Tutorials for Origin 9.0

Table of Contents

1 Introduction Tutorial

Welcome to the Origin 9.0 Tutorial Guide

The material in this guide is designed to provide both new and advanced users with specific instructions on how to perform the most commonly used and powerful features in Origin. The tutorials in this guide handle many specific tasks, so we recommend you look through them at your leisure as you find you need some pointers on specific operations.

A general note before proceeding:

> You will find references to buttons found on various toolbars in many of the tutorials in this guide. These buttons are shortcuts to menu commands. If you don't see the button referenced in a tutorial, it may simply not be shown in your workspace. To open a toolbar, select **View: Toolbars**, click on the checkbox next to the desired toolbar, and then click **Close**.

2 Origin 9

The tutorials in this chapter cover some of the new features in Origin 9. For a full list of features please browse our help files

Topics covered in this section:

1. Data Filter
2. Stacked 3D Surface Plots
3. Parametric Surface with Colormap from Data
4. Colormap from Second Matrix
5. Scatter Matrix
6. IIR Filter

2.1 Data Filter

2.1.1 Summary

The **Data Filter** is a column-based tool to reduce rows of worksheet data, and consequently also hide the undesired rows for relevant data analysis and graphing. Three data formats are supported: numeric, text and date/time.

Minimum Origin Version Required: Origin 9.0 SR0

2.1.2 What you will learn

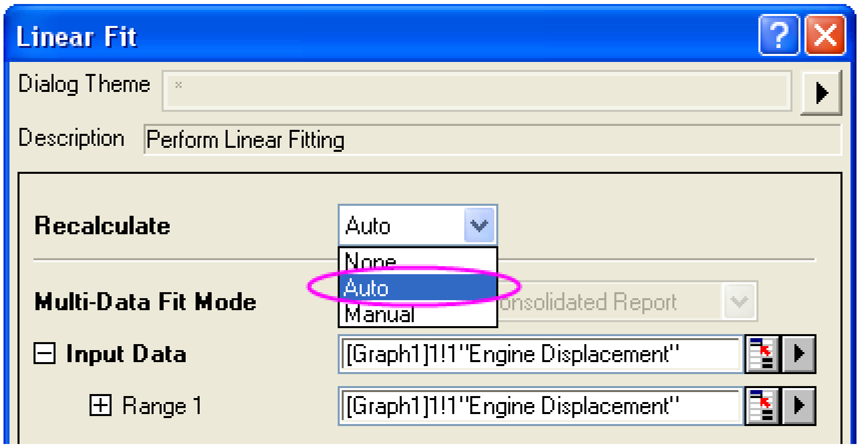
This tutorial will show you how to:

- Use the data filter to reduce worksheet data
- Auto update the graphs and analysis results when apply a column filter
- Add a floating graph to a worksheet.

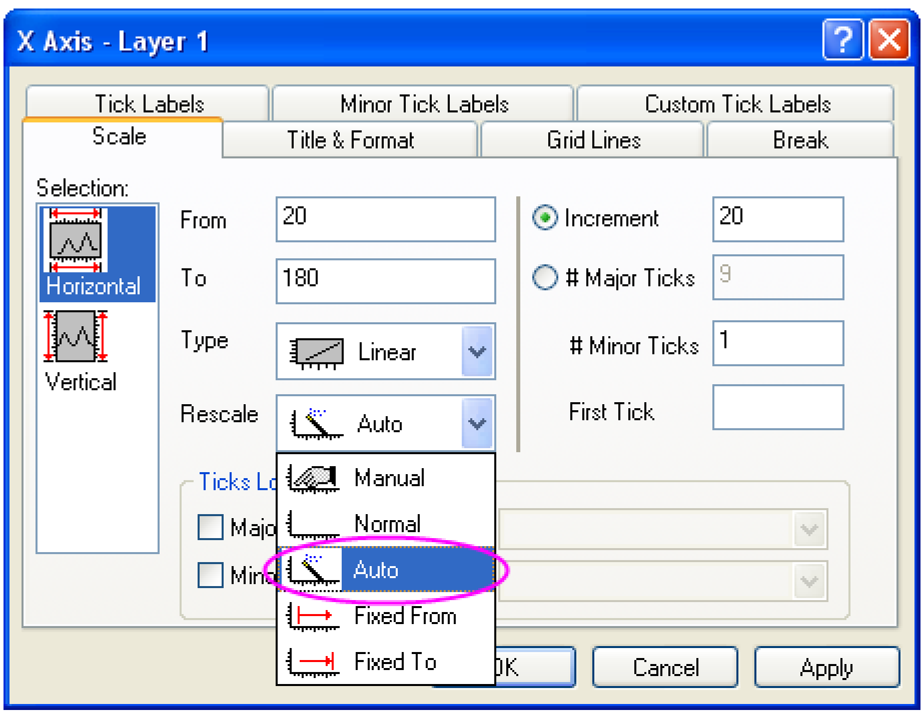
2.1.3 Steps

1. Create a new workbook by clicking the **New Workbook** button . Then click the **Import Single ASCII** button to import the **Automobile.dat** file in the *<Origin Folder>\Samples\Statistics* path. Both buttons are located in the **Standard** toolbar.
2. Highlight column C(Power), right click and choose **Set As:X** in the context menu to set this column as X.
3. Highlight column C and G (hold **Ctrl** key when clicking), click the button on the **2D Graph** toolbar to generate a scatter plot from these two columns.

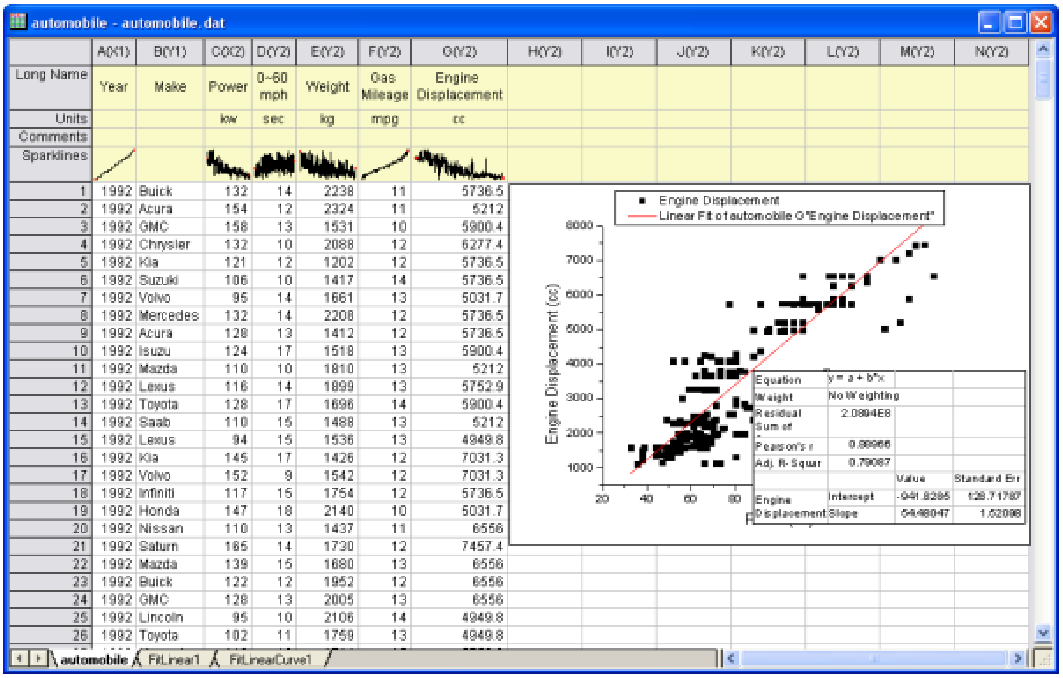
4. Activate the generated graph and select **Analysis:Fitting:Linear Fit** from menu item to open the **Linear Fit** dialog. In this dialog, set **Recalculate** to **Auto** to ensure auto update of the analysis result, accept other settings as default and click **OK** to carry out the analysis.



5. A fitted curve and a result table will be added to the graph, activate the graph again and double click on the X axis to open the **Axis** dialog, in the **Scale** tab, choose **Auto** for **Rescale**. Do the same for the Y axis and also set its rescale mode to **Auto**.

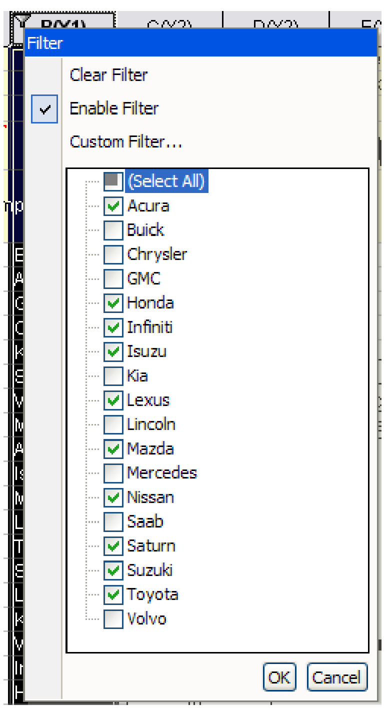


6. Go back to the original worksheet **automobile**, click the **Add New Columns** button seven times to add seven new empty columns to the end of the worksheet to work as background later.
7. Right click in the gray area of the worksheet and select **Add Graph...** in the context menu to open the **Graph Browser**, in this dialog, select the previously generated graph in the left panel and click **OK** to add this graph as a floating chart to the worksheet. Resize and move the floating chart for clearer view.



8. Highlight column A and B and click the **Add / Remove Data Filter** button on the **Worksheet Data** toolbar to add empty data filters to both columns.

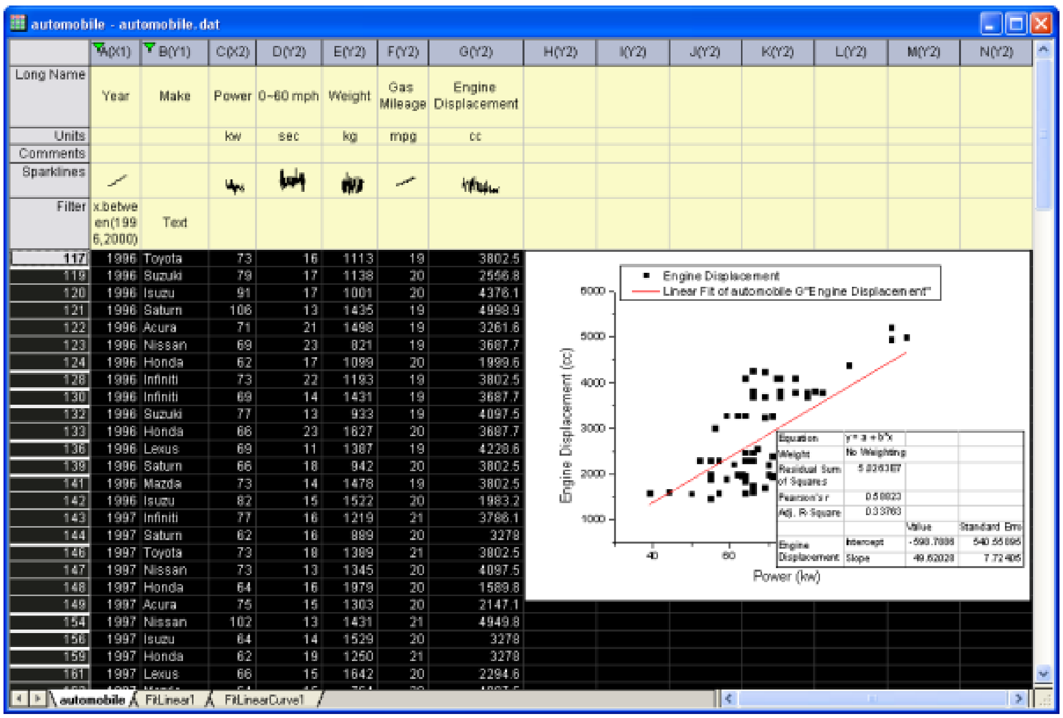
9. Click the **Filter** icon on the column header of column B, clear the check boxes before *Buick*, *Chrysler*, *GMC*, *Kia*, *Lincoln*, *Mercedes*, *Saab*, *Volvo* to hide all rows with these entries, to leave only the Japanese makers. Click **OK** to apply the filter. The worksheet data, graph and analysis result will all be auto updated accordingly.



10. Click the **Filter** icon on the column header of column A and select **Between**, note that the data type of column A is numeric by default from importing. Accept default setting of the **Between** dialog and click **OK**. A data filter is applied to this column.
11. Again click the **Filter** icon on column A and this time choose **Custom Filter** in the context menu to customize the filter, change the **Condition** as **x.between(1996,2000)** to set the **From** and **To** value respectively, click the **Test** button and in the original worksheet, only the rows meet this testing condition will be highlighted, this works as a preview of the data reduction.

> **Notes:** In order to view the whole worksheet at this stage, you can minimize the **Custom Filter** dialog, then scroll up and down the worksheet freely. You can later restore the dialog for further settings.

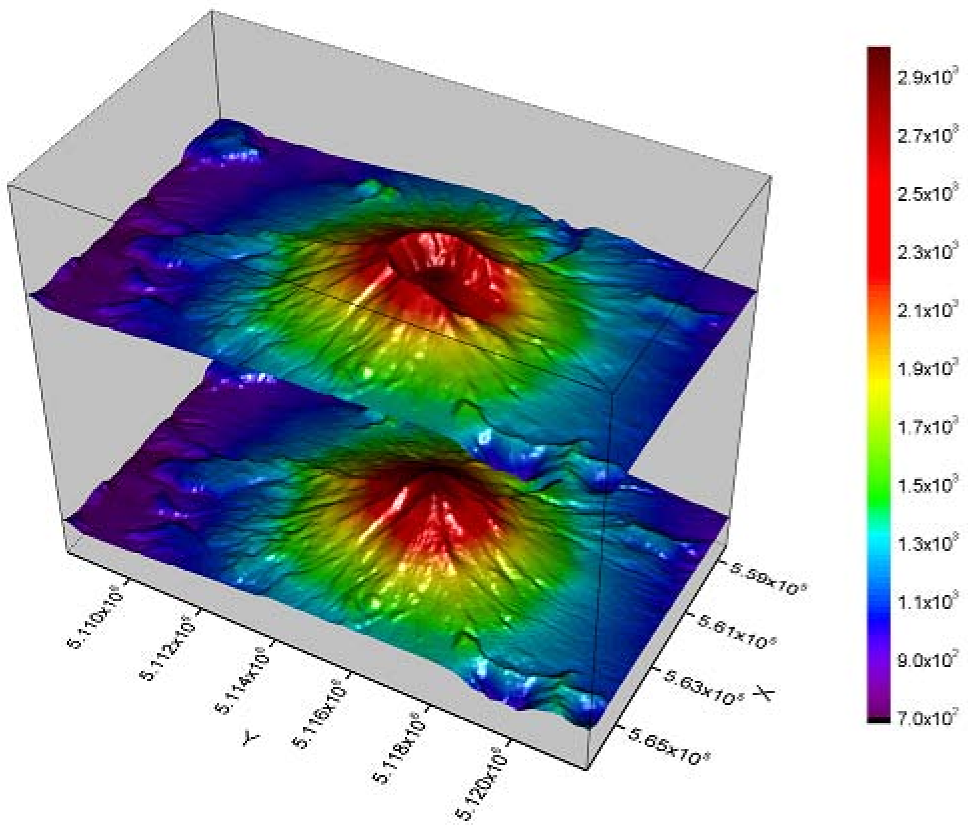
12. Click the **OK** button to apply the new filtering condition and the data, graphs and analysis results are updated and the graph is also auto rescaled.



2.2 Stacked 3D Surface Plots

2.2.1 Summary

This tutorial shows how to create stacked 3D colormap surfaces from different matrix objects. The surfaces in the plot display the topology before and after volcanic eruption. And a graph animation is generated from LabTalk script for the plot rotation.



Minimum Origin Version Required: Origin 9.0 SR0

2.2.2 What you will learn

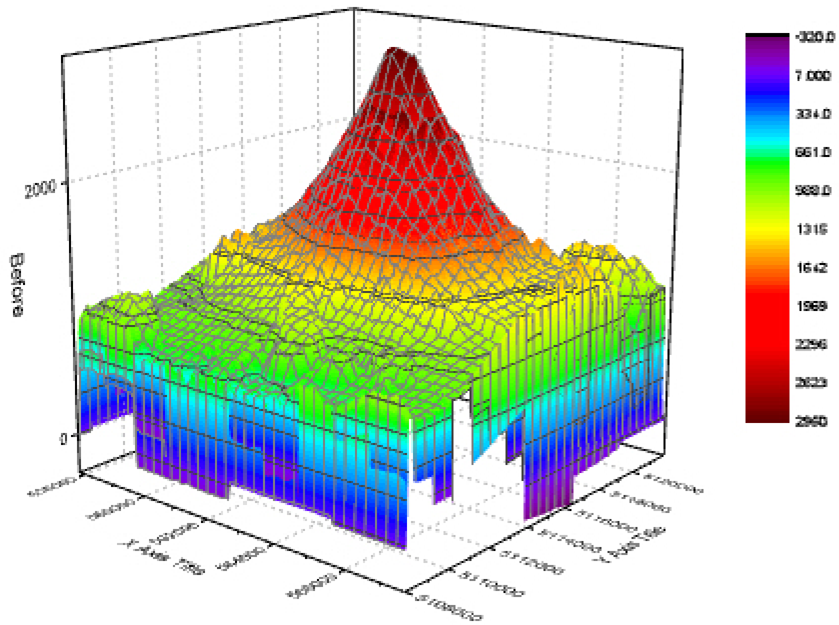
This tutorial will show you how to:

- Create stacked 3D colormap surfaces.
- Customize axes display and layer properties.
- Resize and rotate a 3D plot.

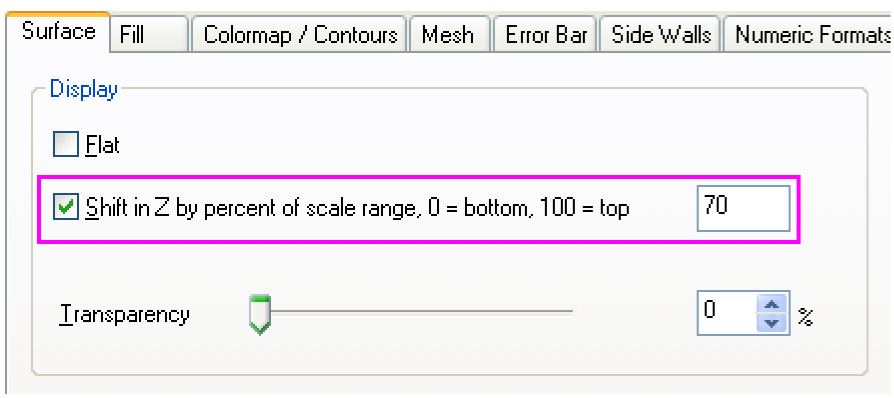
2.2.3 Steps

Create Multiple Colormap Surfaces

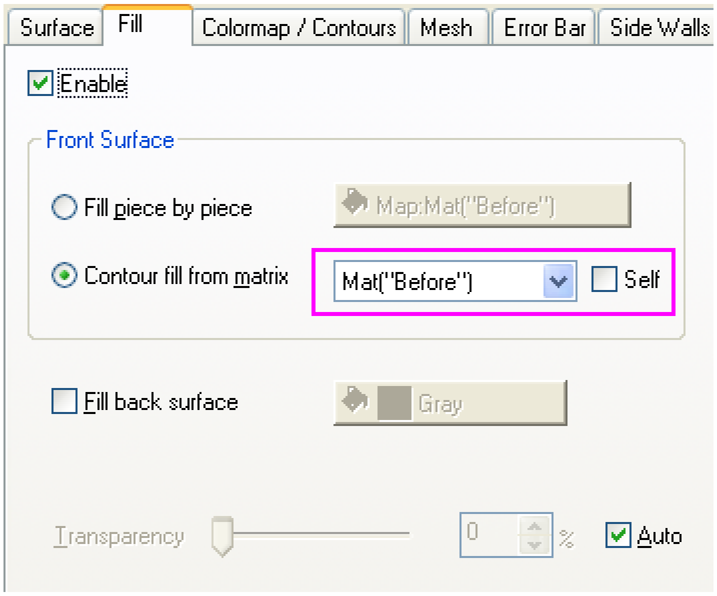
1. Click **File: Open Sample Projects: 3D OpenGL Graphs** from the **Menu** bar to open 3D OpenGL Graphs project. Go to the 3D OpenGL Graphs: 3D Surface: Stacked 3D Surface Plots folder in Project Explorer.
2. Activate the matrix book **Mbook1** which contains two matrix objects, then click **Plot: 3D Surface: Multiple Colormap Surfaces** to create two 3D surfaces from these two matrix objects.



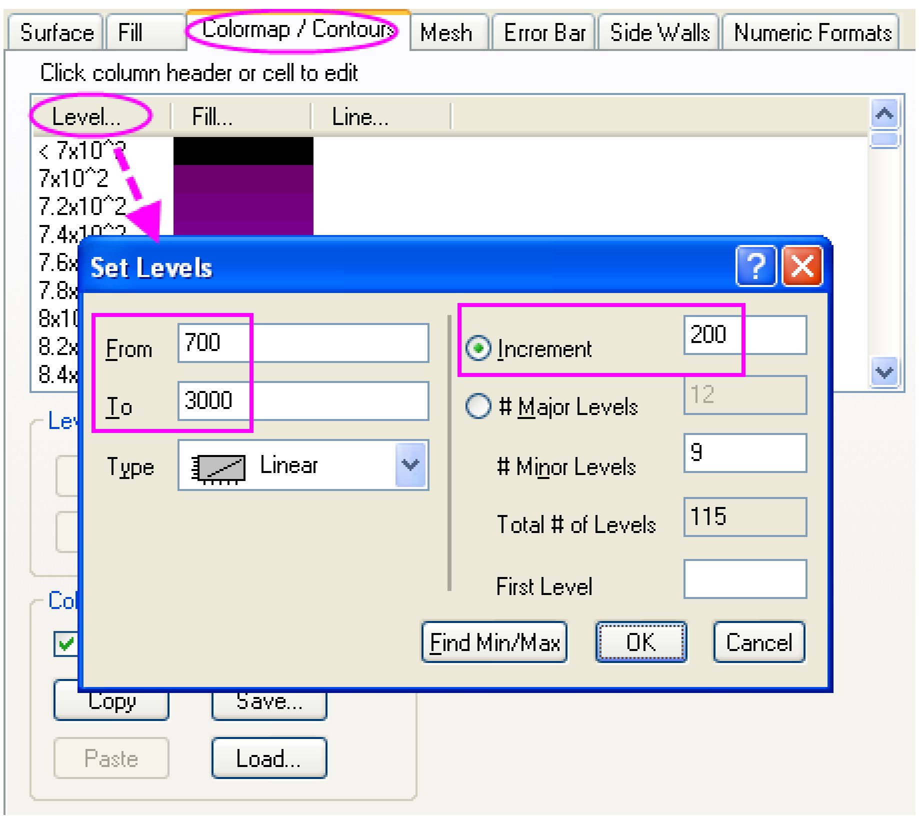
3. Double click on the plot to open the **Plot Details** dialog, you can see that there are two surfaces under the **Layer1** node on the left panel. To shift the "After eruption" surface in Z axis, activate the second plot under **Layer1** on the left panel, and in the right panel, select the **Surface** tab. Then check the box before **Shift in Z by percent of scale range**, and enter **70** in the text box.



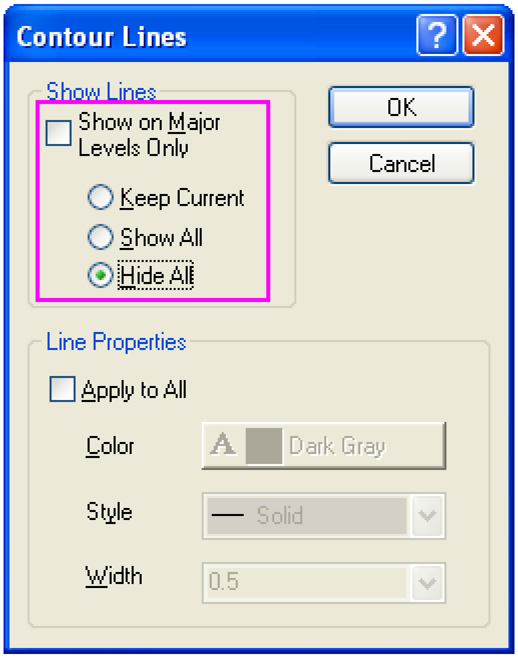
4. Go to the **Fill** tab. In the **Front Surface** section, uncheck the box before **Self** to fill contour by the same matrix object (Mat "Before") as the other surface used.



5. Select the **Colormap / Contours** tab. Click **Level..** to bring up the **Set Levels** dialog. Set the parameters as shown in the following graph and click **OK**.

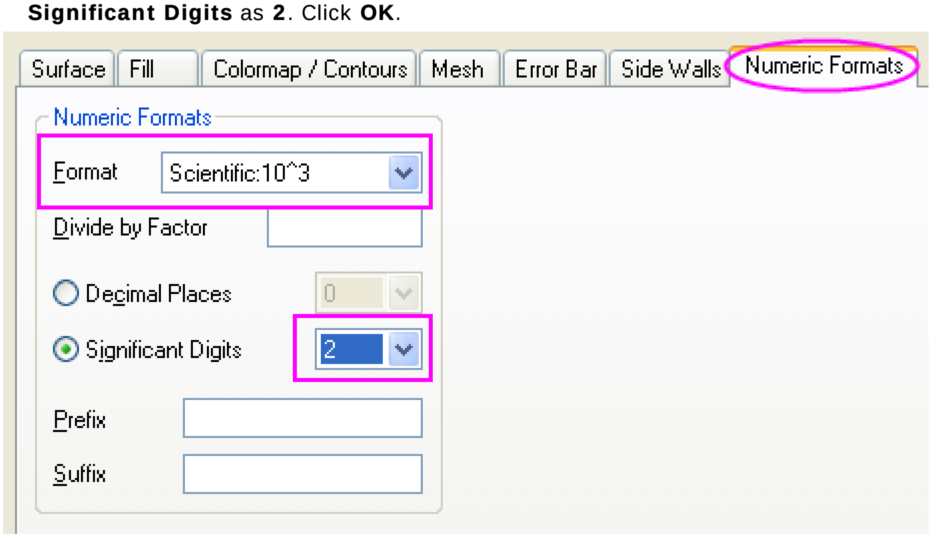


Click **Line...** to open the **Contour Lines** dialog. Uncheck the box before **Show on Major Levels Only** and select **Hide All**. Click **OK**.

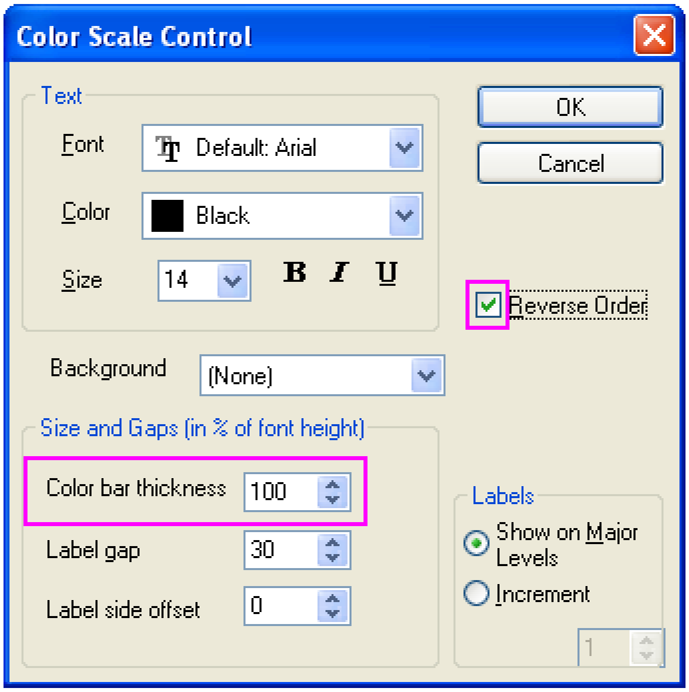


6. Go to the **Mesh** tab, and uncheck the box before **Enable** to disable the mesh line.
7. Repeat steps 5 to 6 for the first plot under the **Layer1** node.
8. In this project the two surfaces use the same matrix as contour fill, so they can share one color scale.

To set the numeric format of the color scale, activate the first plot on the left panel of the **Plot Details** dialog. Then select the **Numeric Formats** tab on the right panel. Select **Scientific: 10^3** from the drop down list next to the **Format**, and set **Significant Digits** as **2**. Click **OK**.



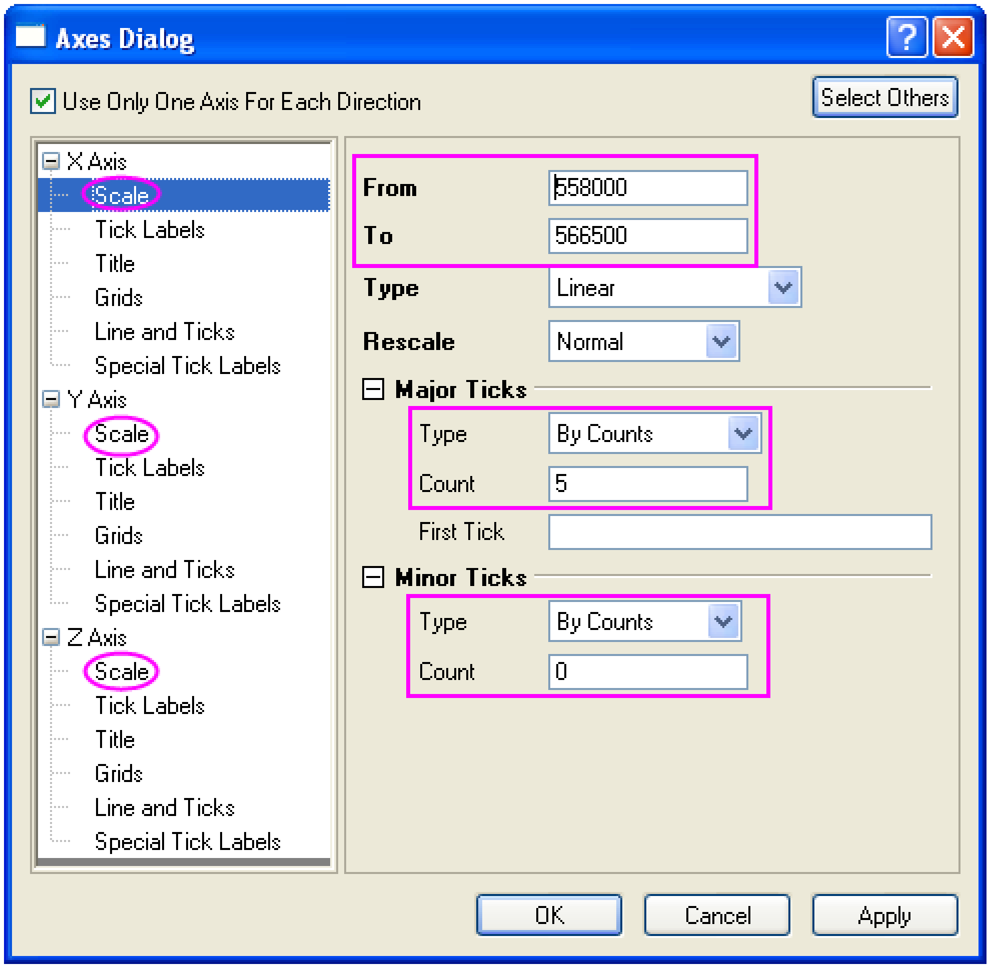
To customize the color scale, double click on the color scale to open the **Color Scale Control** dialog. Check the box before **Reverse Order** and set **Color bar thickness** as **100**. Click **OK**.



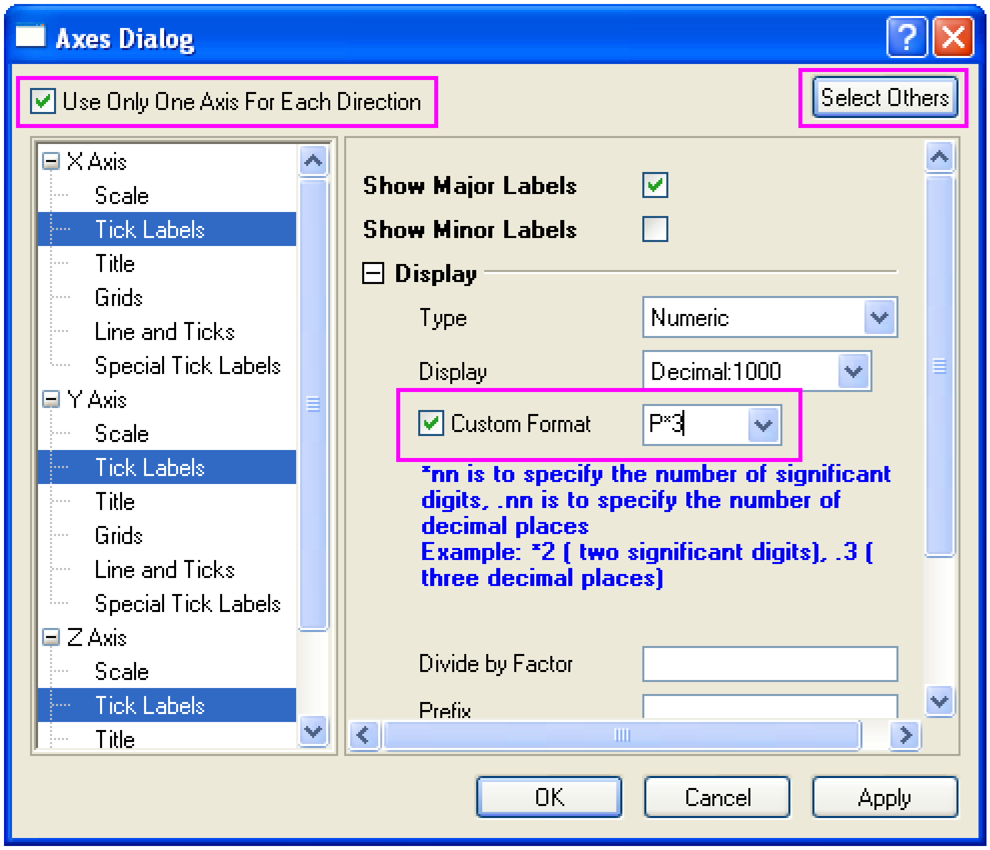
Customize Axes Display

In the **Axes** dialog, you can change the axes scale and tick labels' format. To open this dialog, click **Format: Axes: X Axis...**

1. Settings on the **Scale** node.
   - Set scale from **558000** to **566500** for X Axis, from **5108200** to **5121800** for Y Axis, and from **0** to **10000** for Z axis.
   - For X axis, set **Type** of **Major Ticks** as **By Counts** and set **Count** as **5**. For Y and Z axis, set **Type** of **Major Ticks** as **By Increment** and set **Value** as **2000**. To hide all minor ticks, set **Count** of **Minor Ticks** as **0** for all axes.

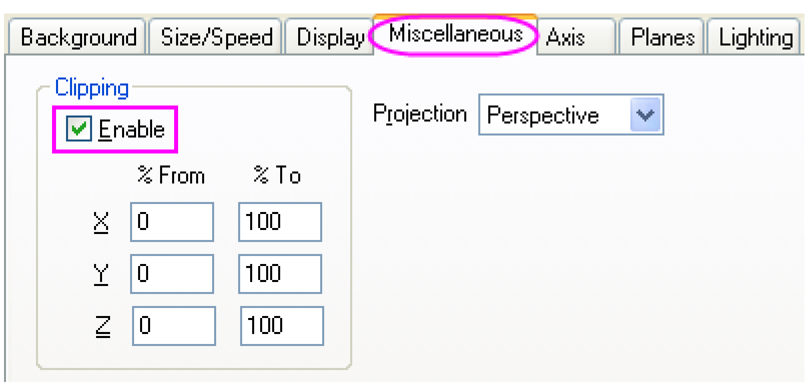


2. Customize the **Tick labels** and **Title**.
   - First, ensure that the **Use Only One Axis for For Each Direction** is enabled in the top of the **Axis** dialog. Therefore, only one axis is listed under each direction in the tree panel. That combined with the **Select Others** button allows you to quickly customize all axes with the same settings.
   - Select the **Tick Labels** node under X Axis. Click the **Select Others** button to select the tick labels of other axes. Check the box before **Custom Format** and select **P*3** from the drop down list to show the tick label as base-10 scientific notation with 3 significant digits. Click **OK**. For more information about the options in this drop down list, please refer to Custom Display Format.

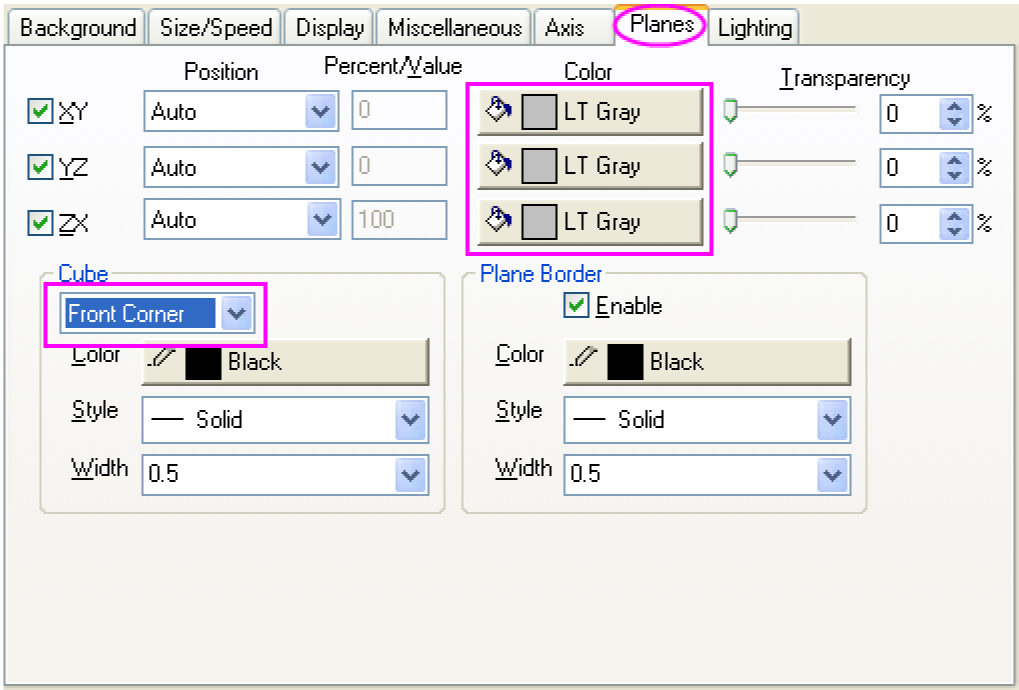


Customize **Layer Properties**

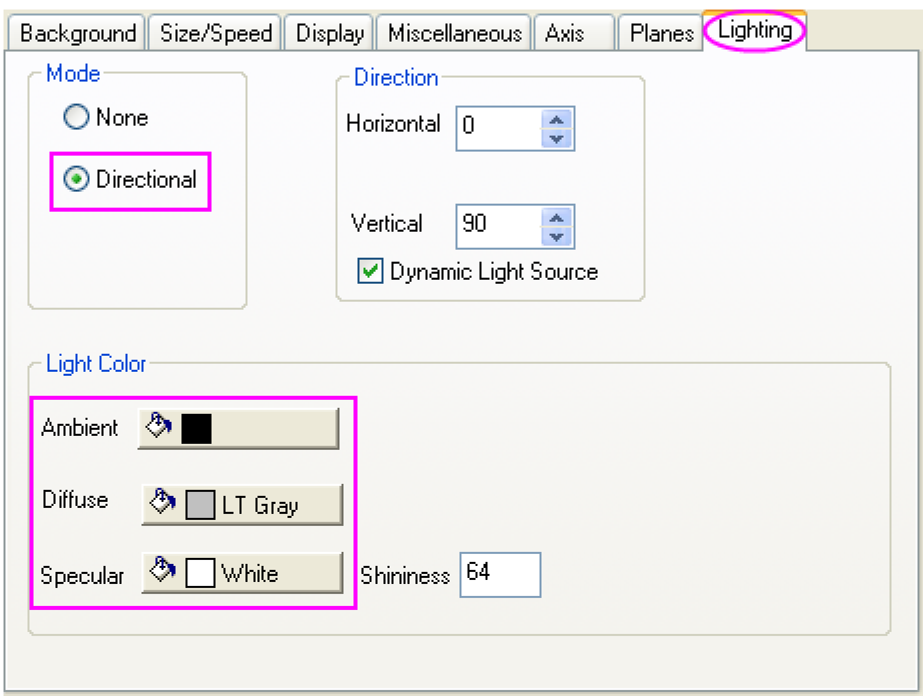
1. Double click on the blank space outside the plots or click **Format: Layer Properties...** from the menu to open **Plot Details - Layer Properties** dialog.
2. Activate the **Miscellaneous** tab on the right panel. Check the box before **Enable** in the **Clipping** section, which will clip the image outside the axes area according to the settings in the **Clipping** section.



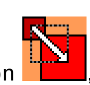
3. Go to the **Planes** tab. Set **Color** as **LT Gray** for all planes. And select **Front Corner** from the drop-down list in the **Cube** section to show the cube's border.



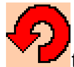
4. Select the **Lighting** tab. In the **Mode** section, choose **Directional** to enable lighting mode. Set **Light Color** as shown in the following graph. Click **OK**.



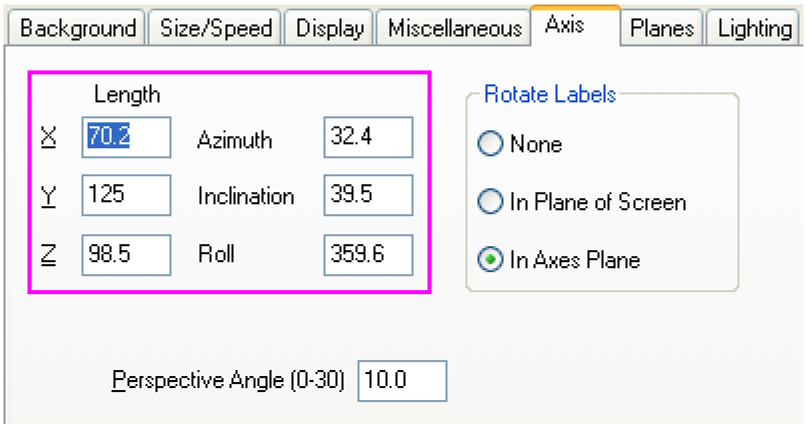
Resize and rotate the plot

1. Click on the cube (not the data plot) to activate the 3D toolbar. Click the Resize button , a 3D Cartesian coordinate will show up. Place the cursor on Y axis, which will then be highlighted,

at this moment drag-and drop the Y axis to stretch the plot in Y axis direction. Do the same to X direction and Z direction.

2. Click the rotate button  to activate rotation mode. A sphere will be displayed at the center of the plot. Rotate the plot to get a better view.

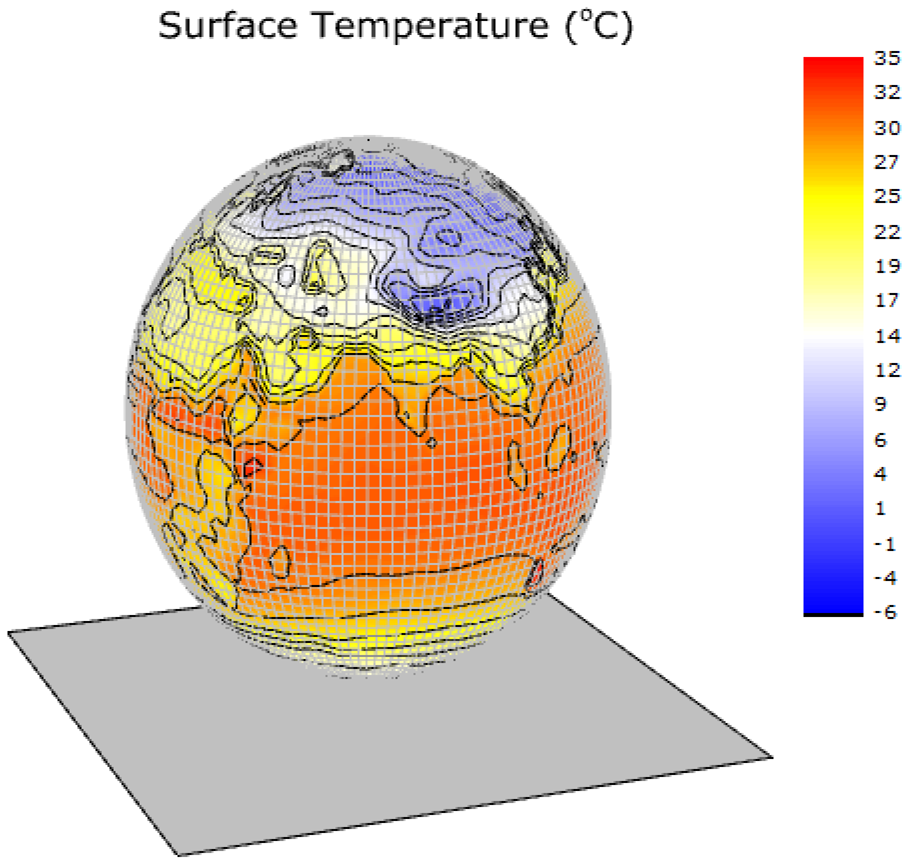
The 3D toolbar allows you to resize and rotate the plot freely. However, you can also achieve the same view as Graph1 in this sample project by setting the value in the **Axis** tab of **Layer Properties** dialog as shown in the following graph.



2.3 Parametric Surface with Colormap from Data

2.3.1 Summary

In this tutorial a 3D sphere is created using the data from three matrices. And the surface is filled to display the surface temperature contour using the data from another matrix.



Minimum Origin Version Required: Origin 9.0 SR0

2.3.2 What you will learn

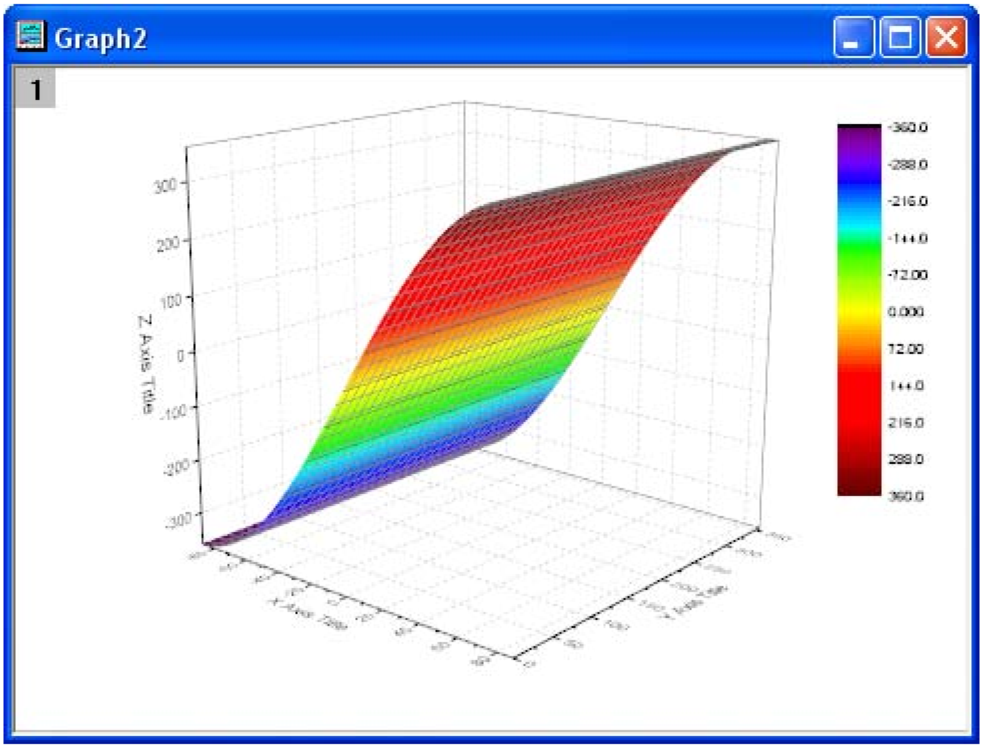
This tutorial will show you how to:

- Create parametric surface from matrix data.
- Set contour fill from another matrix.
- Customize the 3D parametric surface plot.

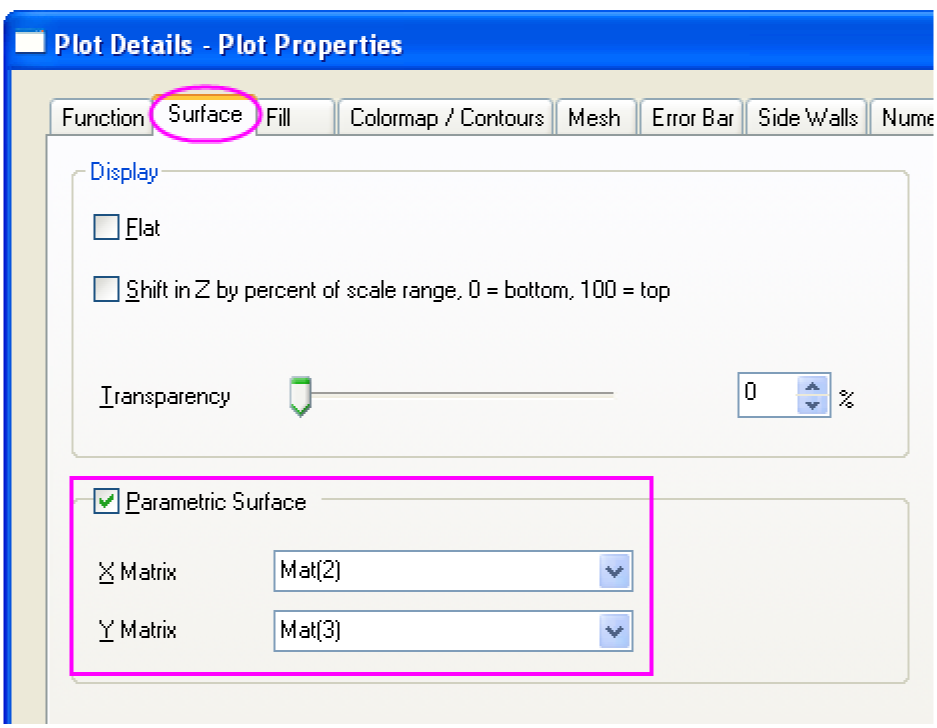
2.3.3 Steps

1. Open the 3D OpenGL Graphs project(*\Samples\3D OpenGL Graphs.opj*), go to the *3D OpenGL Graphs: 3D Function Plot: Parametric Surface with Colormap from Data* folder in Project Explorer.

2. Activate the matrix *FUNCA:1/4*, and click the button on **3D and Contour Graph** toolbar to create a colormap surface as below. You can also create this colormap surface by selecting **Plot: 3D Surface: Color Map Surface**.

