labels are set up as desired.

To set the default attributes:

1. Verify that a selection is not present in the plot window.
2. Click the Text tool and hold down the mouse button to display the pop-up menu.
3. Choose any of the commands in the pop-up menu. Choosing any of the commands except **Erase Background** displays a submenu that contains the options for that item.
4. Make the desired changes in the pop-up menu.

Any text objects that you create with the Text tool have these attributes. The attributes that you set are saved as part of the Style file. You can also use the **Text Options** command (**Format** menu) to set the defaults for standard text labels.

Reference Information

Appendix D

This chapter provides information on the methods and equations used in different areas of the program.

D.1 Box Plot Definition

Each box encloses 50% of the data with the median value of the variable displayed as a line. The top and bottom of the box mark the limits of ± 25% of the variable population. The lines extending from the top and bottom of each box mark the minimum and maximum values within the data set that fall within an acceptable range. Any value outside of this range, called an outlier, is displayed as an individual point.

Note: Outliers are used in the calculations for the Box plot. Removing outliers changes the entire plot. You do not have any control over the marker used to represent outliers.

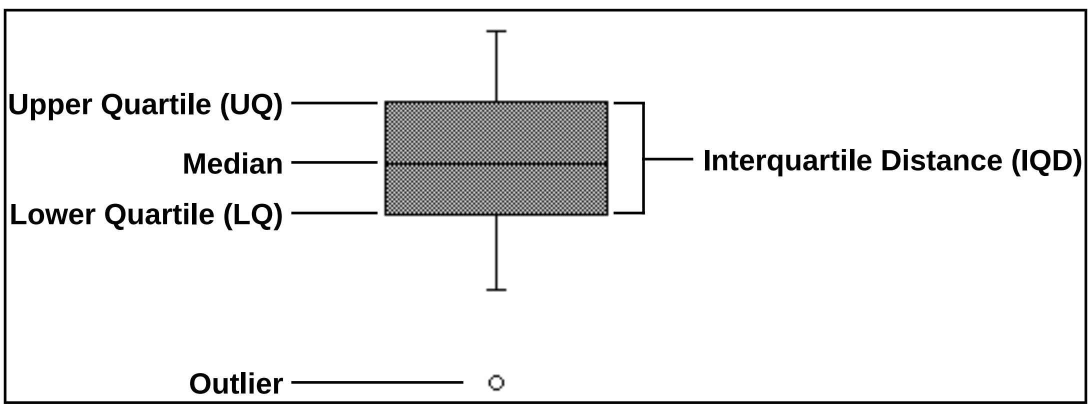


Figure D-1 Box plot terms

- **Median -** The data value located halfway between the smallest and largest values.
- **Upper Quartile (UQ)** - The data value located halfway between the median and the largest data value.
- **Lower Quartile (LQ) -** The data value located halfway between the median and the smallest data value.
- **Interquartile Distance (IQD) -** The distance between the Upper and Lower Quartiles (UQ - LQ).
- **Outliers -** Points whose value is either:

  greater than UQ + 1.5 * IQD or less than LQ - 1.5 * IQD

D.2 Statistics Equations

n = number of data points
x_i = current data value (from 1 to n)
$\bar{x}$ = mean of the data

Mean

$$\bar{x} = \frac{1}{n}\sum_{i=1}^{n} x_i$$

RMS (Root Mean Square)

$$\left[\frac{1}{n}\sum_{i=1}^{n} x_i^2\right]^{1/2}$$

Standard Deviation

$$\sqrt{\frac{\sum_{i=1}^{n} x_i^2 - n(\bar{x})^2}{n-1}}$$

Standard Error

$$\frac{\text{Standard Deviation}}{\sqrt{n}}$$

Variance

$$\frac{\sum_{i=1}^{n} x_i^2 - n(\bar{x})^2}{n-1}$$

Skewness

$$\frac{\frac{1}{n}\sum_{i=1}^{n}(x_i-\bar{x})^3}{\left(\frac{1}{n}\sum_{i=1}^{n}(x_i-\bar{x})^2\right)^{3/2}}$$