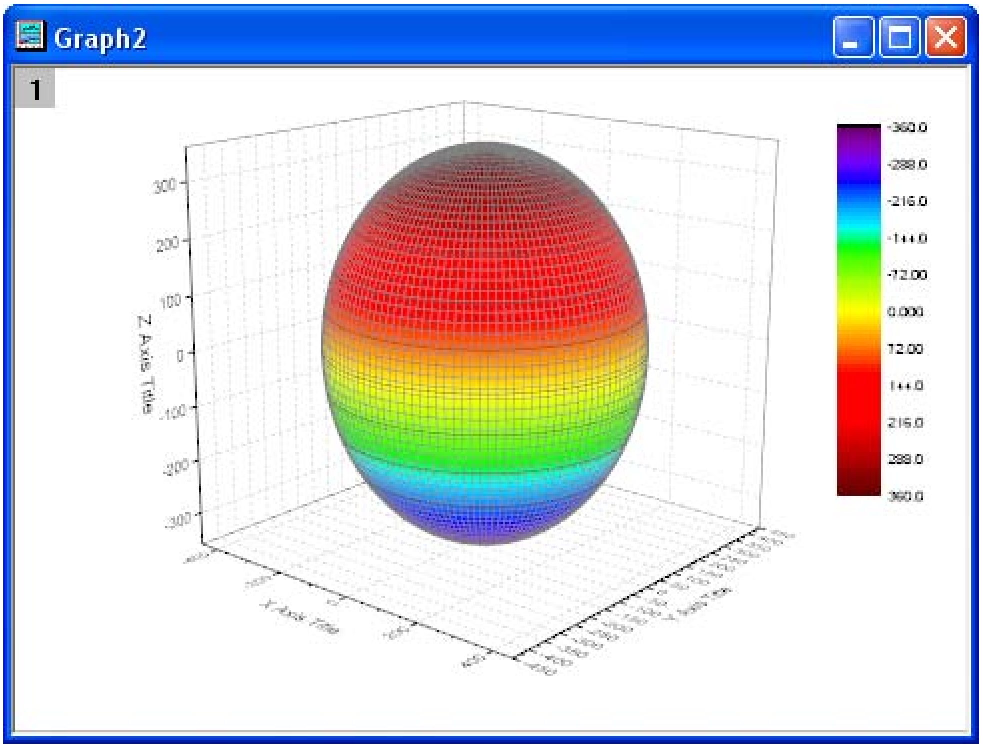


3. Double click on the plot to open the **Plot Details** dialog. Click on the **Surface** tab. Check the box before **Parametric Surface** and set **X Matrix**, **Y Matrix** as **Mat(2)**, **Mat(3)** respectively.

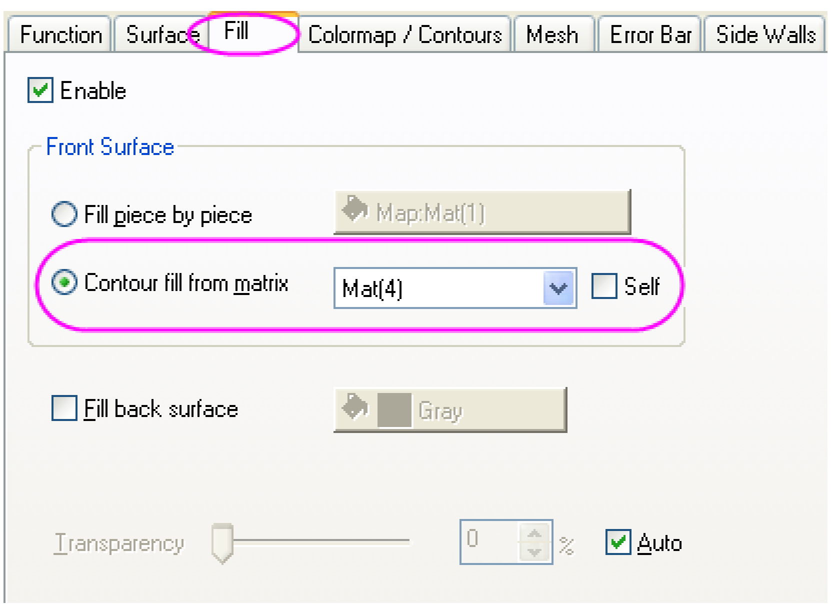


Click **OK** to close the dialog.

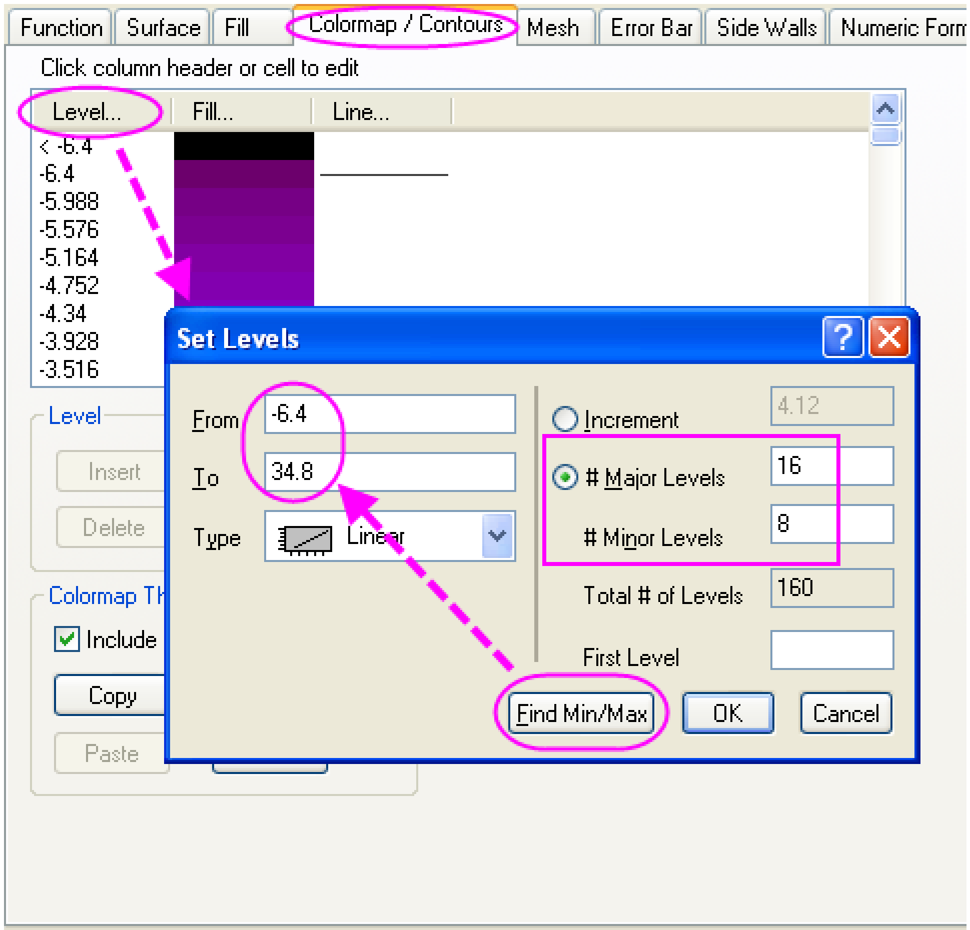
4. In order to show the complete colormap surface click the  button on **Graph** toolbar and the colormap surface should look like the following image:



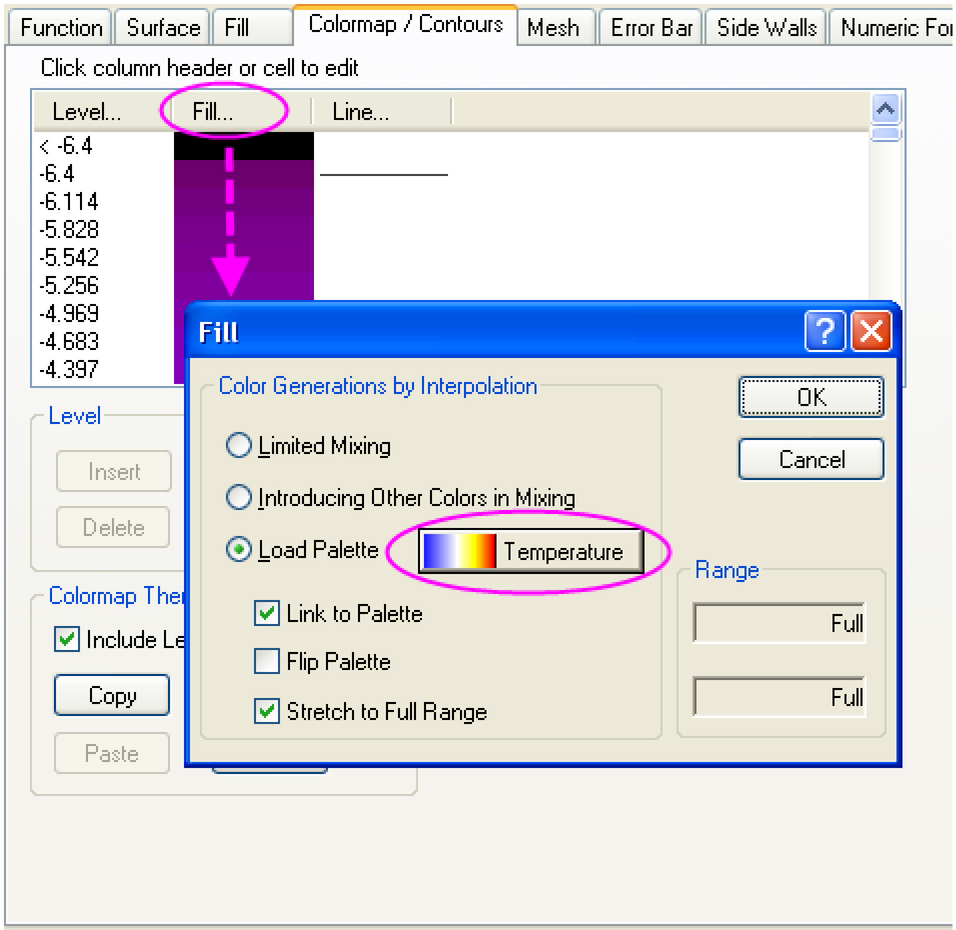
5. Double click on the plot to open **Plot Details** dialog. Go to the **Fill** tab. In Front Surface section uncheck the box before **Self** and set **Contour fill from matrix** as **Mat(4)**. Click **Apply**.



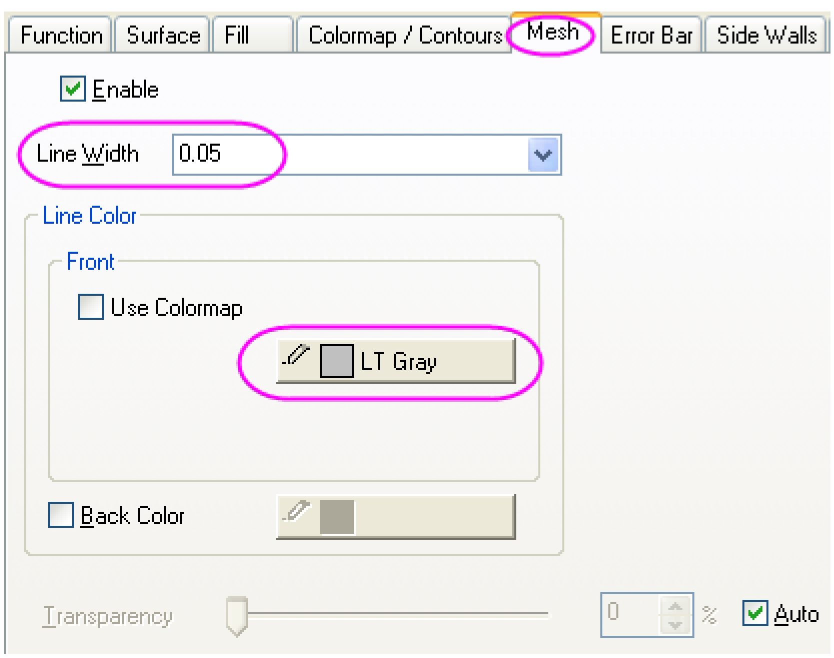
6. Activate the **Colormap / Contours** tab. Click **Level** to open the **Set Levels** dialog. Click **Find Min/Max** and set **Major Levels**, **Minor Levels** as **16**, **8** respectively. Click **OK**.



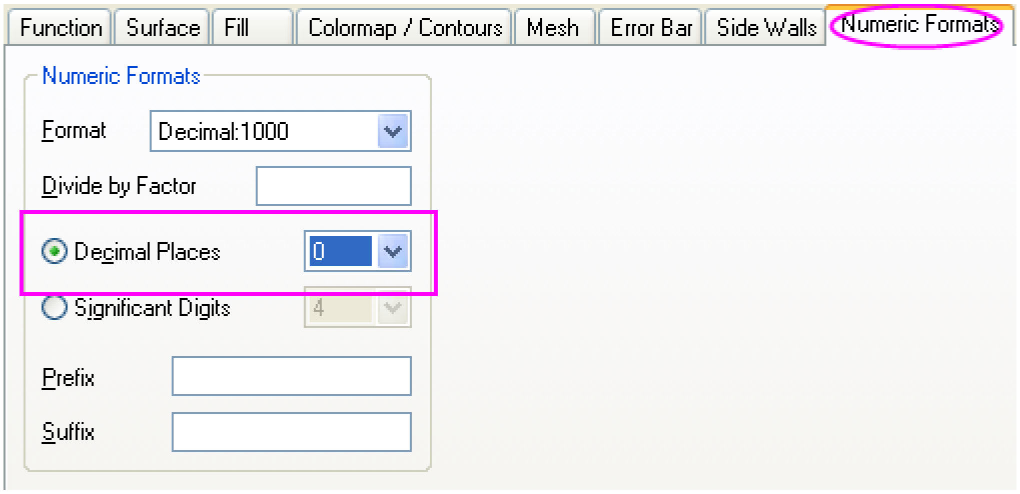
7. Click **Fill** to open the **Fill** dialog. Set **Load Palette** as **Temperature**. Click **OK**.



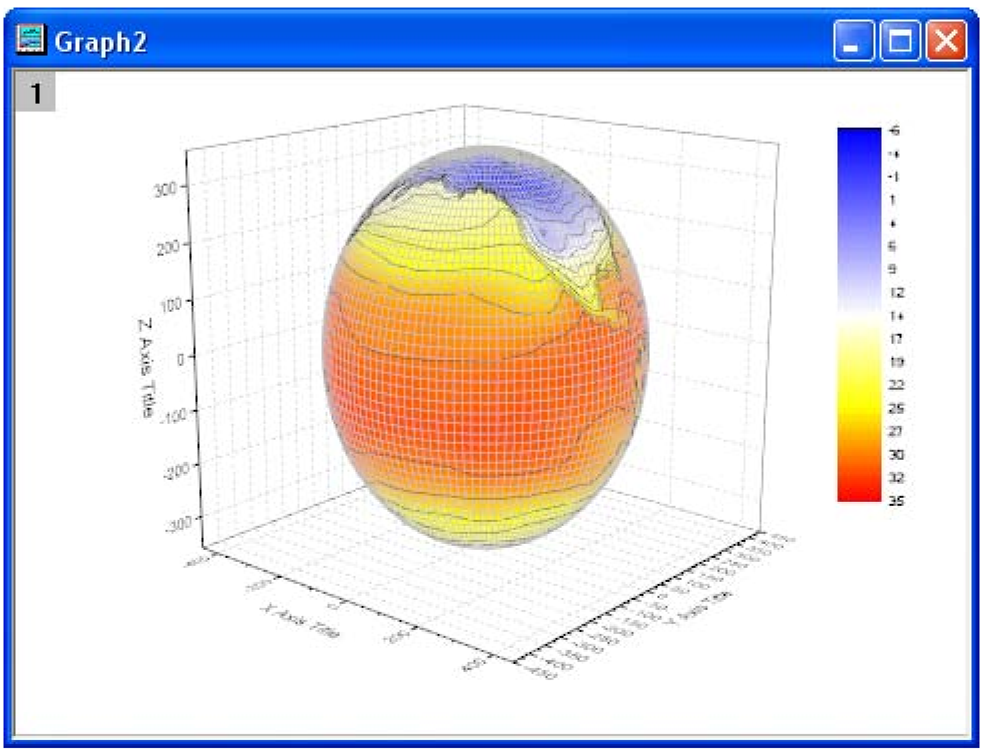
8. Click on the **Mesh** tab. Set **Line Width** as **0.05** and **Line Color** in Font section as **LT Gray**. Click **Apply**.



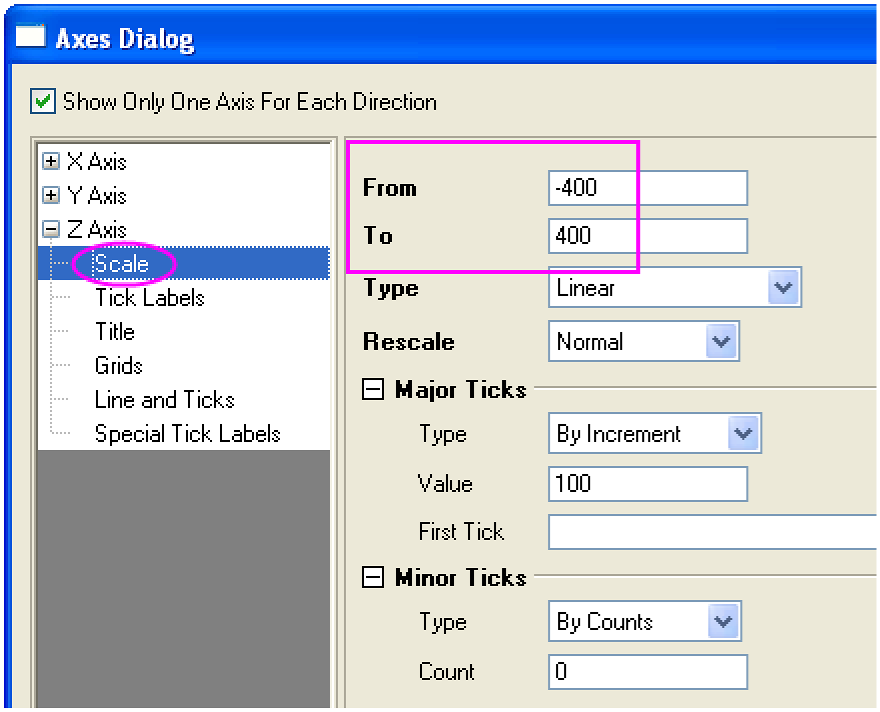
9. Click on the **Numeric Formats** tab. Choose the **Decimal Places radio** button and set its value as **0**.



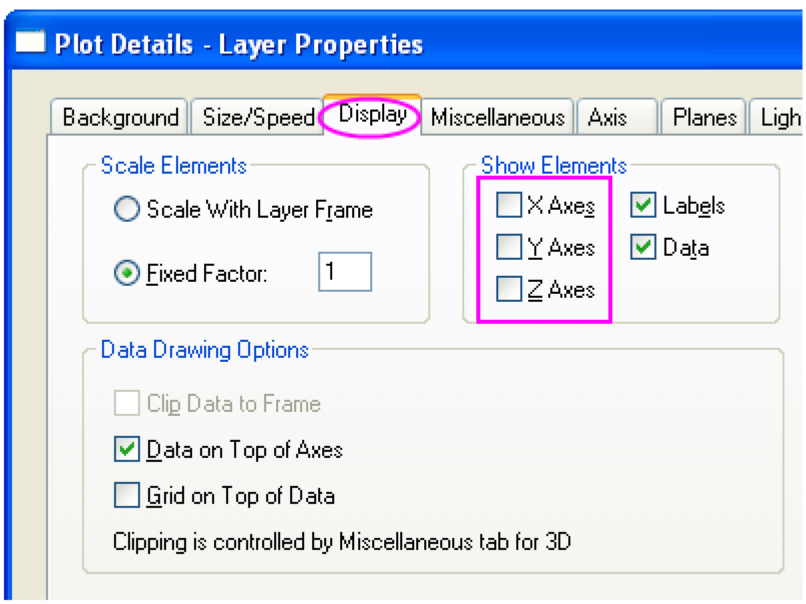
Click **OK** to apply the settings and close the **Plot Details** dialog. The graph should look like the following image.



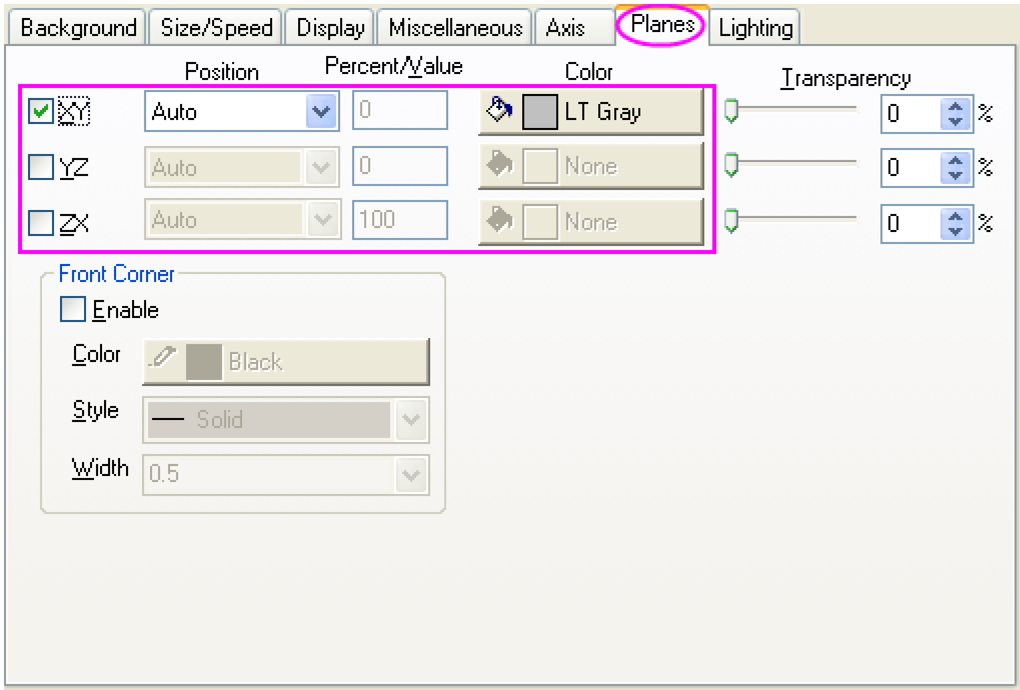
10. Double click on Z axis to open **Axes Dialog**. On the **Scale** node, set the value of **From**, **To** as **-400**, **400** respectively. Click **OK**.



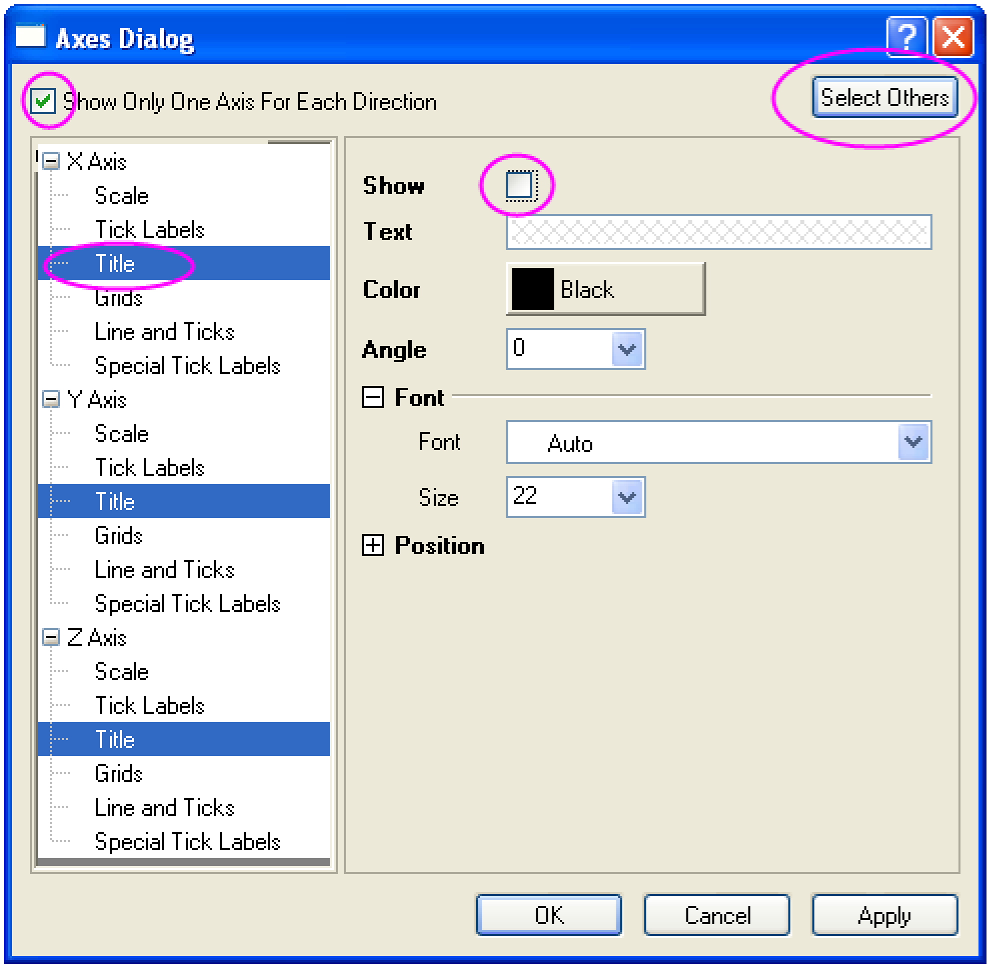
11. Double click on the XY Plane to open **Plot Details - Layer Properties**. Click the **Display** tab, and uncheck the box before **X Axes**, **Y Axes**, **Z Axes** in Show Elements section to hide the axes.



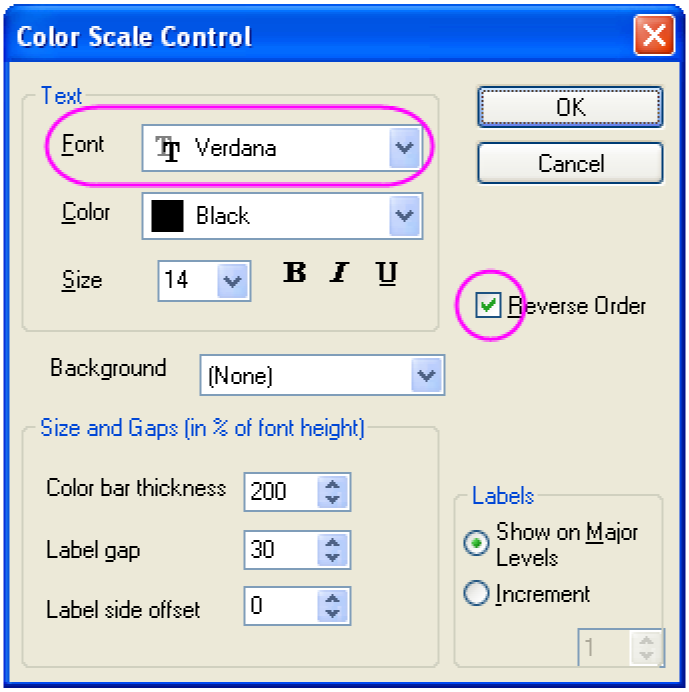
12. Click on the **Planes** tab, and uncheck the boxes before **YZ**, **ZX** to hide YZ and ZX planes. Set **Color** of **XY** as **LT Gray**. Click **OK** to close the dialog.



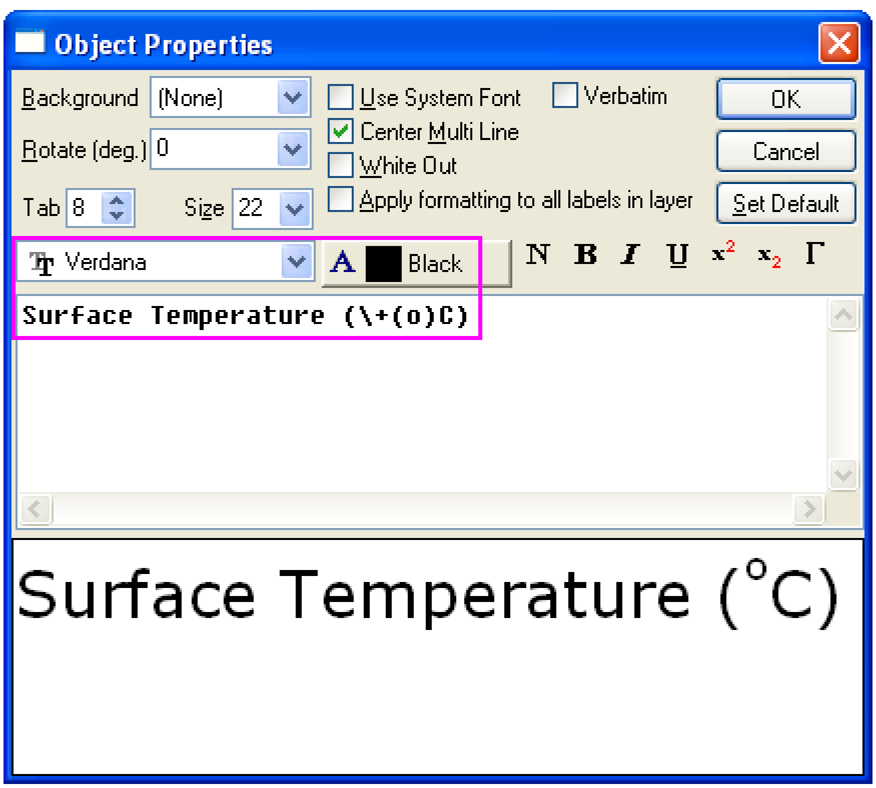
13. Go back to the graph, select **Format:Axes Titles:X axis titles** to open the **Axes** dialog with the **Title** node selected. Click the **Select Others** button. Uncheck the box after **Show** to hide axis title for all axes.



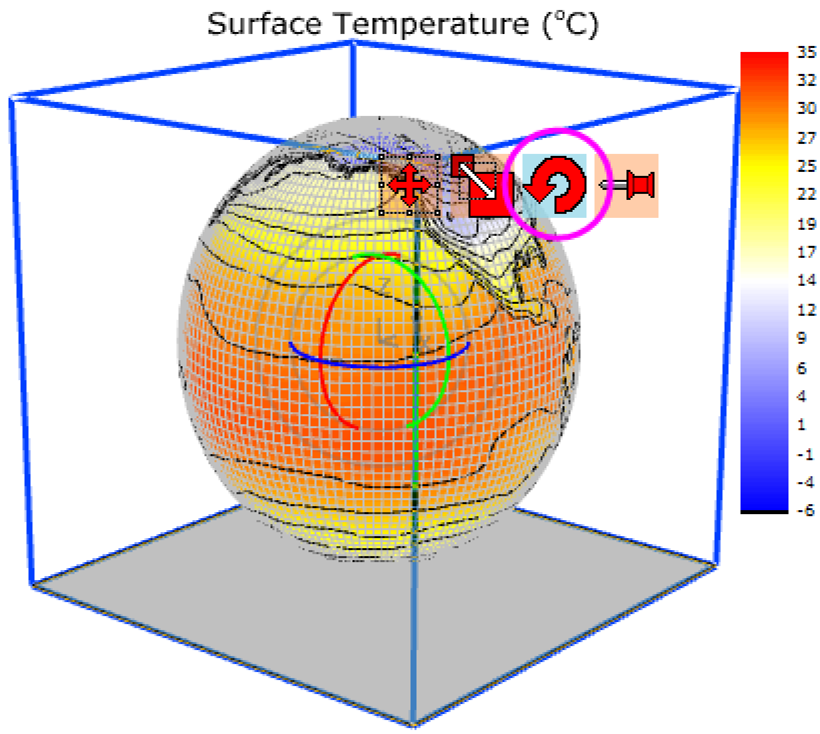
14. Double click on the color scale to open the **Color Scale Control** dialog. Set Text Font as **Verdana**. Check the box before **Reverse Order**. Click **OK** to apply the setting and close dialog. Move the color scale object to a proper place.



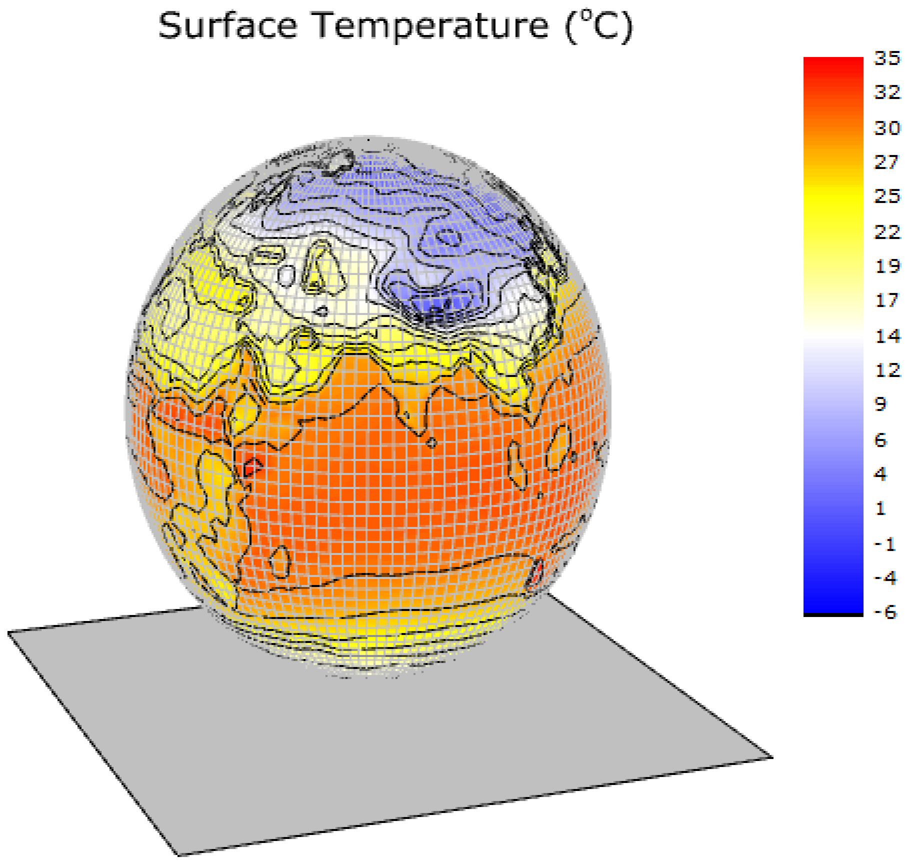
15. Right click on the white area of the graph layer to bring up a context menu and choose **Add / Modify Layer Title**. Select the text object added just now, right-click on it and select **Properties...** on the shortcut menu to open the **Object Properties** dialog. Set text font as **Verdana** and type **Surface Temperature (\+(o)C)** in the content table. Click **OK**.



16. Click on the graph layer within 3D frame (not the data plot), and click the **Rotate** button as shown in the following to activate rotation mode.



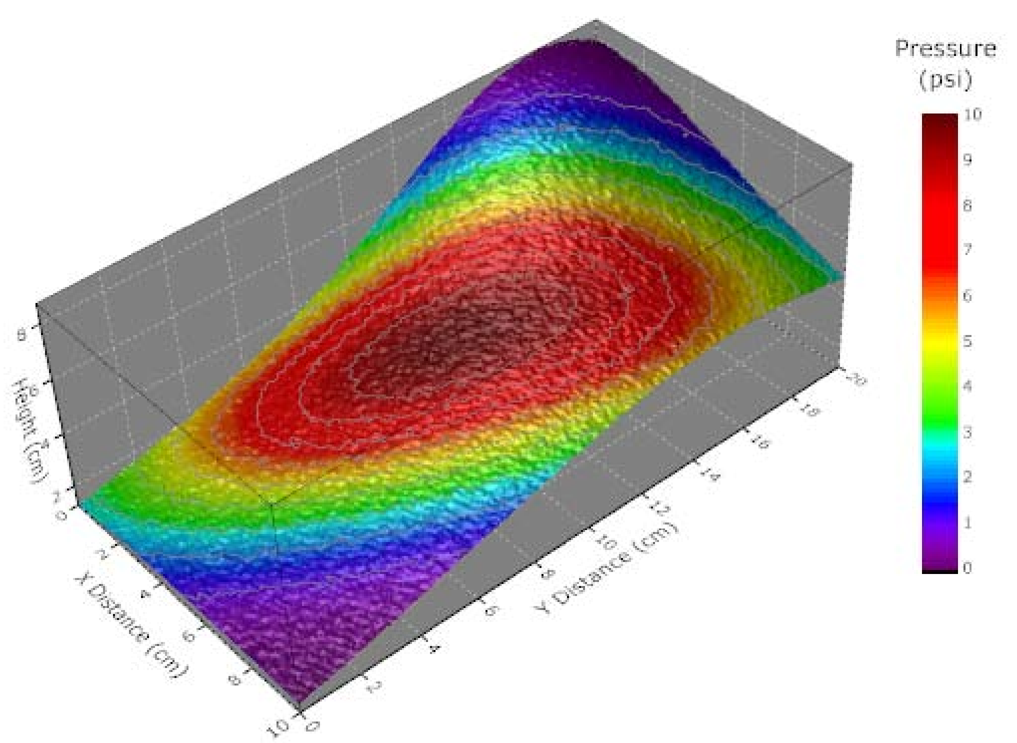
Rotate the plot to get a better view. The graph might look like the following.



2.4 Colormap from Second Matrix

2.4.1 Summary

Origin can represent four-dimensional data by color-mapping a surface plot using a second matrix.



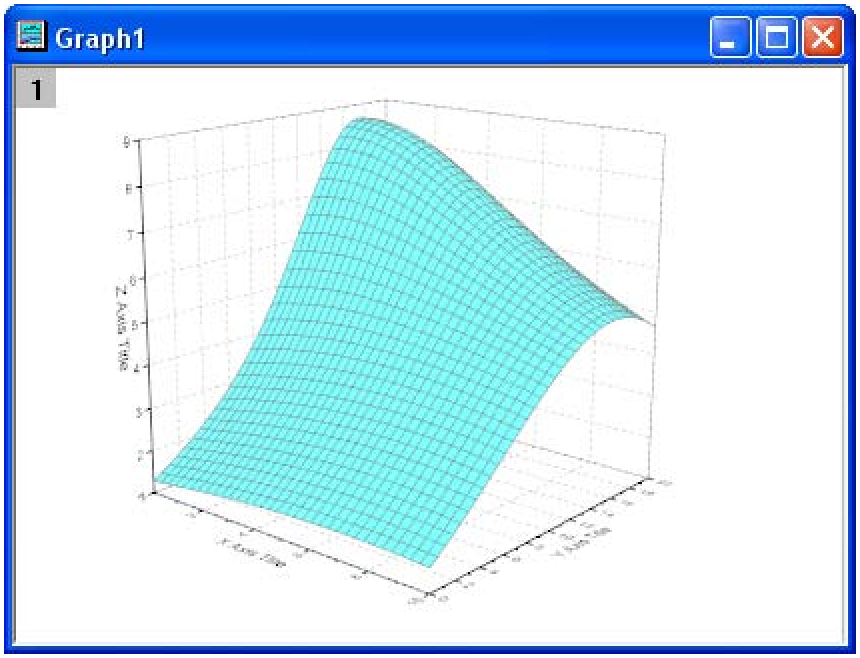
Minimum Origin Version Required: Origin 8.5 SR0

2.4.2 What you will learn

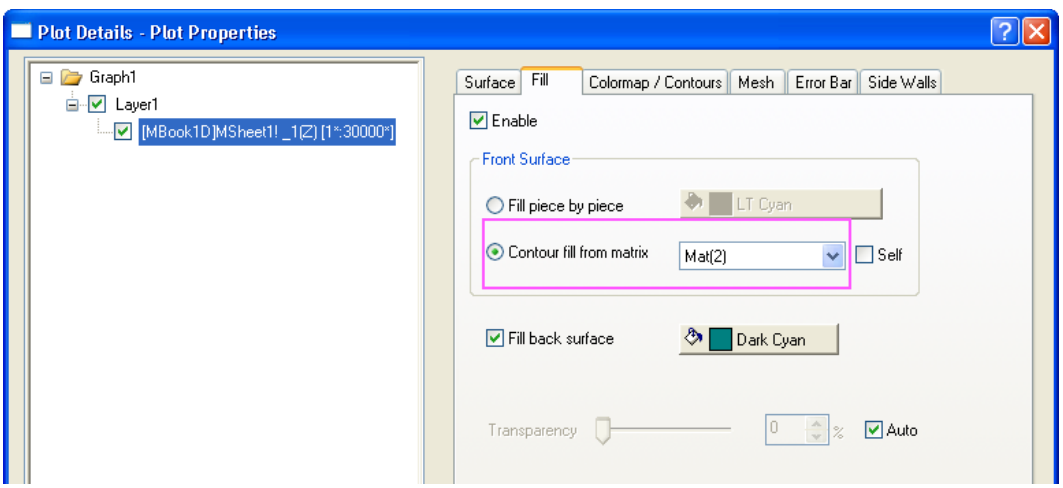
- Create a Color Fill Surface from a matrix.
- Color map a surface plot using a second matrix.
- Customize color map levels and palette.
- Control lighting on the graph (From Origin 9 SR0)

2.4.3 Steps

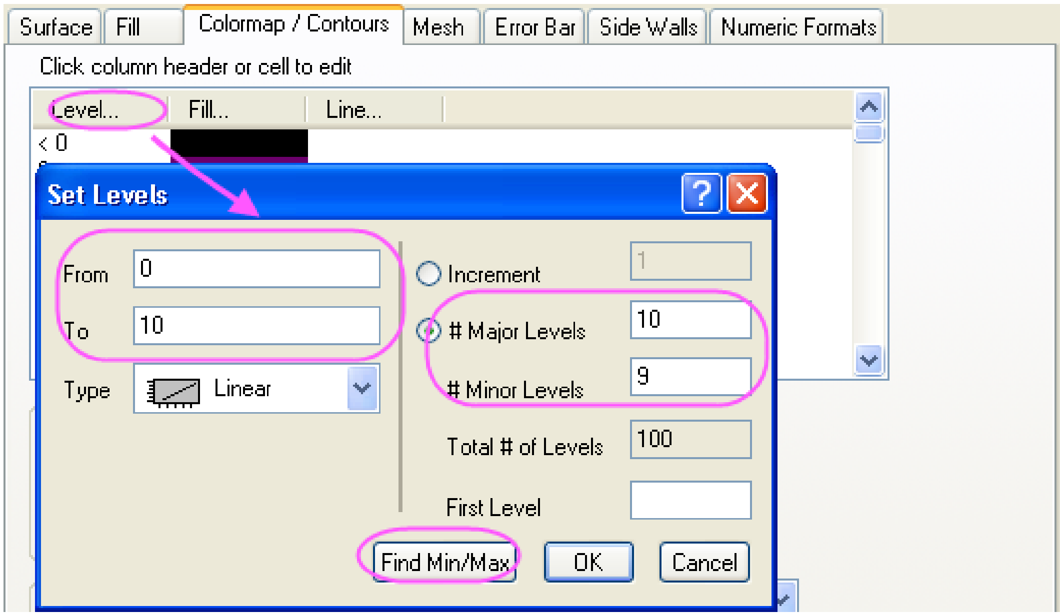
1. Click **File : Open** to open *Colormap_from_Second_Matrix.ogm* under the folder **Sample\Graphing**. You should see two image thumbnails above the matrix data, just under the title bar. (If you do not see image thumbnails, right-click on the matrix title bar and select **Show Image Thumbnails**.) Select image thumbnail **1**.
2. On the main menu, click **Plot**, point to **3D Surface**, then click **Color Fill Surface** to generate a surface plot.



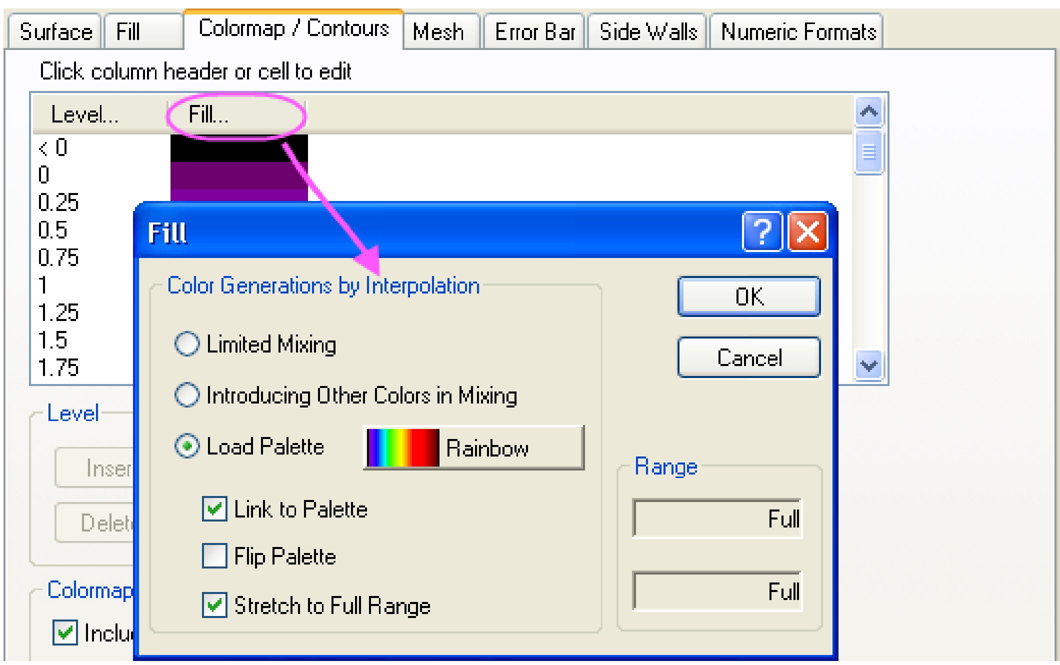
3. Double click on the plot to open the **Plot Details** dialog. In the left panel, select **(MBook1D)MSheet1**. In the right panel, select the **Fill** tab. Choose **Coutour Fill from Matrix**. and select **Mat(2)** for front surface.



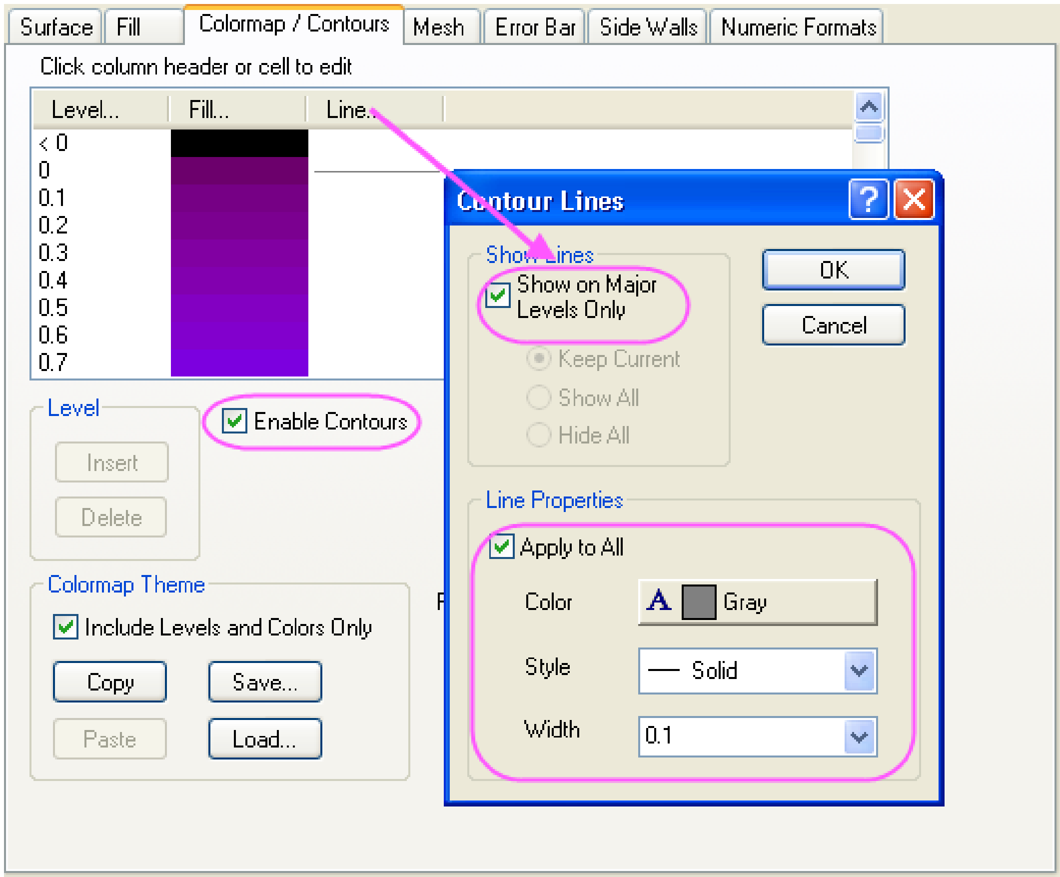
4. Go to **Colormap/Contours** tab, click the **Level...** title to open the Set Levels dialog. In this dialog, click **Find Min/Max** button and set the **#Major Levels** and **#Minor Levels** as *10* and *9*.Click OK button to close the dialog.



5. Click the **Fill...** title to open the **Fill** dialog. select **Load Palette** and then click **Select Palette** button to select **Rainbow** palette.Click OK button to close this dialog.

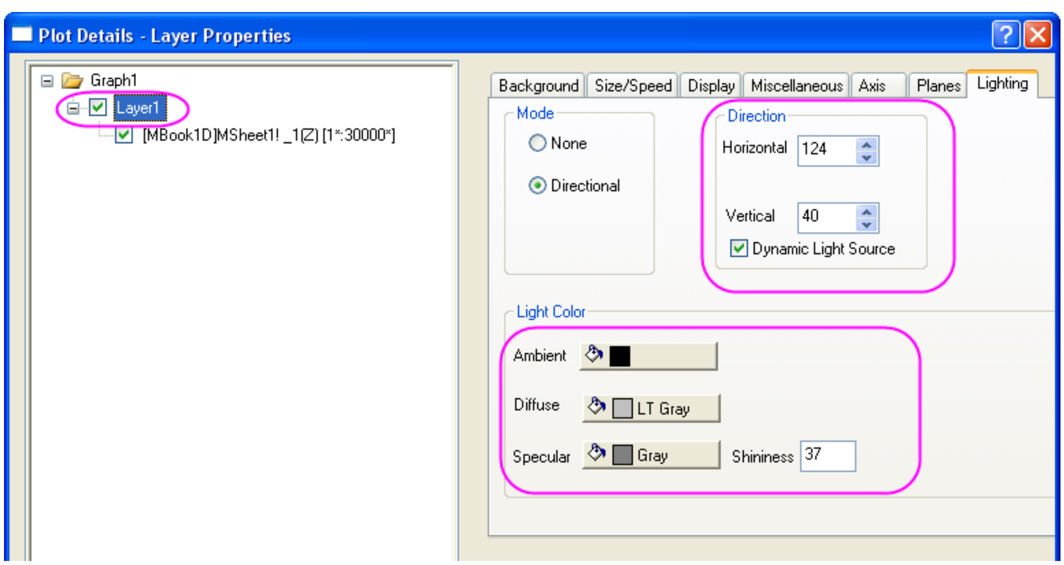


6. Check the **Enable Contours** check box. And then click **Line...** to open the **Contour Lines** dialog. In this dialog, check the **Show on Major Levels Only** check box and set the **Line Properties** as below. Click OK button to close the dialog.

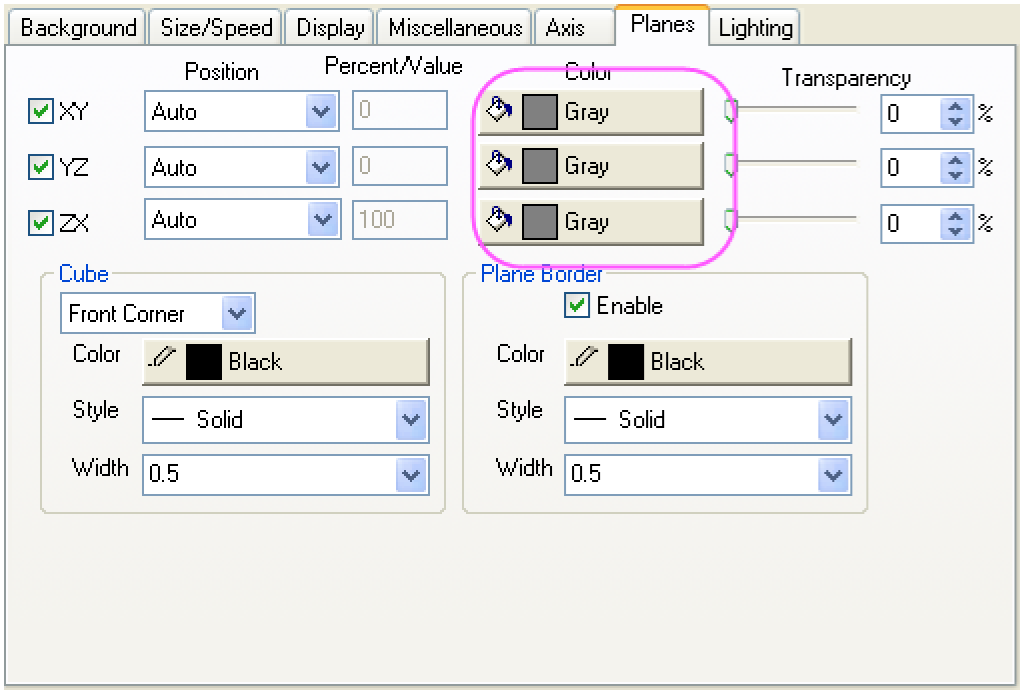


7. Go to the **Mesh** tab,uncheck the **Enable** box to disable the mesh lines.

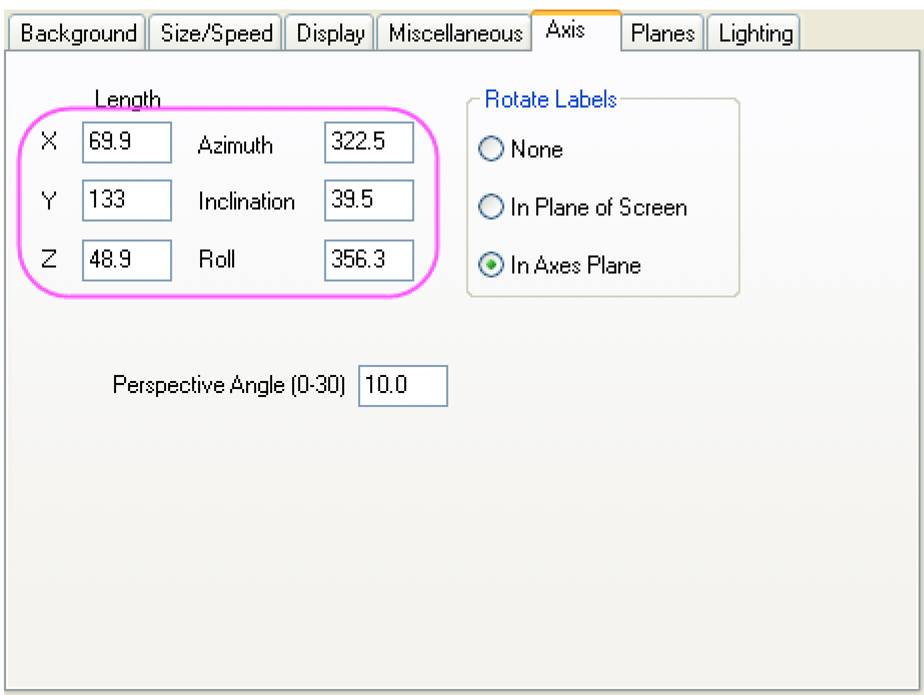
8. Select **Layer1** in the left panel, go to **Lighting** tab in the right panel. Select **Directional** under **Mode**. Set **Horizontal** and **Vertical** as *124* and *40* , and change the color of **Diffuse** as *LT Gray* and **Specular** as **Gray**. Then set the **Shininess** as *37*.



9. Go to the **Planes** tab, set the color as **Gray** and select **Front Corner** for the Cube drop-down menu.

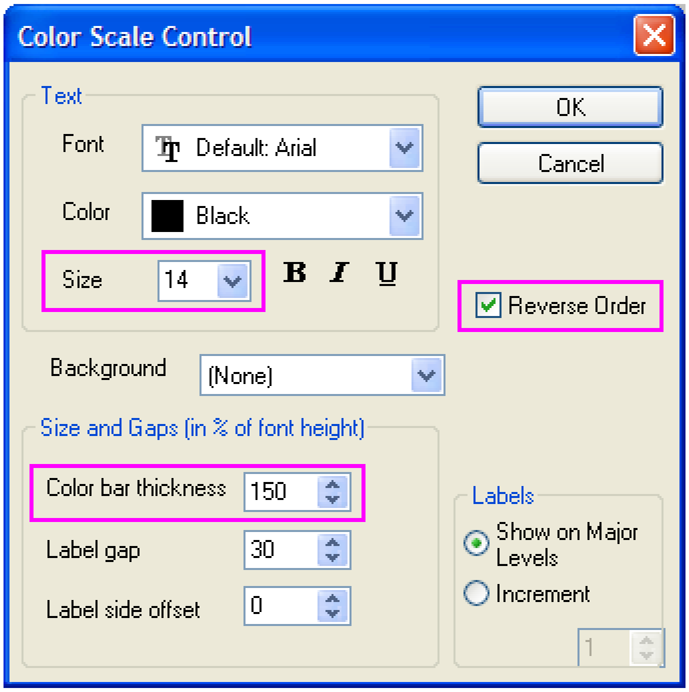


10. Go to Axis tab and do settings as below. Click **OK** button to close this dialog.



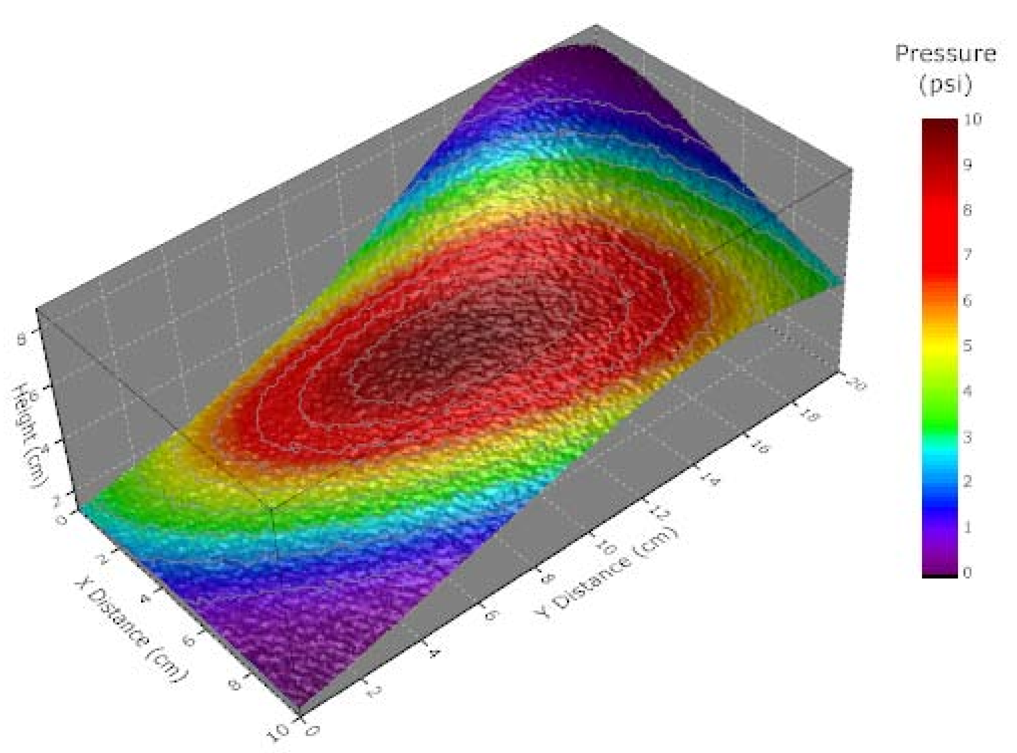
11. Keep the graph window active. Select **Graph: New Color Scale** to add a new color scale in the graph window.

12. Double-click anywhere on the color scale to open the **Color Scale Control** dialog box. Change text **Size** to **14**, and **Color bar thickness** to **150**. Select the **Reverse Order** check box.



Click **OK** to close the **Color Scale Control** dialog box.

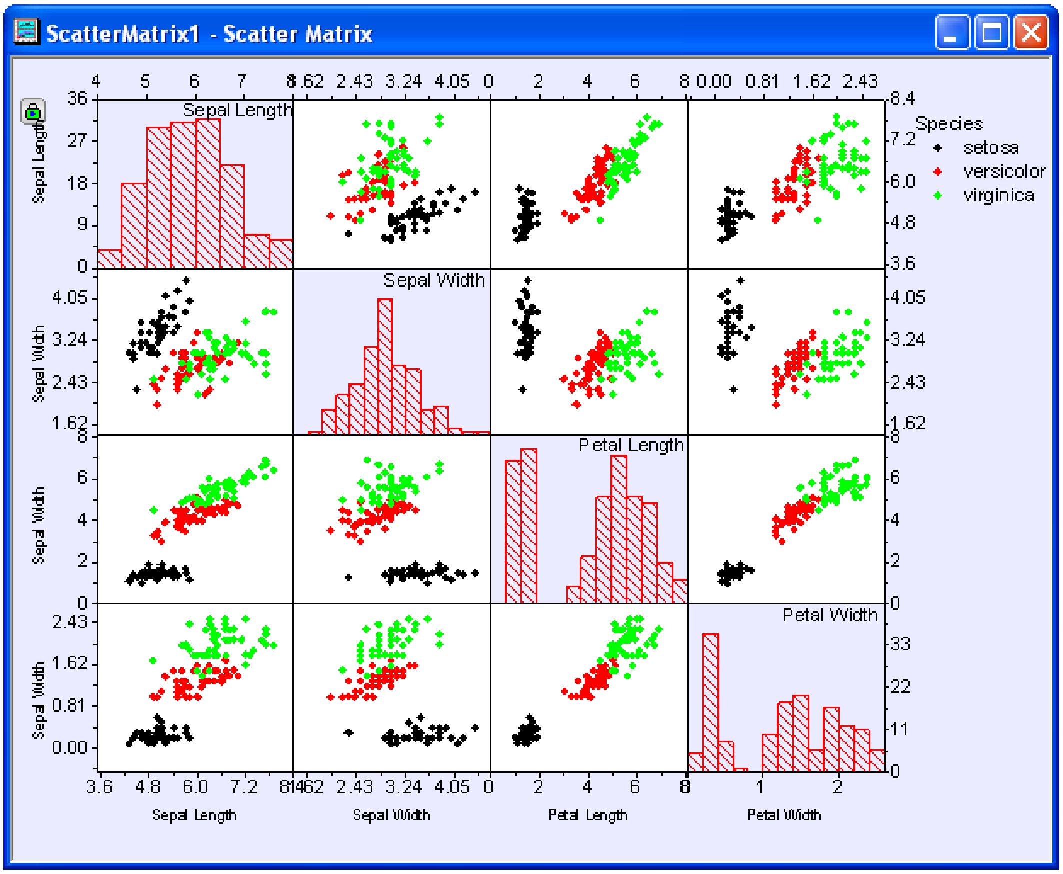
13. Double-click the X, Y and Z axis title and enter "X distance", "Y distance" and "Height". Add a text object "Pressure(psi)" above the color scale.
14. Your final graph should look like this:



2.5 Scatter Matrix

2.5.1 Summary

A scatter matrix is consists of several pair-wise scatter plots of variables presented in a matrix format. It can be used to determine whether the variables are correlated and whether the correlation is positive or negative. This tutorial will show you how to create a Scatter Matrix plot.



Minimum Origin Version Required: Origin 9.0 SR0

2.5.2 What you will learn

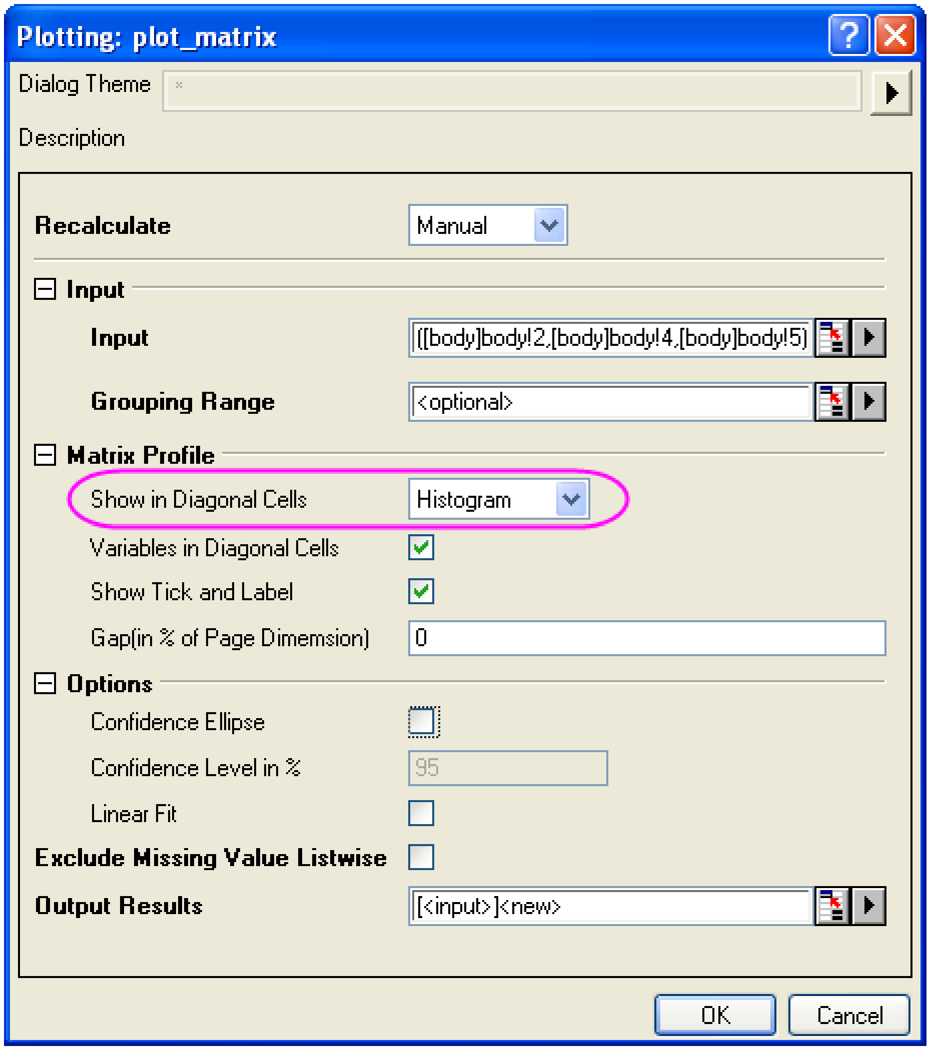
- How to create a Scatter Matrix plot with histogram
- How to customize Scatter Matrix plot
- How to set grouping range for showing color index

2.5.3 Steps

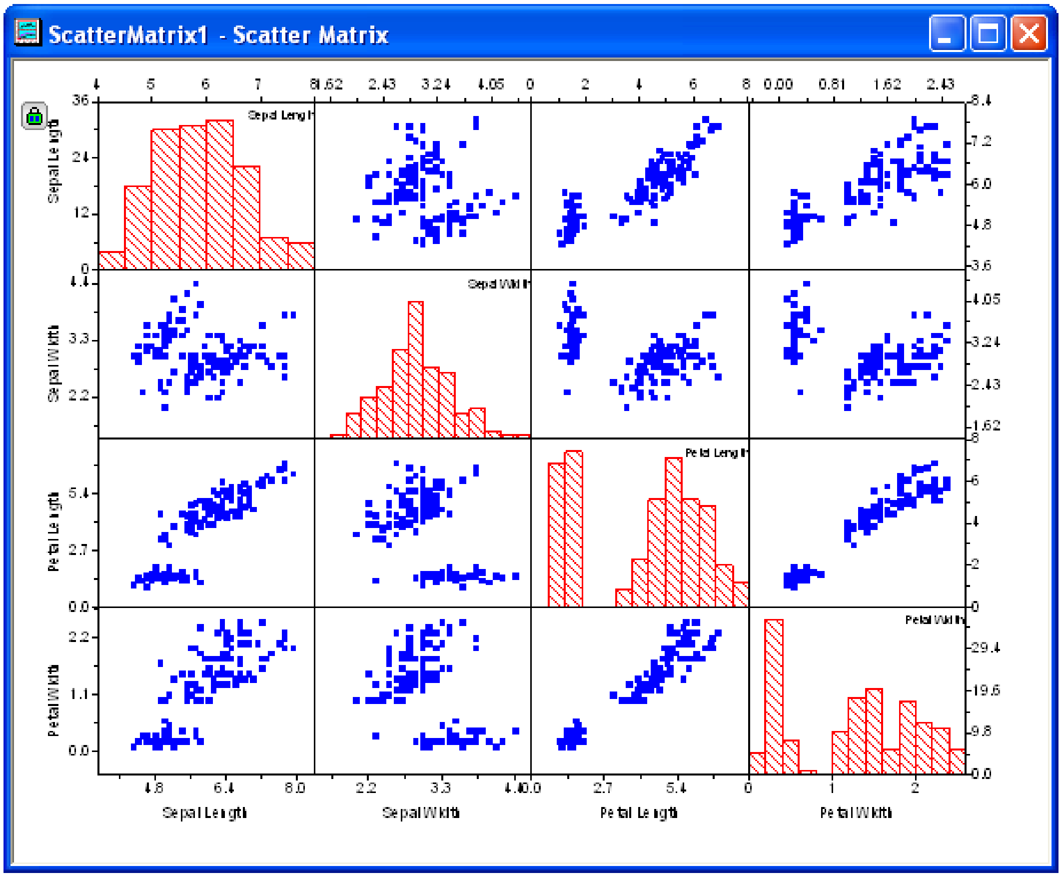
Creating Scatter Matrix plot

1. Start with an empty worksheet, select **File: Import: Single ASCII...** to open the Import Single ASCII dialog, browse to the *\Samples\Statistics* subfolder of the Origin program folder, and import the file *Fisher's Iris Data.dat*.
2. Highlight columns (A)~(D), and then select **Plot: Statistics: Scatter Matrix** from the Origin menu.

3. In the dialog, select **Histogram** in the **Show in Diagonal Cells** drop-down list.



4. Click **OK** to close the dialog. The graph should look like the following:

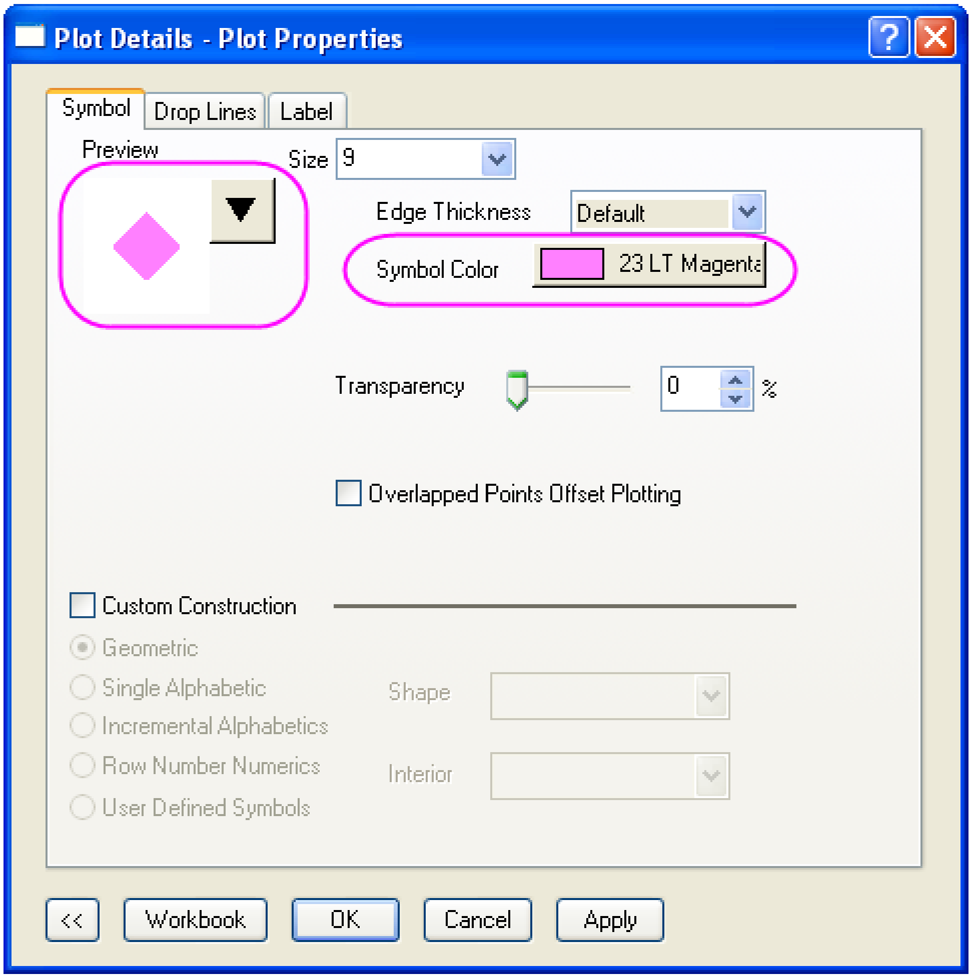


And the **PlotData1** sheet for the scatter matrix plot is generated in the same workbook.

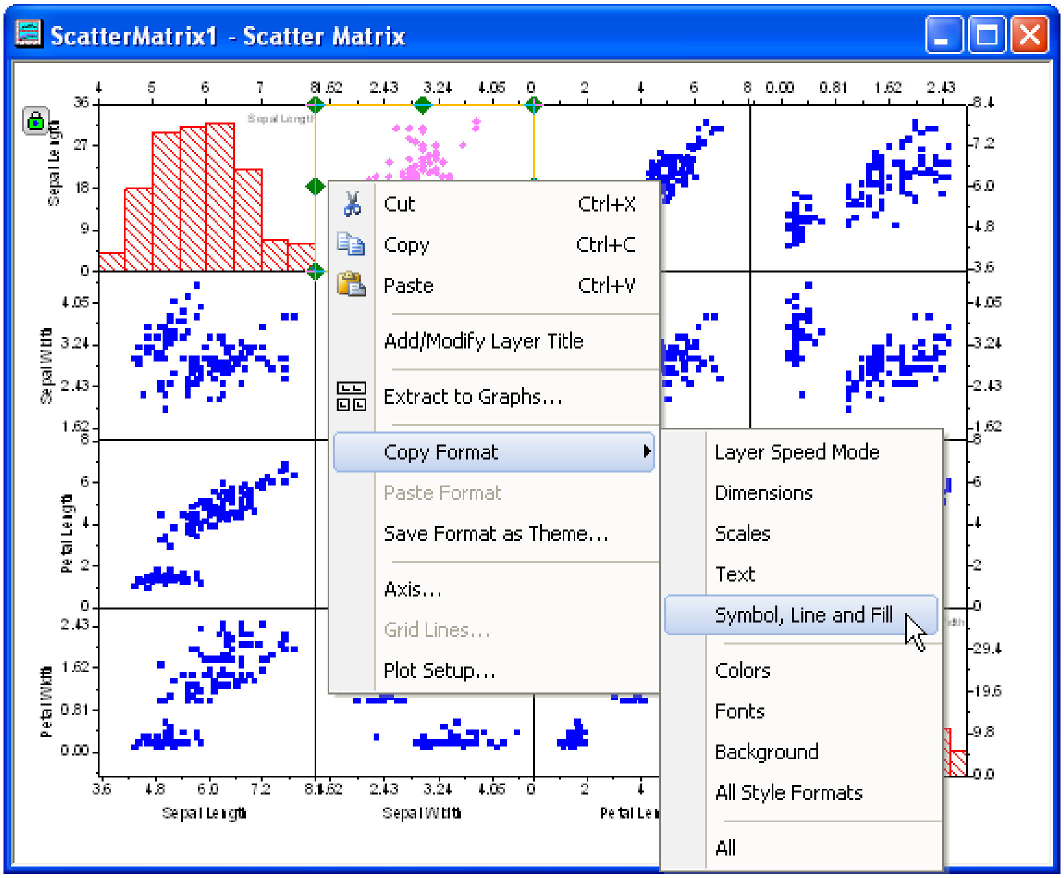
Customizing Scatter Matrix plot

There are multiple layers in the Scatter Matrix graph. This section will show you how to customize the background color, the type and color of a data plot and the tick label of axis of the scatter matrix.

1. Double click on a layer except in the diagonal cells to open the **Plot Details** dialog. Specify the type and color of the symbol as shown in the following image, and click **OK**.



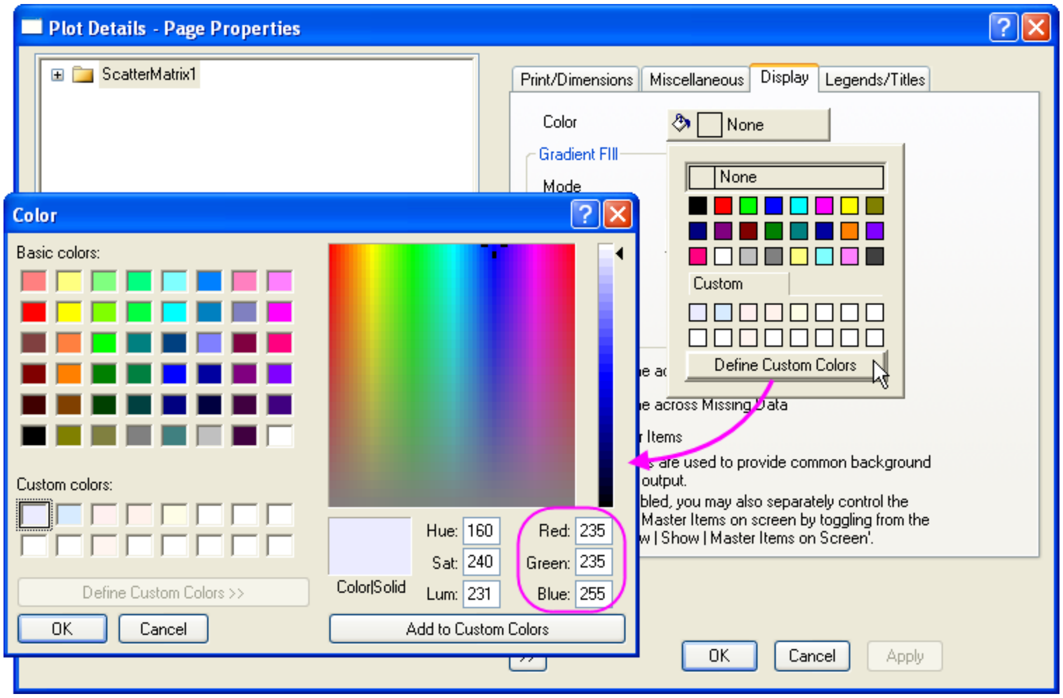
2. Click to select the layer to be updated, right-click on it and select **Copy format: Symbol, Line and Fill**.



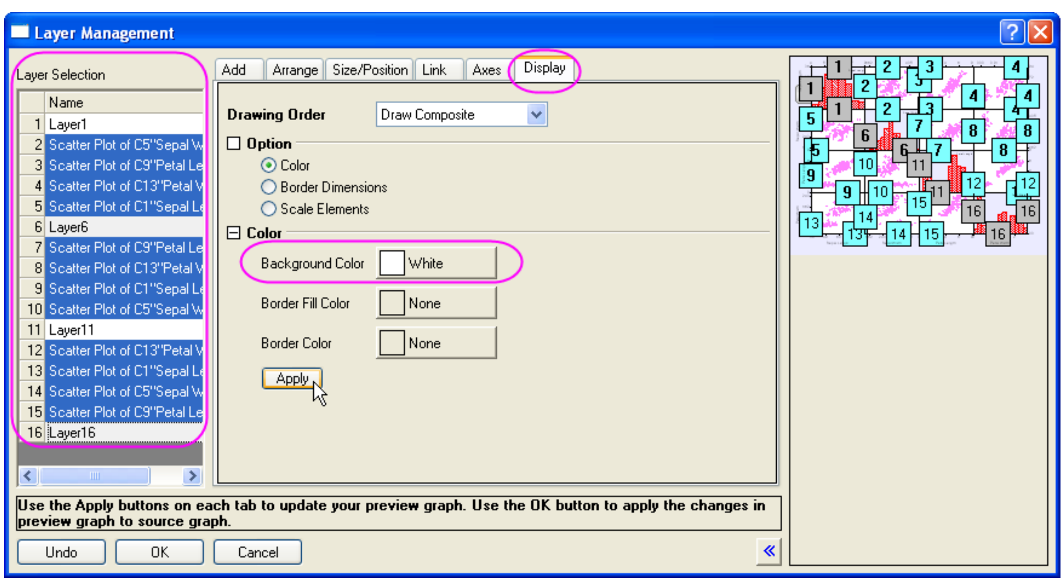
Then click on the white space of the graph, right-click, and select **Paste Format**. You can use the same method to copy the symbol color to other layers.

3. Right-click on that layer again, and select **Copy Format: Colors**. Then click on the white space of the graph, right-click, and select **Paste Format**.
4. Select **Format: Page Properties** to open the **Plot Details** dialog. Go to the **Display** tab, click on the **Define Custom Colors** button for the Color option.

5. In the **Color** dialog, specify the color as Red=235, Green=235, Blue=255. Then add as custom color and click **OK**.



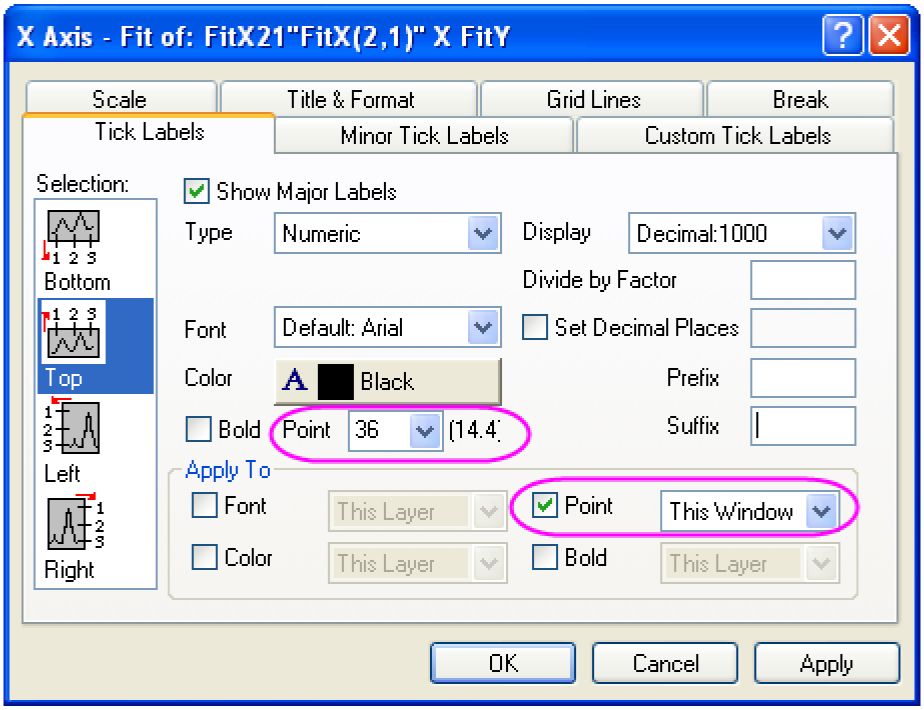
6. To specify background color of the layers with scatter plots, select the **Graph: Layer Management** to open the dialog, and go to the **Display** tab. Highlight all the layers listed in **Layer Selection** except the diagonal cells, and then specify the **Background Color** as **White**. Click on the **Apply** button to preview the change in the right panel of the dialog.



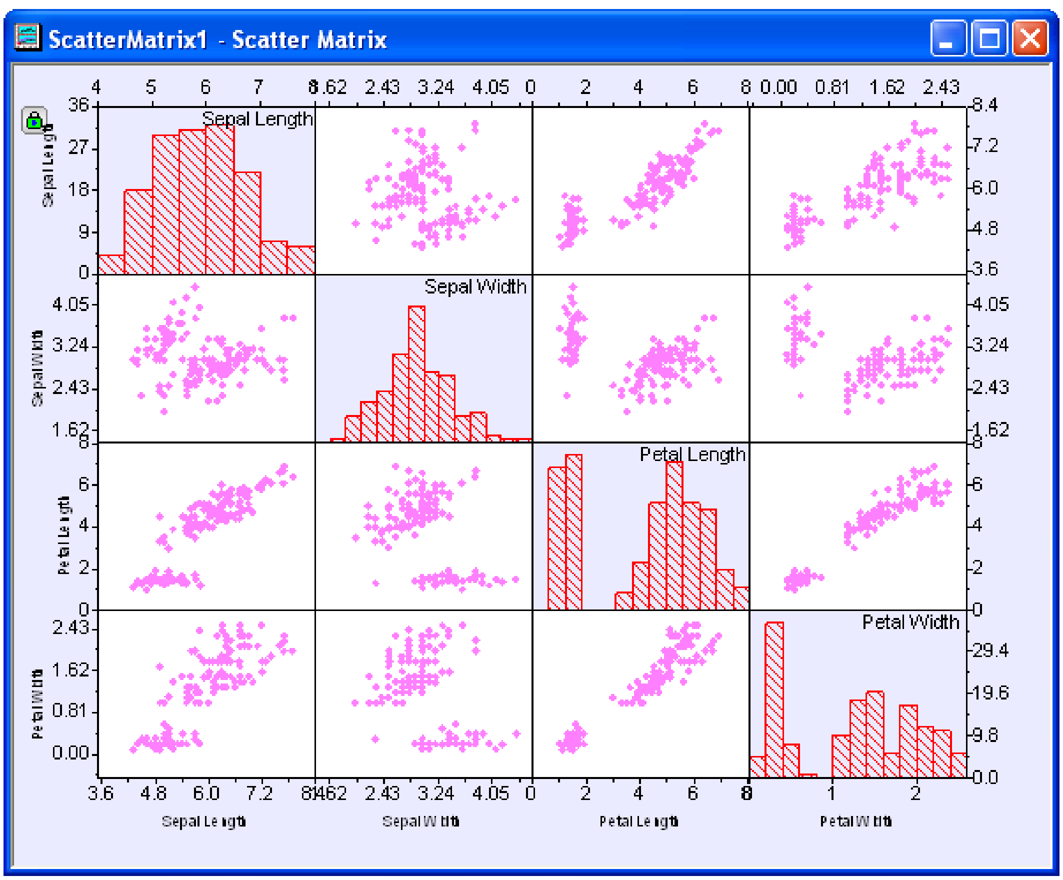
Then click **OK** in the dialog.

7. Double click on a tick label in the graph to open the **Axis** dialog. In the **Tick Labels** tab, specify **Point** to **36**. Then select the **Point** checkbox in the **Apply To** section, and choose **This Window**

in the drop-down list.

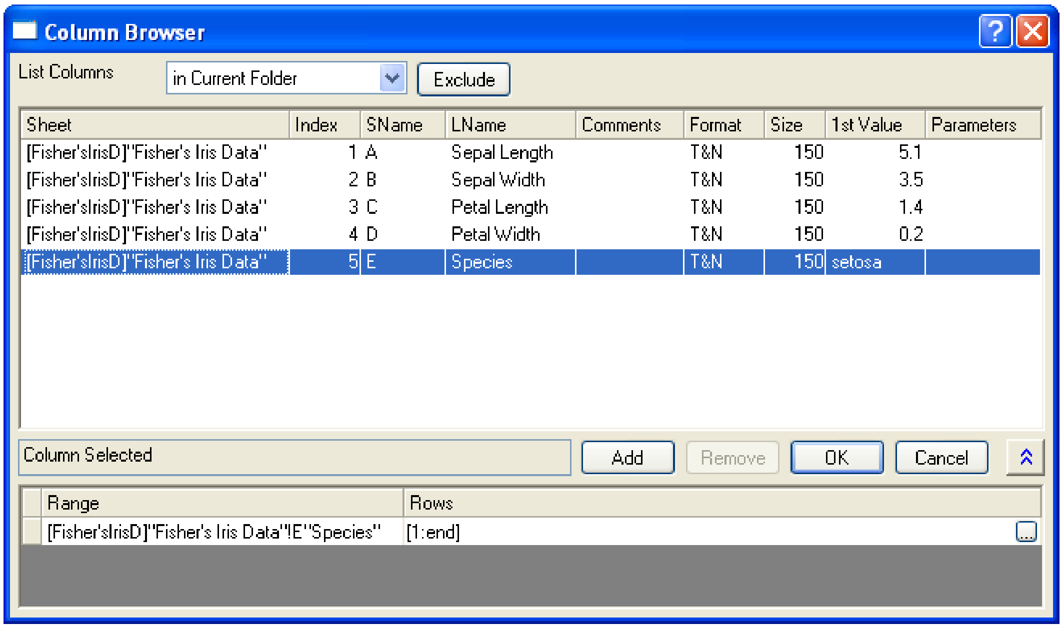


8. Highlight the variables in the diagonal cells, and use the **Size** button  in the **Format** toolbar to set the size to **36**. Then the graph will look like the following:



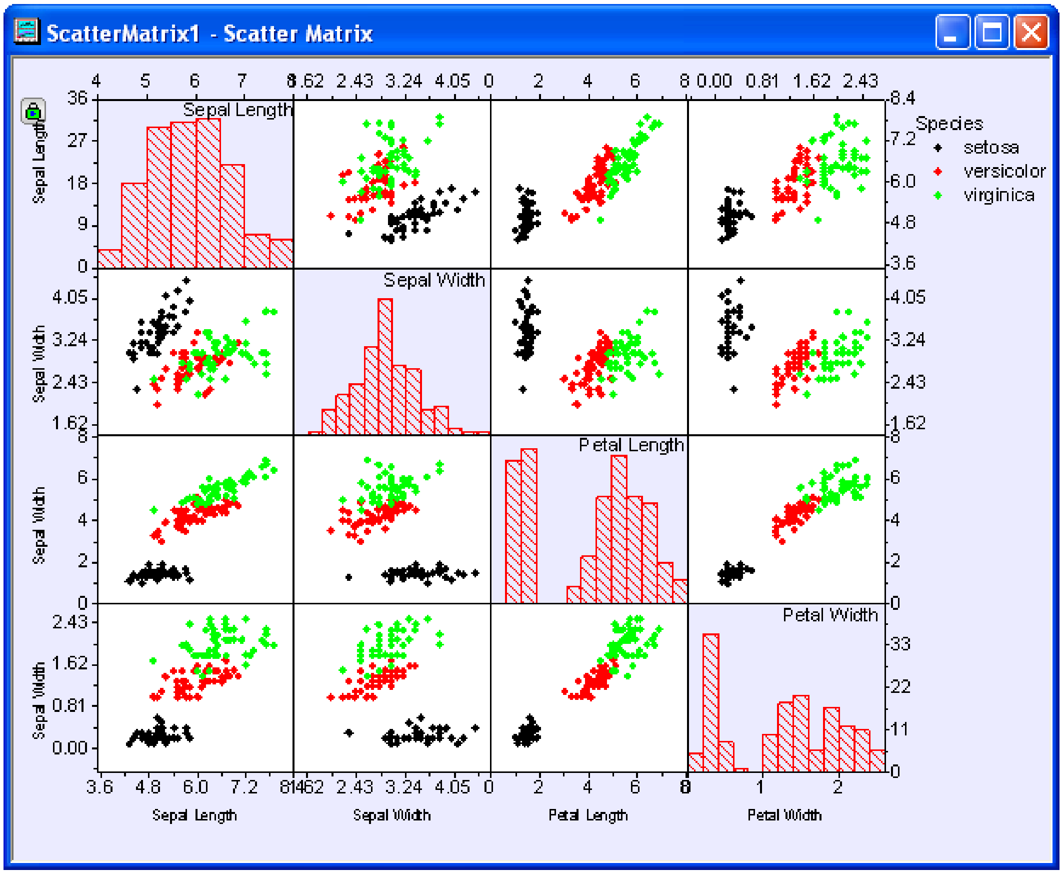
Add Grouping Range

1. To add a **Grouping Range**, click on the green lock icon on the upper-left corner. And then select **Change Parameters** to bring back the **Plotting: plot_matrix** dialog.
2. Click the triangle button next to the **Grouping Range** option. Click on the **Select Columns** to open the **Column Browser** dialog, and then choose column E (Species) as the group range. Click **OK**.



3. Click the **OK** button in the **Plotting: plot_matrix** dialog.

Your final graph should look like this:



2.6 IIR Filter

2.6.1 Summary

In OriginPro, it is possible to design, analyze, and implement IIR (Infinite Impulse Response) digital filters. The IIR filter supports four methods, including **Butterworth**, **Chebyshev Type I**, **Chebyshev Type II**, and **Elliptic**.

This provides users more choices in signal processing.

Minimum Origin Version Required: 9.0 SR0

2.6.2 What You Will Learn

This tutorial will show you:

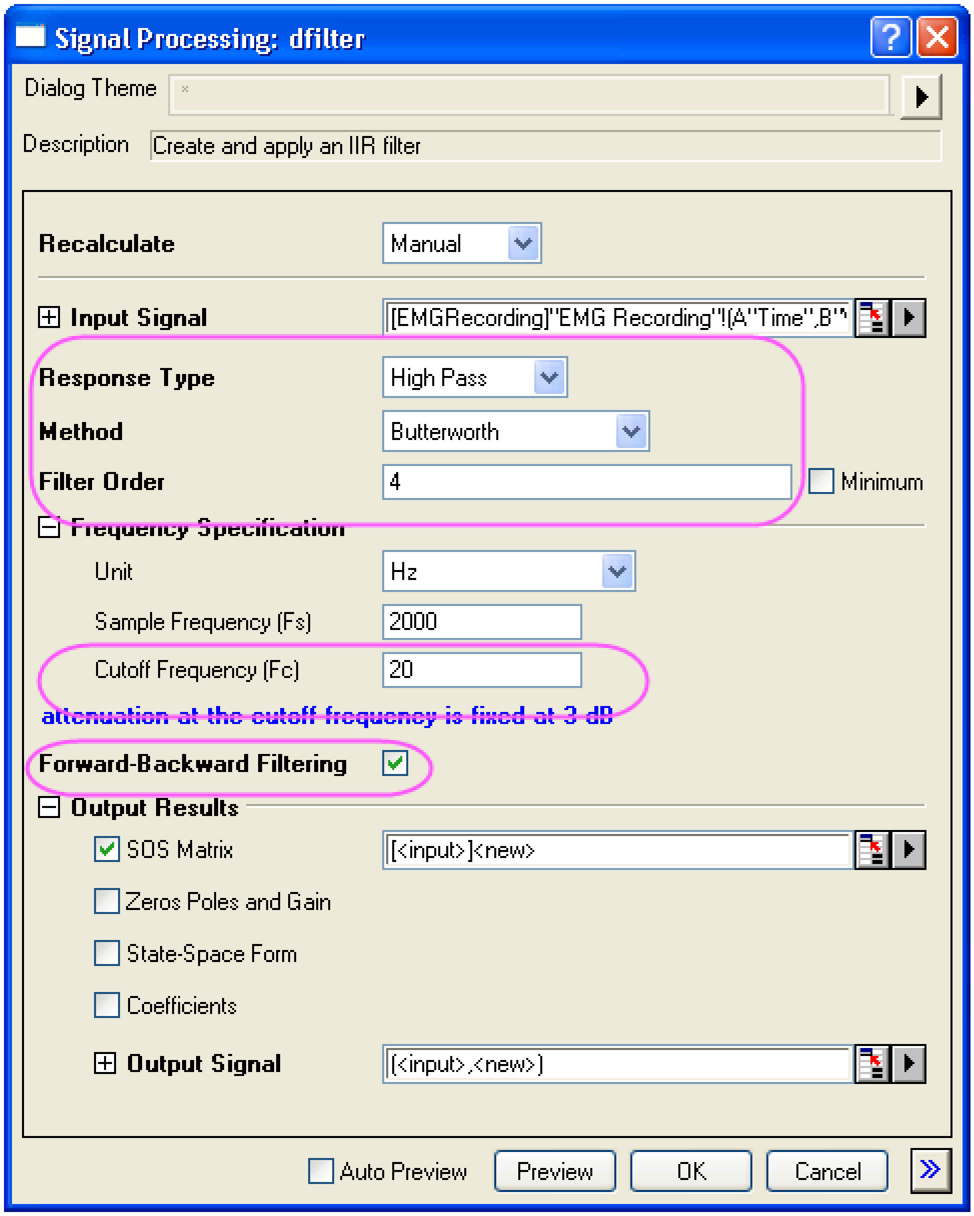
- How to design and apply an IIR filter

- A comparison between IIR filter and FFT filter

2.6.3 Steps

Design and Apply IIR Filter

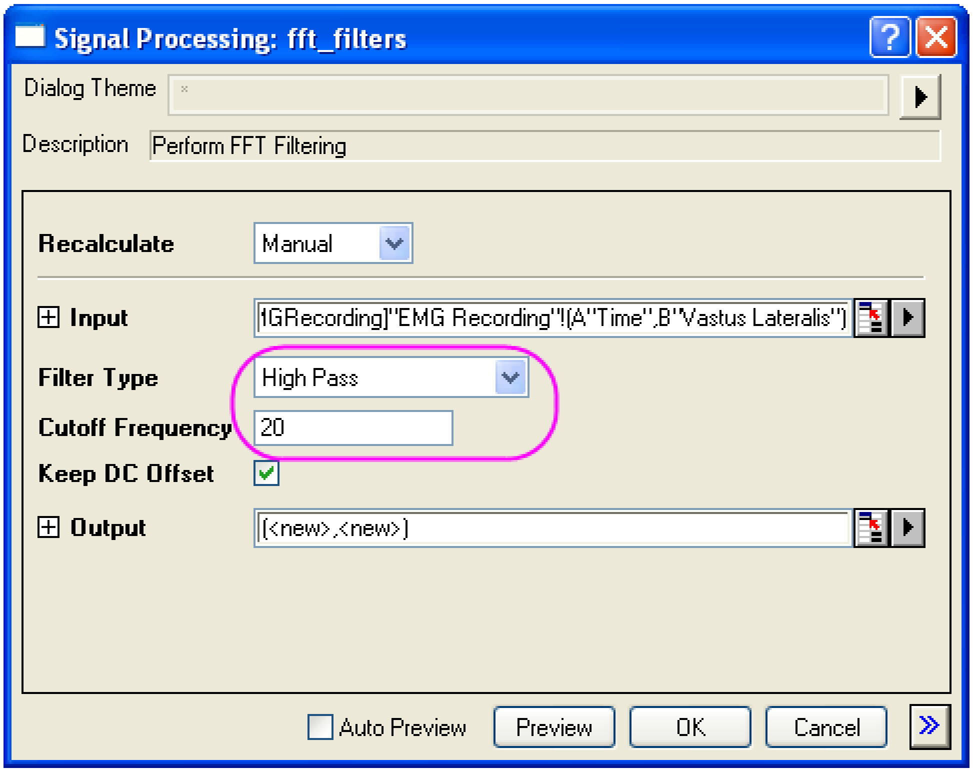
1. Start with a new worksheet and import the **EMG Recording.dat** file from *\Samples\Signal Processing*.
2. Highlight column B and Select **Analysis:Signal Processing:IIR Filter** from the top menu to open the dialog.
3. Change the Response type as **High Pass**, keep the Method as **Butterworth**, uncheck the **Minimum** for Filter Order and set it as **4**. In the Frequency Specification branch, set the Cutoff Frequency(Fc) as **20**, then check the **Forward-Backward Filtering**. The dialog settings should look like the following figure, and the IIR filter is designed.



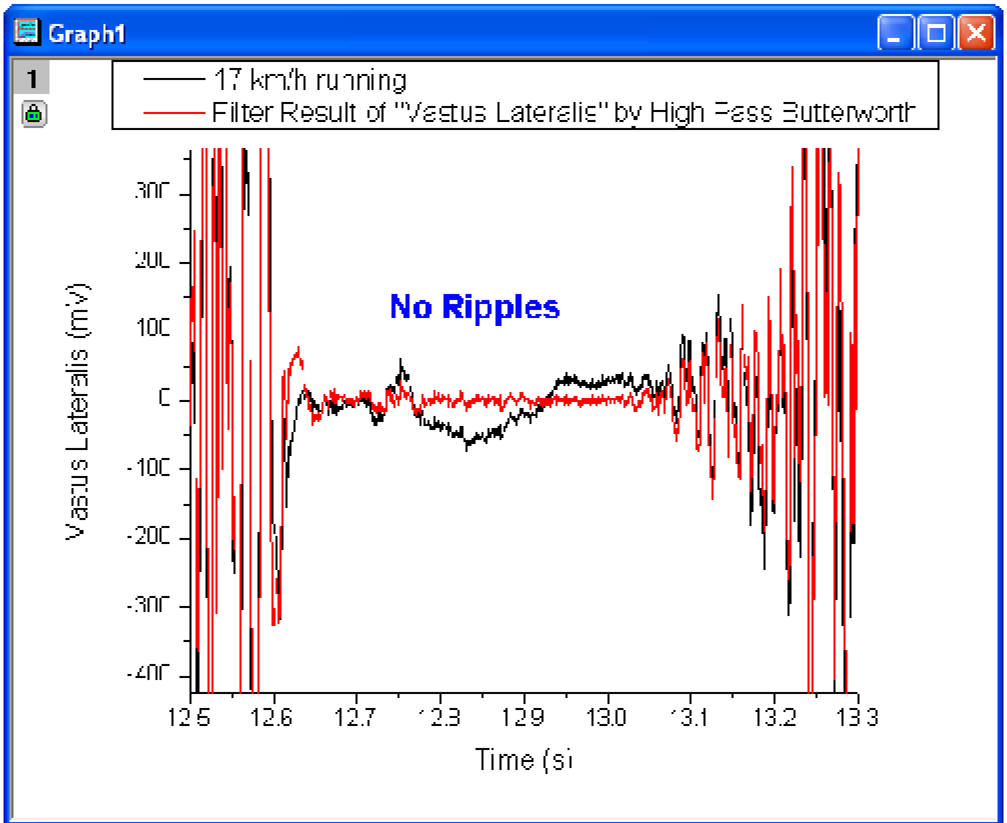
4. Click **OK** to apply the created IIR filter to the input dataset.
5. A new column will be added to the original data as a new column of filtered data and a new SOS Matrix worksheet.

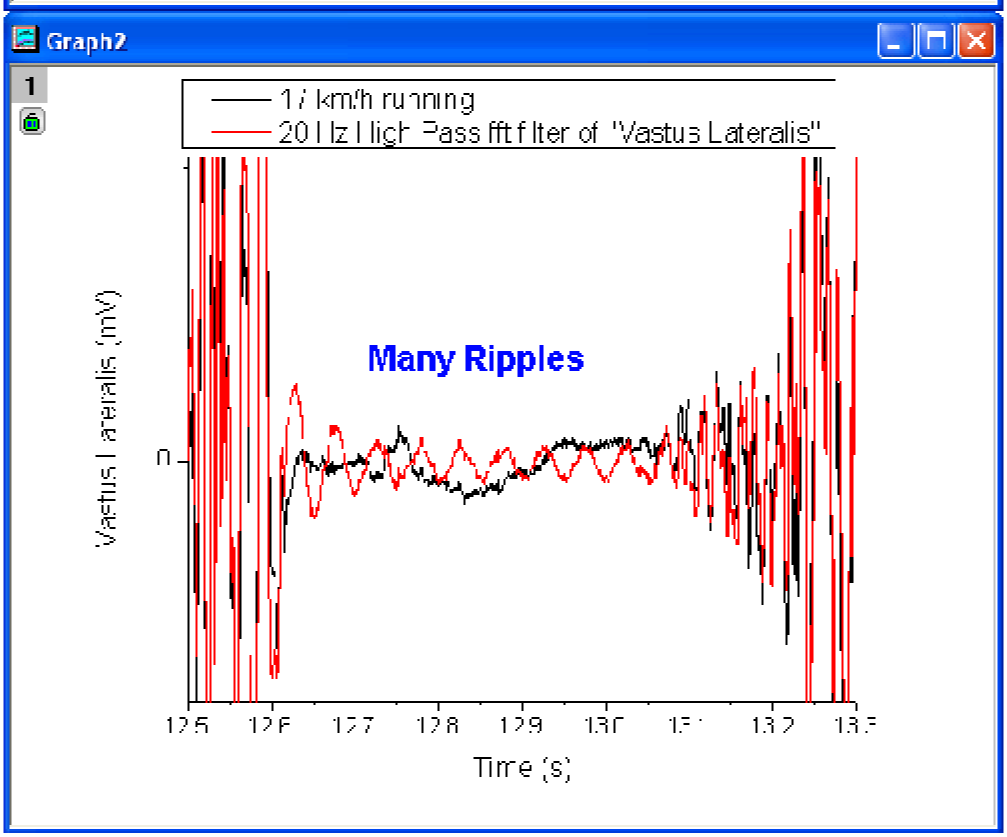
<u>Compare Results with FFT Filter</u>

1. Highlight column B in the original worksheet, perform FFT filter by **Analysis:Signal Processing:FFT Filters**.
2. In the opened dialog, choose **High Pass** for Filter Type and set **20** as Cutoff Frequency.