Kurtosis

$$\frac{\frac{1}{n}\sum_{i=1}^{n}(x_i-\bar{x})^4}{\left(\frac{1}{n}\sum_{i=1}^{n}(x_i-\bar{x})^2\right)^{2}} - 3$$

D.3 Student t-Test Equations

$\bar{x}$ = mean of the data N = number of data points x_i and y_i = current data value (from 1 to N)

t value (single sample)

$$t = \frac{\bar{x} - \text{Test Value}}{\text{Standard Error}(x)}$$

t value (paired data)

$$t = \frac{\overline{x_A} - \overline{x_B}}{s_D}$$

$$s_D = \sqrt{\frac{Var(x_A) + Var(x_B) - 2Cov(x_A, x_B)}{N}}$$

$$Cov(x_A, x_B) \equiv \frac{1}{N-1}\sum_{i=1}^{N}(x_{A_i} - \overline{x_A})(x_{B_i} - \overline{x_B})$$

t value (unpaired data with equal variance)

$$t = \frac{\overline{x_A} - \overline{x_B}}{s_D}$$

$$s_D = \sqrt{\frac{\sum_{i \in A}(x_i - \overline{x_A})^2 + \sum_{i \in B}(x_i - \overline{x_B})^2}{N_A + N_B - 2}\left(\frac{1}{N_A} + \frac{1}{N_B}\right)}$$

t value (unpaired data with unequal variance)

$$t = \frac{\overline{x_A} - \overline{x_B}}{\sqrt{Var(x_A)/N_A + Var(x_B)/N_B}}$$

Correlation

$$\frac{\sum_i (x_i - \bar{x})(y_i - \bar{y})}{\sqrt{\sum_i (x_i - \bar{x})^2}\sqrt{\sum_i (y_i - \bar{y})^2}}$$

F value

$$\frac{\text{Larger Variance}}{\text{Smaller Variance}}$$

D.4 Statistical References

Wilcoxon Signed Rank, Wilcoxon-Mann-Whitney, Kruskal-Wallis, and Friedman Tests

Siegel, S. and Castellan, N. J. Jr. Nonparametric Statistics for the Behavioral Sciences. McGraw-Hill, New York (1988).
Zar, J. H. Biostatistical Analysis. Prentice-Hall, Upper Saddle River, NJ (1999).

One Way ANOVA, Two Way ANOVA, and Tukey HSD Tests

Zar, J. H. Biostatistical Analysis. Prentice-Hall, Upper Saddle River, NJ (1999).
Neter, J., Kutner, M. H., Nachtsheim, C. J., and Wasserman, W. Applied Linear Statistical Models. McGraw-Hill, New York (1996).

Wilcoxon Matched Pairs and Dunnett Tests

Zar, J. H. Biostatistical Analysis. Prentice-Hall, Upper Saddle River, NJ (1999).

One Way Repeated Measures ANOVA, Two Way Repeated Measures ANOVA, Bonferroni, and Holm Tests

Neter, J., Kutner, M. H., Nachtsheim, C. J., and Wasserman, W. Applied Linear Statistical Models. McGraw-Hill, New York (1996).

D.5 Curve Fit References

General Curve Fit

Press, W. H., Flannery, B. P., Teukolsky, S. A., and Vetterling, W. T. *Numerical Recipes in C.* Cambridge University Press, New York (1988).

Linear, Polynomial, Exponential, Logarithmic, and Power Curve Fits

Beyer, W. H. *Standard Mathematical Tables*. CRC Press, Cleveland (1976).

Smooth Curve Fit

Stineman, R. W. A consistently well-behaved method of interpolation. *Creative Computing*, July (1980).

Weighted Curve Fit

Chambers, J. M., Cleveland, W. S., Kleiner, B., and Tukey, P. A. *Graphical Methods for Data Analysis*. Duxbury Press, Boston (1983).