3. Column C in the EMGRecording worksheet is the filtered result of the previously designed IIR filter, highlight column B and column C to generate a line plot with the button (Graph 1).
4. Use the scale in button to zoom the area between 12.5s and 13.3s.
5. Column E in the EMGRecording worksheet is the filtered result of the FFT filter, highlight column B and column E to generate a line plot with the button (Graph 2).
6. Also use the scale in button to zoom the area between 12.5s and 13.3s, the two graphs could be used for visualized comparison.

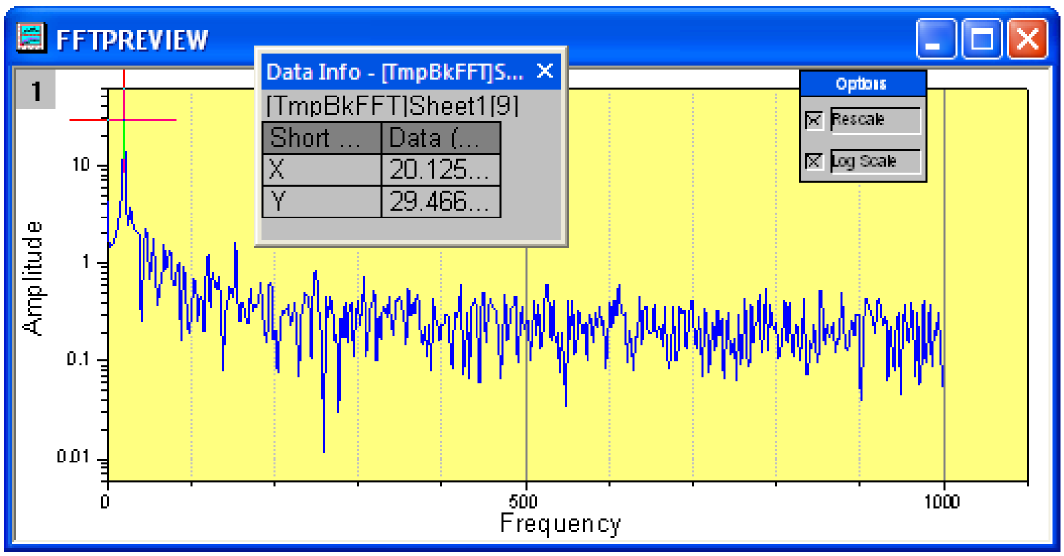

Graph1
17 km/h running
Filter Result of "Vastus Lateralis" by High Pass Butterworth
No Ripples
Vastus Lateralis (mV)
Time (s)


Graph2
17 km/h running
20 Hz High Pass fft filter of "Vastus Lateralis"
Many Ripples
Vastus Lateralis (mV)
Time (s)

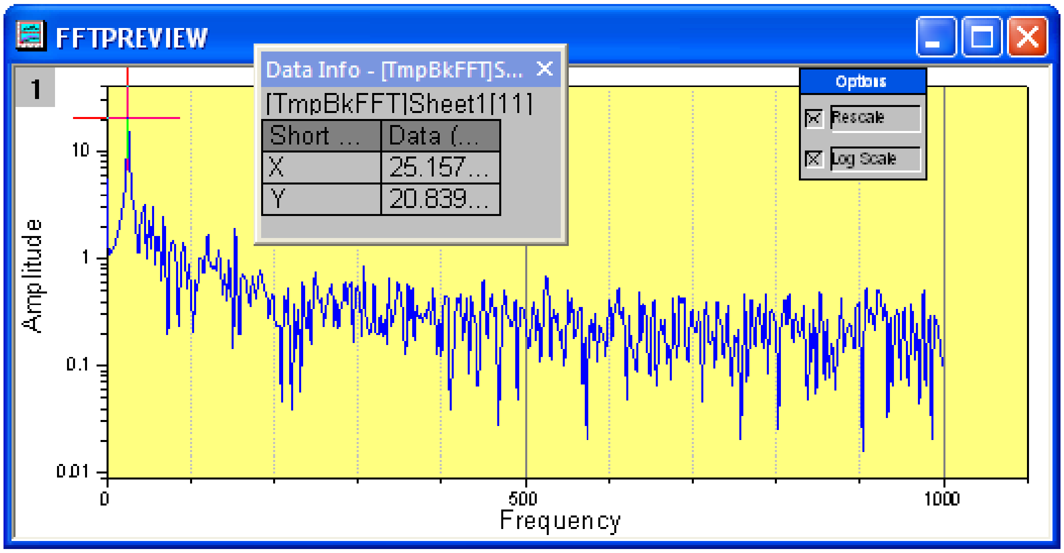
- Note that there are many ripples in the FFT filter result, but almost no ripples in the IIR filter result.

Ripples in FFT Filter

1. Highlight column E and click on the  button to create a line plot (Graph 3).
2. Activate Graph 3, choose **Gadget:FFT** and set the X Scale as From **12.664** To **13.052**.
3. Click OK to bring up the preview window, in which ripples are almost pure 20.125 Hz sine.



4. Now we would try to remove the ripples at 20.125 Hz by applying another high pass filter at 25 Hz, keep highlighting column E and select **Analysis:Signal Processing:FFT Filters**.
5. Select **High Pass** for Filter Type and set Cutoff Frequency as **25**.
6. The result is listed in Column G, highlight column G and click on the  button to create a line plot(Graph 4).
7. Activate Graph 4, choose **Gadget:FFT** and set the X Scale as From **12.664** To **13.052**, in the preview window, there are still ripples, and they are shifted from 20.125 Hz to 25.157 Hz.



- Note that the ripples could not be removed by FFT filter for this dataset.

3 Data Analysis

Topics covered in this section:

3.1 Gadgets

3.1.1 Quick Sigmoidal Fit Gadget

Summary

The Quick Sigmoidal Fit gadget can be used to quickly perform a sigmoidal fit in the ROI (Region of Interest) range.

Minimum Origin Version Required: Origin 8.6 SR0

What you will learn

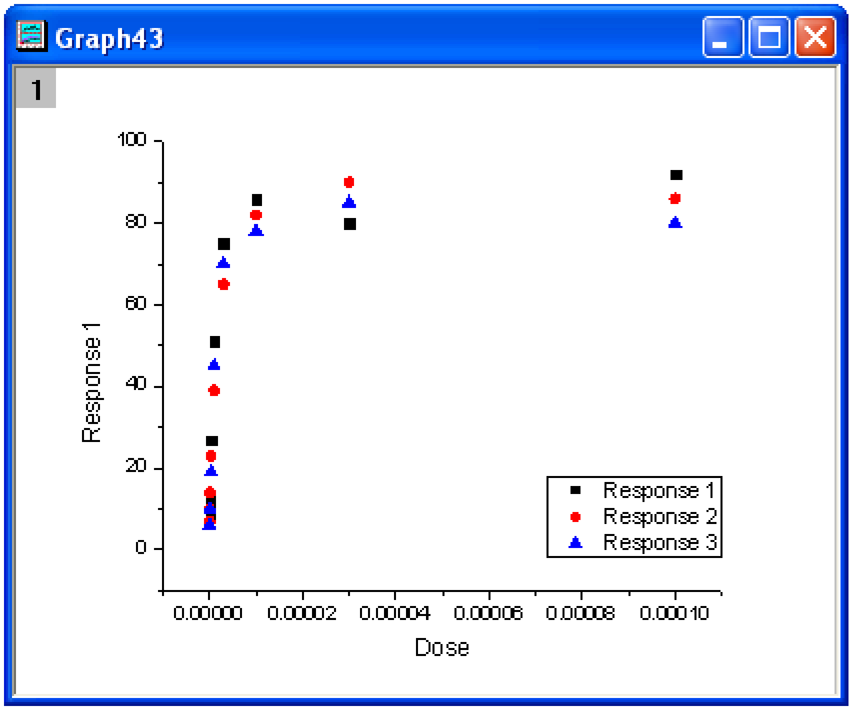
- How to use the Quick Sigmoidal Fit gadget on a graph.
- How to do the settings for fitting.
- How to switch to NLFit dialog.
- How to output the fitting result.
- How to find X/Y values on the fit curve.

Steps

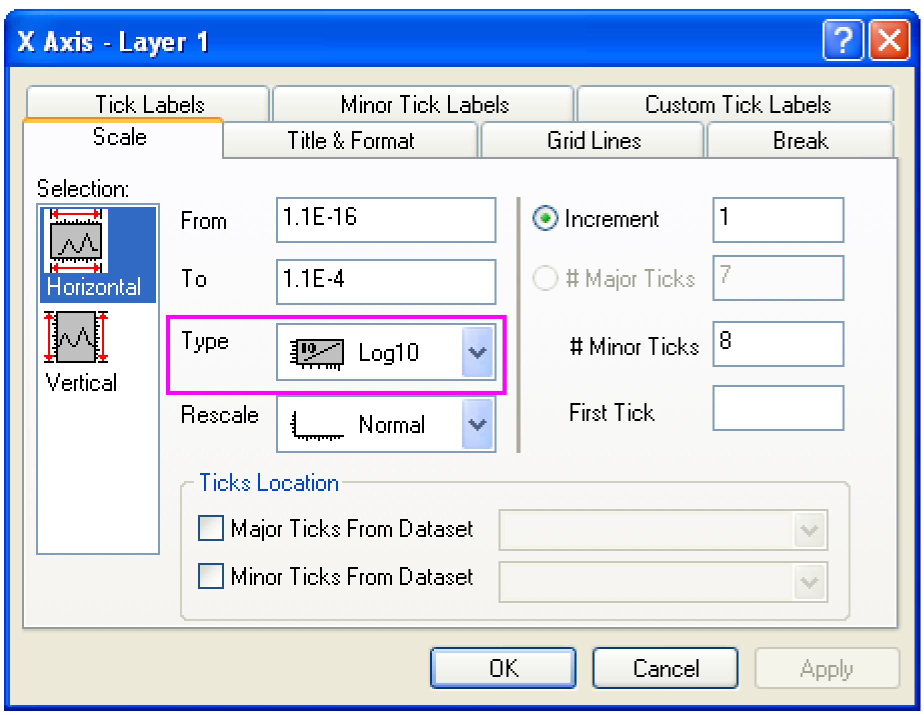
This tutorial is associated with the **Analysis: Quick Sigmoidal Fit Gadget** folder in the Analysis project (\Samples\Analysis.opj) which can be opened by selecting **File: Open Sample Projects: Analysis** from the main menu.

Quick Fit

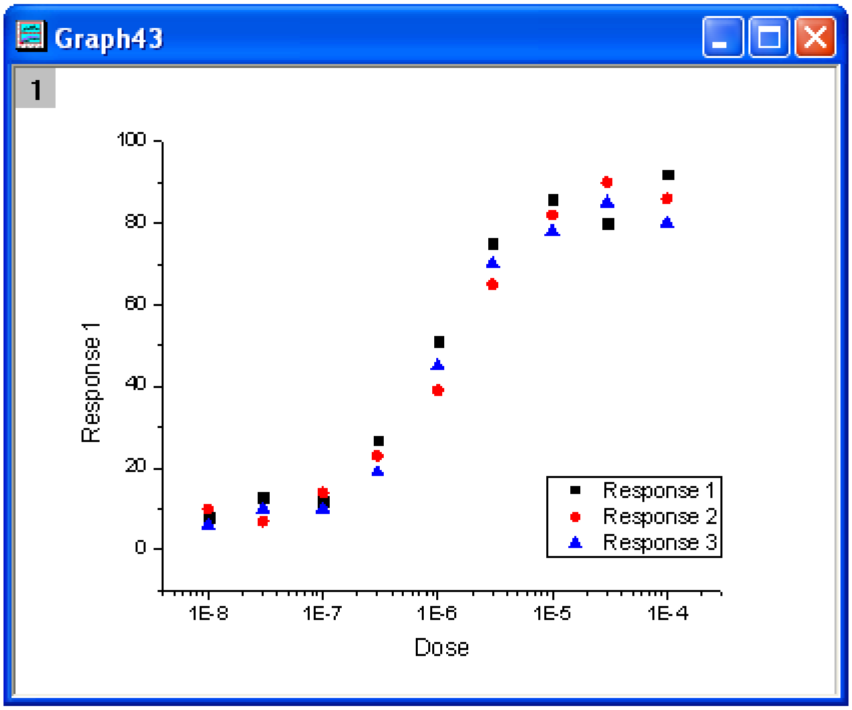
1. Highlight the Col(A)~col(D) in workbook to plot a scatter graph.



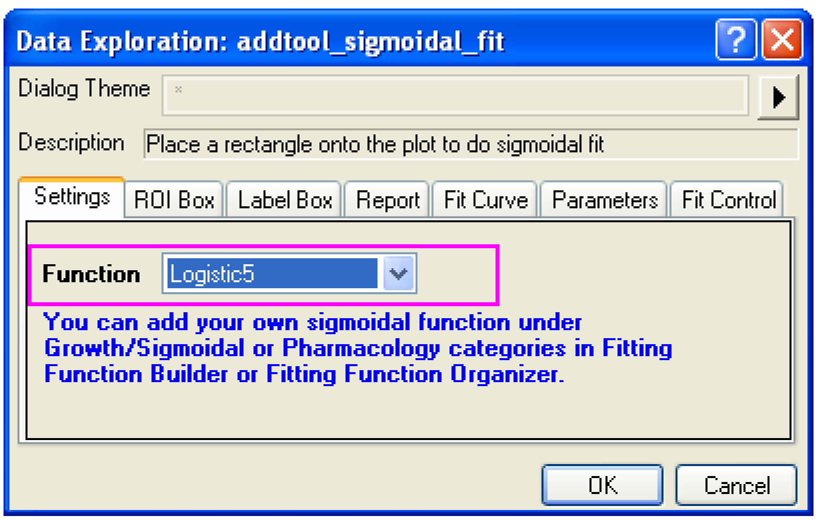
2. Double-click the X axis to open **Axis** dialog. Set the **Type** as *Log10* and click **OK** button to close the dialog.



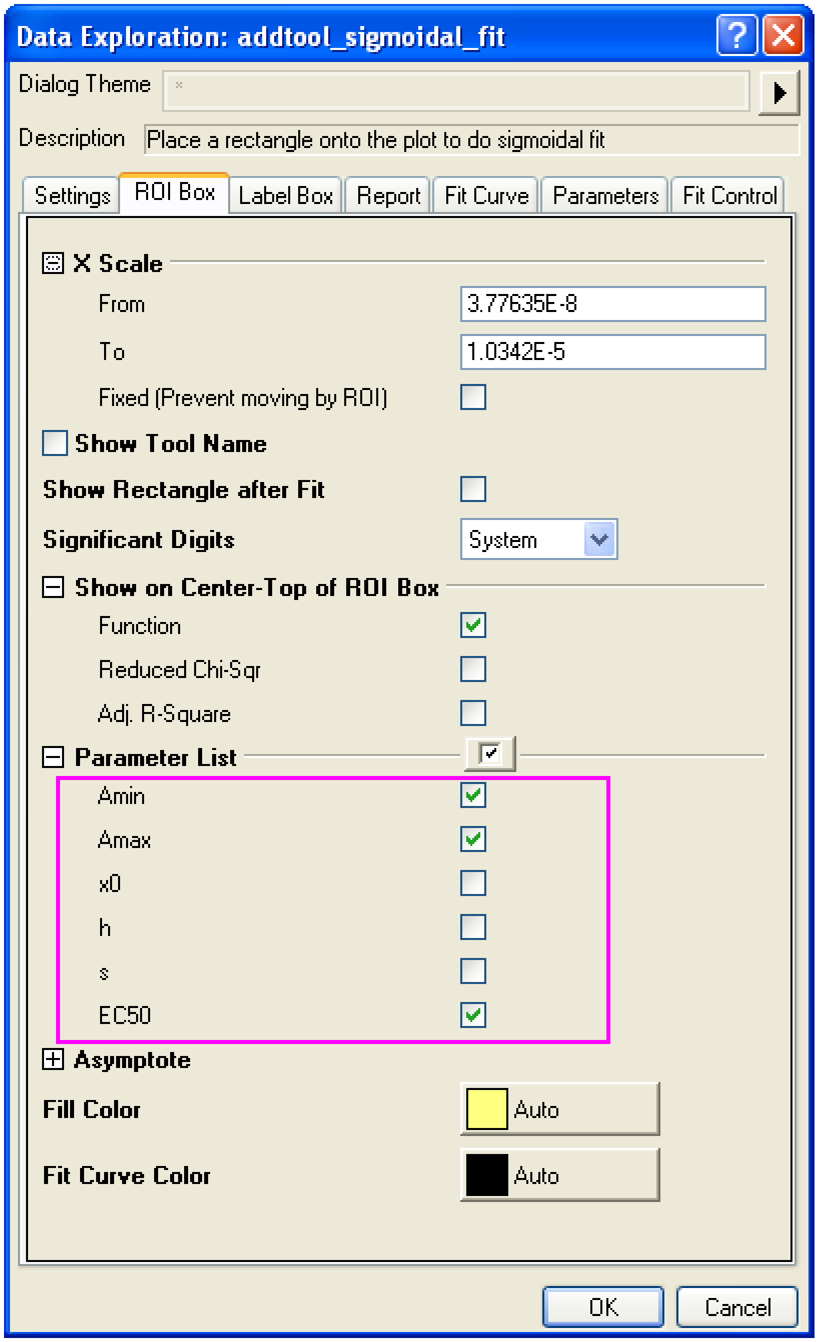
3. Rescale the graph by clicking the **Rescale** button to show the plot in right scale.



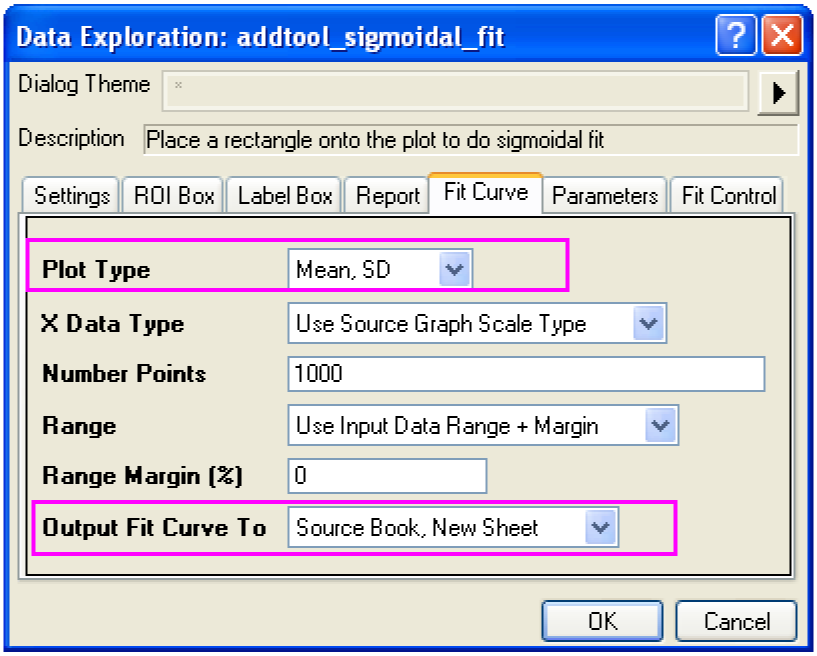
4. Select **Gadgets: Quick Sigmoidal Fit...** from the main menu to open the **addtool_sigmoidal_fit** dialog. Select the function *Logistic5* from the **Function** drop-down list in the **Settings** tab.



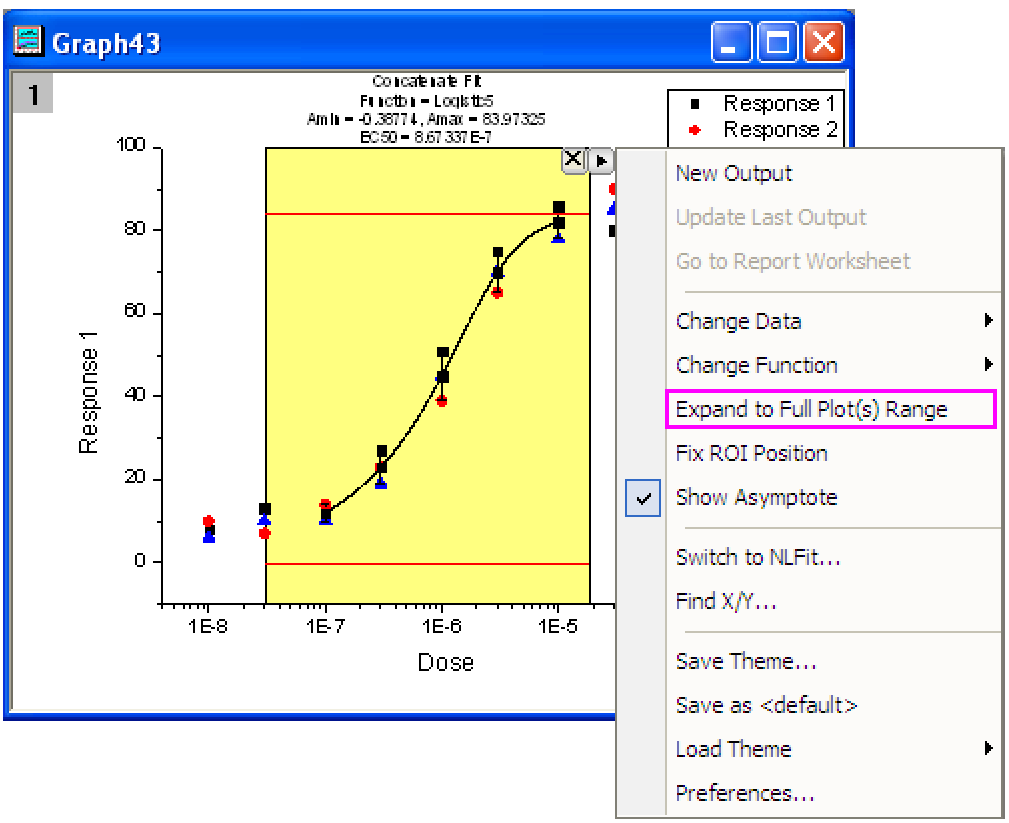
5. Go to the **ROI Box** tab, uncheck the parameter *x0*, *h*, and *s* under the **Parameter List** branch.



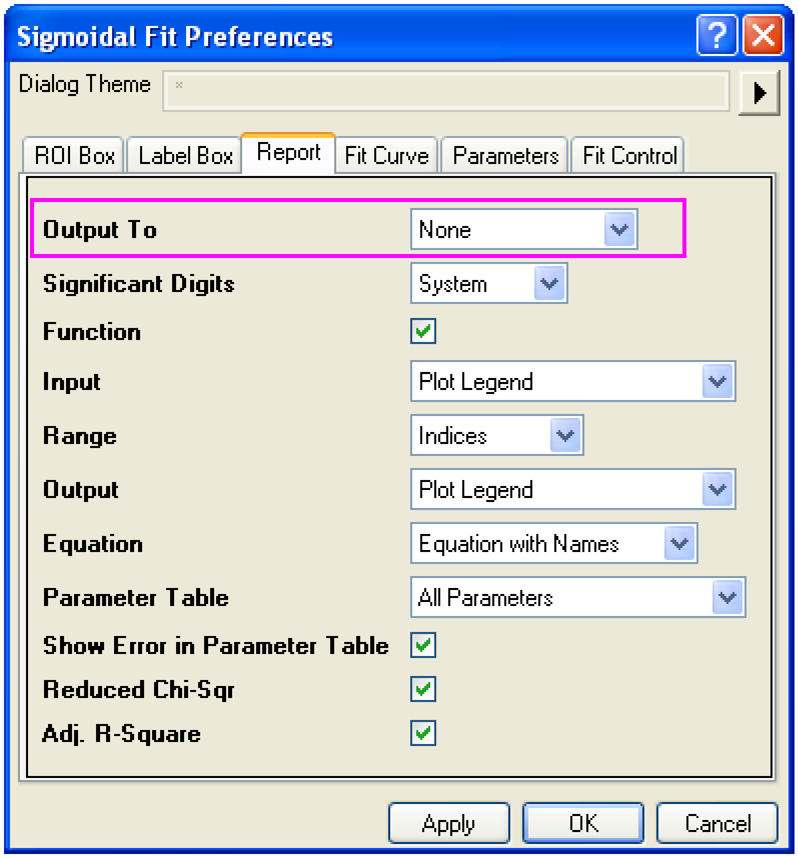
6. Go to the **Fit Curve** tab, select **Mean, SD** from the **Plot Type** drop-down list and **Source Book, New Sheet** from the **Output Fit Curve To** drop-down list.



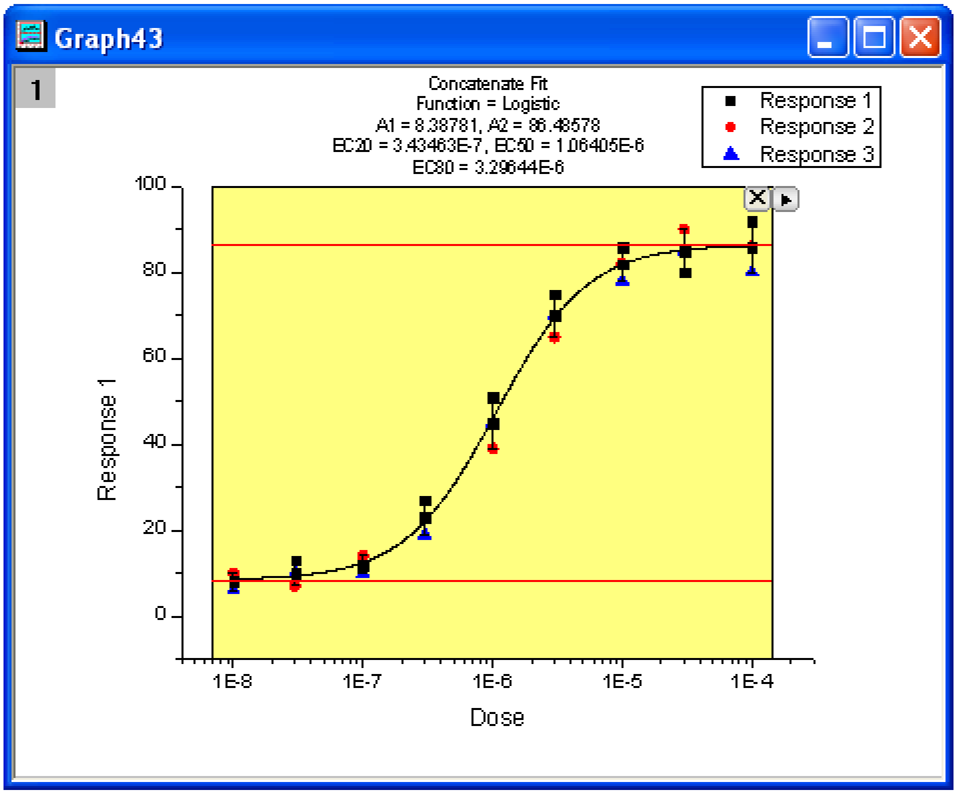
7. Click **OK** button to add the ROI box on the graph. At the top right corner of the ROI box, click the Arrorw button to select **Expand to the Full Plots Range** from the context menu. The ROI box will cover the full data range on the graph.



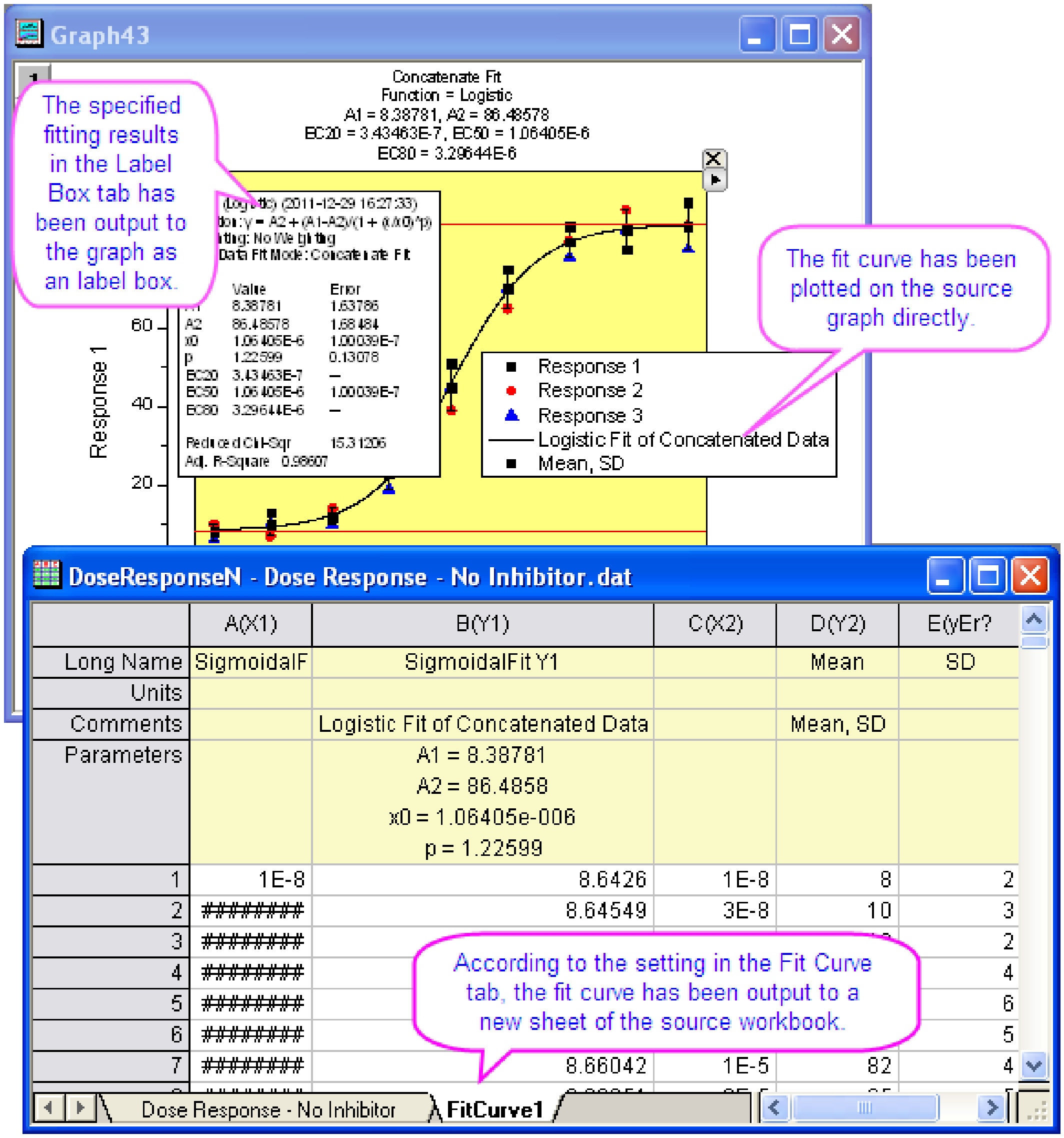
8. Click the arrow button again to select **Preferences...** from the menu to open the **Sigmodial Fit Preferences** dialog. In this dialog, go to the **Report** tab and set **Output To** as *None*.



9. Click **OK** button to close the dialog. Then click the arrow button to select **Change Function: Logistic** to change the fitting function as *Logistic*. The label text on top of the ROI box will be updated automatically.



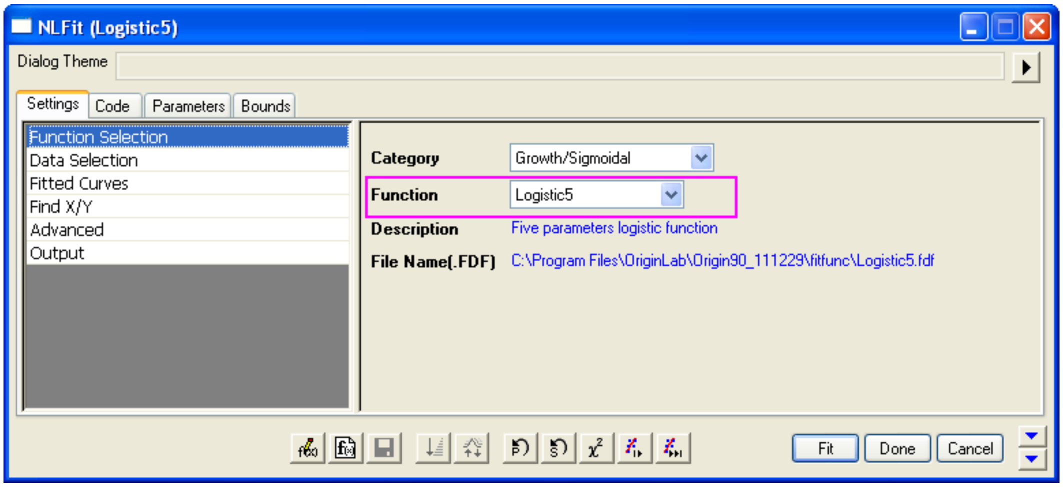
10. Then click the arrow button to select **New Output** to output the fitting result to the worksheet and graph.

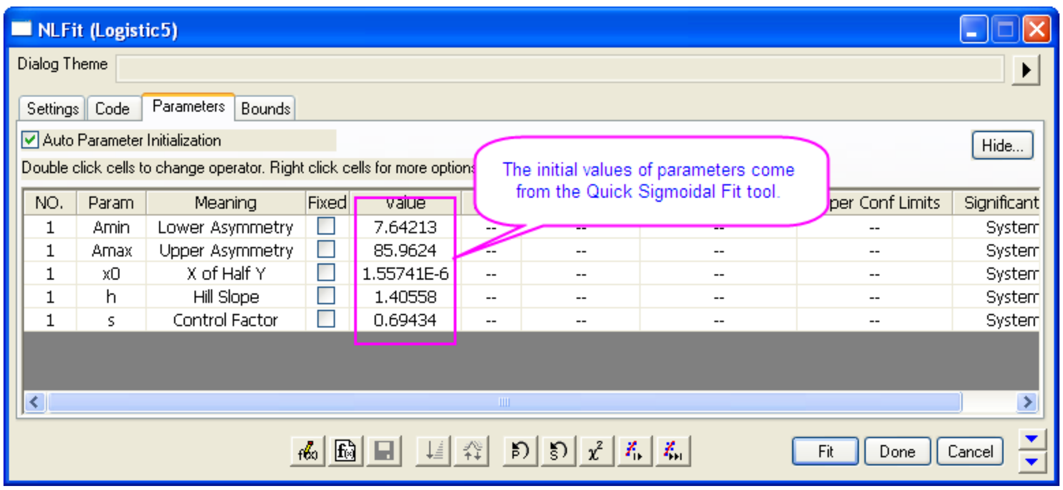


Connect to NLFit

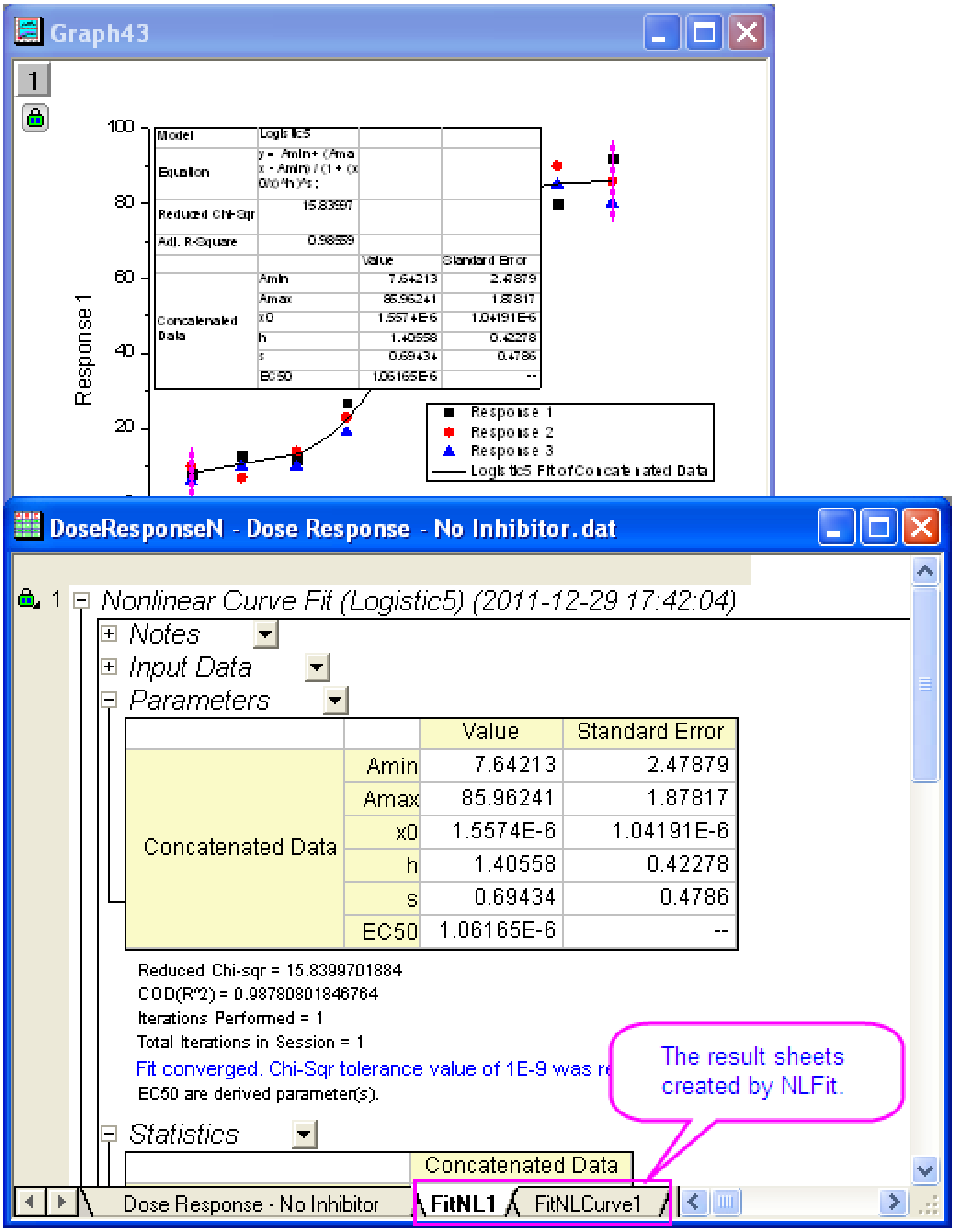
Quick Sigmoidal Fit gadget provides ability to switch to NLFit tool. With this feature, you can get a detailed fit report with the current fitting settings in the gadget.

1. Redo step 1 through step 8 as shown above.
2. Click the arrow button to select **Switch to NLFit...** to open the **NLFit** dialog with the inherited settings.





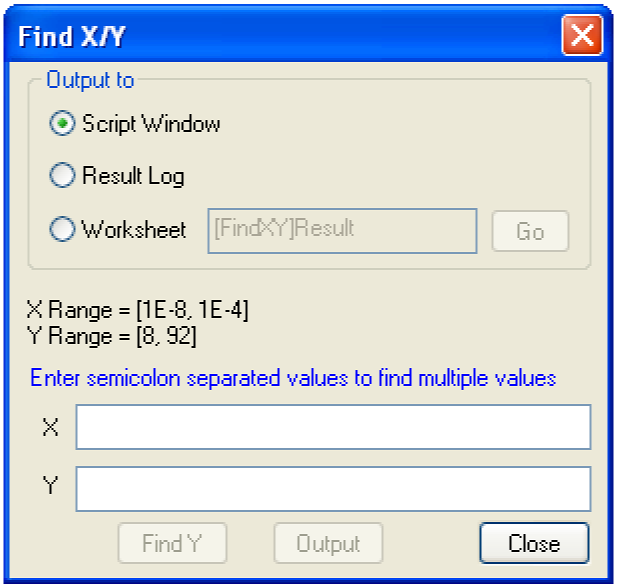
3. Click **Fit** button to do fitting. The results will be output to the result sheets and the source graph.



Find X/Y

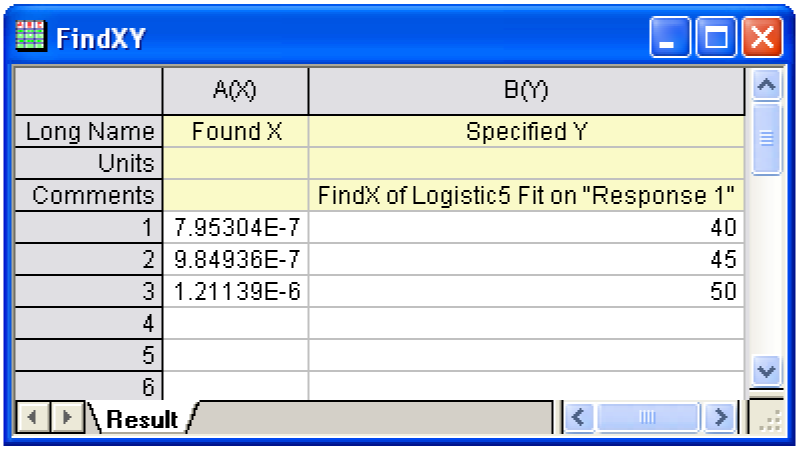
This gadget also provides ability to find X/Y value on the fitted curve quickly.

1. Redo step 1 through step 8 as shown in the first section.
2. Click the arrow button to select **Find X/Y...** from the menu to open the **Find X/Y** dialog.



3. Select the **Worksheet** radio box and enter *40;45;50* for **Y**, then click **Find X** to show the corresponding X values in the blank X box. After that, click the **Output** button, the X and Y values will be output to the specified worksheet.

4. Click **Go** button after the **Worksheet** radio box to open the **Find X/Y** workbook. Then click the **Close** button to close the **Find X/Y** dialog.



3.1.2 Integrate Gadget

Summary

The Integrate Gadget performs numerical integration of a data plot to calculate the area under the curve. You can select an arbitrary range of the data plot using the region of interest (ROI) object displayed in the graph.

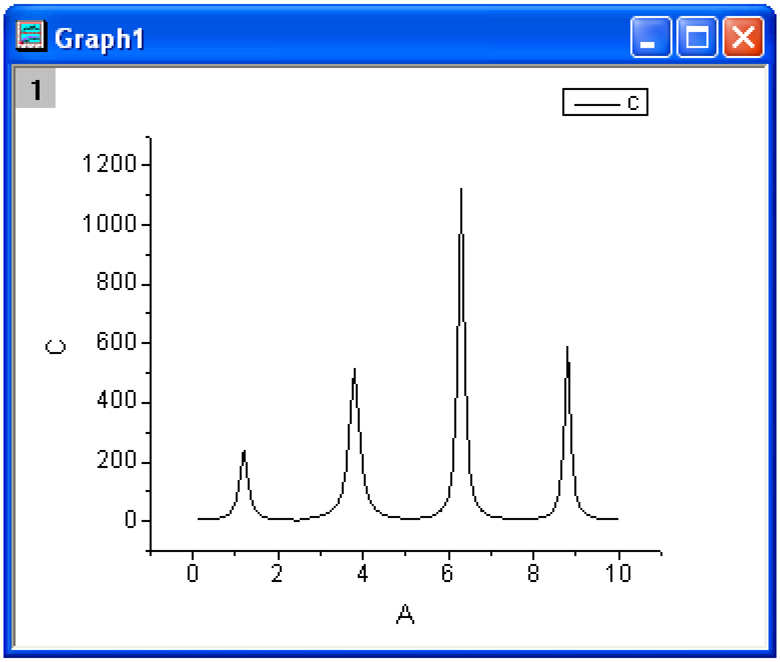
What you will learn

- How to easily integrate a data plot on a rectangular region.
- How to specify the integration limits and baseline.
- How to display the integral curve inside the ROI.
- How to calculate quantities including: peak area, peak height, peak center, and FWHM.

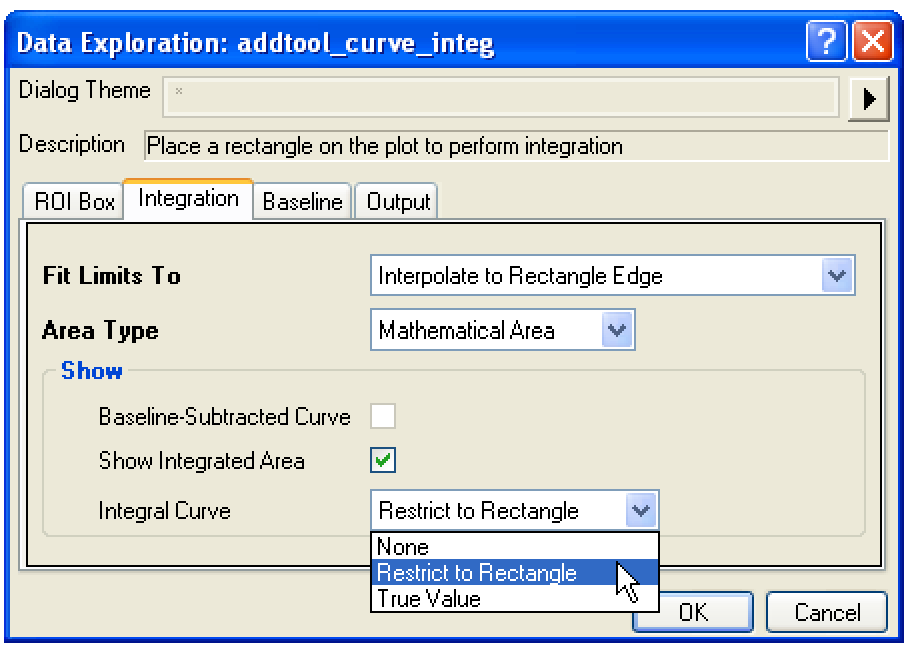
Integrate and output the quantities

1. Start with a new workbook and import the Origin sample data *Multiple Peaks.DAT* which is located in *<Origin Program Folder> \Samples\Curve fitting*.

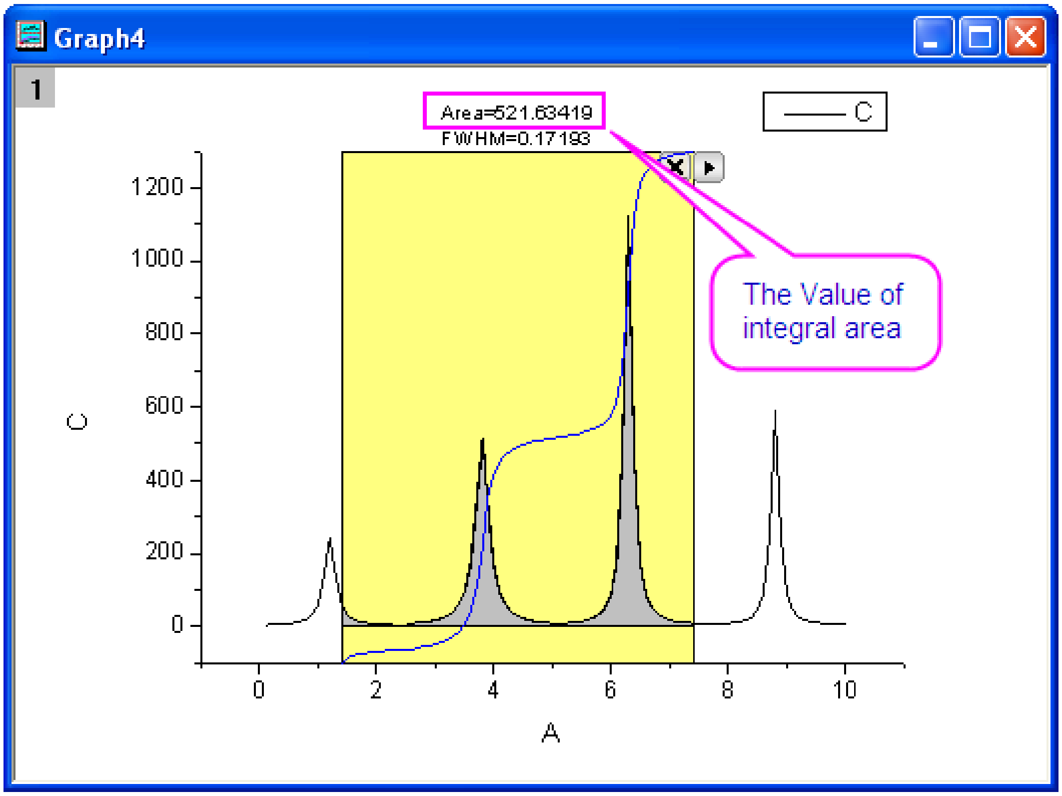
2. Highlight the Col(C) and select **Plot: Line: Line** from the Origin menu to draw a graph.



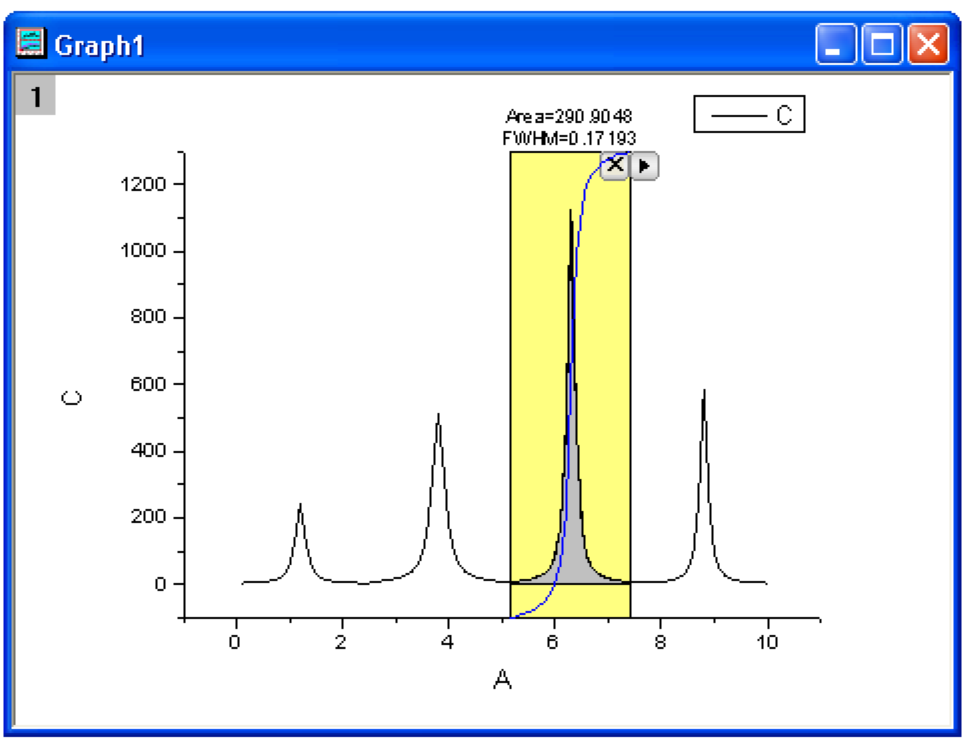
3. Select **Gadgets: Integrate** from the Origin menu when a graph is active, to bring up the **Data Exploration: addtool_curve_integ** dialog box.
In **Integration** tab, choose **Restrict to Rectangle** in the **Integral Curve** drop-down list to plot the integral curve within the rectangle.