Cubic Spline Curve Fit

Press, W. H., Flannery, B. P., Teukolsky, S. A., and Vetterling, W. T. *Numerical Recipes in C.* Cambridge University Press, New York (1988).

Interpolate Curve Fit

Stineman, R. W. A consistently well-behaved method of interpolation. *Creative Computing*, July (1980).

D.6 Pearson's R and Chi Square Equations

χ^2 = Chi Square — y_i = actual value — x_i = calculated value

σ_i = weight — $\bar{y}$ = mean of actual values — $\bar{x}$ = mean of calculated values

Pearson's R (Least Squares curve fits)

$$R = \frac{\sum_i (x_i - \bar{x})(y_i - \bar{y})}{\sqrt{\sum_i (x_i - \bar{x})^2}\ \sqrt{\sum_i (y_i - \bar{y})^2}}$$

Pearson's R (General curve fit)

$$R = \sqrt{1 - \frac{\chi^2}{\sum_i \sigma_i (y_i - \bar{y})^2}}$$

Chi Square

$$\chi^2 = \sum_i \left(\frac{y_i - x_i}{\sigma_i}\right)^2$$

Glossary

active window: The frontmost window on the desktop. The active window's title bar is highlighted.

Align tool: A tool used to align a group of selected objects in the plot or layout window.

analysis of variance (ANOVA): A statistical test that compares the means of two or more groups for a statistically significant difference. KaleidaGraph provides four ANOVA tests: one way, two way, one way repeated measures, and two way repeated measures.

anchor point: An endpoint on a segment of a Bezier curve. Anchor points contain direction points and tangent lines. The position of the anchor points and their direction points determine the shape of the curve segment.

anchor tick: The anchor tick specifies the value from which all other major tick marks are calculated. When placing major ticks at a specific interval, the interval is counted from the value of the anchor tick. This tick does not have to be at the edge of the axis; it can be placed anywhere.

AppleScript: A system-level scripting language available on the Macintosh. It is used to automate processes within programs that support AppleScript.

Apple Event: A system-level command sent between programs. The command tells the destination program to perform a specific operation.

Arc tool: A tool used to draw arcs in the plot or layout window.

axis: One of the horizontal or vertical lines in a plot along which values or categories are plotted.

axis labels: Labels that display the values associated with the major tick marks on the plot.

axis titles: Titles that are used to identify which variable is plotted along an axis. The titles are taken from the column names being plotted. If multiple variables are plotted on the same axis, the title comes from the first column being plotted.

background color: The color KaleidaGraph uses to represent white pixels in fill and pen patterns.

Bezier curve: A curve made up of one or more segments. Each segment is defined by four points: two anchor points, which determine the endpoints, and two direction points, which determine its shape.

Boolean: A value that results from evaluating an expression. The value can be either true (non-zero) or false (zero).

Box tool: A tool used to draw squares and rectangles in the plot or layout window.

cell: The basic unit of a data window where values are entered and stored.

Clipboard: A window that contains the last selection that was cut or copied. The contents remain until the next time you cut or copy a selection.

column: In the data window, a vertical line of cells. A maximum of 1000 columns are available. In a column plot, columns are the vertical bars that represent the data values.

column title: The name identifying a column in the data window.

column numbers: The numbers located to the left of the column titles, used to identify each column. The column numbers are used by Formula Entry and the Macro Calculator during calculations. These numbers change when a selection is made in the data window.

Command key: This key (⌘) enables you to use keyboard shortcuts to choose commands on a Macintosh computer.

comments: Notes added to a macro or function to make it easier to follow.

Ctrl key: This key enables you to use keyboard shortcuts to choose commands under Windows.

curve fitting: The process of applying one of 10 different types of fits to a variable in the plot window. The fits can be used to find an equation for the data, to smooth the curve between the points, or to see how closely the data compares to a specific equation.

Data Selection tool: A tool used to select a portion of the data on a plot for further plotting and analysis.

data window: A window containing a spreadsheet used to enter and store data for plotting and analysis.

default settings: Settings that KaleidaGraph uses unless you specify different settings. You can save the current settings as default settings by choosing **File** > **Export** > **Style**.

degrees of freedom: This is a measure of the sample size, which affects the power of the statistical test.

delimiter: A character used to separate data within a text file; for example, tabs, spaces, or commas.

Dynamic Data Exchange (DDE): A feature that is used to create a link between data in a source document and a KaleidaGraph for Windows data window. If the data in the source document changes, the data window updates to reflect the changes.

Eraser tool: A tool used to delete text and other objects (such as lines, boxes, and pictures) from the plot or layout window.

error bars: Vertical or horizontal lines on a plot that illustrate the statistical error of the data.

extract: To retrieve the original data from a plot which was saved with data. You can use either the **Extract Data** command or the Find Data button ( ) to display the plot's data.

fill pattern: A pattern chosen from the pattern palette to fill a selected object.

Find Data button: This button ( ) is located in the upper-right corner of the plot window. Clicking this button displays data windows referenced by the plot. If the data is archived in the plot, the data windows are extracted and displayed.

foreground color: The color KaleidaGraph uses to represent black pixels in fill and pen patterns.

format: How the data in a column is displayed. Each column has its own format, which controls the way its values are displayed and used in a plot.

formula: A sequence of values, column references, memory references, functions, or operators that generate data from existing data.

Formula Entry: A window that is used to enter and execute algebraic formulas which operate on the active data window.

formula script: A program written in the Posted Note of the Formula Entry window. Formula scripts can automate the process of opening data, creating plots, and exporting plots to disk or to a printer.

frame: The line surrounding a bar, column, pie wedge, legend, or object in the plot or layout window.

Gaussian equation: An equation of the form **m1+m2*exp(–(m0–m3)*(m0–m3)/(m4*m4))**. A good starting value for m1 is the value that the tails of the curve approach for large values of |x|. The maximum of the curve can be used for m2. The mean of the data is a good guess for m3. The bell-width is a good guess for m4.

grid lines: Horizontal or vertical lines that may be displayed at the major and minor tick marks.

group: To combine two or more objects so that they act as a single object.

handles: Points at the corners of an object's boundary which can be used to modify the size and shape of the object.

Home button: This button ( ) is located in the upper-left corner of the data window. Click this button to return the data window to its origin (row 0, column 0).

Identify tool: A tool used to find coordinates of points on a plot.

indexed data: A type of data format where a factor column contains the group names, and one or more columns contain the corresponding data values for each group.

insertion point: The blinking vertical line that appears whenever you click within a cell or a text dialog.

justification: The method used to align text and labels. You can left, center, or right justify text and labels.

layer: A separate part of the plot window used to organize objects. You can create plots with several layers, with each layer containing different objects.

Layout file: A settings file that contains the cell positions and background, as well as any text or graphic objects in the active layout window.

Layout window: A window used to place multiple plots on a single page. Text, tables, and other objects may be placed anywhere in the layout window. The page may be copied to the Clipboard, saved in a variety of formats, or printed.

legend: The portion of the plot that contains symbols and labels used to identify each variable in a plot.

library: Used to create and store shorthand references for functions, curve fits, constants, and variables.

Line/Arrow tool: A tool used to add lines or arrows to the plot or layout window.

macro: An RPN program that performs a series of calculations on the active data window.

Macro Calculator: An RPN calculator which can be used to write and execute macros that operate on the active data window.

Macros file: A settings file that contains the macros in the **Macros** menu, the contents of the library, any formulas assigned to the function buttons in Formula Entry, the settings in the Text File Input Format dialog, the keyboard shortcuts, and the curve fits in the **General** submenu.

major ticks: Marks that display the major divisions along an axis.

marker: A symbol which represents a single data point in the plot window.