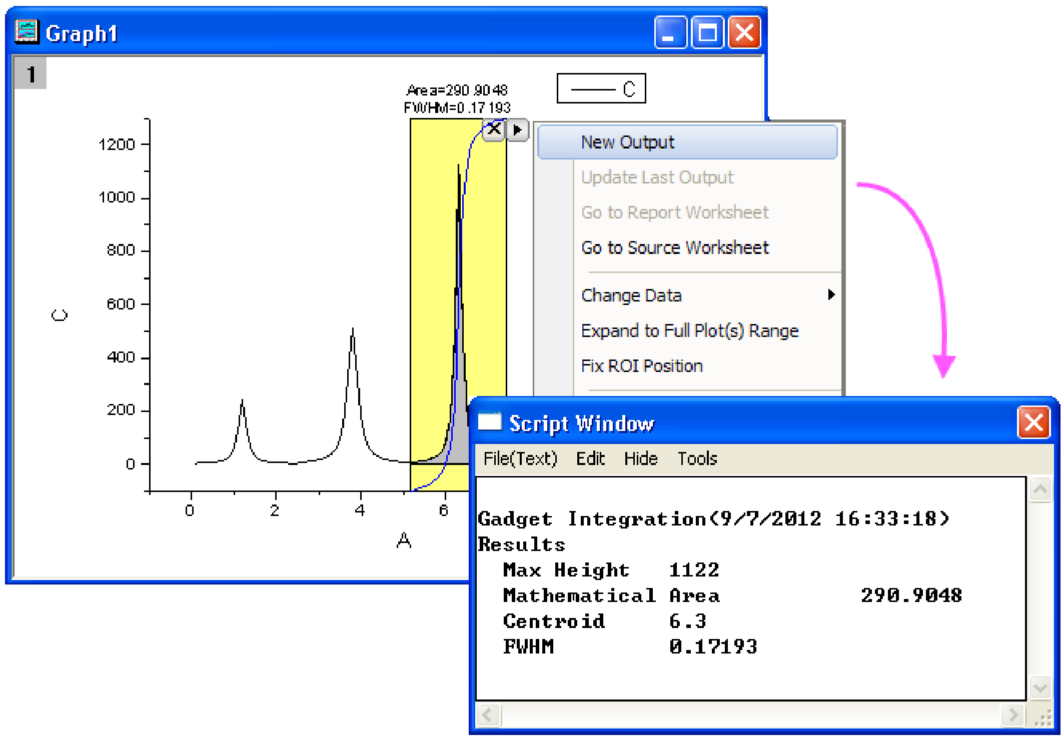
4. Click **OK** button. You will see that a yellow rectangle and a blue integral curve are added to the plot. The integral area is filled with gray, and the value is shown at the top of the rectangle.



5. Move the yellow region of interest rectangle to set the region for the single peak you want to integrate.



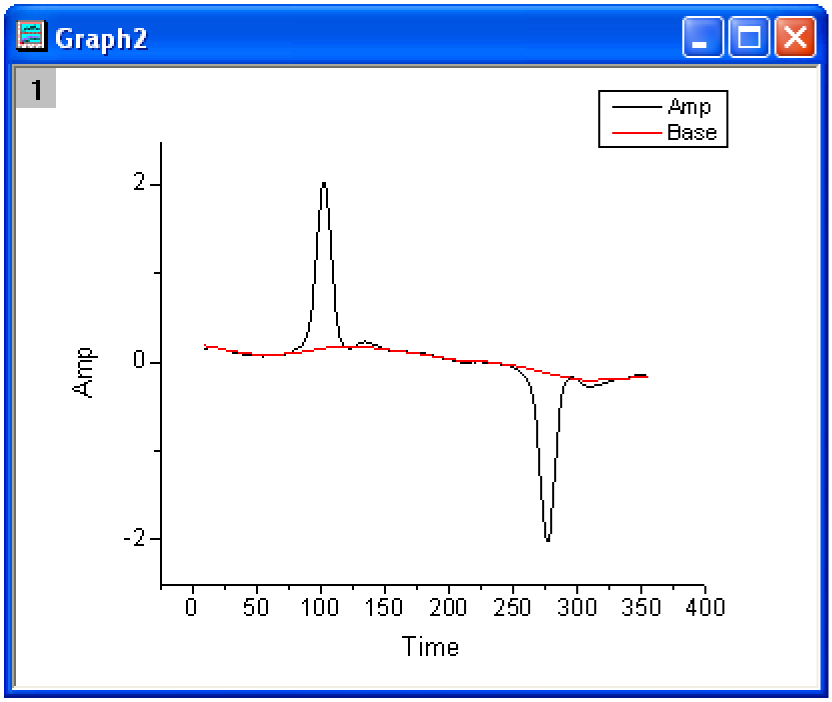
6. Click the triangle button  at the top right corner of the ROI tool to open the fly-out menu and choose **New Output**. Then the **Gadget Integration Results** will display in the **Script Window**.



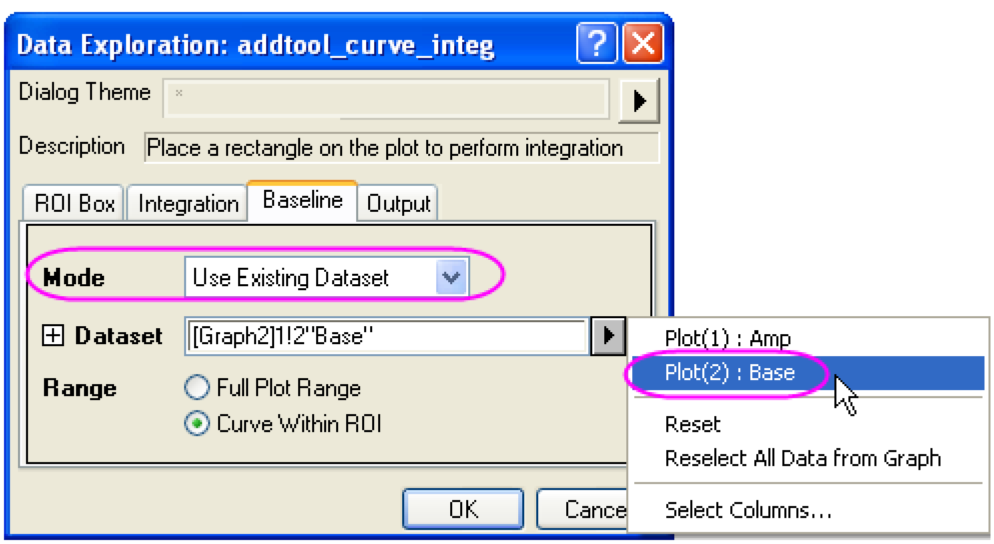
Integrate with a data plot baseline

1. Start with a new workbook and import the file *<Origin Program Folder>\Samples\Spectroscopy\Peaks with Base.DAT*

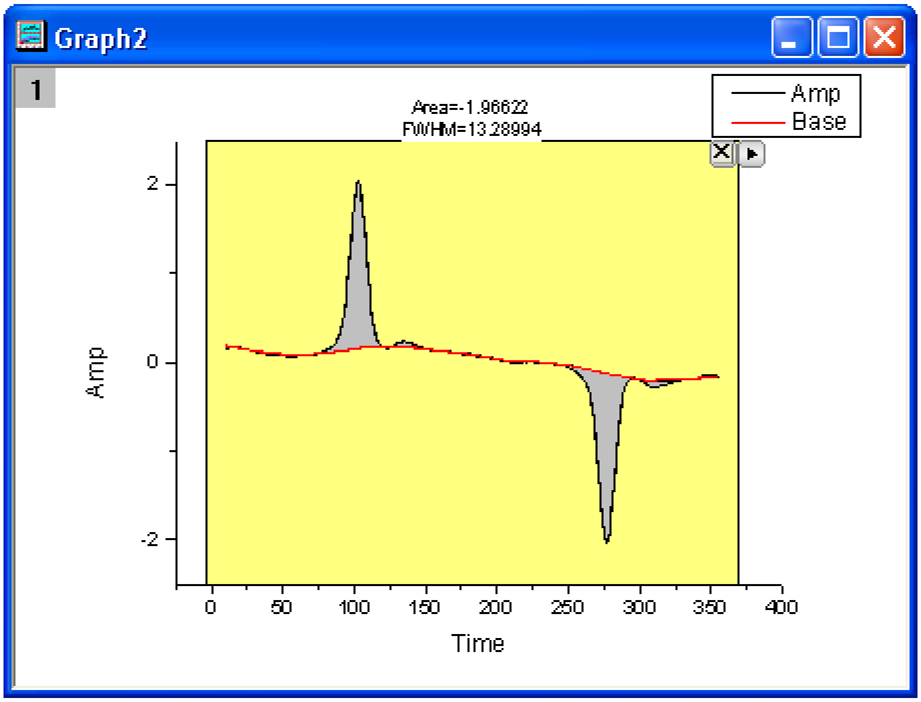
2. Highlight Col(B) and Col(C) and select **Plot: Line: Line** from the Origin menu to draw a graph.



3. Select **Gadgets: Integrate** from the Origin menu when a graph is active, to bring up the **Data Exploration: addtool_curve_integ** dialog box.
4. In **Baseline** tab, choose **Use Existing Dataset** for the **Mode**. Then select *Plot(2): Base* as **Dataset** and click **OK** button.



5. Click the triangle button and select **Expand to Full Plot(s) Range** in the fly-out menu to integrate the total area for the curve.



3.1.3 Curve Intersection Gadget

Summary

When there is more than one curve in a graph layer, you might want to calculate the intersection data points of these curves. Since Origin 8.6, a new gadget **Intersect** is available to calculate the intersection points of the input curves on the graph.

Minimum Origin Version Required: Origin 8.6 SR0

What you will learn

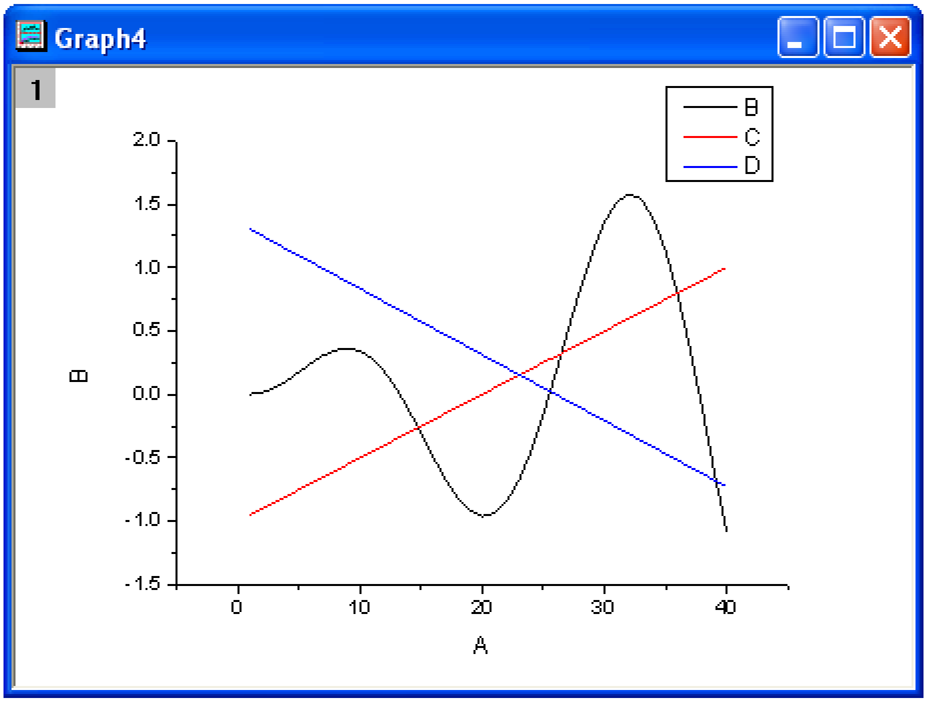
This tutorial will show you how to:

- Use the Intersect gadget on a graph.
- Tag the intersection points.
- Output the intersection points to worksheet.

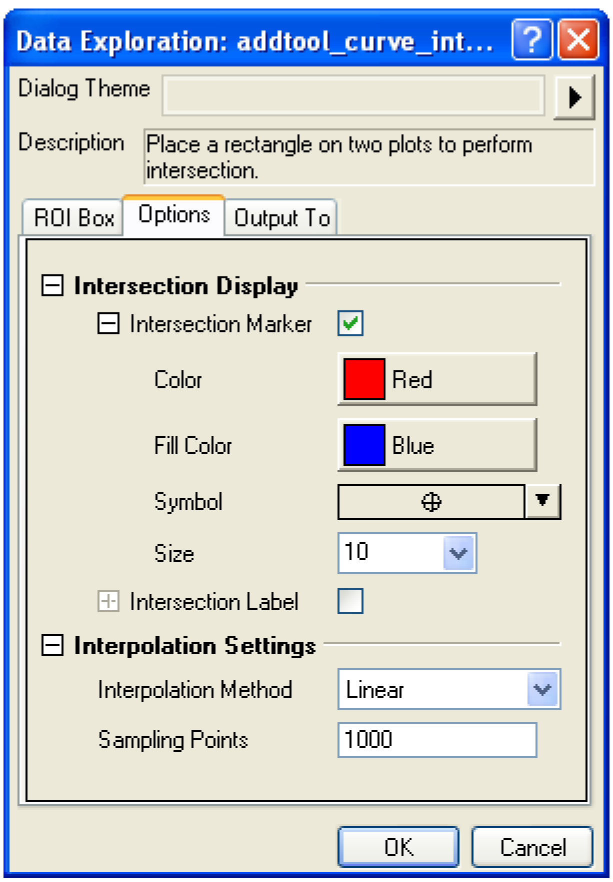
Steps

This tutorial is associated with the **Analysis: Curve Intersection Gadget** folder in the **Analysis** project (\Samples\Analysis.opj) which can be opened by selecting **File: Open Sample Projects: Analysis** from the main menu.

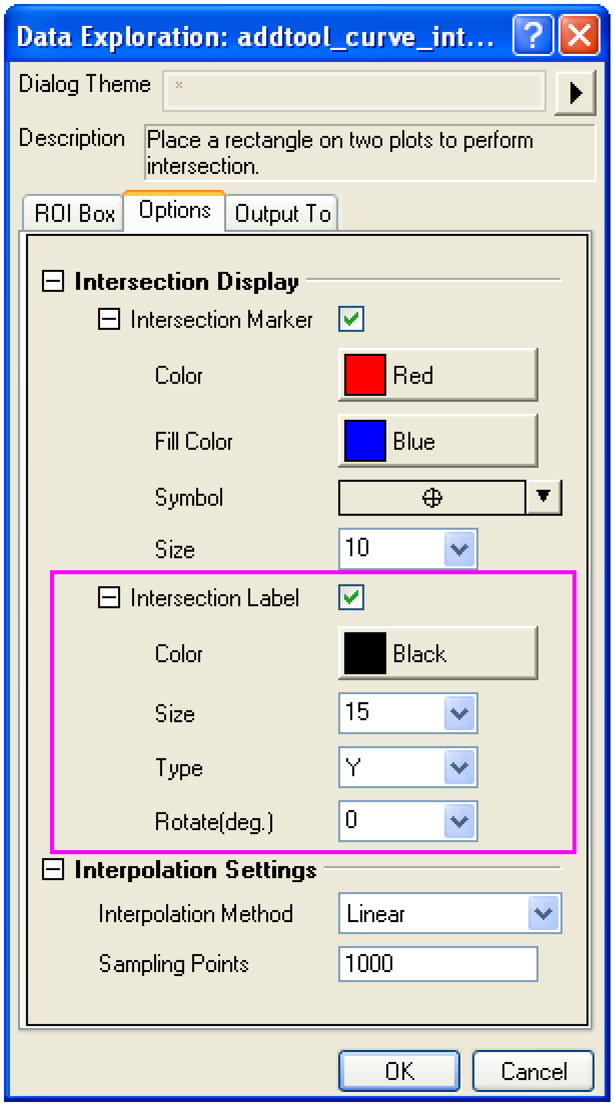
1. Highlight the Col(A)~col(D) in workbook **Book6**, and then click **Plot:Line:Line** to plot a line graph.



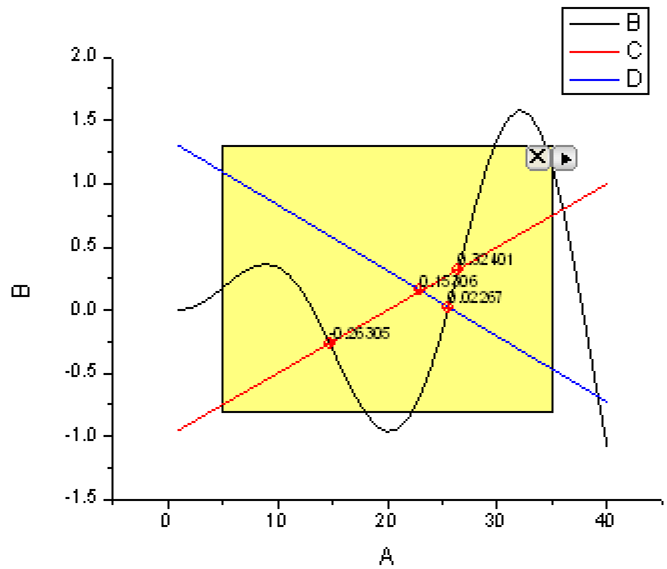
2. Select **Gadgets: Intersect...** from the main menu to open the dialog. Go to the **Options** tab.



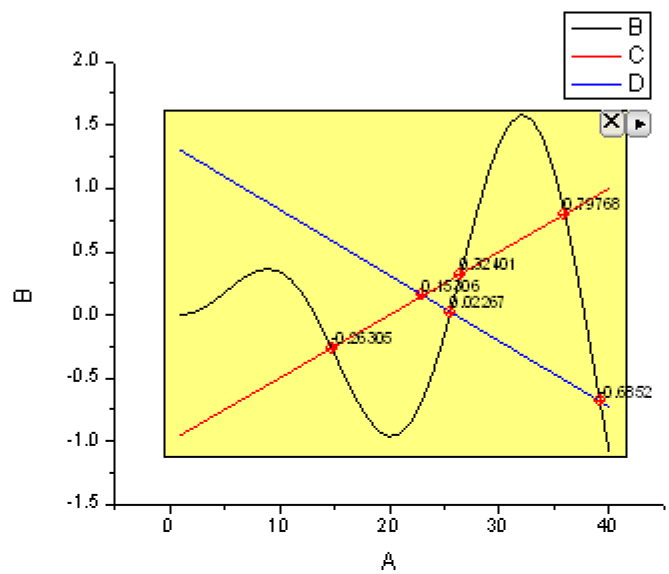
3. Check the **Intersection Label** check box in the **Options** tab, set **Size** as *15*, set **Type** as *Y* and set **Rotate(deg.)** as *0*.



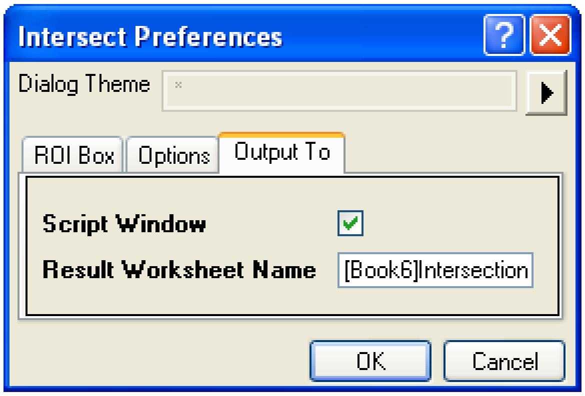
4. Click **OK** button to go back to the graph window. The yellow ROI box will be added onto the graph.



5. Click the Arrow button at the top right of the ROI box, select **Expand to Full Plot(s) Range** from the context menu. The ROI box will be expanded to cover full plot range as shown below.

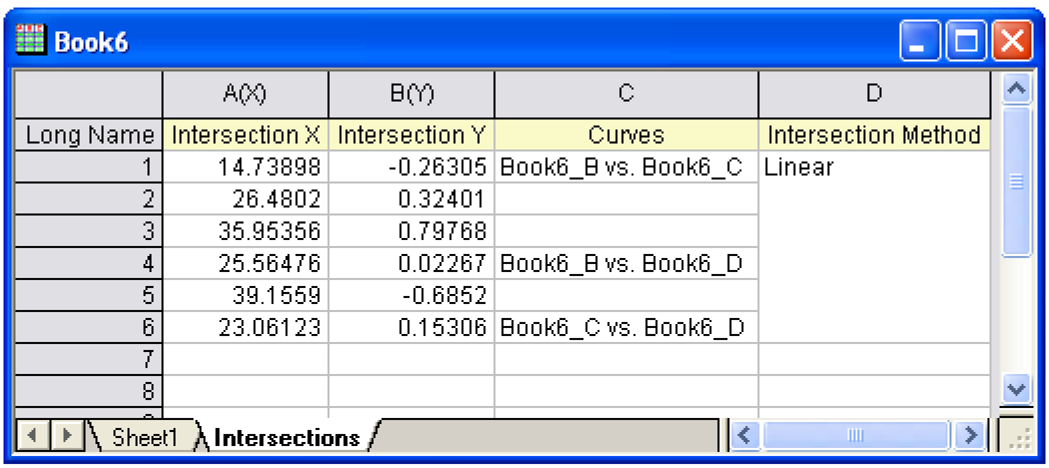


6. Click the Arrow button at the top right of the ROI box. Select **Preferences...** from the context menu to open the **Intersect Preferences** dialog. Go to the **Output To** tab and input *[Book6]Intersections* in the **Result Worksheet Name** box.



7. Click **OK** button to go back to the graph window. Click the arrow button at the top right of the ROI box, select **New Output** from the context menu.

8. The results will be output to the **Script Window**. Click the arrow button again, select **Go To Report Worksheet** from the menu. The X and Y coordinates of the intersection points will be listed in the results worksheet.



3.1.4 Rise Time Gadget

Summary

The Rise Time Gadget can be used to analyze the rising and falling stages of a step-like signal in a graph. With this gadget, you can intuitively select an area on the graph with a rectangle, and then calculate the rise time or fall time within that area.

What you will learn

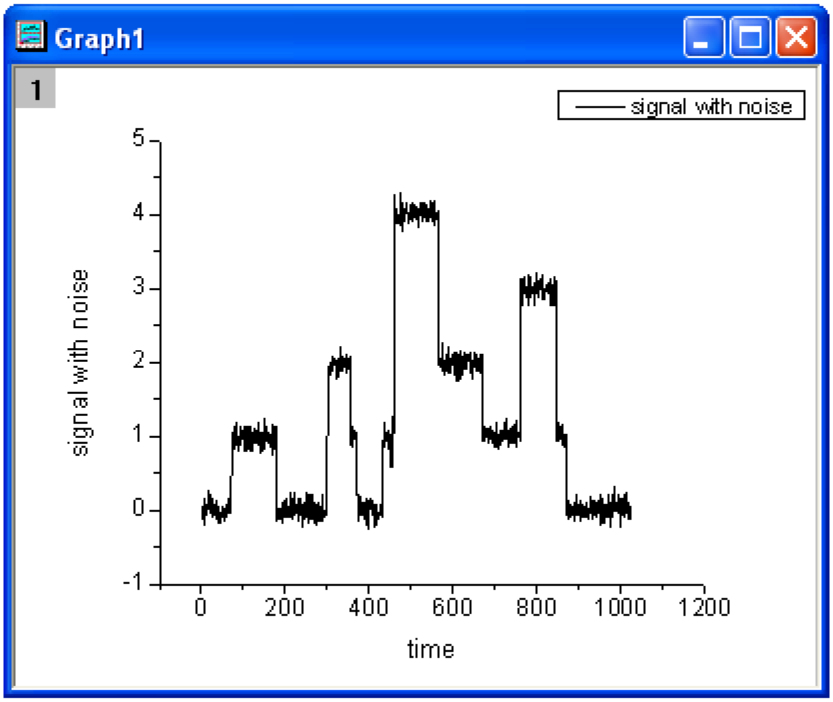
- How to select a specific region of the signal by moving and resizing a region of interest (ROI).
- How to mark **Rise Time** (**Fall Time**) and **Rise Range** (**Fall Range**) in the graph.
- How to switch the tool between **Rise Time** and **Fall Time**.

- How to output analysis results of **Rise Time** (**Fall Time**).

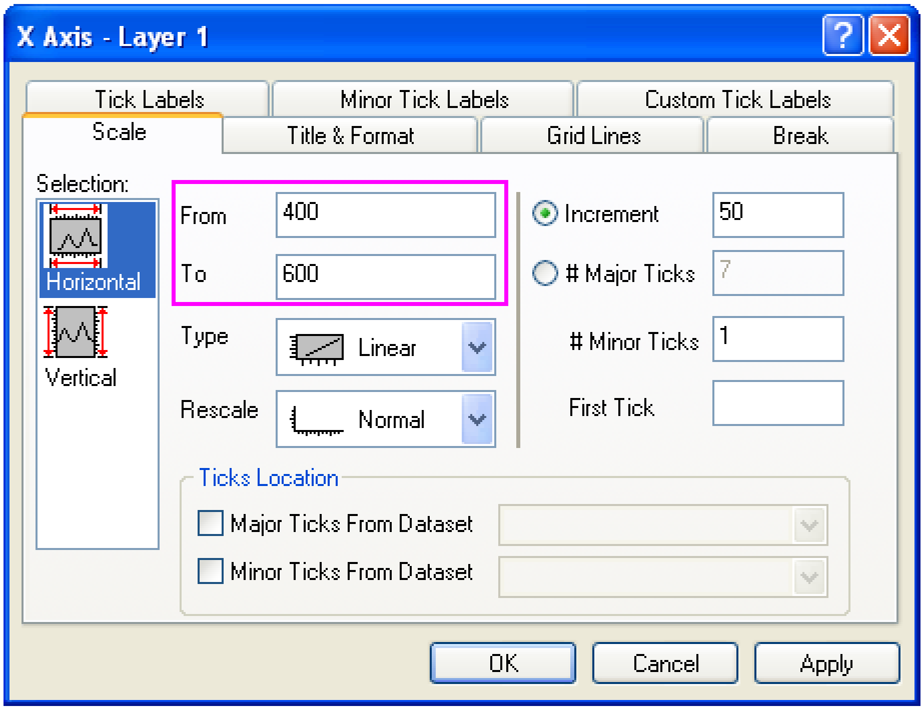
Analyze Rise Time

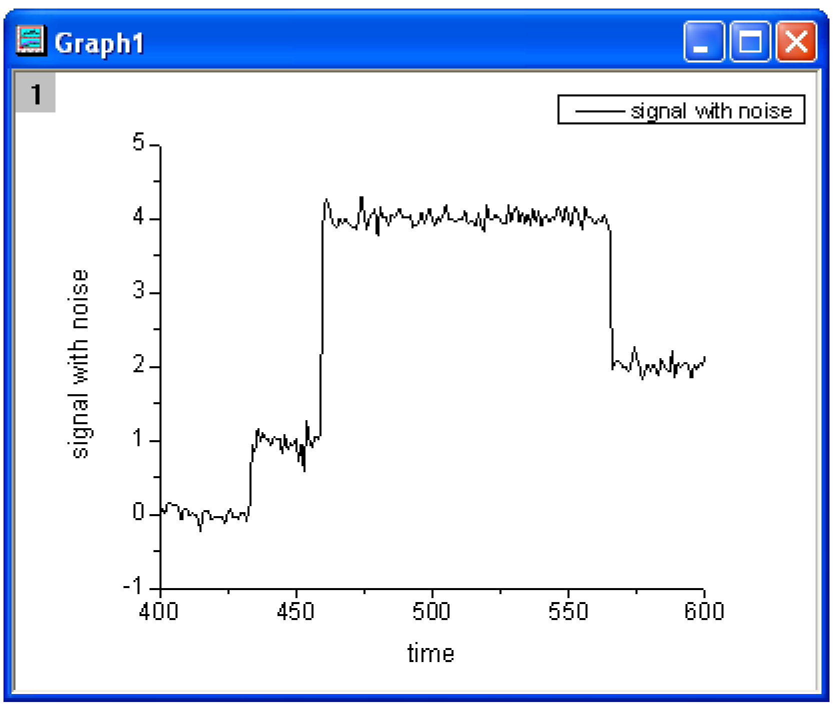
This tutorial is associated with the **Analysis - OriginPro: Rise Time (Pro Only)** folder in the **Analysis** project (\Samples\Analysis.opj) which can be opened by selecting **File: Open Sample Projects: Analysis** from the main menu.

1. Highlight Column B and select **Plot: Line: Line** in the menu to create a graph.

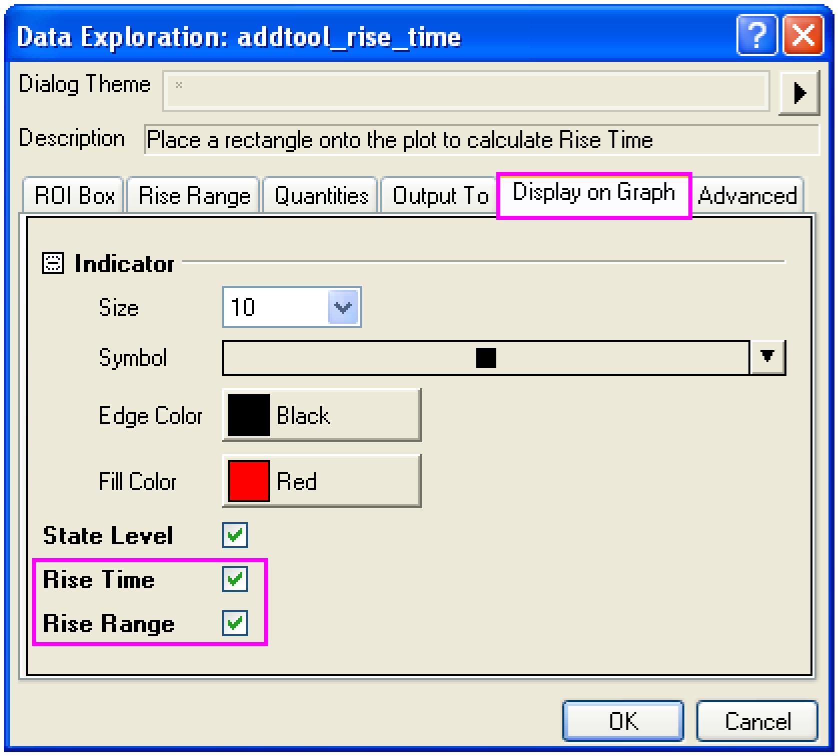


2. Double-click the X axis to open the **Axis** dialog box. Go to the **Scale** tab, select **Horizontal** from the **Selection** box, and set the axis range from 400 to 600. Set **Increment** as 50. Then click **Ok** button.

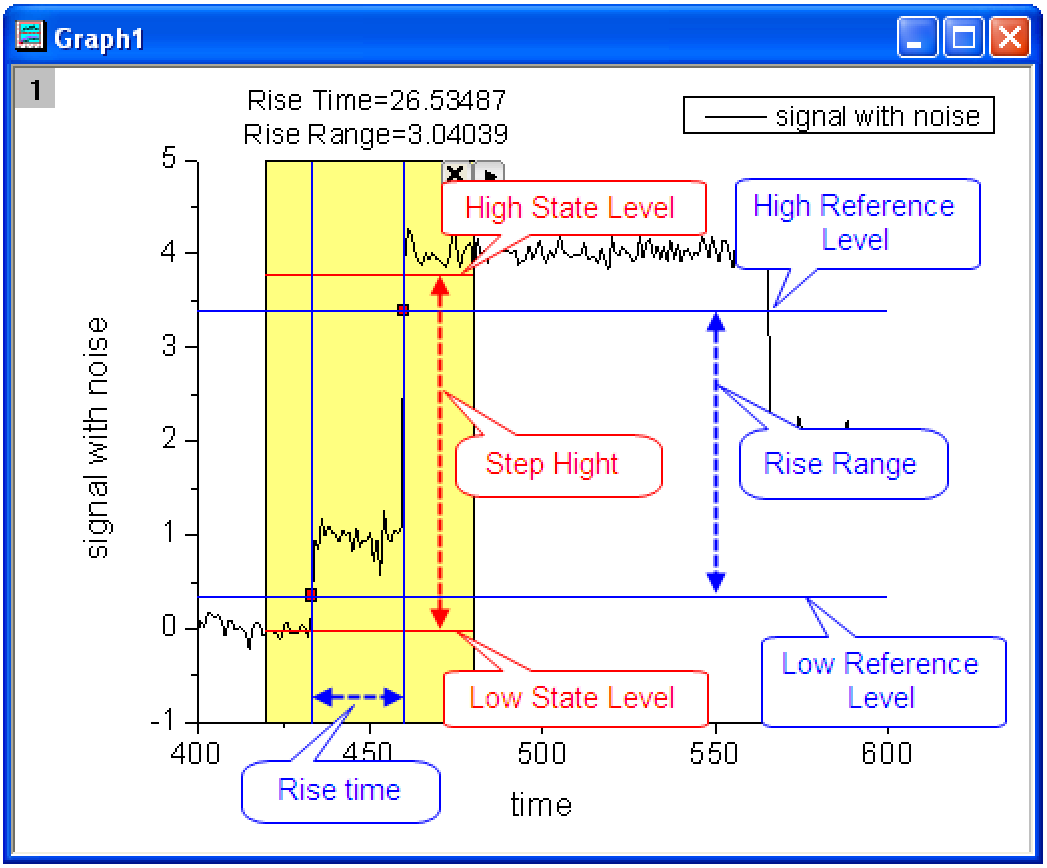




3. Select **Gadgets: Rise Time** from the Origin menu to open the **Data Exploration: addtool_rise_time** dialog box. Go to the **Display on Graph** tab, and click the **Rise Time** and **Rise Range** check boxes.



4. Click **OK** button, and you will see that a yellow rectangle is added to the plot. Move the rectangle horizontally on the rise signal step.



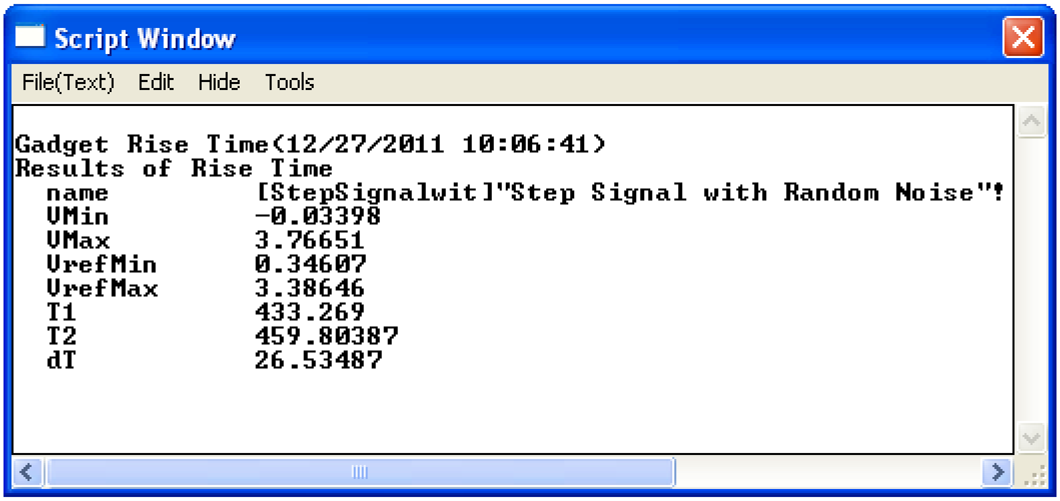
In the graph, you can see the **Rise Time** and **Rise Range** values are shown at the top of the rectangle.

The two blue vertical lines go through the two indicators marking the **Rise Time**.

The two blue horizontal lines that display the **Low Reference Level** and **High Reference Level** go through the two indicators marking the **Rise Range**.

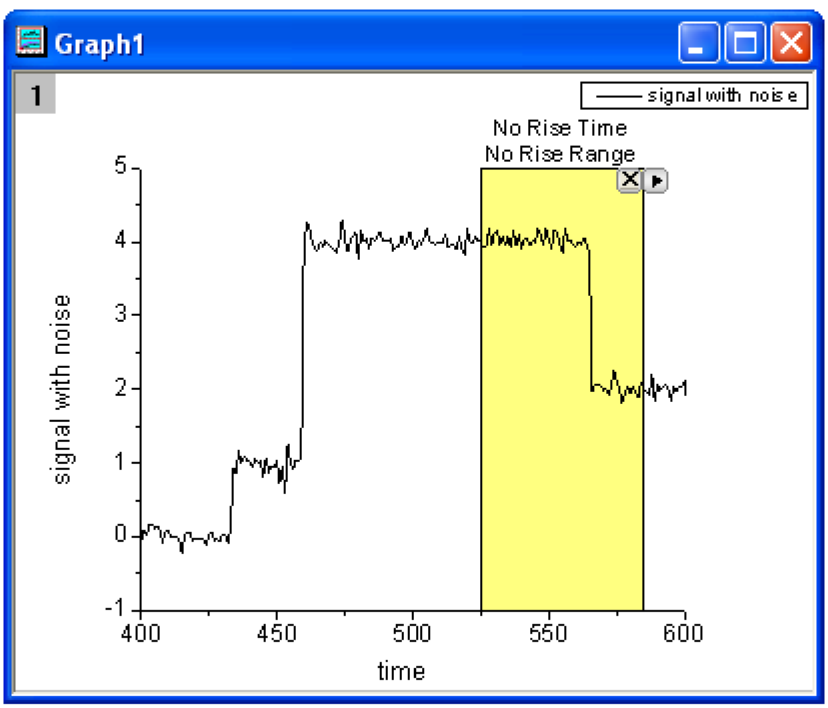
The two red horizontal lines display **Low State Level** and **High State Level**.

5. Click the triangle button  at the top right corner of the rectangle. Then select **New Output**. The results will be output to the **Script Window**.

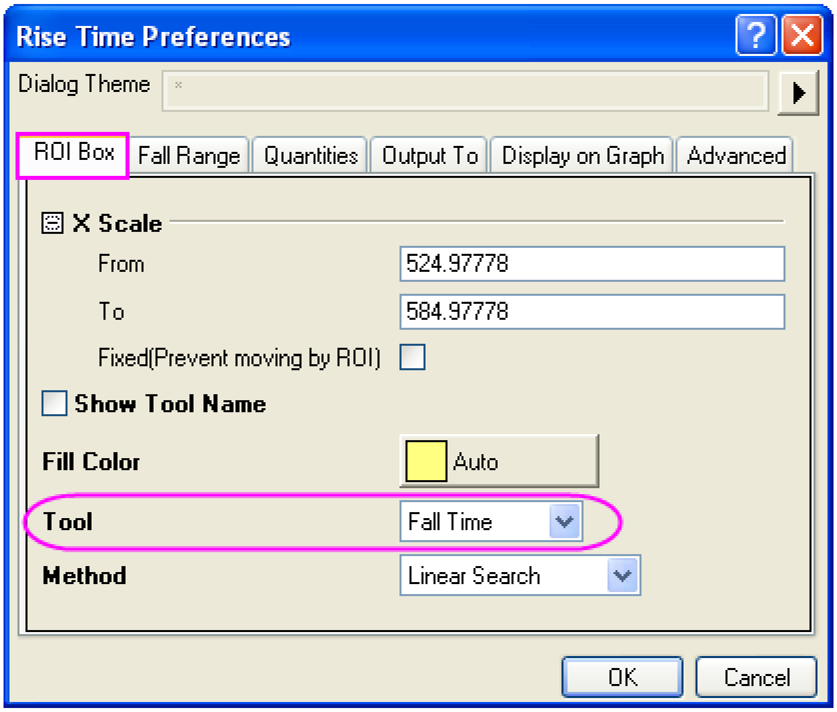


Analyze Fall Time

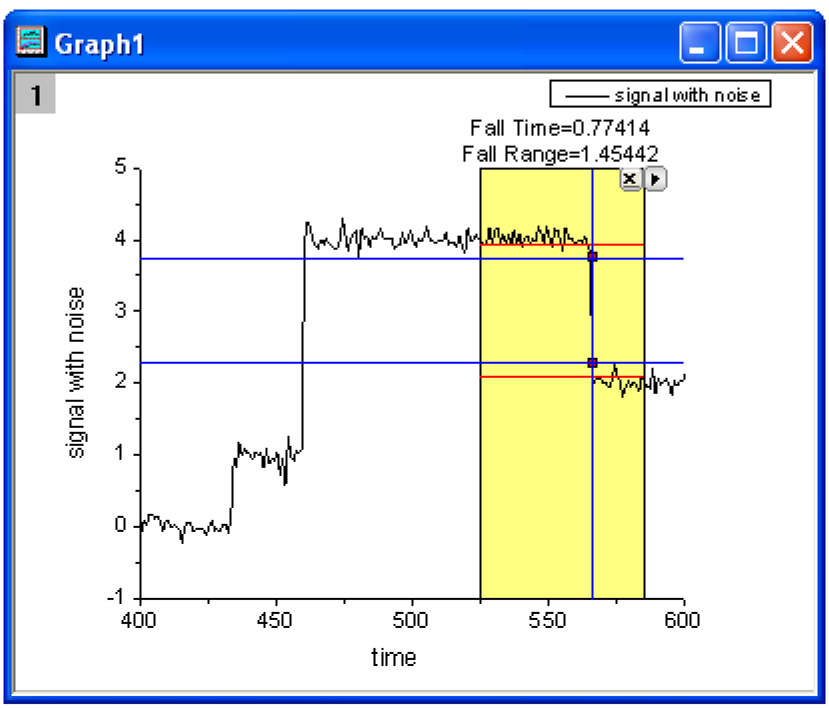
1. This gadget also allows you to get the fall time and fall range in a graph. Move the rectangle horizontally on the fall signal step.



2. Click the triangle button  and select **Preferences** in the fly-out menu to open the **Rise Time Preferences** dialog. Go to the **ROI Box** tab and select **Fall Time** in the **Tool** drop-down list.



3. Click the **OK** button, the **Fall Time** and **Fall Range** values are shown at the top of the rectangle.



4. Click the triangle button and select **New Output**. The results will be output to the **Script Window**.

3.1.5 Interpolate Gadget

Summary

Origin supports the interpolate gadget to perform quick interpolation on a ROI (Region of Interest) range when a graph is active. And you can easily to change the interpolate region by moving the ROI.

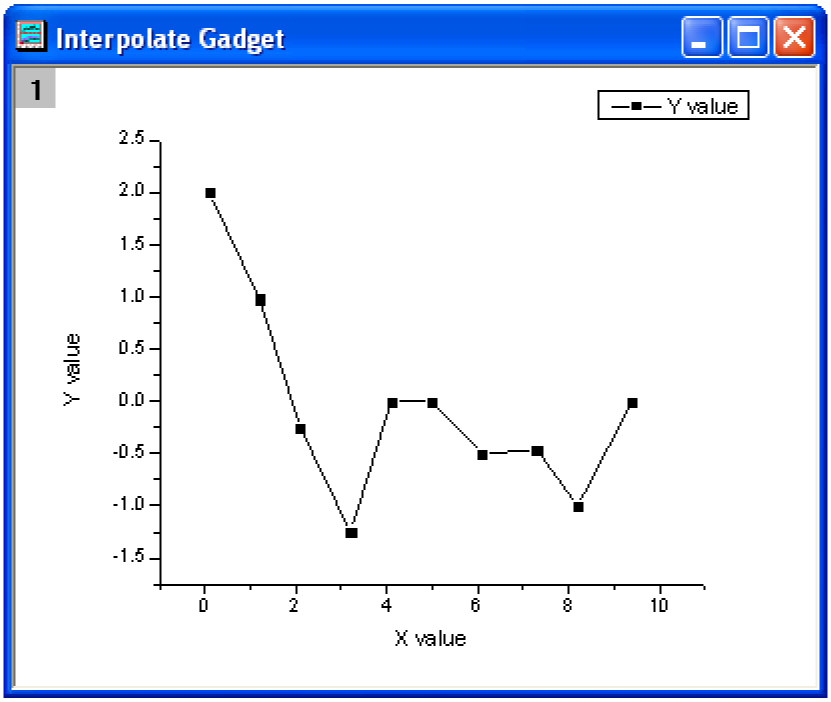
What you will learn

- How to easily interpolate data points on a rectangular region.
- How to quickly find out an interpolated Y value from any given X value.
- How to output the interpolated values to **Script Window**, **Result Log**, or a specified worksheet.

Steps

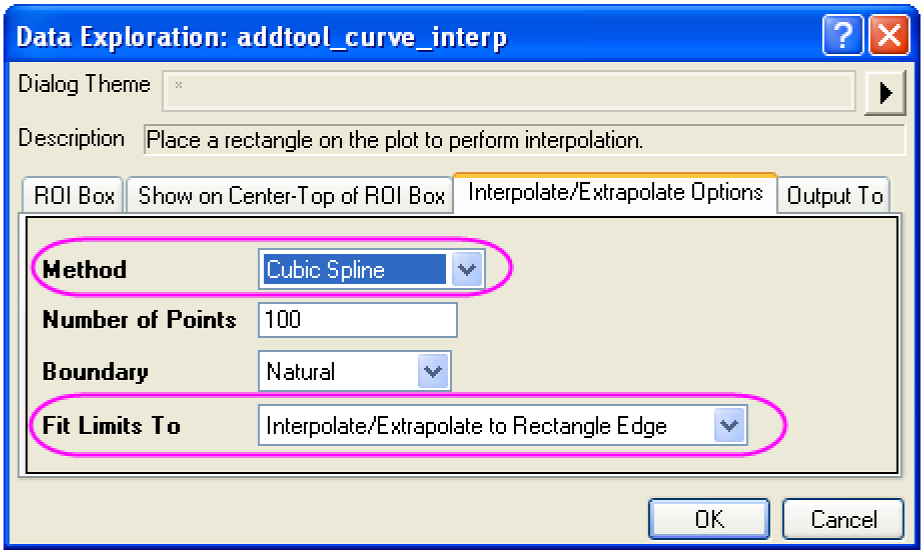
This tutorial is associated with the **Analysis: Interpolate Gadget** folder in the **Analysis** project (\Samples\Analysis.opj) which can be opened by selecting **File: Open Sample Projects: Analysis** from the main menu.

1. Highlight the Col(A) and col(B) in workbook Book1R and plot a **Line +Symbol** graph.

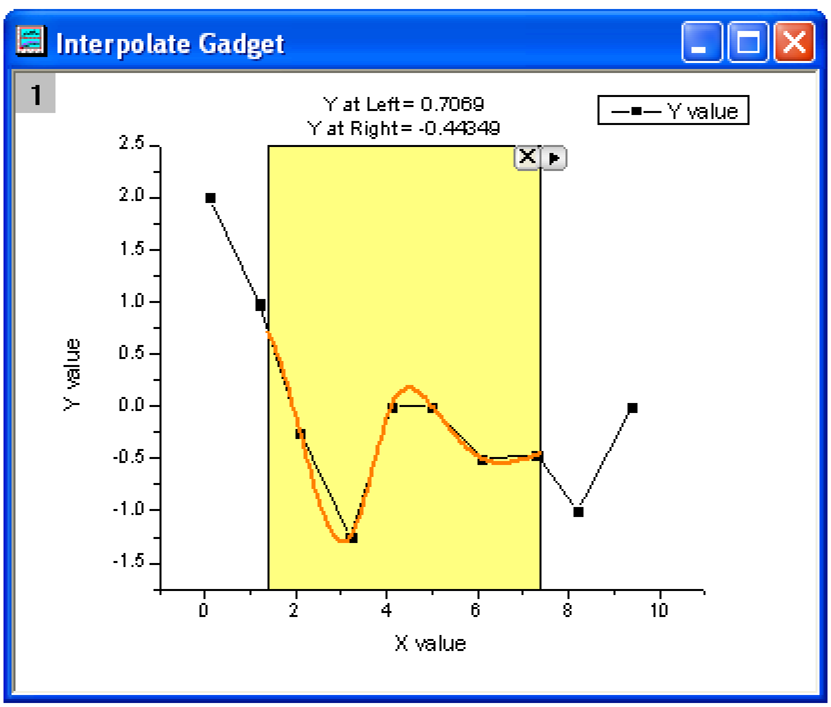


2. Select **Gadgets: Interpolate** from the Origin menu when a graph is active, to bring up the **Data Exploration: addtool_curve_intep** dialog box.

3. Go to the **Interpolate/Exterpolate Options** tab. Choose **Cubic Spline** for the **Method**, and then select **Interpolate/Extrapolate to Rectangle Edge** for **Fit Limits To**.

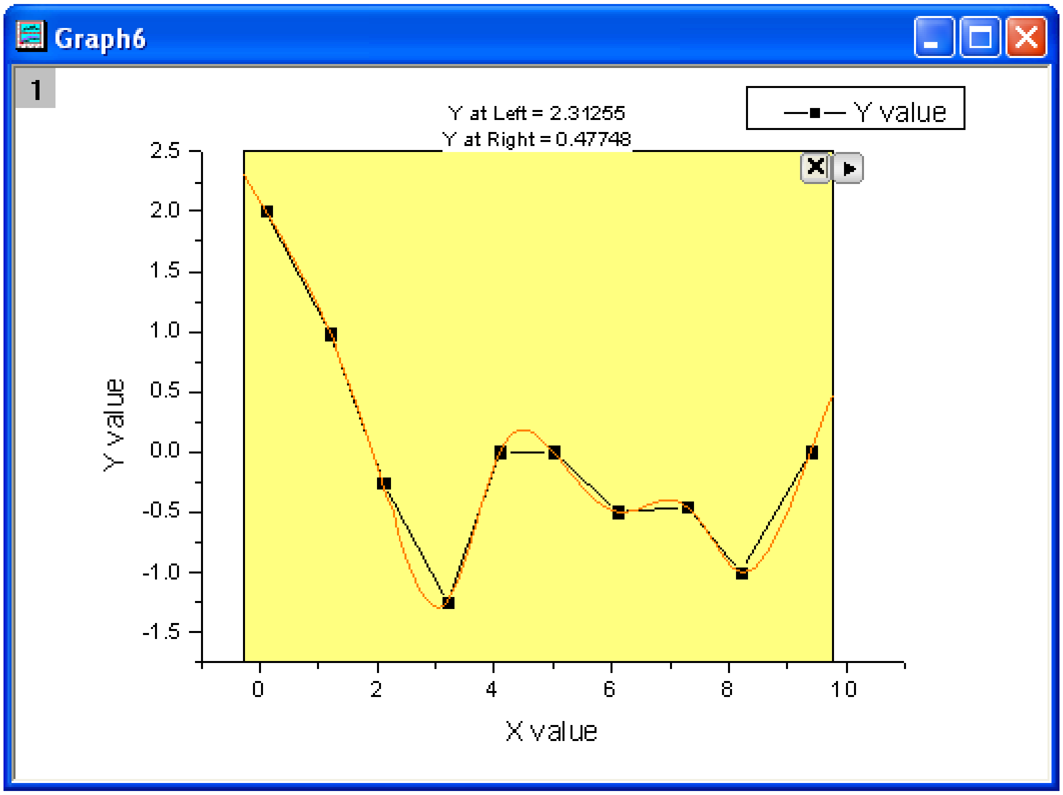


4. Click **OK** button. It will add an interpolation line onto the plot. And the Y values of interpolation line at right and left are shown at the top of the rectangle region.

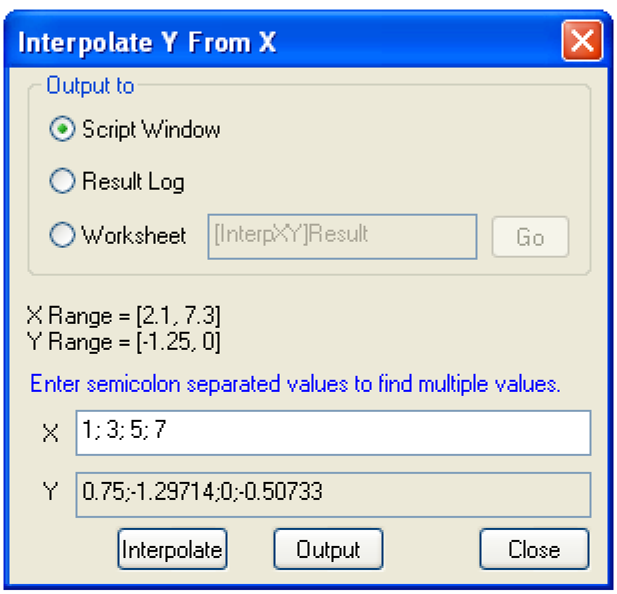


5. You can change the data range by moving or resizing the yellow region of interest rectangle, the interpolated curve displayed will update as the ROI is being moved.

Click the triangle button and select **Expand to Full Plot(s) Range** in the fly-out menu to interpolate the total area for the curve.



6. This gadget also allows you to find Y values from a given X value. Select **Interpolate X/Y** in fly-out menu to open the **Interpolate Y from X** dialog. You can enter multiple X values and click the **interpolate** button. This tool will output the interpolated Y values for each of the X values.



7. The interpolated Y values can be output to **Script Window**, **Result Log**, or a specified worksheet.

3.1.6 Cluster Gadget

Summary

Origin supports the Cluster Gadget to perform simple statistics on a region of interest (ROI) in a graph. The gadget can also be used to edit, clear, or mask data points. The statistics results are dynamically updated as the ROI object is moved or resized.