Metafile: A Windows file format used by graphics programs to transfer documents to different programs.

minor ticks: Minor ticks display the divisions between the major tick marks.

nonparametric tests: Statistical tests that do not require the data be normally distributed. These tests operate on the ranks of the values. Examples of nonparametric tests include the Wilcoxon, Kruskal-Wallis, and Friedman tests.

object handles: Small squares that appear when an object is selected. The object can be resized by dragging these handles.

Object Linking and Embedding (OLE): A feature that can be used to embed a KaleidaGraph for Windows plot in another application. To edit an embedded plot, double-click the plot to start KaleidaGraph and make changes.

Oval tool: A tool used to draw circles and ovals in the plot or layout window.

overlay: To place multiple plots on top of each other in the layout window to get the appearance of a single plot.

P value: This value is a probability, with a value that ranges from zero to one. It is used to determine if there is a statistically significant difference in the groups being analyzed. If this value is below a certain level (usually 0.05), the conclusion is that there is a difference between the groups.

parametric tests: Statistical tests that assume the samples are drawn from populations that are normally distributed. Examples of parametric tests include the t-test and analysis of variance (ANOVA).

pen pattern: A pattern given to the line segments of an object.

PICT: A Macintosh file format used by graphics programs to transfer documents to different programs.

plot script: A series of data selections from any number of data windows that are used to create multiple plots at once. A plot script is created in the Plot Script window.

plot title: The name of the graph. By default, its name is taken from the name of the data window.

plot window: Any window that contains a plot. Plots are created by choosing a plot type from the Gallery menu. When the plot window is active, certain menus are displayed that only affect the plot window.

Polygon tool: A tool used to draw an object with any number of sides in the plot or layout window.

post hoc tests: Post hoc tests (also known as multiple comparisons) are used to isolate the differences between groups when performing a one way or one way repeated measures ANOVA. KaleidaGraph provides the following post hoc tests: Tukey HSD, Bonferroni, Holm, and Dunnett.

Posted Note: A text editor available in both the data and Formula Entry windows. In the data window, it is used to enter information about the current data set. In Formula Entry, it can be used to write a multi-line formula or a script to automate the plotting process.

raw data: A type of data format where the data values for each group are placed in separate data columns.

residual: The difference between an actual data value and a mean or other estimated value.

Rounded Rectangle tool: A tool used to draw squares and rectangles which have rounded corners.

row: A horizontal line of cells. A maximum of 1 million rows can be displayed in the data window.

row numbers: The column of numbers that appear on the left side of every data window.

Script file: A settings file that contains the settings and group selections from the Plot Script window.

Selection Arrow: A tool used to select, move, and resize objects. This tool is also used to select an object's properties from pop-up menus.

shift-click: To click the mouse button while pressing **Shift**. This technique is used to select multiple objects or deselect individual objects.

sort key: A column that contains the values you want to sort by. You can specify three sort keys at a time for each sort.

Style file: A settings file that contains the settings for most of KaleidaGraph's dialogs. Any changes made to the original defaults, such as text attributes, plot layout, and dialog settings, are saved in the Style file.

Table tool: A tool used to create tables of data within the plot or layout window.

template: Any plot that is used as a base for creating new plots. The new plot has the same characteristics as the template.

Text tool: A tool used to create and edit labels in the plot or layout window.

tick marks: The lines which indicate the major or minor increments along an axis.

toolbox The floating window which displays the tools available to create and edit items in the plot or layout window.

ungroup: To separate a grouped object into its original components.

Update Plot button: This button ( ) is located in the upper-right corner of the data window. Click this button to force an immediate update of the plot that is currently linked to it.

X axis: The horizontal axis in a plot, which usually represents the independent variable.

Y axis: The vertical axis in a plot, which usually represents the dependent variable.

Zoom Selection tool: A tool that can be used to quickly change the axis limits.

Index

B

C

D

E

F

G

H

I

P

Q

R

S

X

Y

Z