Minimum Origin Version Required: Origin 8.5.1 (Pro only)

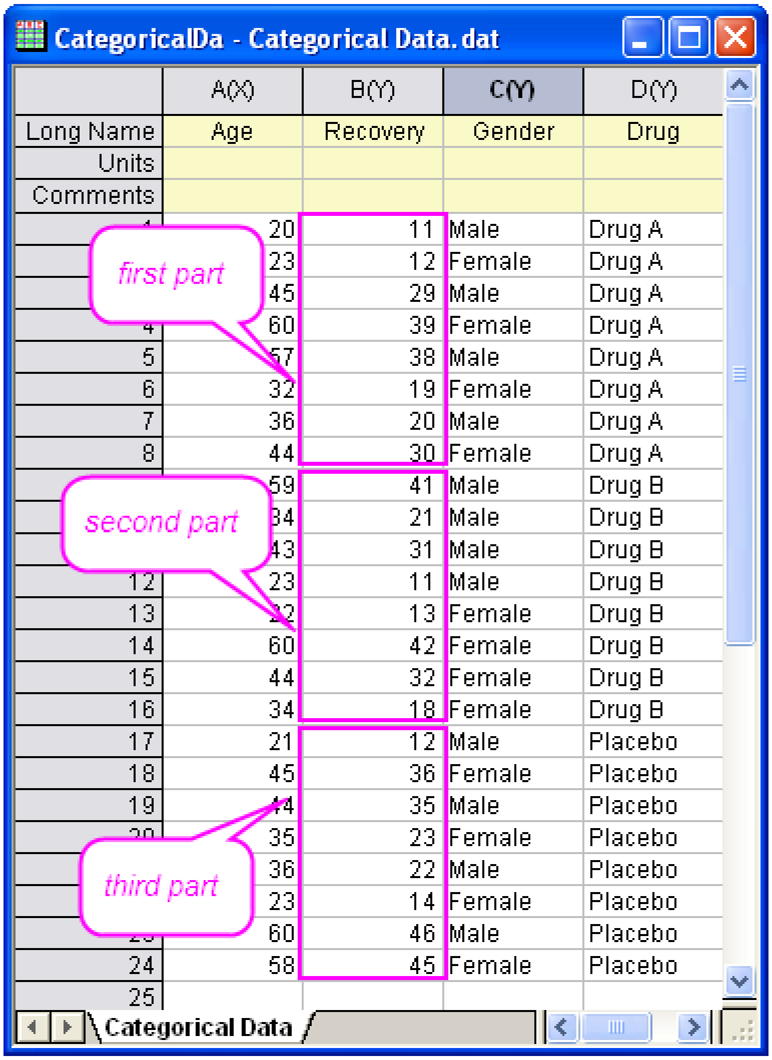
What you will learn

- How to perform simple statistics on a region of interest (ROI) in a graph.
- How to edit the data points such as clear, or mask points in graph using menu options or buttons.
- How to view or output the statistic for points inside and outside of the ROI.

Perform basic statistics

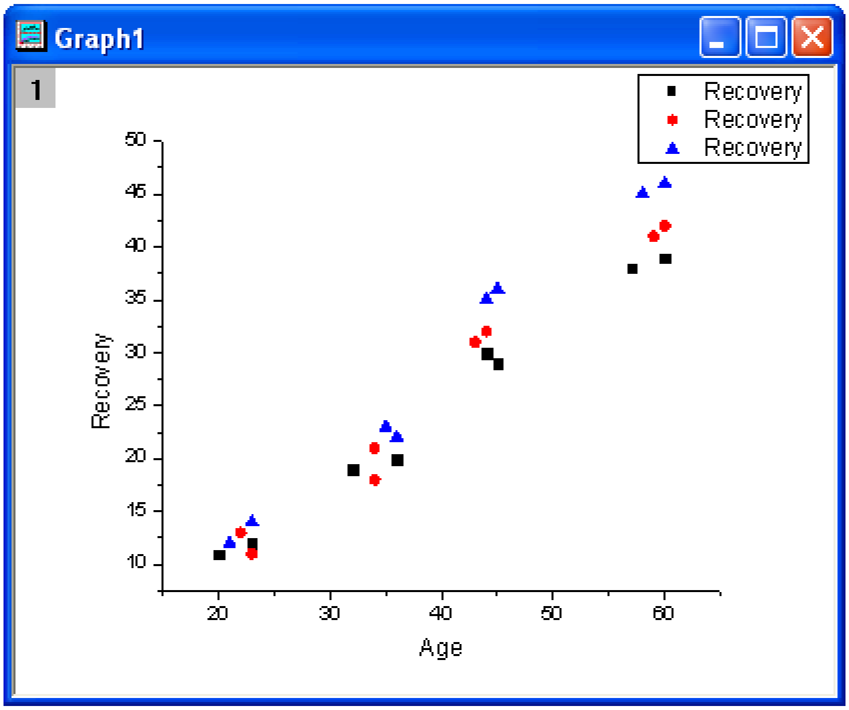
1. Start with a new workbook and import the Origin sample data *Categorical Data.dat* which is located in *<Origin Program Folder> \Samples\Graphing*.
   Right-click on Col(D) and select **Sort Worksheet: Ascending**. Then you will see the worksheet is sorted by the category of *Drug*.

2. Pressing **Ctrl** key, highlight three parts of the Col(B) separately.

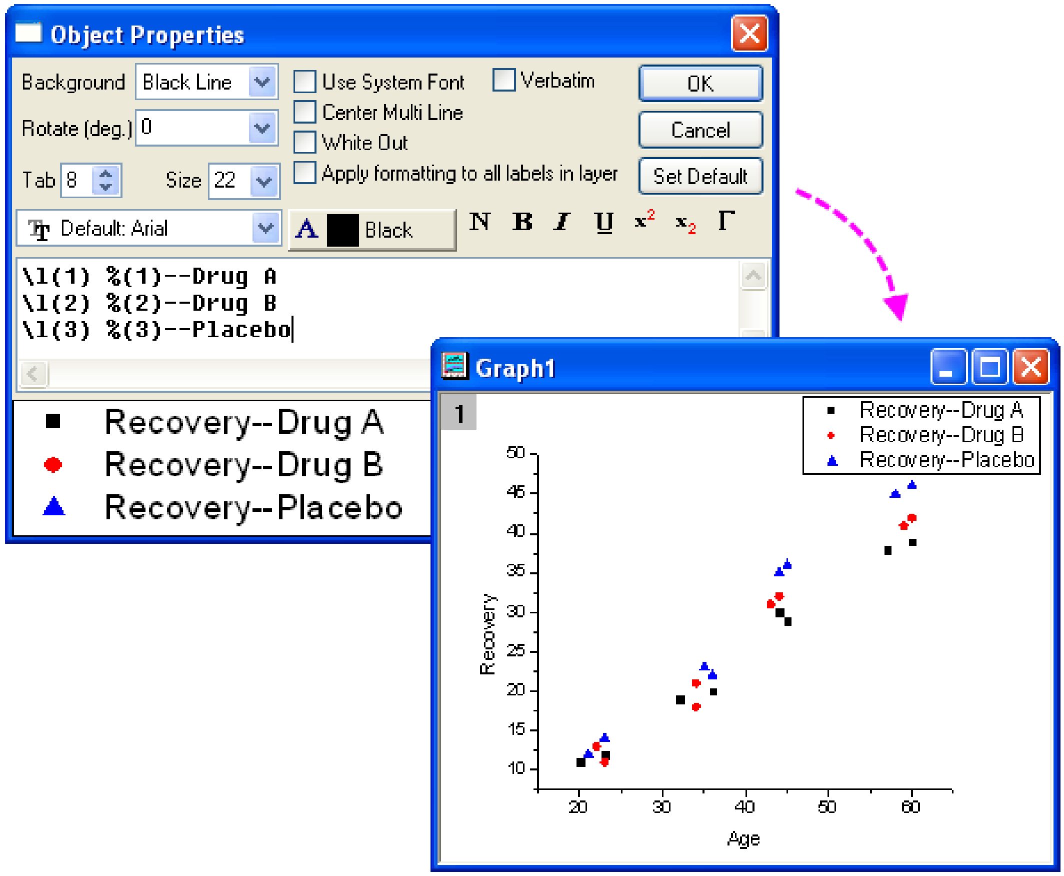


	A(X)	B(Y)	C(Y)	D(Y)
Long Name	Age	Recovery	Gender	Drug
Units				
Comments				
1	20	11	Male	Drug A
2	23	12	Female	Drug A
3	45	29	Male	Drug A
4	60	39	Female	Drug A
5	57	38	Male	Drug A
6	32	19	Female	Drug A
7	36	20	Male	Drug A
8	44	30	Female	Drug A
9	59	41	Male	Drug B
10	34	21	Male	Drug B
11	43	31	Male	Drug B
12	23	11	Male	Drug B
13	22	13	Female	Drug B
14	60	42	Female	Drug B
15	44	32	Female	Drug B
16	34	18	Female	Drug B
17	21	12	Male	Placebo
18	45	36	Female	Placebo
19	44	35	Male	Placebo
20	35	23	Female	Placebo
21	36	22	Male	Placebo
22	23	14	Female	Placebo
23	60	46	Male	Placebo
24	58	45	Female	Placebo
25				

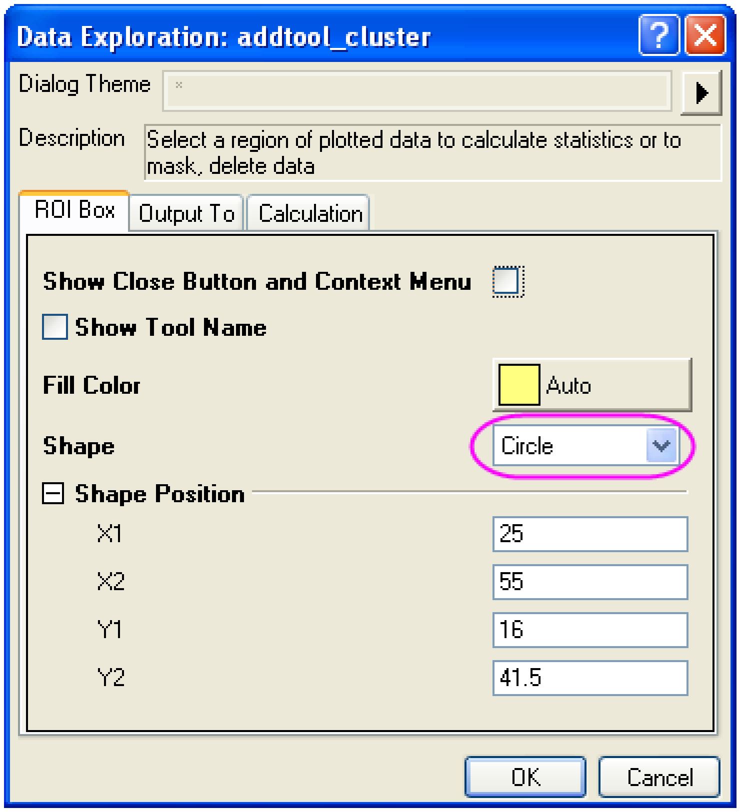
3. Select **Plot: Symbol: Scatter** from the main menu to create a graph. The graph with three plots display the recovery for each drug.



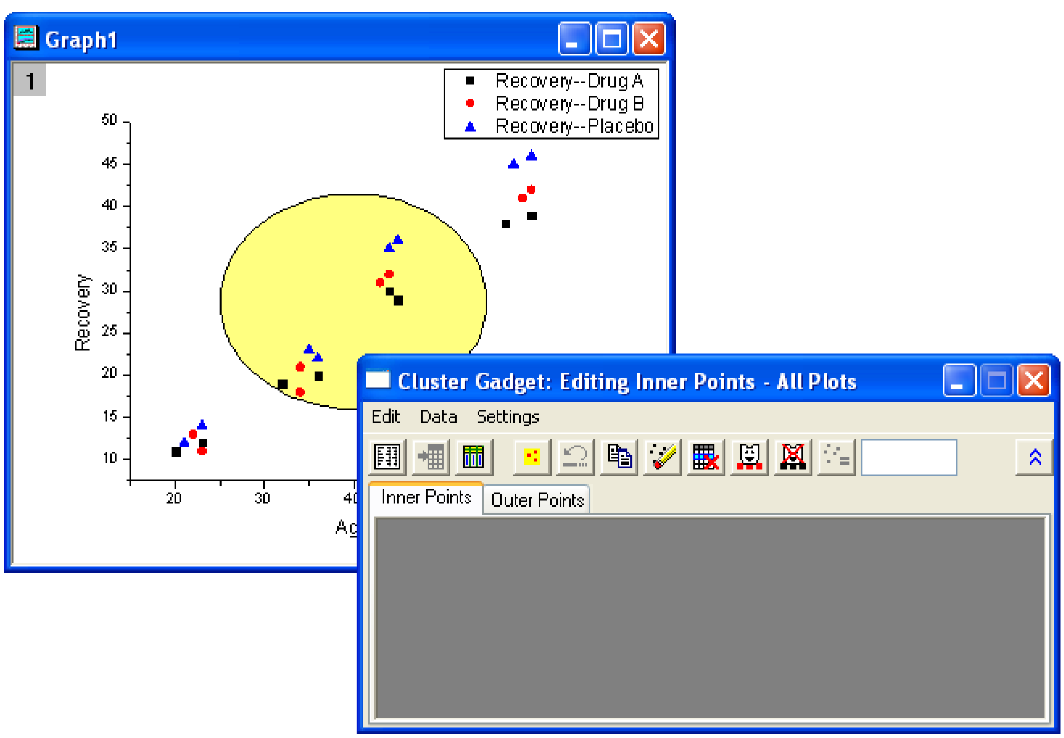
4. Right-click on the graph legend and select **Properties** to open **Object Properties** dialog. Then edit the legend as shown below. Click **OK** button.



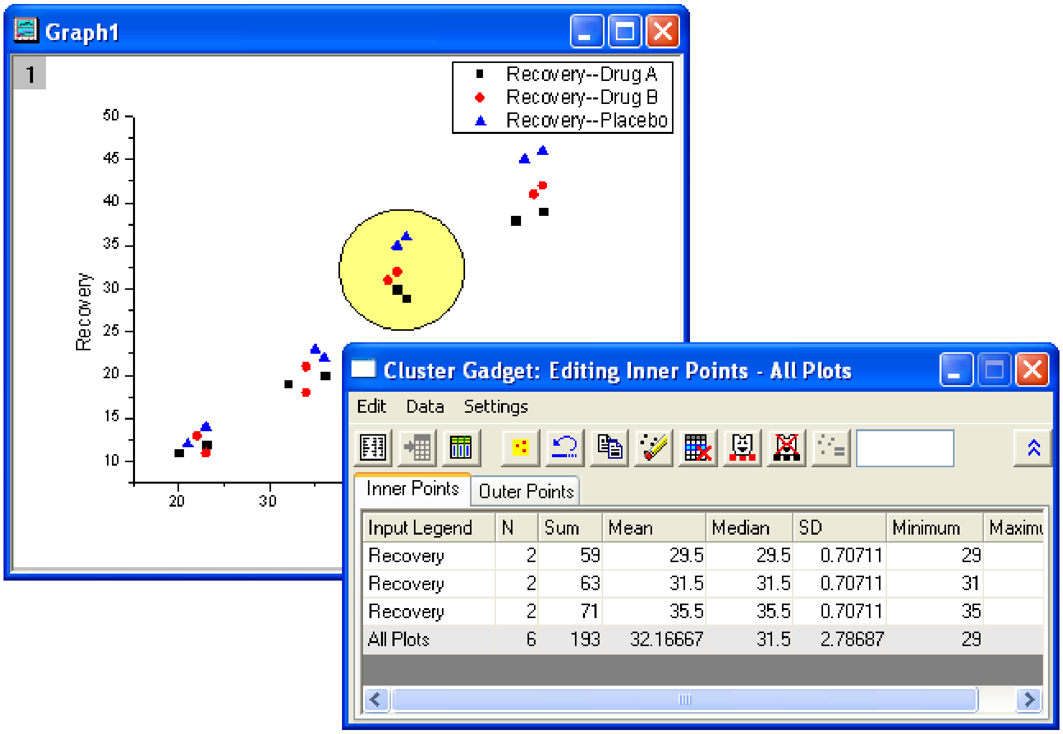
5. Select **Gadgets: Cluster** from the Origin menu when a graph is active, to bring up the **Data Exploration: addtool_cluster** dialog box. Choose **Circle** in the **Shape** drop-down list of the **ROI Box** tab.



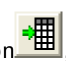
6. Click **OK** button. It will add a yellow circle for **ROI** on the plot and bring up the **Cluster Gadget** dialog.

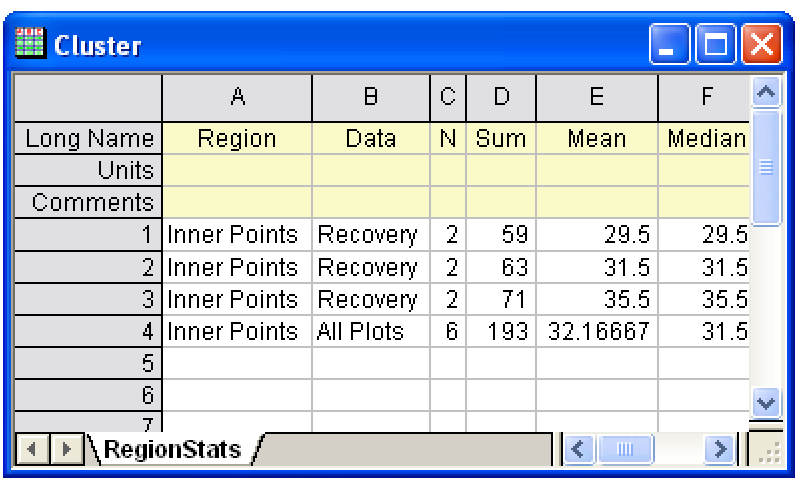


7. Move the yellow circle to the region for which you want to get statistics and see the results shown on the **Inner Points** tab of the dialog.



8. Click the **Output Statistics Report** button. The results are output to the Result Log, Script Window, and the Cluster workbook.

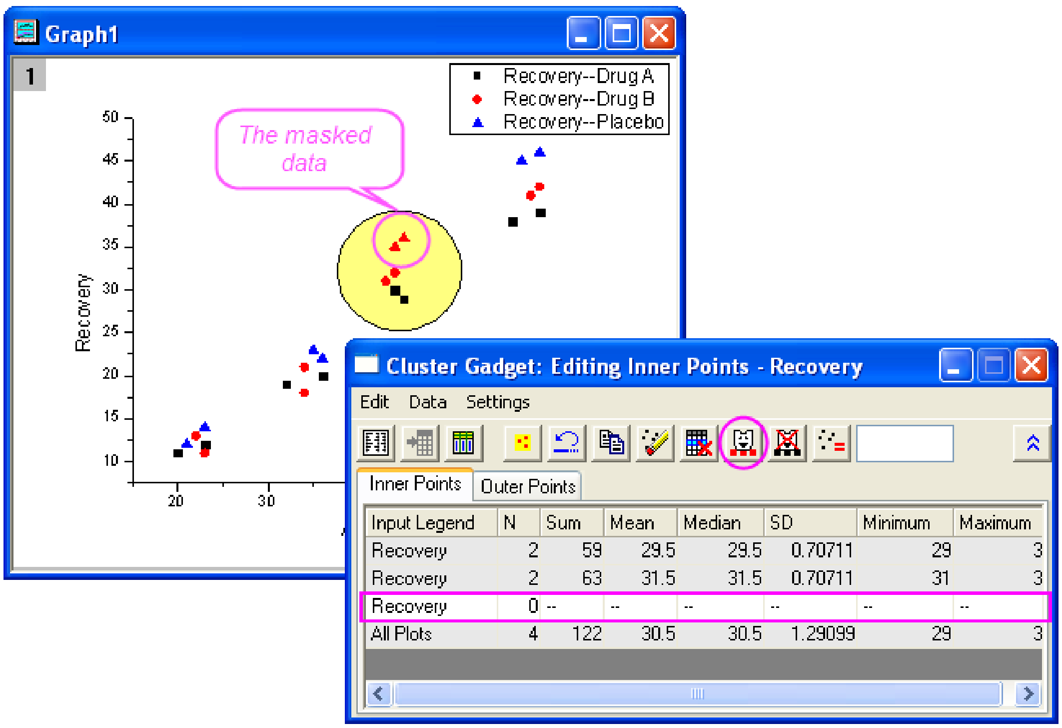
9. Click the **Go to Report Worksheet** button, then the **Cluster** workbook will be shown.



Exclude the data points in a cluster

In this section, we will show you how to exclude a specific plot from the cluster. Based on the example above, we will do simple statistics on *Recovery of Drug A* and *Recovery of Drug B*, excluding *Recovery of Placebo*.

1. Click the **Data** menu in the **Cluster Gadget** dialog and uncheck **Plot(1)** and **Plot(2)**. The first and second rows become gray in the lower panel, and they can no longer be manipulated by the buttons in the dialog.
2. Click **Mask Data Points** button. The *Recovery of Placebo* data points in the ROI are masked and the color becomes red. At the same time the statistics results for *Recovery of Placebo* become missing values.

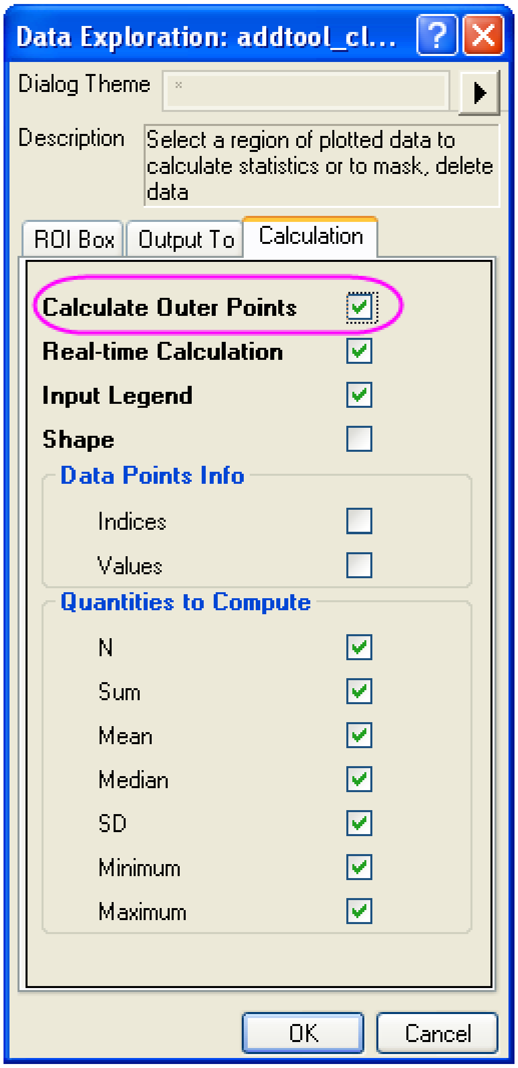


3. Click the **Output Statistics Report** button. The results are output to the Result Log, Script Window, and the Cluster workbook.

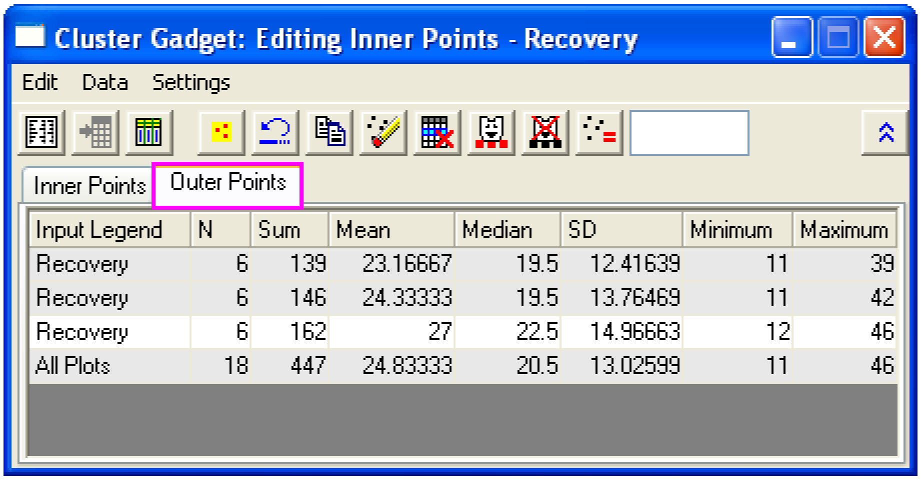
Get statistics results for points outside of the ROI

1. Click **Settings: Preferences** from the **Cluster Gadget** dialog's menu to open the **Cluster Manipulation Preferences** dialog.

2. Go to the **Calculation** tab. Click the **Calculate Outer Points** check box.



3. Click the **OK** button, the statistics results for points out of the ROI are shown on the **Outer Points** tab of the dialog.



4. Click the **Output Statistics Report** button . The results for inner and outer points are shown in the Result Log, Script Window, and the Cluster workbook.

3.2 Curve Fitting

Topics covered in this section:

1. Linear and Polynomial Fitting (Tutorials)
2. Nonlinear Fitting (Tutorials)

3.2.1 Linear and Polynomial Fitting

- Tutorial:Linear_Fitting

Linear Fitting and Outlier Removal

Summary

An outlier is typically described as a data point or observation in a collection of data points that is "very distant" from the other points and thus could be due to, for example, some fault in the measurement procedure. Identification and removal of outliers is often controversial, and is typically "more acceptable" in situations where the model used to describe the data is well known and well accepted.

Minimum Origin Version Required: Origin 8.1 SR2

What you will learn

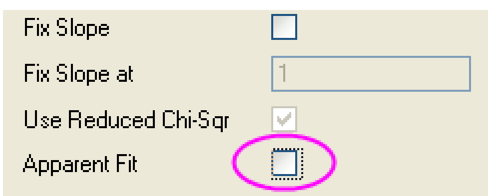
This tutorial will show you how to:

- Perform linear regression on a set of data points
- Examine the Residuals Table in the output and "identify" outliers
- Use the Masking Tool to remove the outlier points
- Use the Recalculation mechanism to automatically update the result after outlier removal

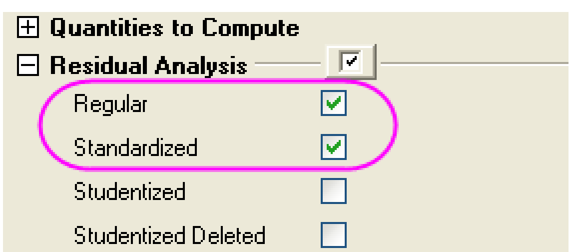
The procedure described in this tutorial is also applicable to other fitting tools such as Polynomial and Nonlinear Fitting

Steps

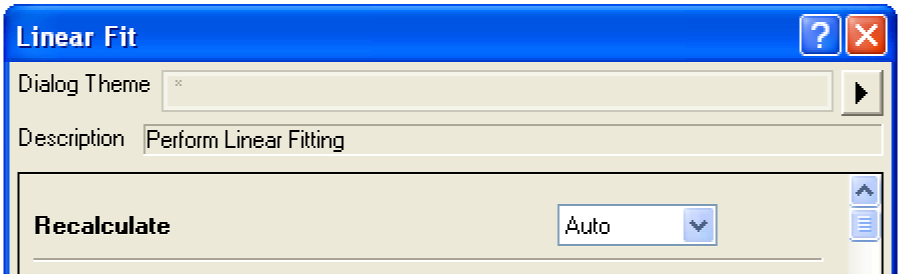
1. Start with a new workbook and import the file *\Samples\Curve Fitting\Outlier.dat*.
2. Click and select the second column and use the menu item **Plot: Symbol: Scatter** to create a scatter plot.
3. With the graph active, use the menu item **Analysis: Fitting: Fit Linear** to bring up the Linear Fit dialog. Note that if you have used the Linear Fit dialog before, there will be a fly-out menu and you need to select the **Open Dialog...** sub menu.
4. Under the **Fit Options** branch, clear the **Apparent Fit** check box.



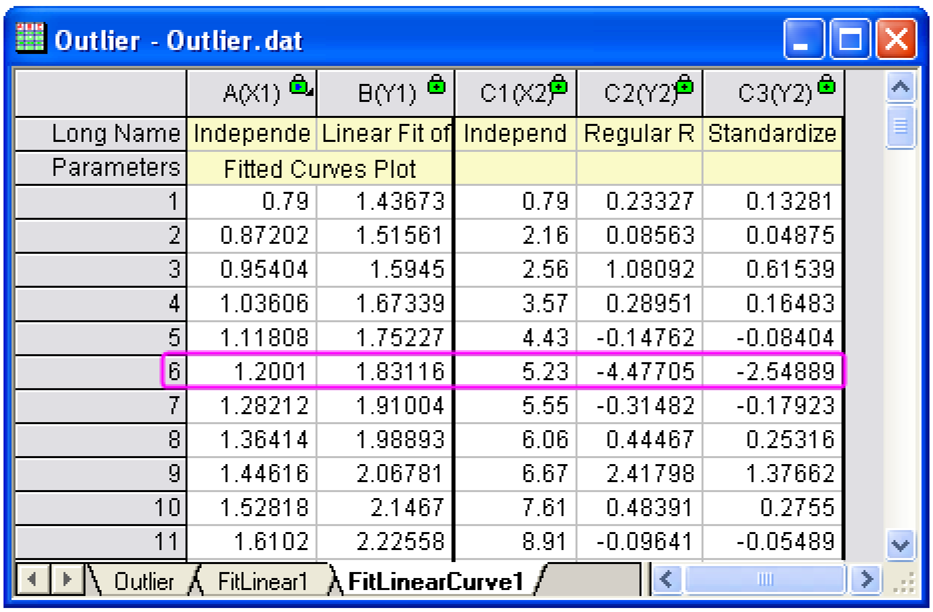
5. Expand the **Residual Analysis** tree node in the dialog, and check the **Standardized** check box.



6. Change the **Recalculate** drop-down at the top of the dialog to **Auto** and press the OK button at the bottom of the dialog. The dialog will close and linear regression will be performed on the data.

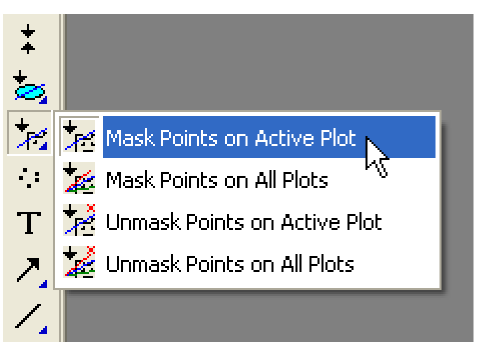


7. Select the **FitLinearCurve1** result sheet in the data workbook and scroll to the right side to view the **Standardized Residual** column. You will note that the value in row 6 in this columns is -2.54889:

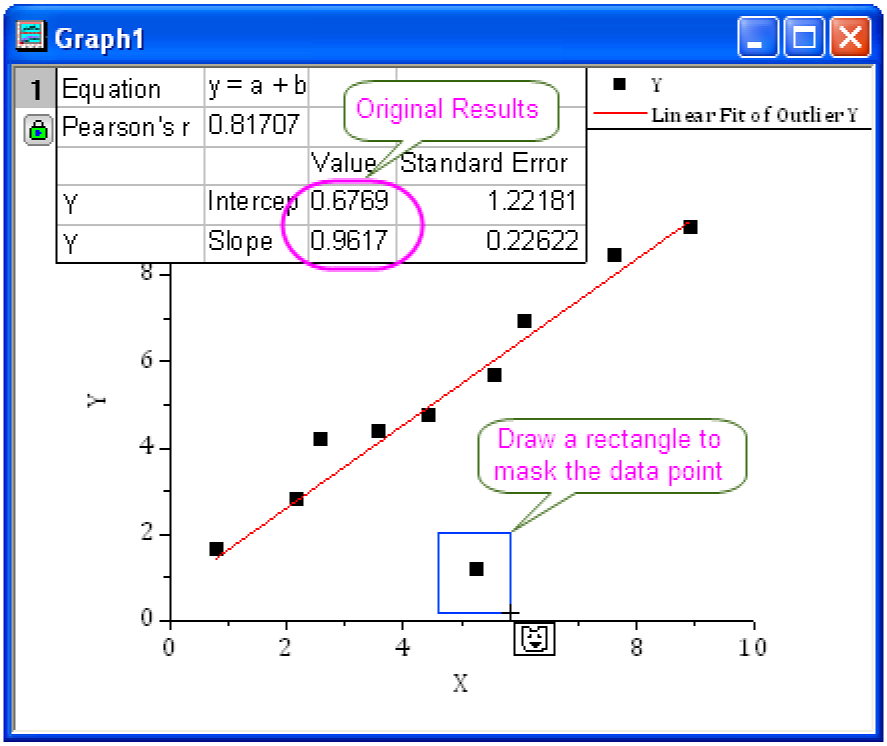


	A(X1)	B(Y1)	C1(X2)	C2(Y2)	C3(Y2)
Long Name	Independe	Linear Fit of	Independ	Regular R	Standardize
Parameters	Fitted Curves Plot				
1	0.79	1.43673	0.79	0.23327	0.13281
2	0.87202	1.51561	2.16	0.08563	0.04875
3	0.95404	1.5945	2.56	1.08092	0.61539
4	1.03606	1.67339	3.57	0.28951	0.16483
5	1.11808	1.75227	4.43	-0.14762	-0.08404
6	1.2001	1.83116	5.23	-4.47705	-2.54889
7	1.28212	1.91004	5.55	-0.31482	-0.17923
8	1.36414	1.98893	6.06	0.44467	0.25316
9	1.44616	2.06781	6.67	2.41798	1.37662
10	1.52818	2.1467	7.61	0.48391	0.2755
11	1.6102	2.22558	8.91	-0.09641	-0.05489

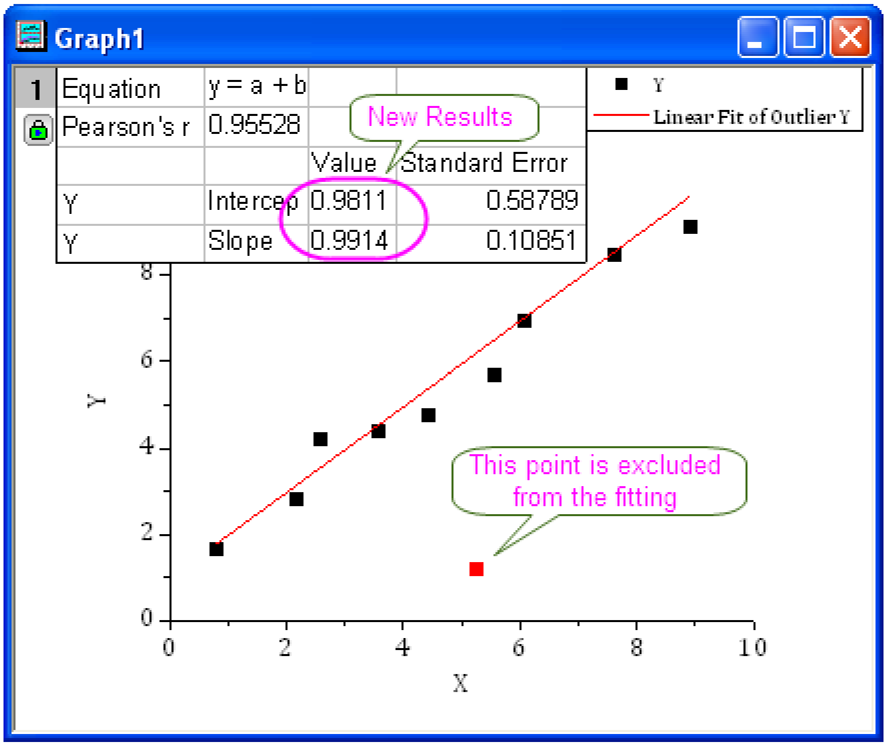
8. Make the graph active and then click and hold down the mouse left button on the "Regional Mask Tool" button in the Tools toolbar. Select the "Masked Points on Active Plot" submenu which will be the first item in the fly-out menu:



9. With the above submenu selected, go to the graph and click on the 6th data point to mask the point.



This changes the input data to the linear fit operation and the auto update mechanism will trigger. The linear fit will be repeated with this particular masked point left out. The fit curve in the graph and the pasted parameters will automatically update. Your result graph should then look like below:



3.2.2 Nonlinear Fitting

Topics covered in this section:

1. Nonlinear Curve Fit Tool
2. NLFIT Built In
3. Global Fitting
4. User Defined Fitting Function using OC
5. Fitting Datasets
6. Fitting with Multiple Independent Variables
7. User Defined Fitting Funciton using GNU Scientific Library
8. Fitting with NAG Special Function
9. Fitting Integral Function with parametric limit using NAG Library
10. Fitting with Integral using NAG Library
11. Fitting with Integral using LabTalk Function
12. Fitting with Two Integrals using LabTalk Function
13. Fitting with Summation
14. Fitting Complex Function
15. Fitting With Convolution
16. Quoting Built in Functions in Your New Function
17. Fit Function with Non-constant Background
18. Fitting with Piecewise Functions

Nonlinear Curve Fit Tool

Summary

Nonlinear fitting in Origin is performed using the NonLinear Fitting (NLFit) dialog box. The NLFit tool contains more than 200 built-in fitting functions used in many different disciplines.

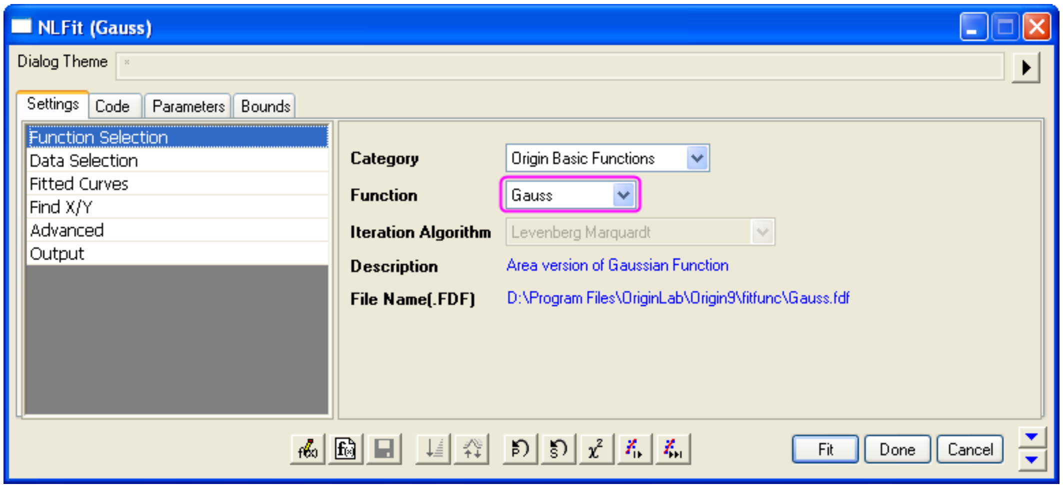
Minimum Origin Version Required: 8.5

What you will learn

- How to fit with a built-in fitting function.
- How to change NLFit setting using Recalculate.
- How to define and fit with a user-defined function.

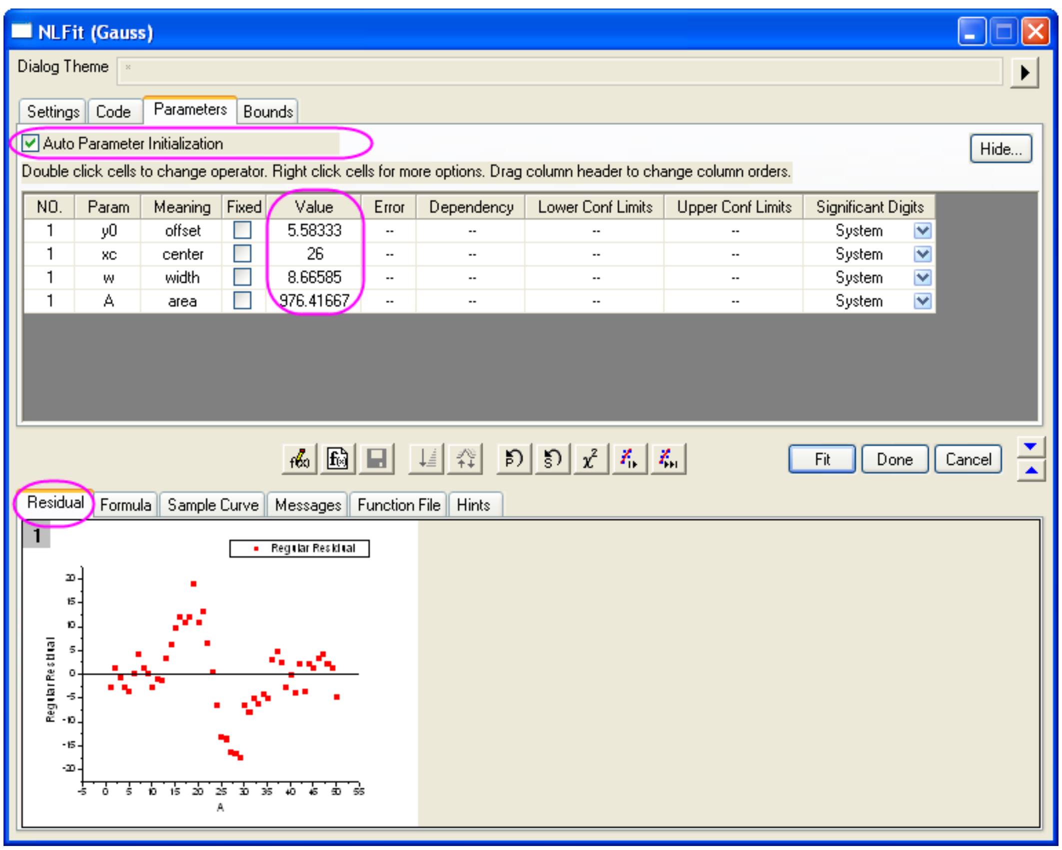
Use a built-in function to fit the data

1. Click **File:Open...** to open the **Intro_to_Nonlinear Curve Fit Tool.opj** from the **\Samples\Curve Fitting** folder and select the **Built-In Function** folder from the Project Explorer window.
2. With the **Graph1** active, select the menu item **Analysis: Fitting: Nonlinear Curve Fit** to bring up the **NLFit** dialog, and then select **Gauss** from the **Function** drop-down list:

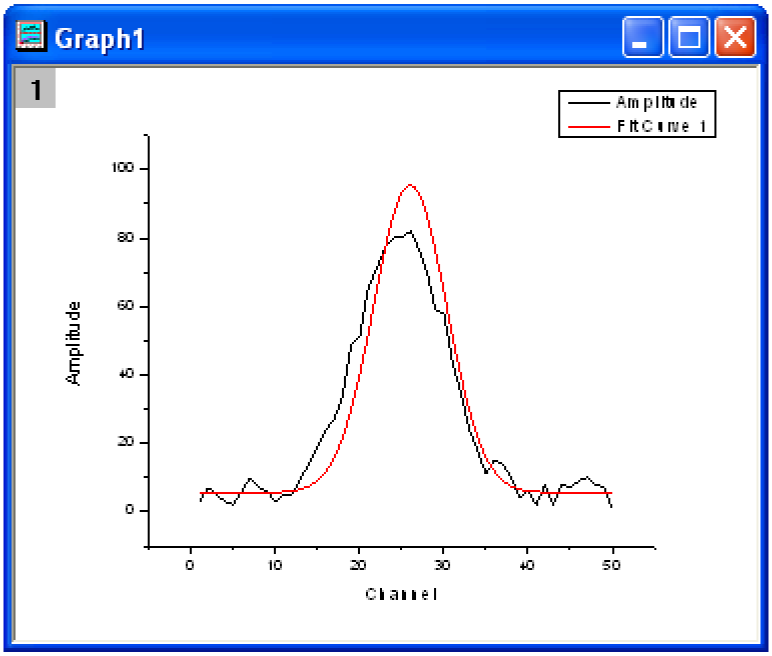


3. In the **Parameter** tab, initial parameter values are automatically assigned, because the built-in functions have parameter initialization code.

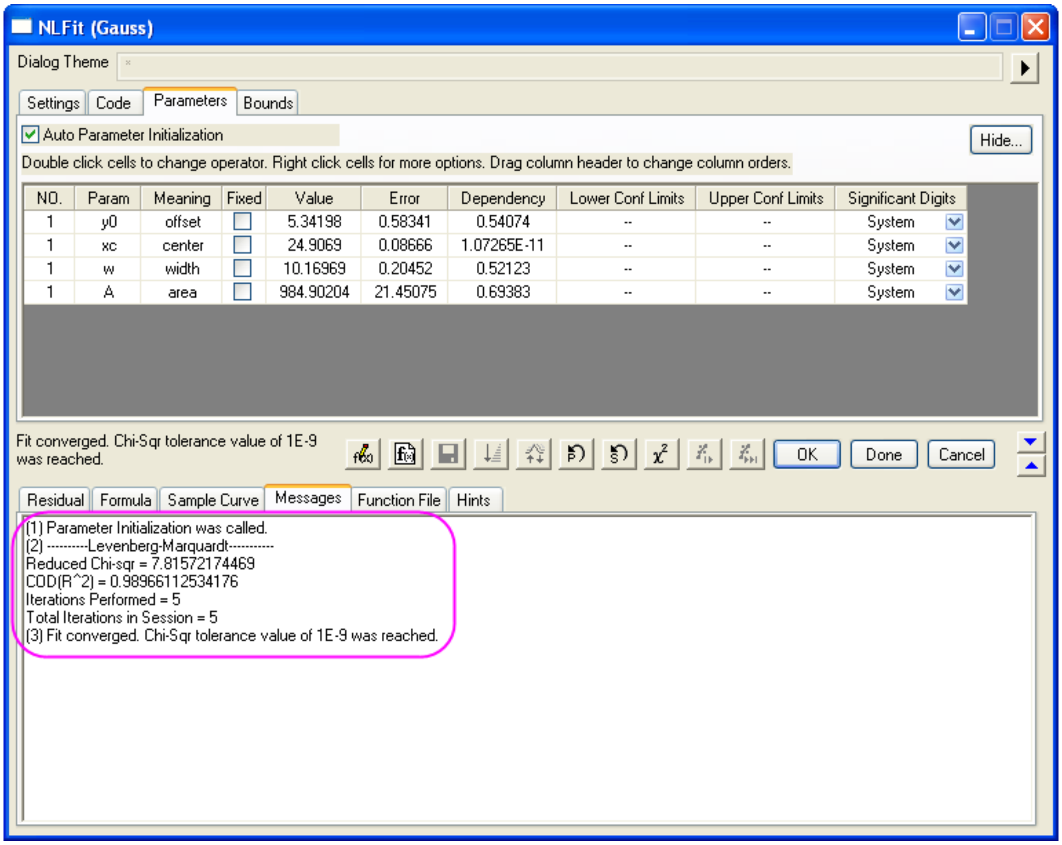
   If you go to the **Residual** tab, you can see the current residuals and you can judge whether the current fit results is good.



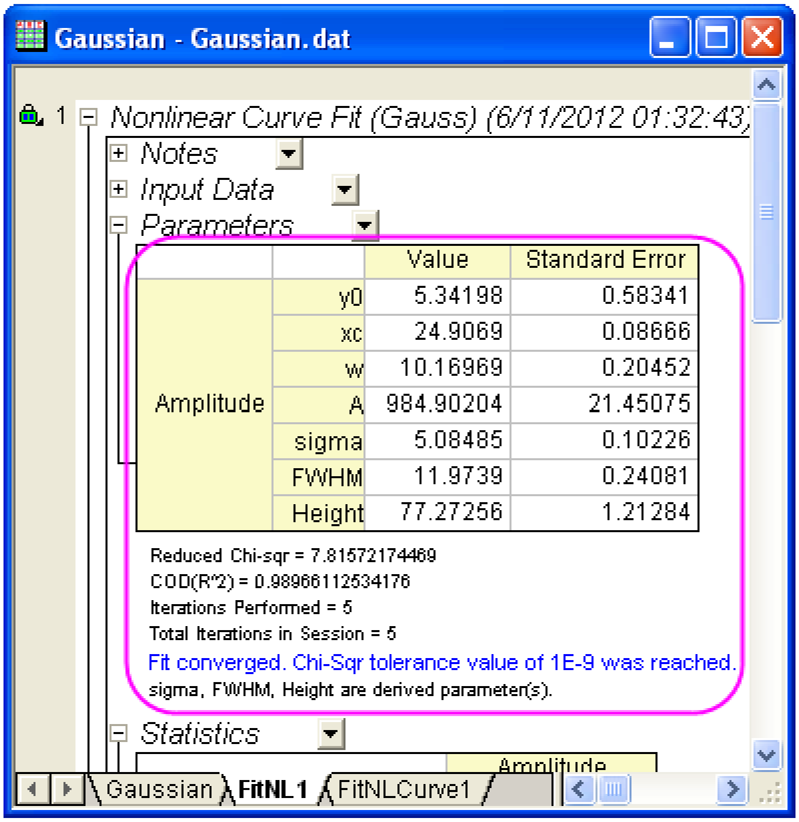
4. A fitted curve determined by the initial values of the parameters is shown in the graph.



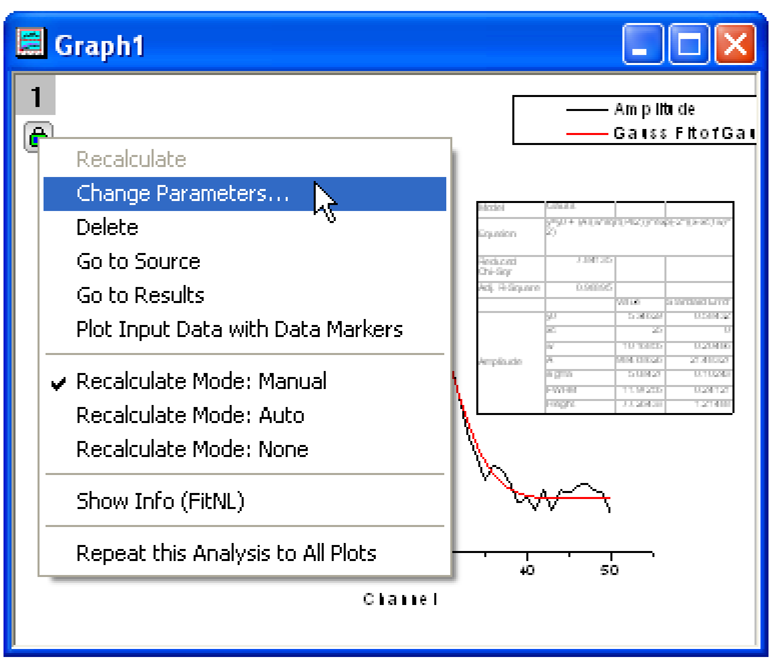
5. Click the **Fit until converged** button , then the **Messages** tab displays number of iterations, reduced chi-sqr, and R^2 values.



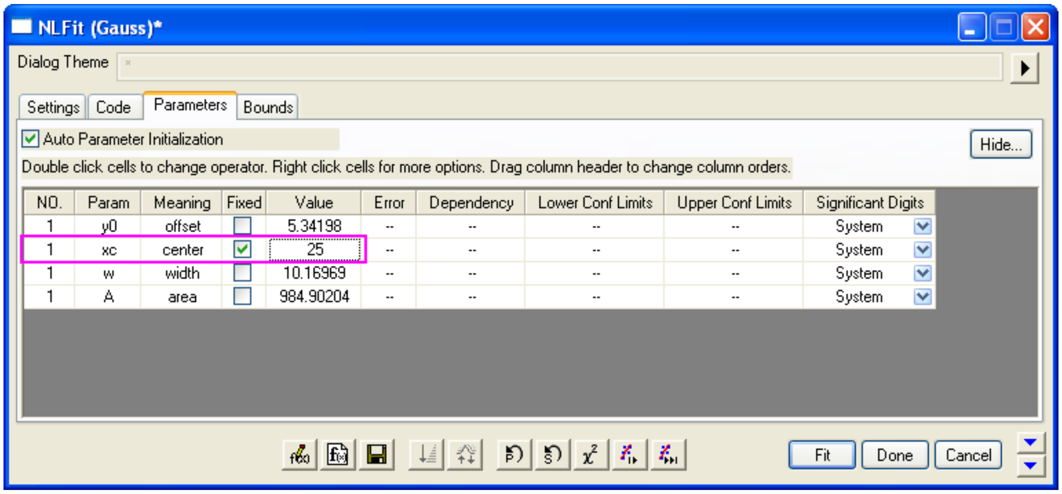
6. Click on the **OK** button, the **FitNL1** report sheet is created with fit results such as parameter values and fit statistics.



7. Reopen the Nonlinear Curve Fitting dialog. You can click on the green lock icon on the upper left corner of the graph and select **Change Parameters** to open it.

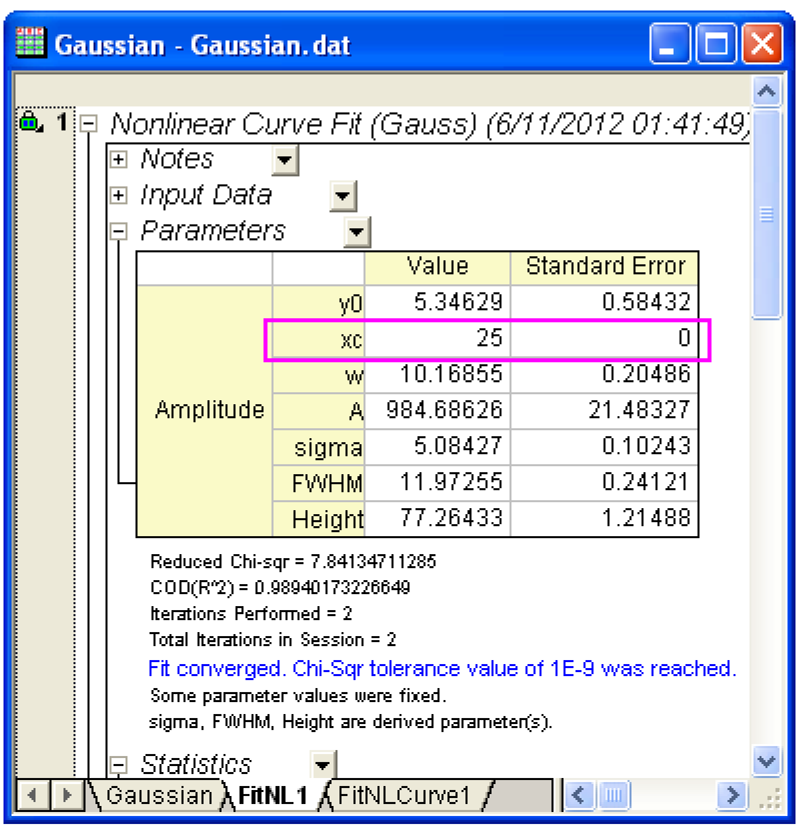


8. Go to the **Parameter** tab, change *xc* value to *25* and check **Fixed** checkbox.



9. Click the **Fit until converged** button and **OK** button again.

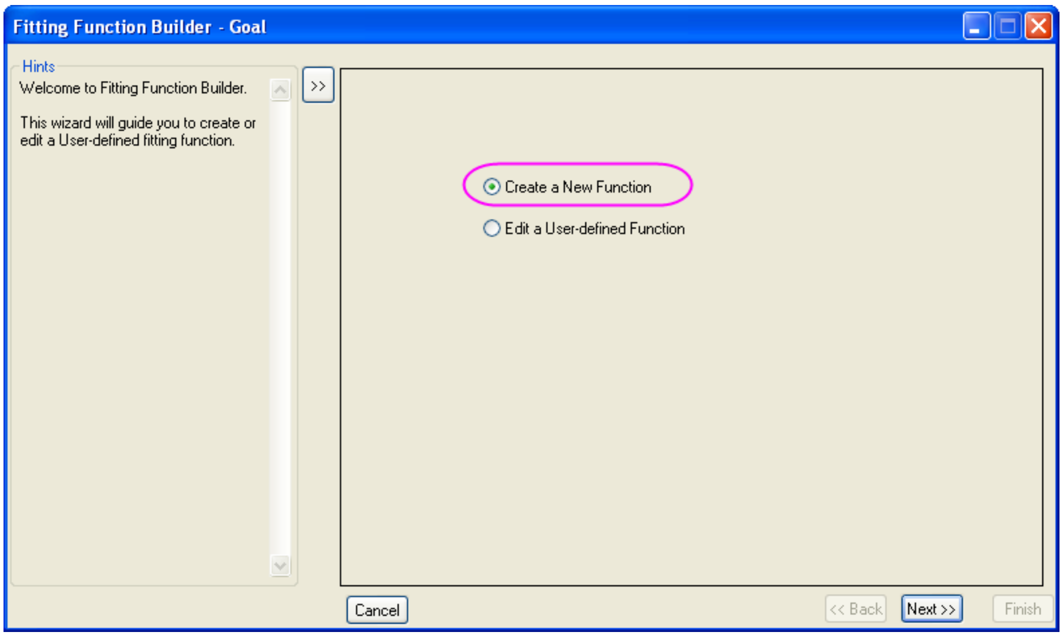
10. In the report sheet, you can see the error value of *xc* is zero because parameter value was fixed.



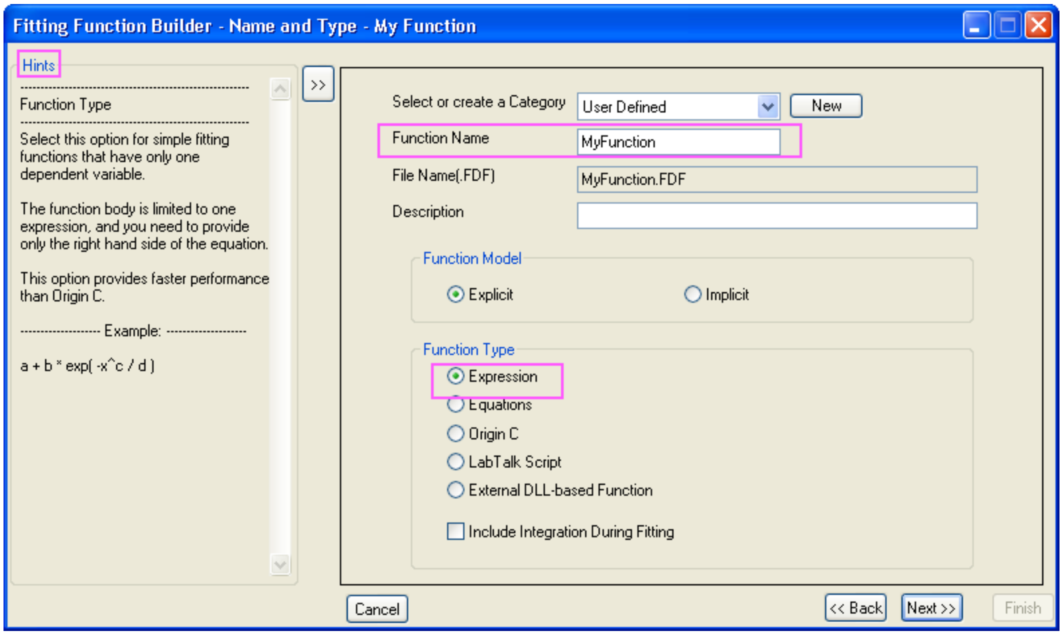
Define and fit with a user-defined function

In this section, we will show you how to define and fit with the following fitting function:
y=y0+a*exp(-b*x)

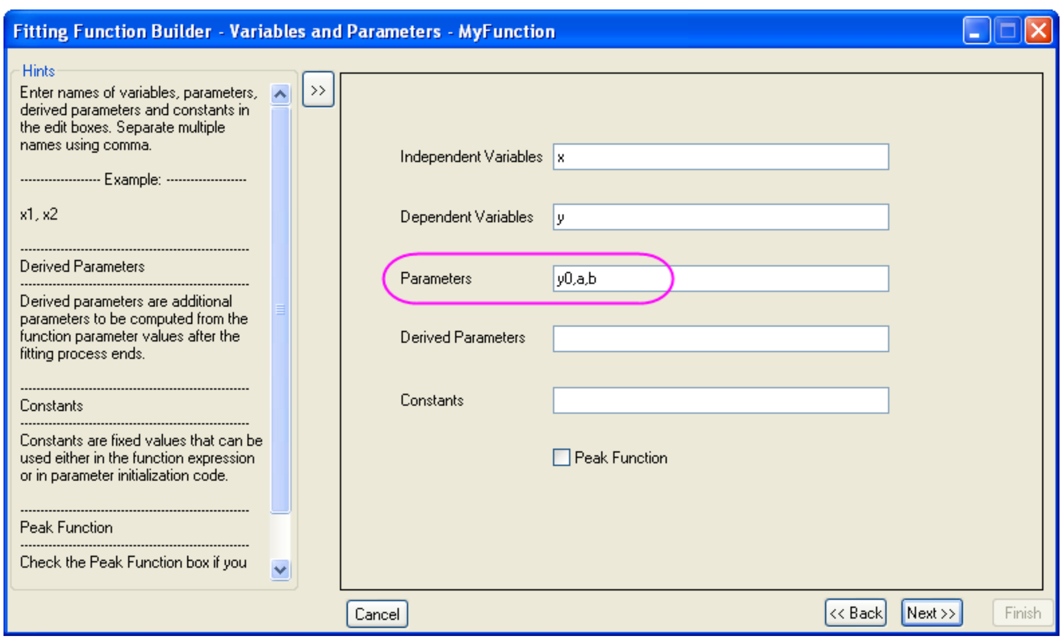
1. Expand **Project Explorer**, and go to **User-Defined Function** folder.
2. In the menu, select **Tools: Fitting Function Builder** to bring up the dialog.
3. Click **Create a New Function** on the **Goal** page, and then click **Next** button to go to the **Name and Type** page.



4. On the **Name and Type** page, you can create a category for the new fitting function, then name the function and select a Function Type.
   - Set **MyFunction** as the **Function Name**.
   - Select **Expression** from the **Function Type list**. You can find Hints in the left panel.
   - Click the **Next** button to go to the **Variables and Parameters** page.

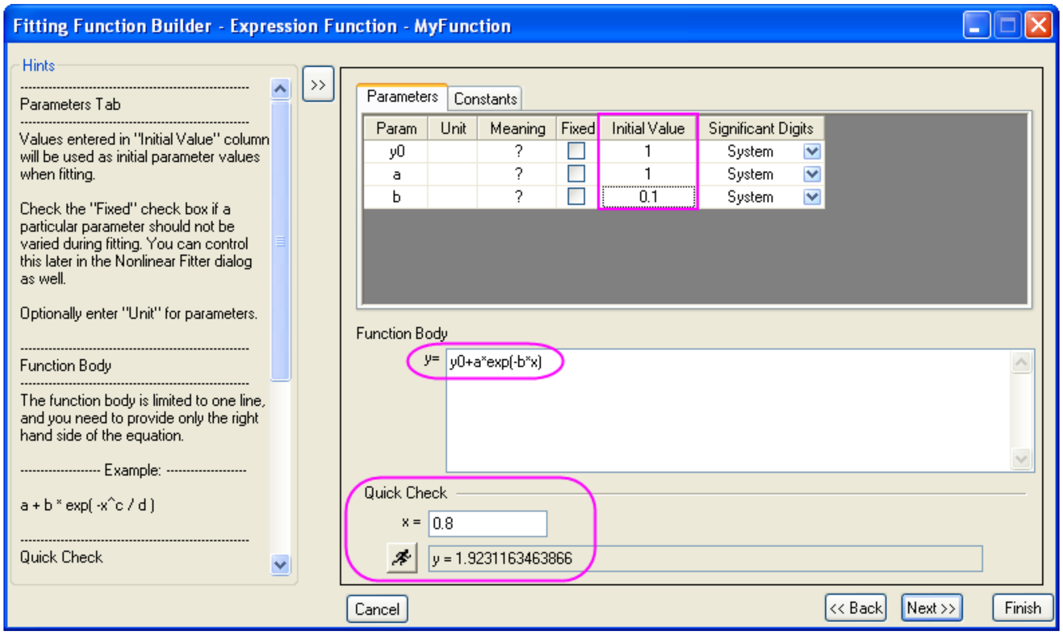


5. On the **Variables and Parameters** page, make sure **Independent Variables** is *x* and **Dependent Variables** is *y*. Then input *y0, a, b* into the **Parameters** textbox. Click the **Next** button.

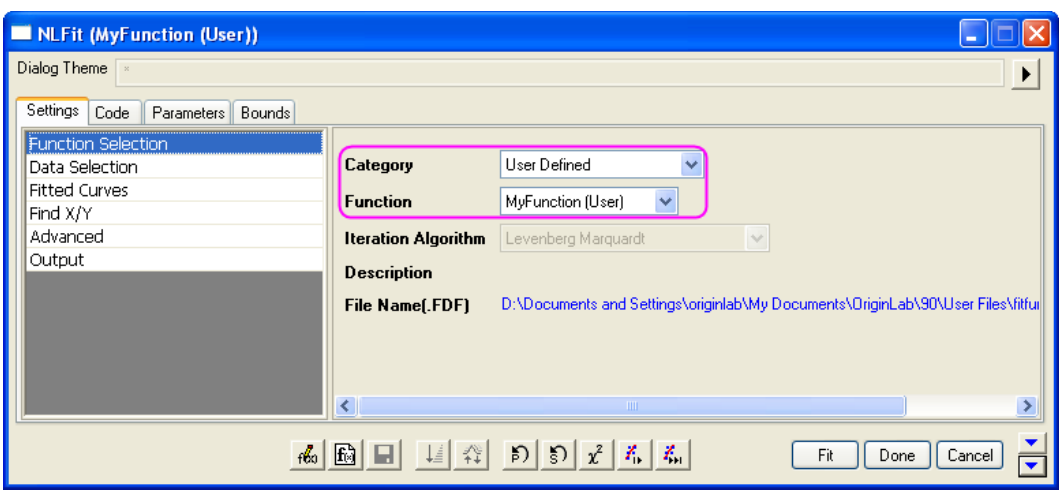


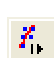
6. On the **Function Body** page, you can:
   - Set the **Initial Values** for parameters.
   - In the **Function body** edit box, enter *y0+a*exp(-b*x)*.

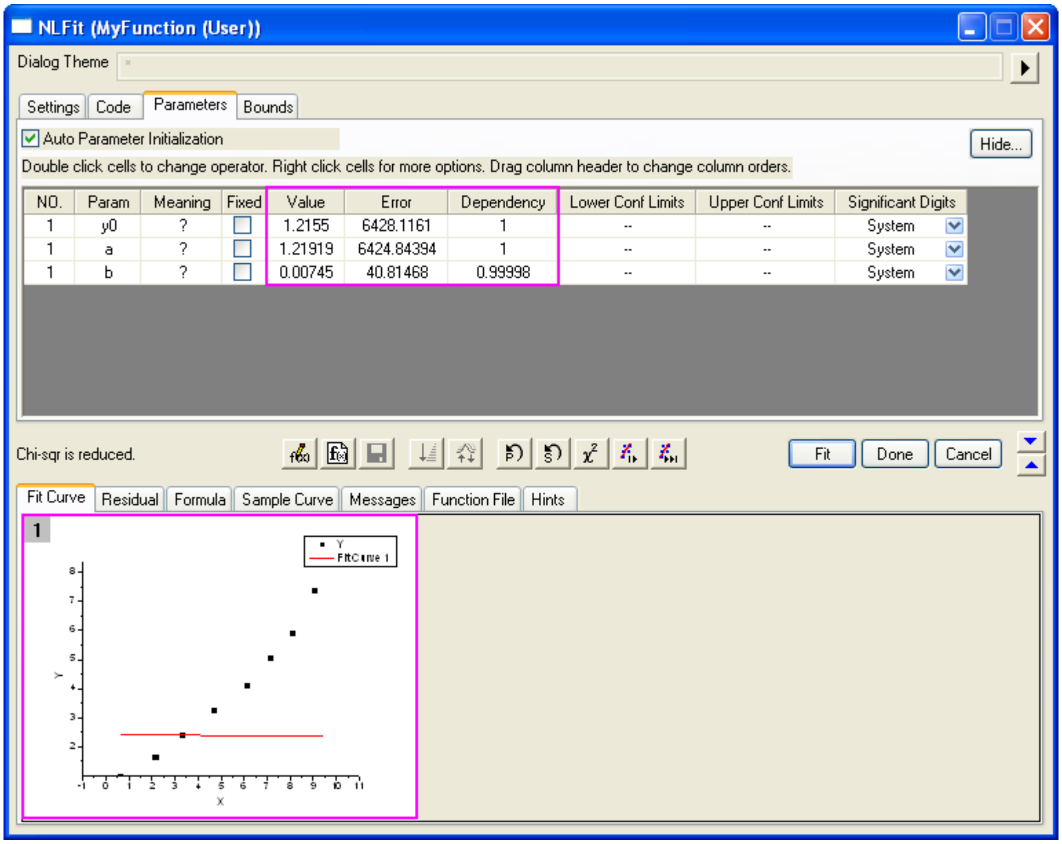
- Use **Quick Check** to check the validity of a function. After entering value for the independent variables, click the **Evaluate** button to quickly check the fitting function.

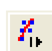


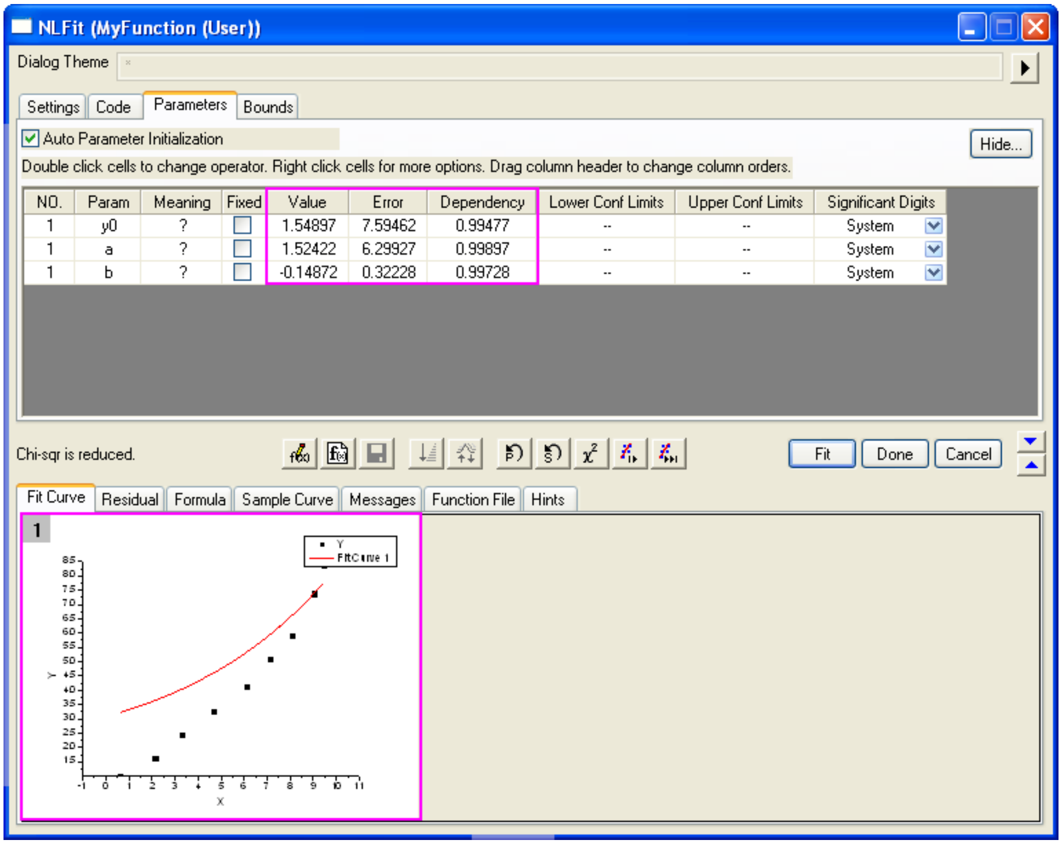
7. Click **Finish** button.
8. Highlight column A and B, select the menu item **Analysis: Fitting: Nonlinear Curve Fit** to bring up the **NLFit** dialog. Select the function **MyFunction** on the **Function Selection** page, under the **Settings** tab:

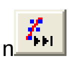


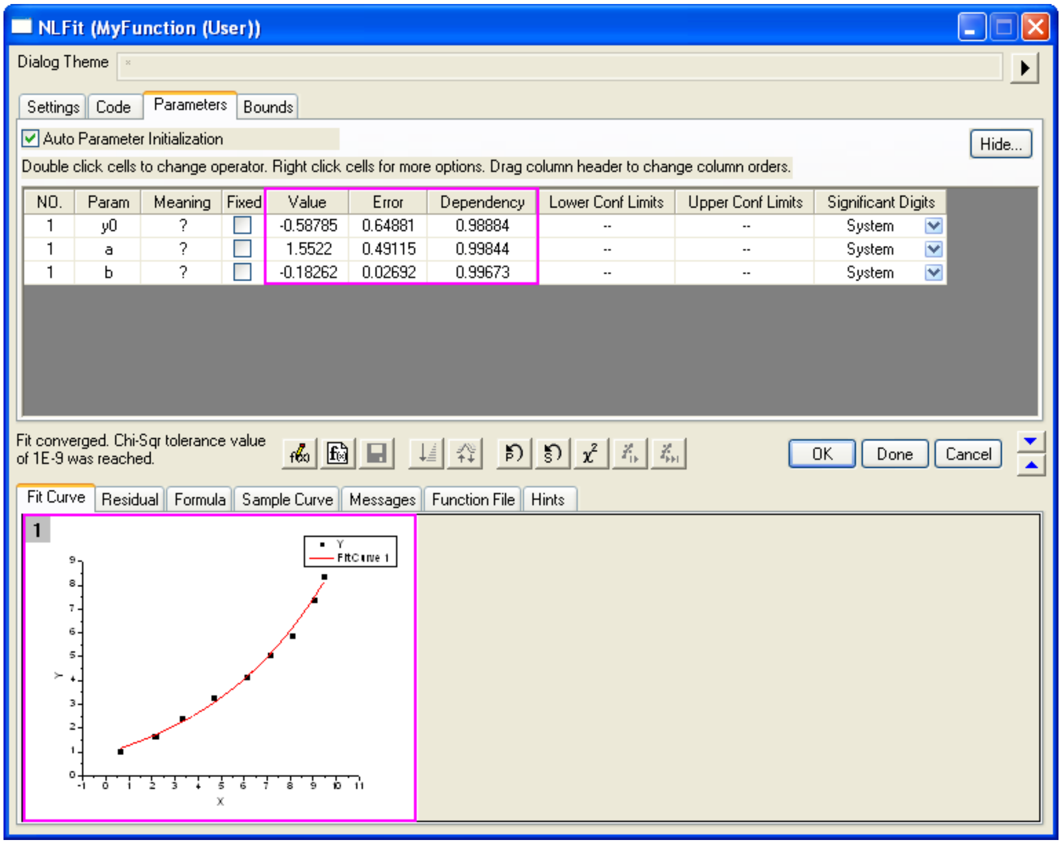
9. Fitting three times by following the steps, you can see the change of parameter values as well as the fitted curve.
   - Click the "1 Iteration" button .



o Click the "1 Iteration" button  again.



- Click the **Fit until converged** button 



10. Click the **OK** button, the FitNL1 report sheet is created with fit results such as parameter values and fit statistics.

Nonlinear Fitting with System Function

Summary

The NLFit dialog is an interactive tool which allows you to monitor the fitting procedure during the non-linear fitting process. This tutorial fits the Michaelis-Menten function, which is a basic model in Enzyme Kinetics, and shows you some basic features of the NLFit dialog. During the fitting, we will illustrate how to perform a Global Fit, which allows you to fit two datasets simultaneously and share some parameter values.

Minimum Origin Version Required: Origin 8.0 SR6

What you will learn

This tutorial will show you how to:

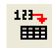
- Import a single ASCII file
- Perform a global fit with shared parameters
- Select a fitting range and fit part of the data

- Use the Command Window to perform simple calculation

Steps

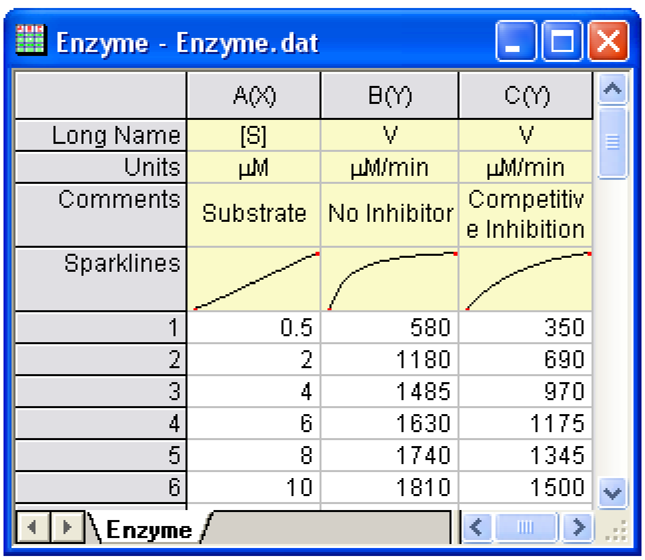
Import the file

- Open a new workbook.

- Click the *Import Single ASCII* button  to bring up the **Open** dialog. Browse to *\Samples\Curve Fitting* folder and select the file *Enzyme.dat*. Make sure to check the **Show Options Dialog** checkbox at the bottom of the dialog, and then click **Open**.

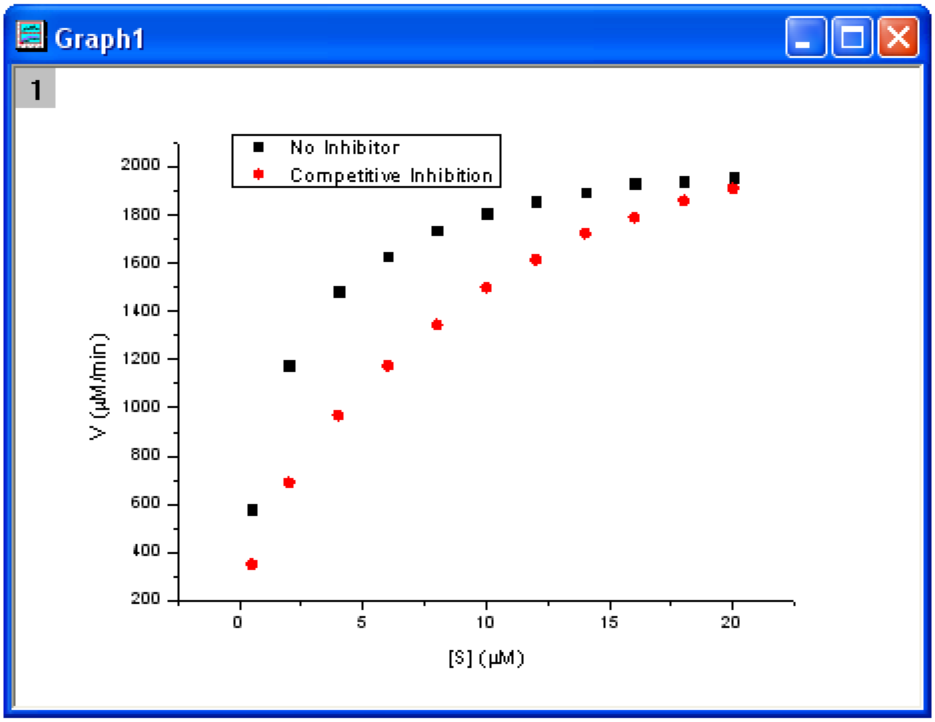
- In the **impASC** dialog, expand **Import Options: Header Lines** nodes, and select **3** from *Comments From* drop down.

- Click OK to import the file.



Plotting the Data

- Highlight columns B & C and plot as a scatter plot by clicking the  button.



Fitting Michaelis-Menten Function

The single-substrate Michaelis-Menten function:

$$v = \frac{V_{max}[S]}{K_m + [S]}$$

is a basic model in enzyme kinetics study, where *v* is the reaction velocity, [*S*] is the substrate concentration, V_{max} is the maximal velocity and K_m represents the Michaelis constant. We can determine the V_{max} and K_m value, which are important enzyme properties, by fitting M-M function on *v* vs. [*S*] curve.

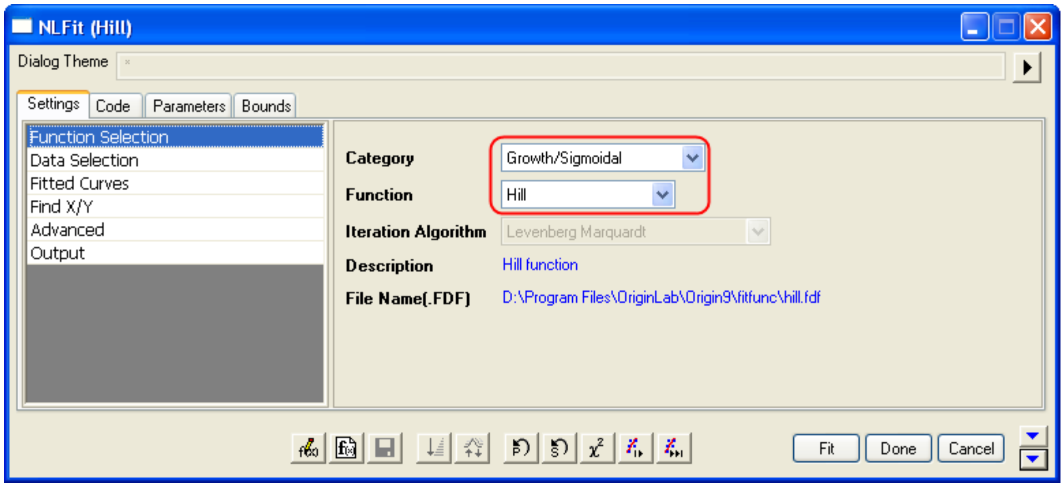
There is no M-M fitting function in Origin; however, we can use a more general model, the built-in *Hill* function to fit:

$$v = V_{max}\frac{x^n}{k^n + x^n}$$

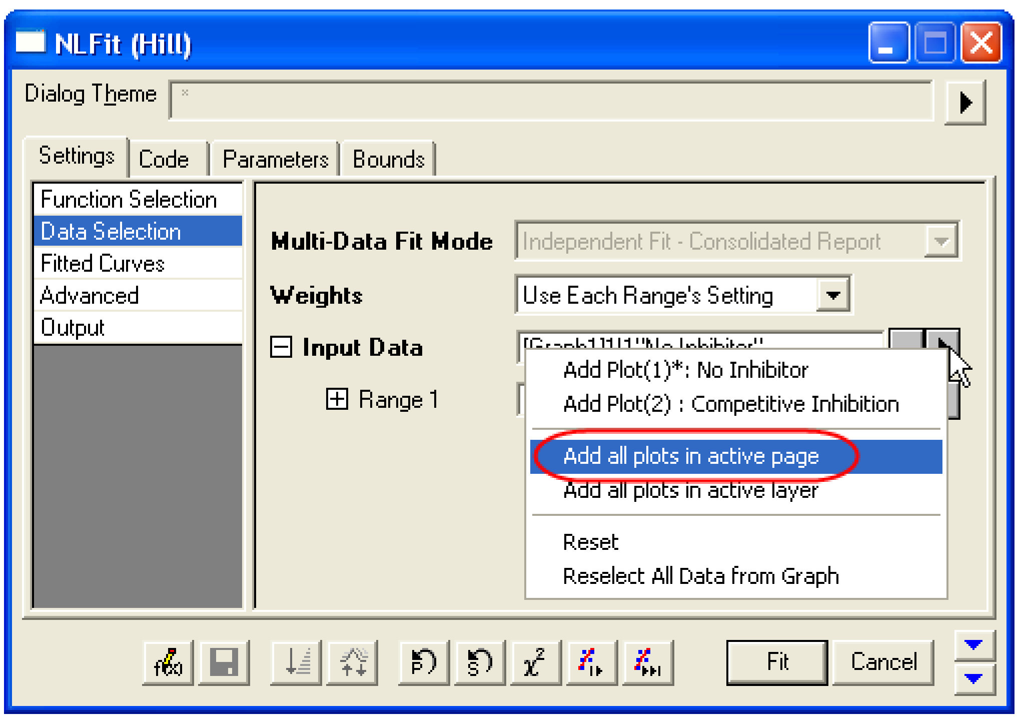
where *n* means the cooperative sites. For single-substrate model, we can just fix *n* = 1 during fitting and it will become the simplest form, the M-M function.

There are two curves, reaction without Inhibitor and reaction with Competitive Inhibitor in the graph, and the NLFit tool can fit these two curves simultaneously. Since for competitive inhibition reaction, the maximum velocity is the same with no inhibition reaction, we can share the V_{max} value during the fitting procedure, which can be implemented by a *Global Fit*.

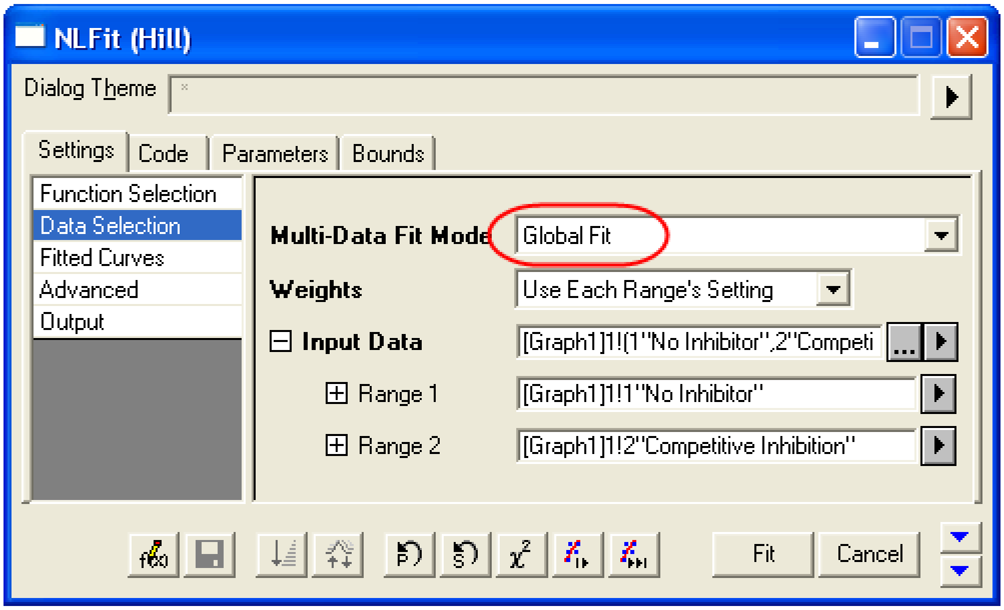
- With the graph active, select the menu item **Analysis: Fitting: Nonlinear Curve Fit** to bring up the NLFit dialog. Select *Hill* function from *Growth/Sigmoidal* category on the **Settings: Function Selection** page.



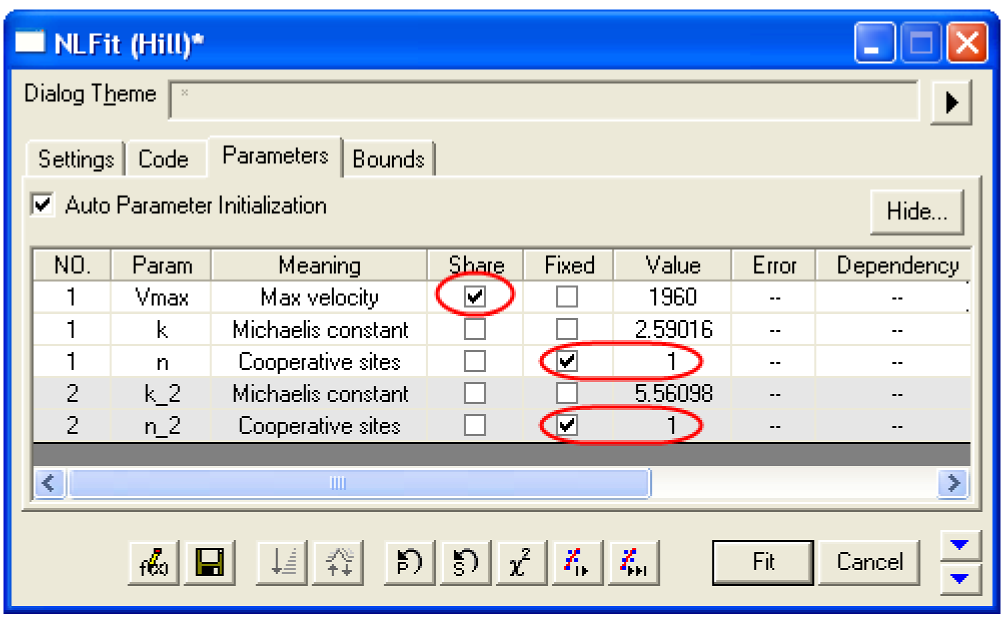
- On **Settings: Data Selection** page, click the triangular button next to the **Input Data** and choose **Add all plots in active page** to set the data.



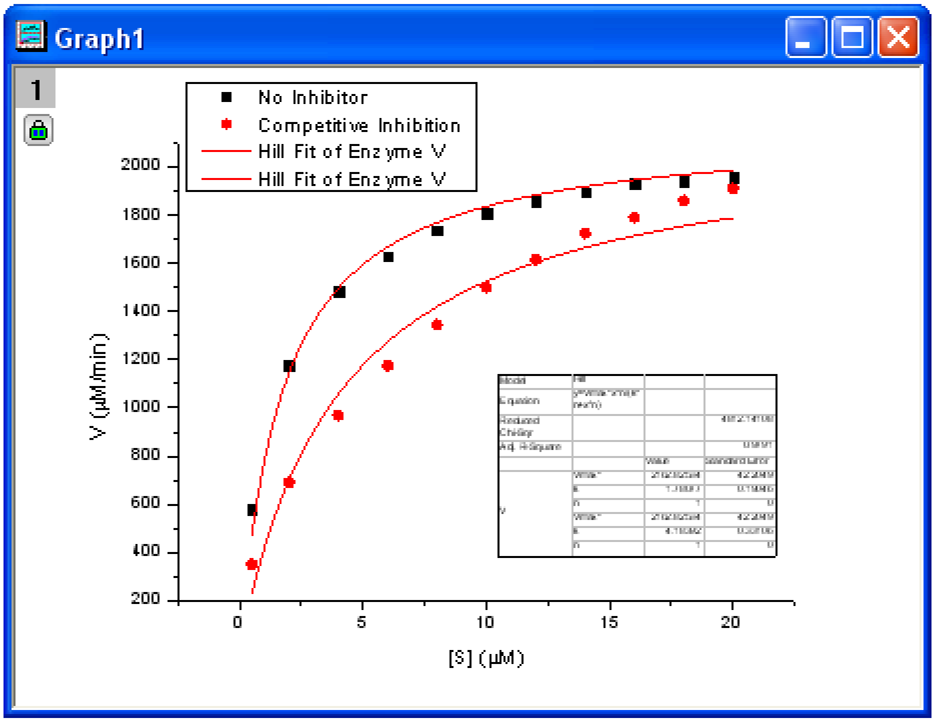
- Select *Global Fit* from **Multi-Data Fit Mode** drop-down list on the **Settings: Data Selection** page.



- Switch to the **Parameters** tab, check the **Share** box on the Vmax row. These *Share* check boxes are only available when using *Global Fit* mode. Check the **Fixed** box for n and n_2, and make sure their values are 1.



After that, click the *Fit* button to generate reports. The fit result will also be pasted on the original graph. (We just show the parameter values in the following figure.)



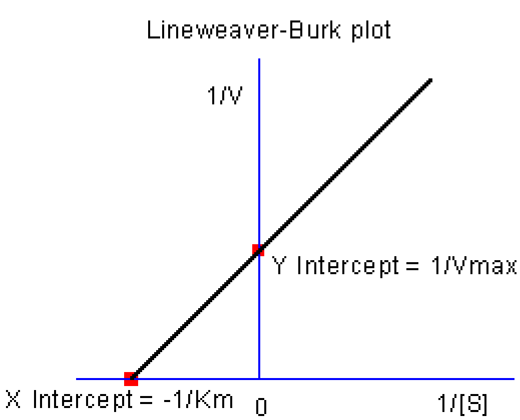
From the fit result, we can conclude that the maximum velocity is 2162.8 µ*M* / *min*. and K_m for no inhibitor and competitive inhibitor model is 1.78µ*M* and 4.18µ*M*, respectively.

Fitting Lineweaver-Burk Plot

As we know, the model parameters can also be estimated by the Lineweaver?Burk or double-reciprocal plot. The Lineweaver?Burk plot takes the reciprocal of both sides of the M-M function and plots by 1/v vs. 1/[S]:

$$\frac{1}{v} = \frac{1}{V_{max}} + \frac{K_m}{V_{max}[S]}$$

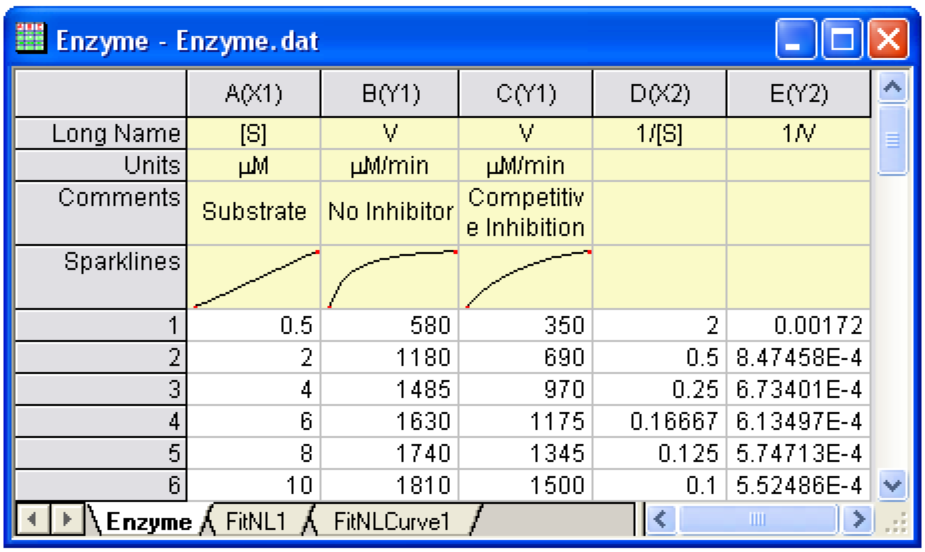
This is actually a linear function:



We will use the No Inhibitor data to illustrate how to calculate K_m and V_{max} by L-B plot.

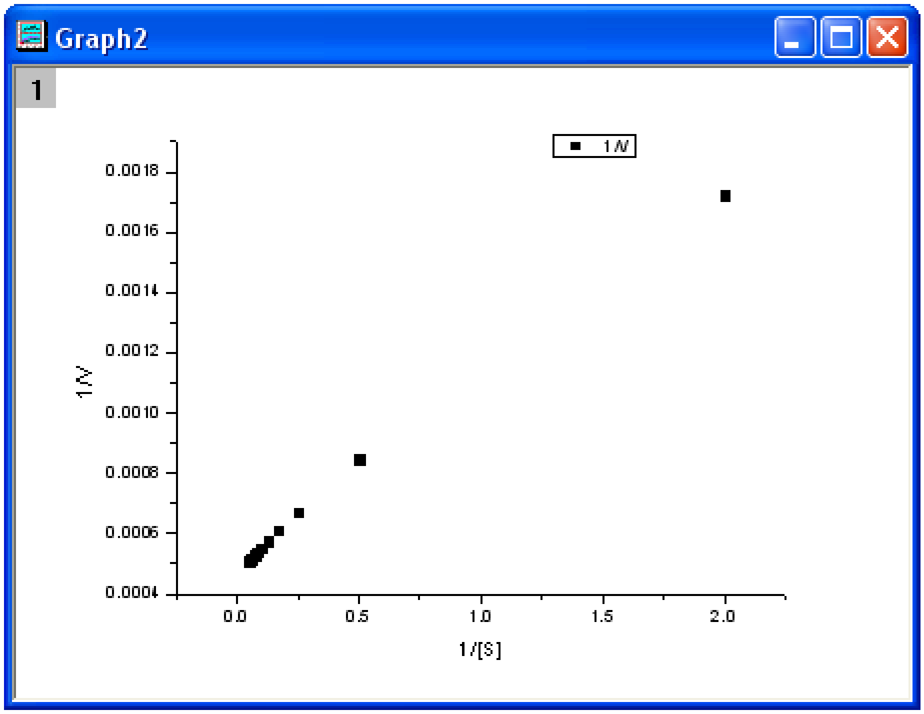
- Go back to the raw data worksheet and add two more columns by clicking the button. Right-click on column D and select **Set As: X** from the context fly-out menu to set it as an X column. Right-click on column D again and select **Set Column Values** to bring up the **Set Values** dialog. In the dialog edit box, enter: `1/Col(A)` and set the **Recalculate** mode as *None*, since we don't need to auto update the reciprocal values in this example.

Similarly, set column E's values as `1/Col(B)`. Enter the long name for column D & E as 1 / [*S*] & 1 / *V*, respectively. And then we have:



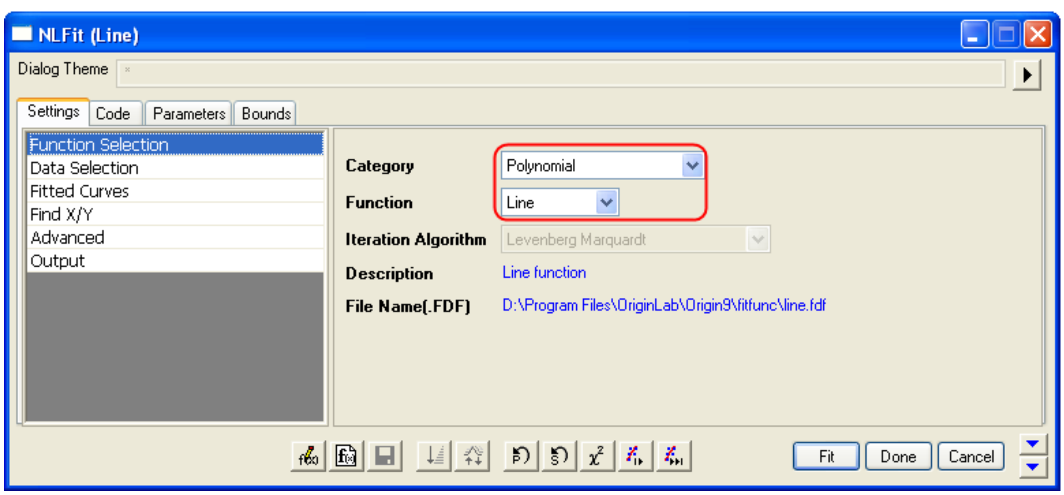
	A(X1)	B(Y1)	C(Y1)	D(X2)	E(Y2)
Long Name	[S]	V	V	1/[S]	1/V
Units	µM	µM/min	µM/min		
Comments	Substrate	No Inhibitor	Competitive Inhibition		
Sparklines					
1	0.5	580	350	2	0.00172
2	2	1180	690	0.5	8.47458E-4
3	4	1485	970	0.25	6.73401E-4
4	6	1630	1175	0.16667	6.13497E-4
5	8	1740	1345	0.125	5.74713E-4
6	10	1810	1500	0.1	5.52486E-4

- Highlight columns D & E and click button to create a scatter plot.

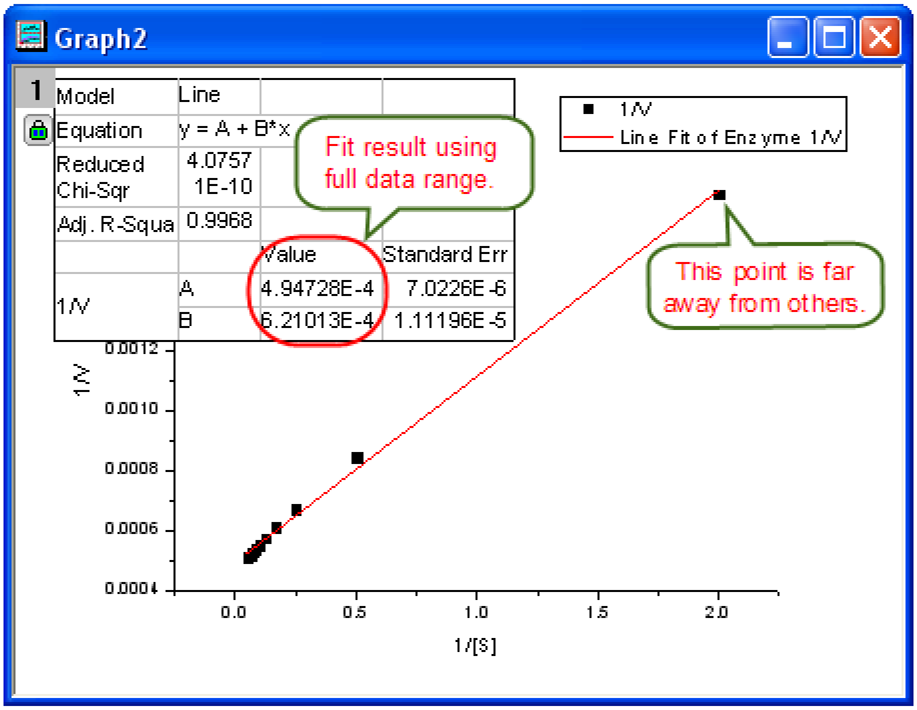


From the above equation, we know there is a linear relationship between 1/v and 1/[S], so we can use the NLFit tool to fit a straight line on this plot. (You can also use the Fit Linear tool from **Analysis: Fitting: Fit Linear**)

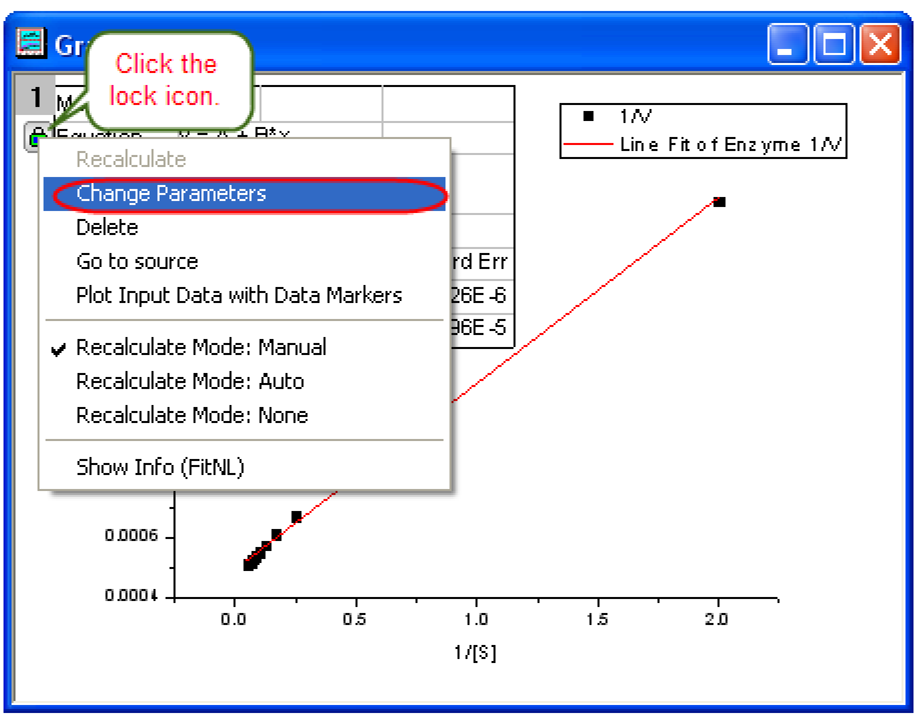
- Bring up the NLFit dialog again, select *Line* function from *Polynomial* category, and then click the *Fit* button Fit directly to generate results.



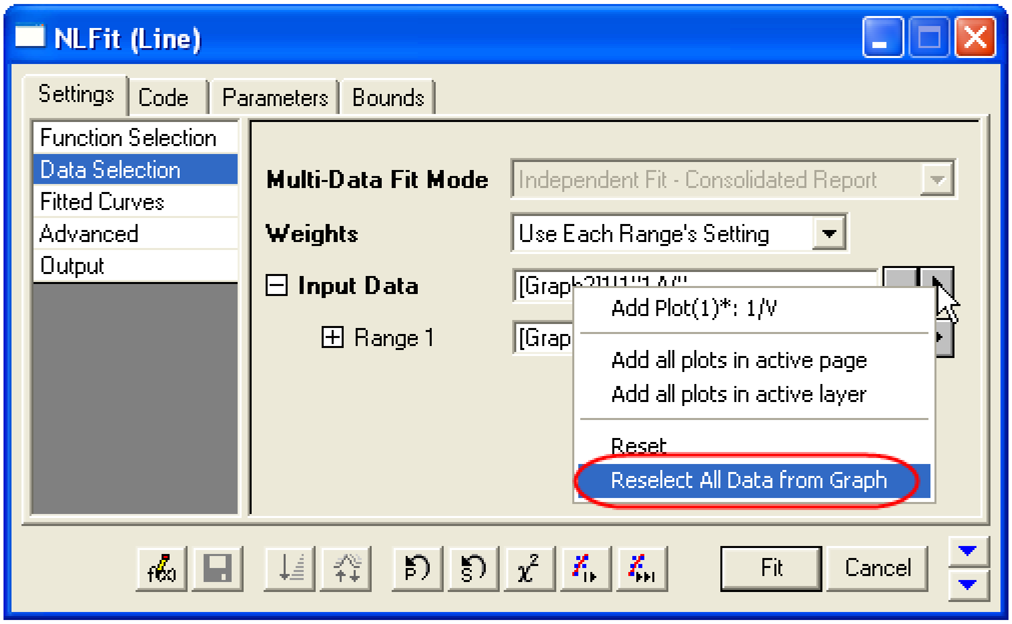
From the plot, one may doubt that this is the best fit curve since there is a point located far away. Actually, the right side of L-B plot is low substrate concentrations area, the measurement error may be large, so we'd better exclude these points during fitting.

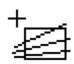


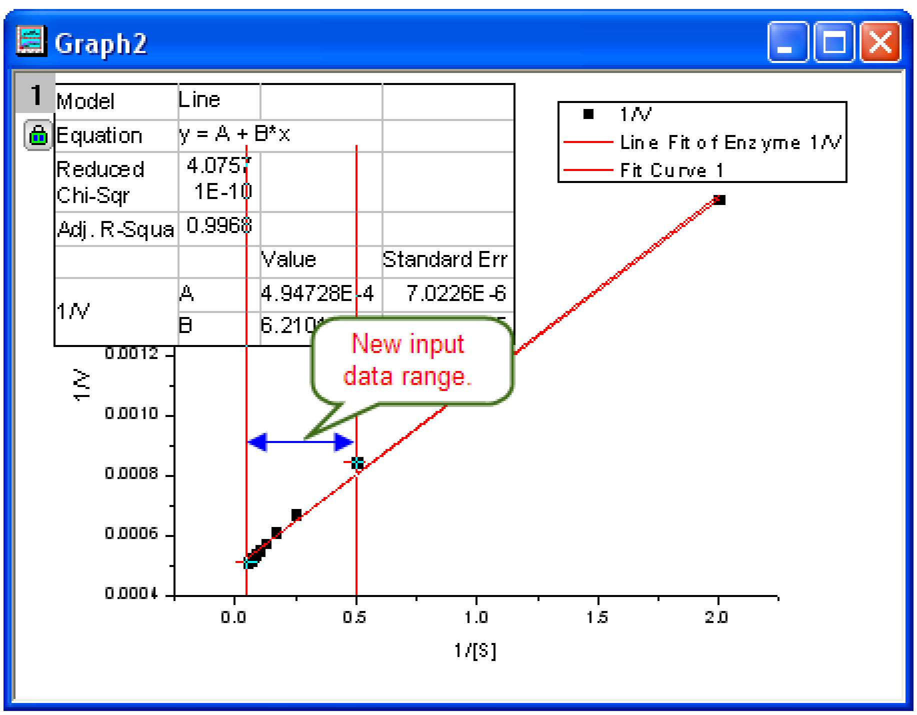
- Click the lock icon on the graph upper-left corner, and select *Change Parameters* to bring back NLFit dialog.



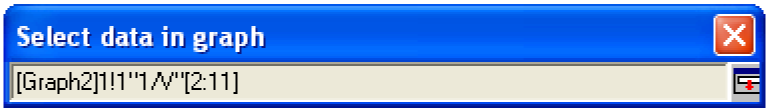
In **Settings: Data Selection** page, click the ▶ button on **Input Data** node, and then choose *Reselect All Data from Graph* from fly-out menu.



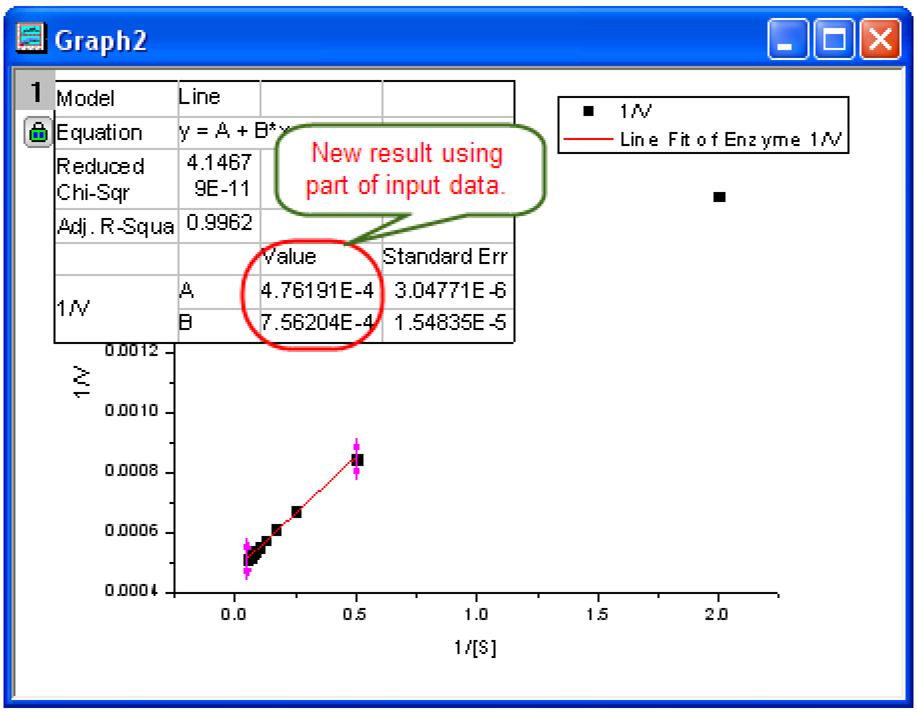
Then the NLFit dialog rolls up and your cursors become  when you move to the graph page. Click and draw a rectangle to select data points you want to fit. The input range is labeled by vertical lines. You can also click-and-move these lines to change the input range.



Click the  button on **Select Data in Graph** window to go back to NLFit dialog.



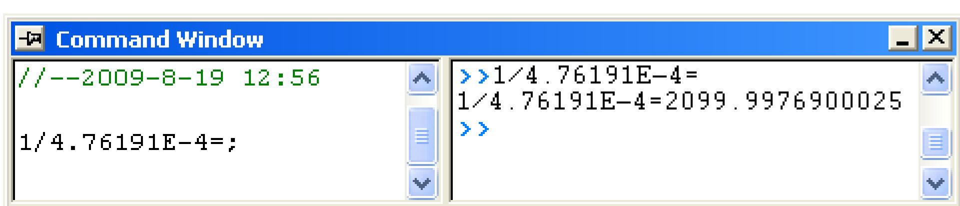
- Click the *Fit* button on the NLFit dialog to recalculate the result. You can see from the graph that the report table was updated.



- Since the intercept of the fitted curve is 1 / V_{max}, it is equal to 4.76191E-4 in this example. To get the V_{max} value, select **Window: Command Window** to open the command window, type

```
1/4.76191E-4 =
```

and press ENTER:



Origin returns the value 2099, which is close to what we got above, 2160. (When fitting the hill function above, we shared V_{max} when fitting two datasets. If you fit the No Inhibitor data only, this value will be closer.)

Global Fitting with Parameter Sharing

Summary

Global fit is one of the fit modes in Origin when fitting multiple curves. It will fit all datasets simultaneously, allowing parameter sharing. Compared to concatenate fit, which combine all datasets into one, global fitting performs chi-square minimization in a combined parameter space, so the parameter errors, DOF, npts and even parameter values may be different from a concatenated fit. Therefore global fitting is only appropriate/necessary if you want to share parameters.

Minimum Origin Version Required: Origin 8.0 SR6

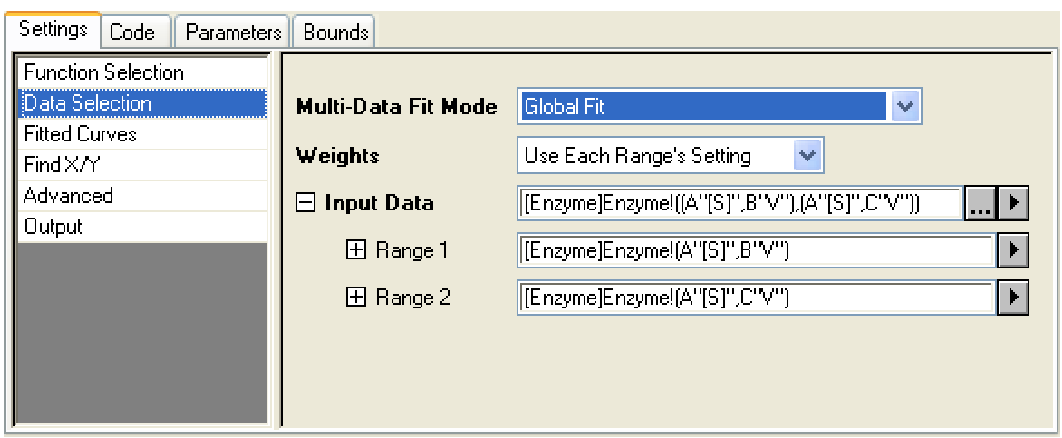
What you will learn

This tutorial will show you how to:

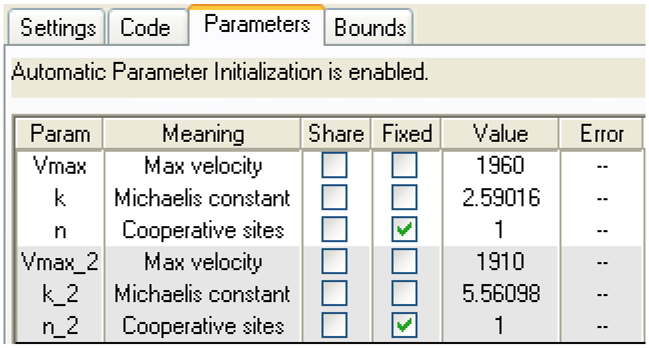
- Select multiple datasets for fitting.
- Select different fitting modes.
- Perform global fit with parameter sharing.

Steps

1. Start with a new workbook and import the file *\Samples\Curve Fitting\Enzyme.dat*.
2. Highlight column B and C and bring up the **NLFit** dialog from **Analysis: Fitting: Nonlinear Curve Fitting**. In the **Function Selection** page of **NLFit** dialog, choose **Hill** function from **Growth/Sigmoidal** category. Go to **Data Selection** page, and select **Global Fit** mode from **Multi-Data Fit Mode** drop-down list:



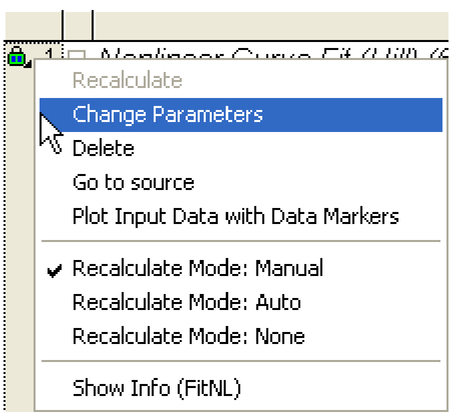
   Then make sure the **Recalculate** mode is **Manual** in the **Advanced** page.
3. Active the **Parameters** tab. Check the **Fixed** checkbox for *n* and *n_2* to fix their values to 1.



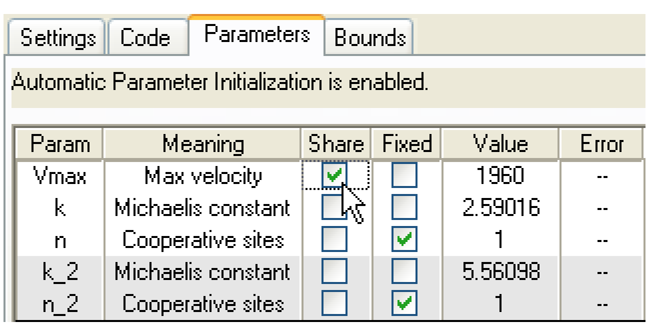
Click the **Fit** button to fit curves. You can see these results from the report worksheet:

		Value	Standard Error
V	Vmax	2091.96109	33.62032
	k	1.52432	0.13786
	n	1	0
	Vmax	2428.49265	81.97136
	k	5.86377	0.56737
	n	1	0

4. Since the maximum velocity, *Vmax* in this case, maybe the same. We now want to share this parameter value to fit. Click the lock icon in the report worksheet and select **Change Parameters** to bring back the **NLFit** dialog.



5. In the **Parameters** tab, check the **Shared** checkbox for *Vmax*.



And then click the **Fit** button again to generate new results, you can see the *Vmax* values for both curves are the same. The asterisk in parameter name means that this parameter is shared:

		Value	Standard Error
V	Vmax*	2162.82534	42.2049
	k	1.78077	0.19046
	n	1	0
	Vmax*	2162.82534	42.2049
	k	4.18392	0.33106
	n	1	0

User Defined Fitting Function using Origin C

Summary

All the fitting functions in Origin are organized by the **Fitting Function Organizer**. Besides the built-in functions, you can also create user-defined functions in the **Fitting Function Organizer** or the **Fitting Function Builder**. Once a function is created, it can be accessed in the **NLFit** dialog. We will illustrate how to fit with a user-defined function below.

Minimum Origin Version Required: Origin 8.5 SR1

What you will learn

- How to create a user-defined fitting function using the Fitting Function Organizer.
- How to create a user-defined fitting function using the Fitting Function Builder.

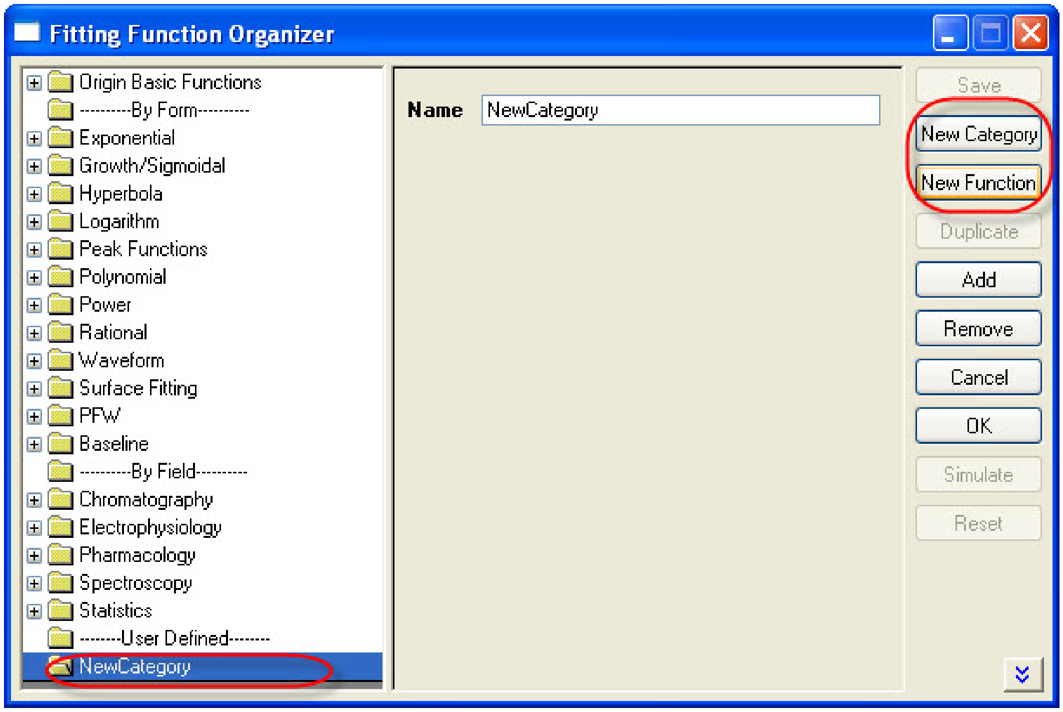
Example

We will illustrate how to define the following fitting function:

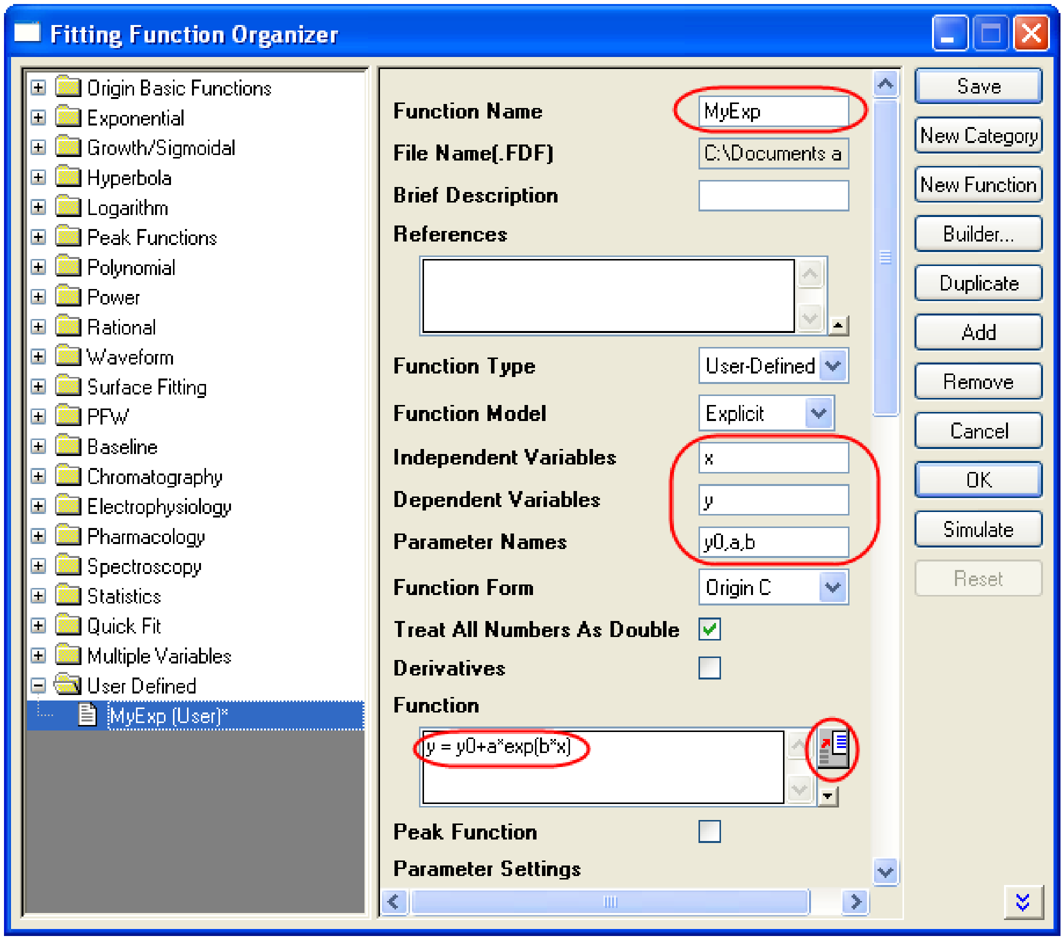
$y = y0 + ae^{bx}$

Define the function in Fitting Function Organizer

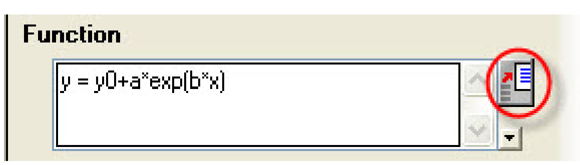
1. Select **Tools: Fitting Function Organizer** from the menu (or press **F9**) to open the function organizer. Click the **New Category** button to create a function category, rename it as *User-Defined*, for example. Then press the **New Function** button to create a **new function** under this category:


Fitting Function Organizer
Origin Basic Functions
---------By Form---------
Exponential
Growth/Sigmoidal
Hyperbola
Logarithm
Peak Functions
Polynomial
Power
Rational
Waveform
Surface Fitting
PFW
Baseline
---------By Field---------
Chromatography
Electrophysiology
Pharmacology
Spectroscopy
Statistics
--------User Defined-------
NewCategory
Name
NewCategory
Save
New Category
New Function
Duplicate
Add
Remove
Cancel
OK
Simulate
Reset

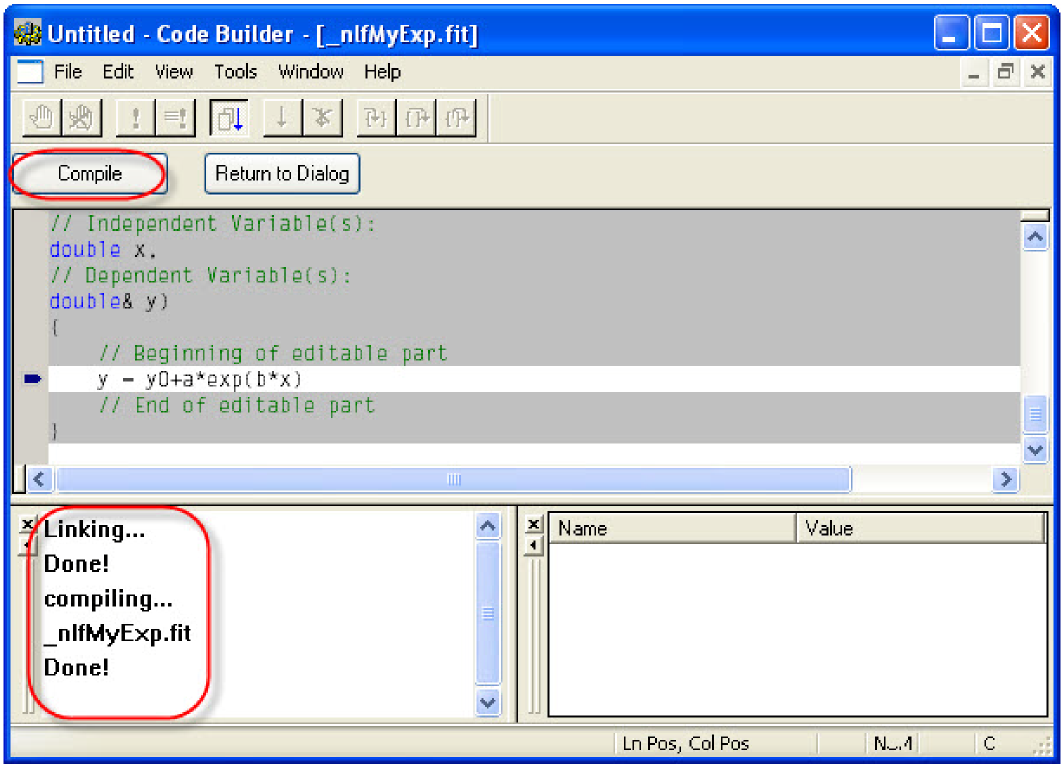
2. Enter function definition as shown in the following image and **Save**:



3. To verify the correctness of the function, click the button beside the **Function** box to open the Origin Code Builder:



In the **Code Builder**, click the **Compile** button to compile the function. If passed, click the **Return to Dialog** button to return to the **Fitting Function Organizer**.

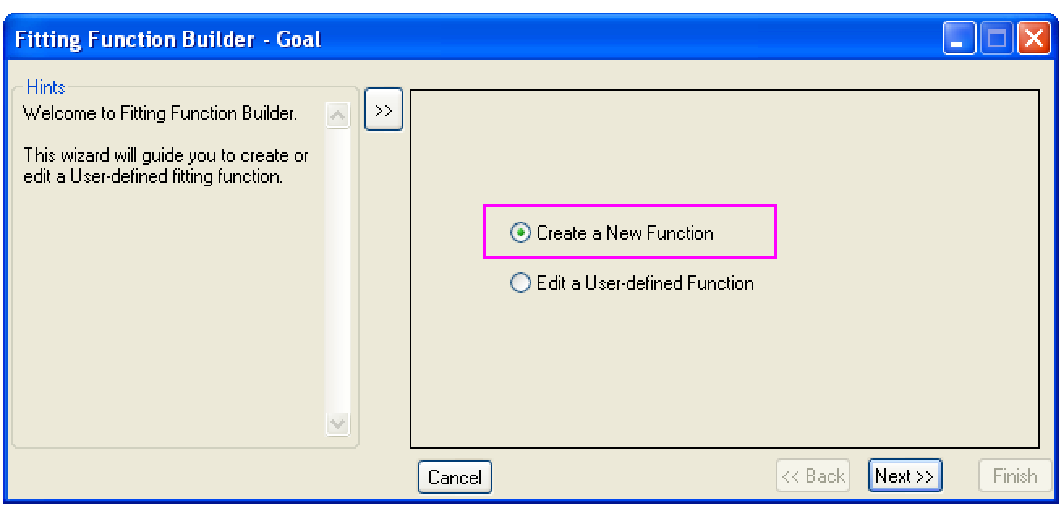


4. Click **Save** and **OK** to save the function and quit the **Fitting Function Organizer**.

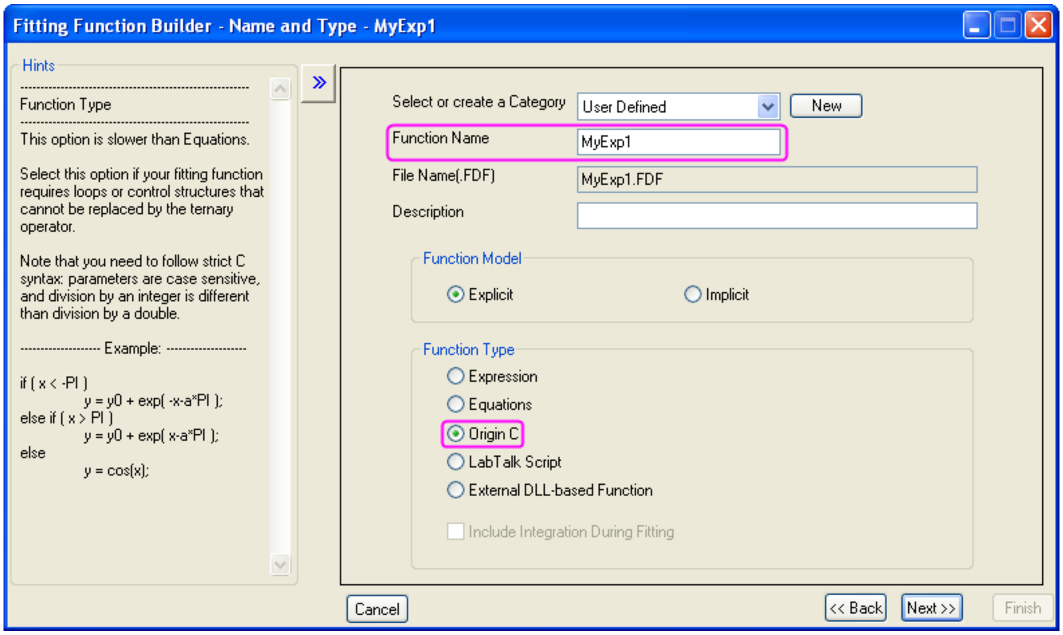
Define the function in Fitting Function Builder

Since Origin 8.5, a new fitting function wizard **Fitting Function Builder** has been added, which can be used to create or edit a user-defined function. Below, let's define the fitting function using this wizard.

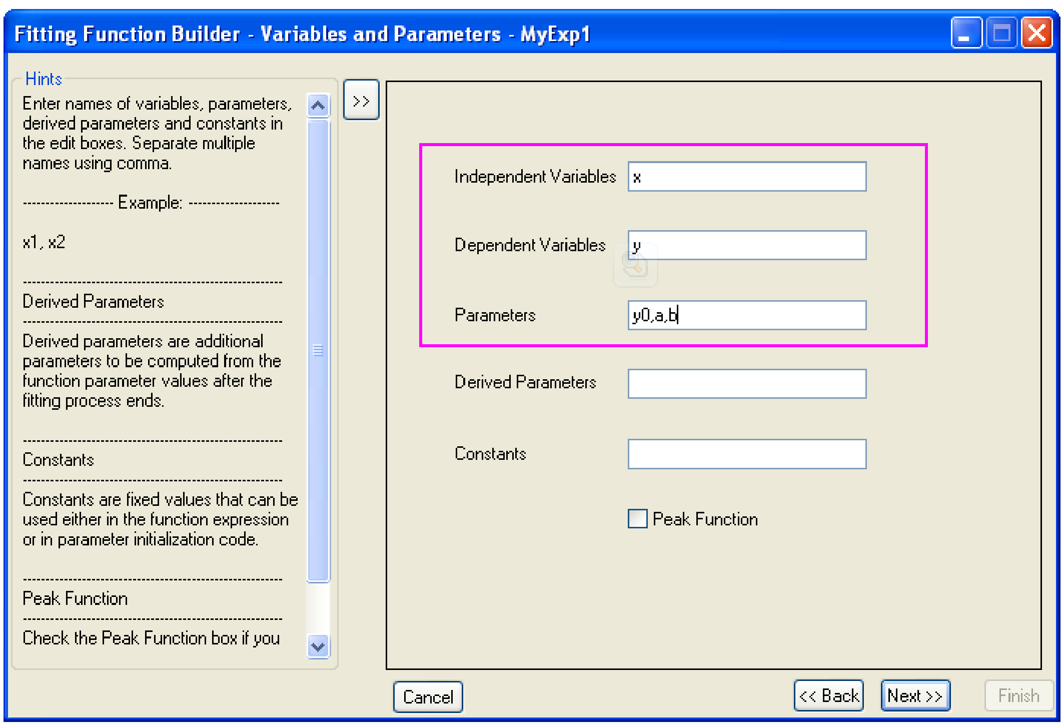
1. Select **Tools: Fitting Function Builder...** from the main menu (or press **F8**) to open the **Fitting Function Builder**. Select the **Create a New Function** radio box.



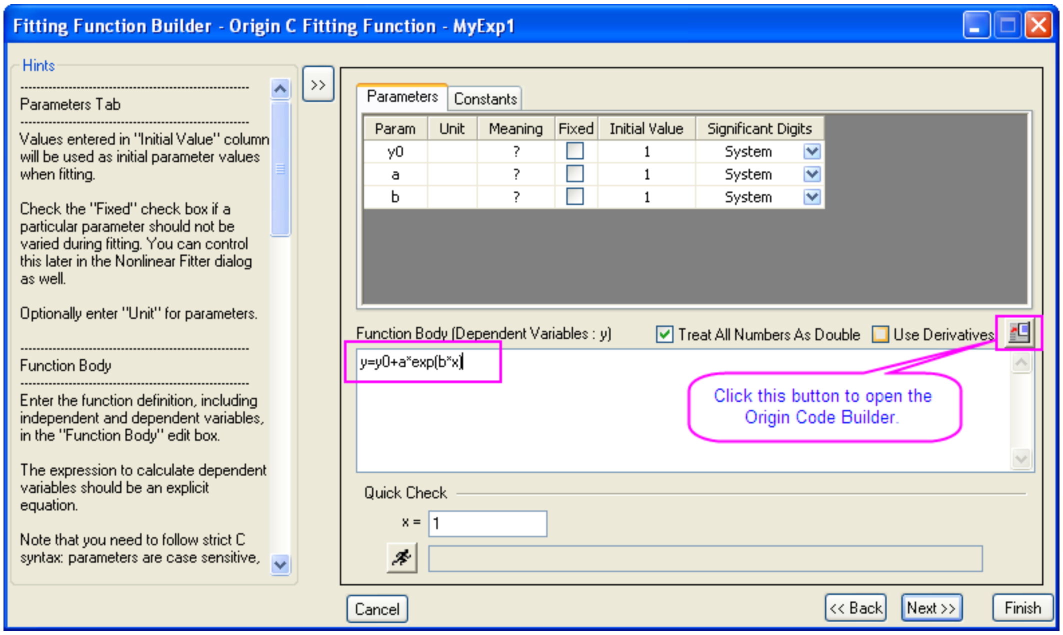
2. Click the **Next** button to go to the **Name and Type** page. Enter *MyExp1* in the **Function Name** box and select the **Origin C** radio box for the **Function Type**.



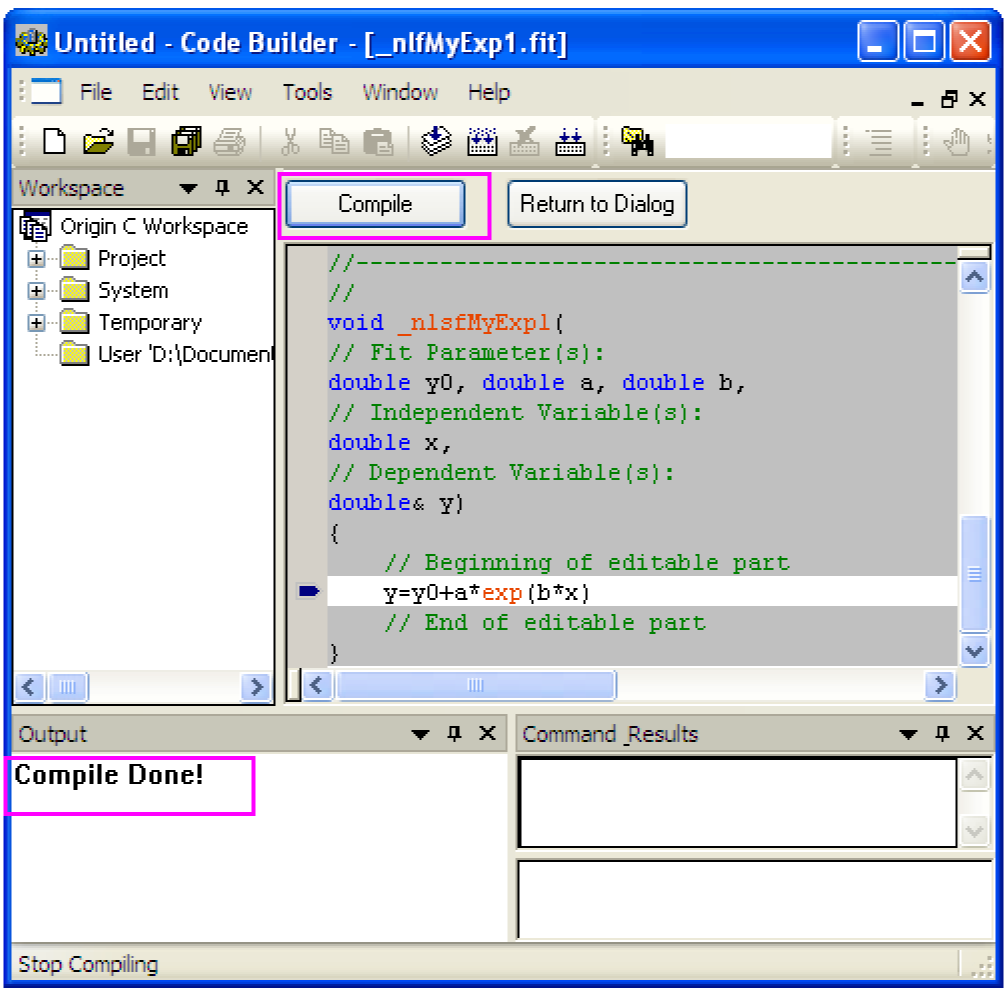
3. Click the **Next** button to go to the **Variables and Parameters** page. Enter the variables and parameters as shown in the following screenshot:



4. Click the **Next** button to go to the **Origin C Fitting Function** page. Enter the function body in the **Function Body** box. To verify the correctness of the function, click the button beside the Function box to open the Origin Code Builder.



In the **Code Builder**, click the **Compile** button to compile the function. If passed, click the **Return to Dialog** button to return to the **Fitting Function Builder**.



5. Click the **Finish** button to finish defining this function.

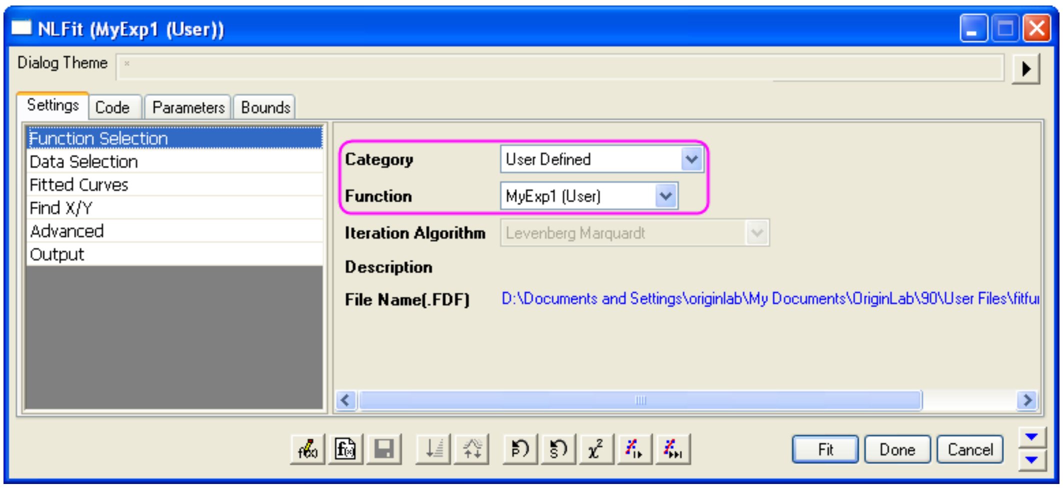
Fit data with the function:

Let's use the above user-defined function to do fitting.

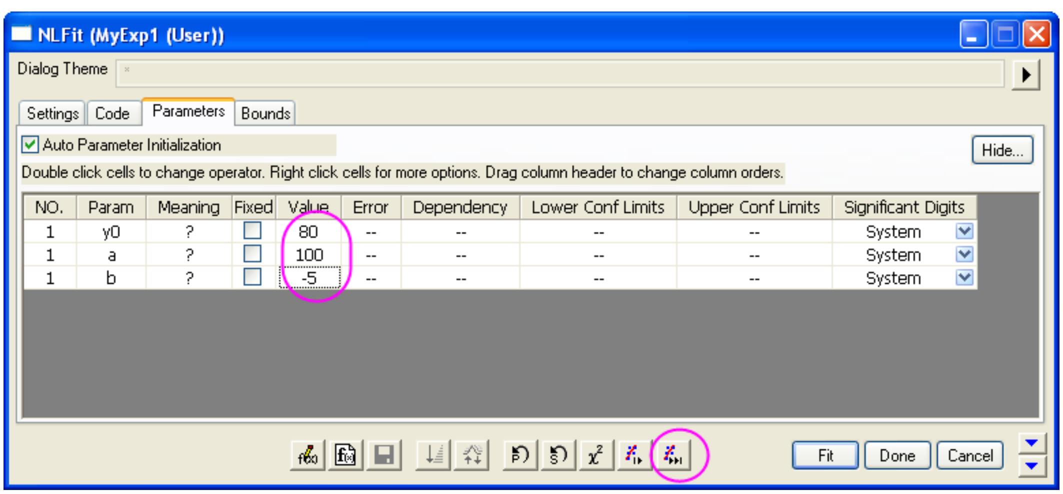
1. Import *\Samples\Curve Fitting\Exponential Decay.dat* to Origin worksheet.

2. Highlight column B and select **Analysis: Fitting: Non-linear Curve Fit** from the menu to bring up the **NLFit** dialog.

3. Select the function just defined in the **Settings** tab, **Function Selection** page:

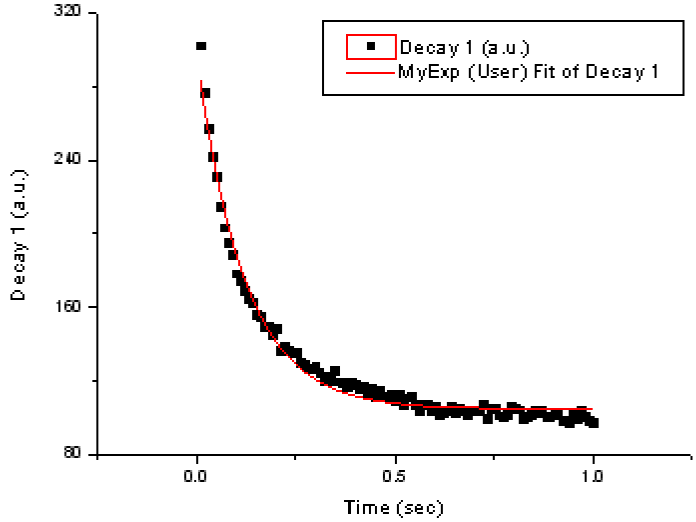


4. Switch to the **Parameters** tab, enter *80*, *100*, *-5* on the **Value** column as initial values for *y0*, *a*, *b*. Click the **Fit until Converge** button to fit the curve:



5. When the fit converges, click **OK** button to generate fitting reports.

From the Fitted Curves Plot, we see the fitting is fine.



And the fitting function is $y = 104.85968 + 193.3244 * exp(- 8.273 * x)$

Parameters

		Value	Standard Error
Decay 1	y0	104.85968	0.69005
	a	193.3244	3.10614
	b	-8.273	0.21726

Reduced Chi-sqr = 24.6340584677
COD(R^2) = 0.98505461857169
Iterations Performed = 4
Total Iterations in Session = 4
Fit converged. Chi-Sqr tolerance value of 1E-9 was reached.

Fitting One Dataset as a Function of Other Datasets

Summary

Sometimes, one may want to perform "Dataset Fitting", that is, the output may be composed of one or several datasets, like:

$Output = A_1 * Dataset_1 + A_2 * Dataset_2$

For example, you may want to analyze a composite spectrum to find the contributions/ratio from individual component spectra. This can be accomplished either by defining multiple independent variables or by calculating the "combination" inside the fitting function.

Minimum Origin Version Required: Origin 8.0 SR6

What you will learn

This tutorial will show you how to:

- Perform "Dataset Fitting"
- Define multiple independent variable fitting function

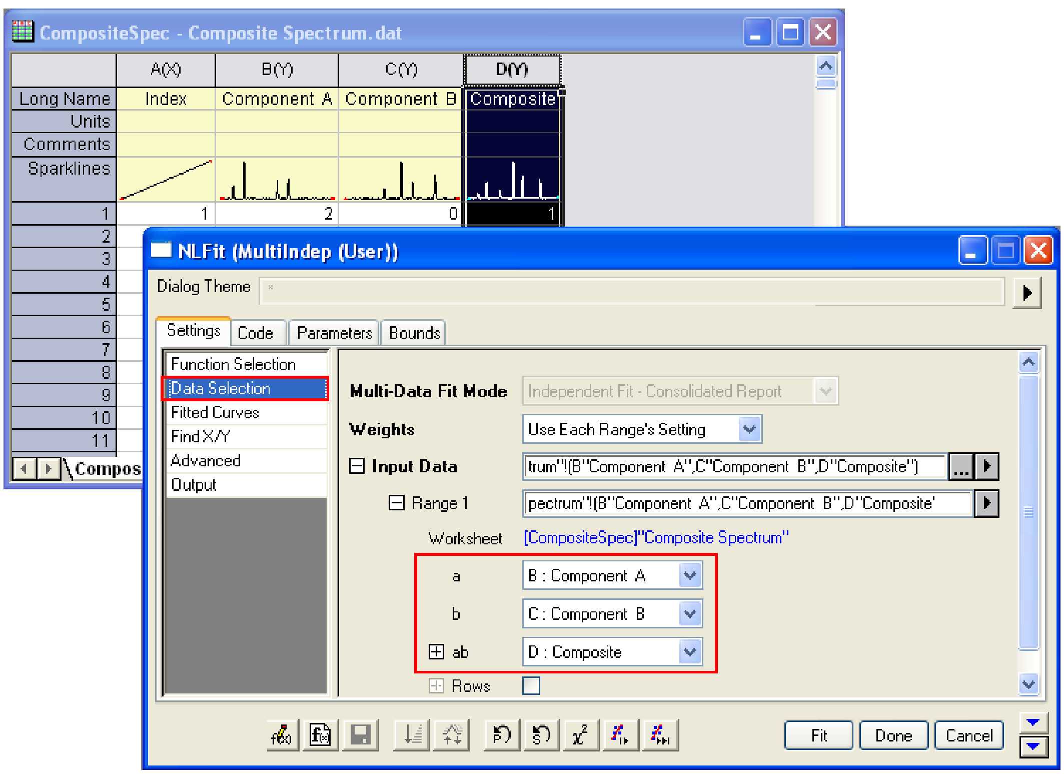
Steps

Import the *Composite Spectrum.dat* file from the *\Samples\Curve Fitting* folder. In this sample data, we can see that column A is the index, columns B and C are the values for the spectrum of components A and B. Column D contains values obtained after reading a composite spectrum of components A and B. By fitting column D to an equation determined by the component spectra of the pure forms of columns B and C, the coefficients for the contributions of B and C (call them c1 and c2 respectively) can be found. (Note: In this case, we supposed the independent and dependent variables have the same size. If not, interpolation is need.)

Bring up the **Fitting Function Organizer** and define a new fitting function as follow:

Function Name:	MultiIndep
Function Type:	User-Defined
Independent Variables:	a, b
Dependent Variables:	ab
Parameter Names:	C1, C2
Function Form:	Origin C
Function:	ab = C1*a + C2*b;

Initialize both C1 and C2 to 1 in the **Parameter Initialization** edit box by entering:
C1=1;
C2=1;
Save the fitting function and close **Fitting Function Organizer**. Highlight **ONLY** Column D and bring up the **NLFit** dialog, specify the input datasets in the **Data Selection** page as follow:



Then you can click the **Fit** button to generate results.

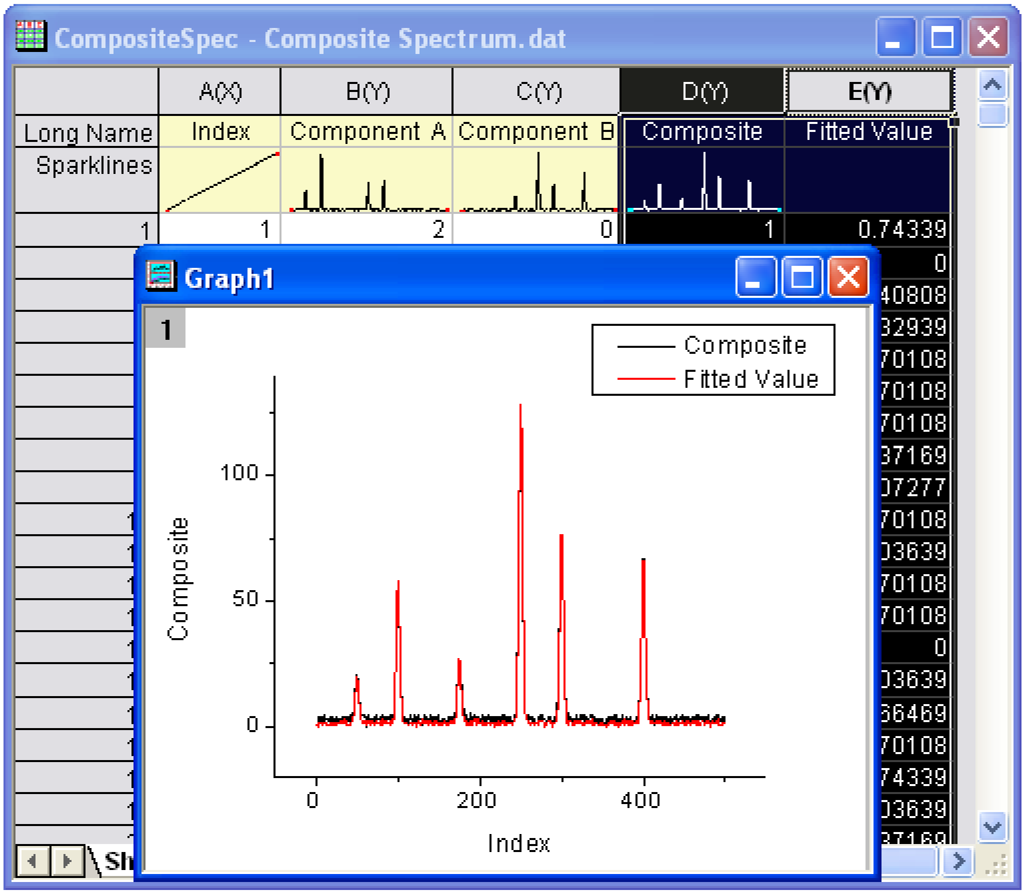
Results

You are supposed to get these results:

	Value	Standard Error

C1	0.37169	0.00483
C2	0.66469	0.0047

To verify the fitted results, you can add a new column and Copy + Paste the fitted value, which comes from the fitted Y in the worksheet *FitNLCurve1*, into it. Then Highlight the *Composite* and the fitted data and plot a line graph to see how good the fit is:



Fitting With Multiple Independent Variables

Summary

The **Function Organizer** tool can be used to create user-defined functions with more than one independent or dependent variable. The **NLFit dialog** can then be used to fit with such functions. The preview window in the fitter dialog is capable of plotting only one quantity versus another, however even if the preview does not *make sense*, the fitting process will correctly proceed once proper data and parameter assignments have been made.

Note that if you wish to fit multiple independent variables with an equation of the type
*y = A0 + A1 * x1 + A2 * x2 + ...*
you can make use of the Multiple Regression tool instead of the nonlinear fitter dialog.

Minimum Origin Version Required: Origin 8.0 SR6

What you will learn

This tutorial will show you how to:

- Create a user-defined fitting function with two independent variables and one dependent variable
- Fit with that function in NLFit

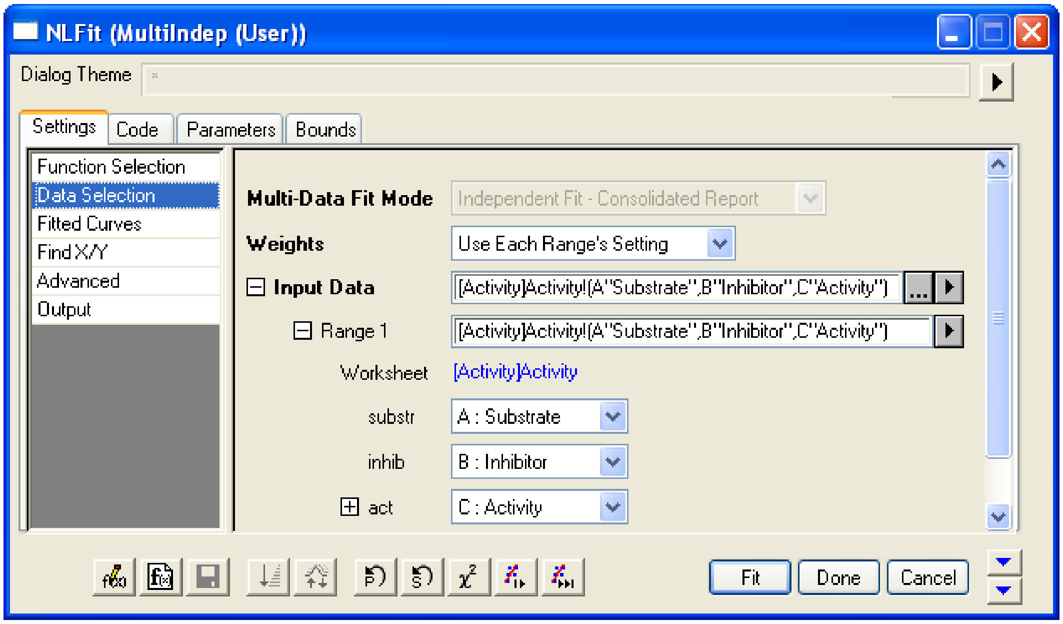
Steps

1. Start with a new workbook and import the file *\Samples\Curve Fitting\Activity.dat*.
2. Select **Tools: Fitting Function Organizer** from menu (or press **F9**) to bring up the **Fitting Function Organizer** and define a new fitting function named **MultiIndep** in **NewCategory** (create the category if not exist) as follow:

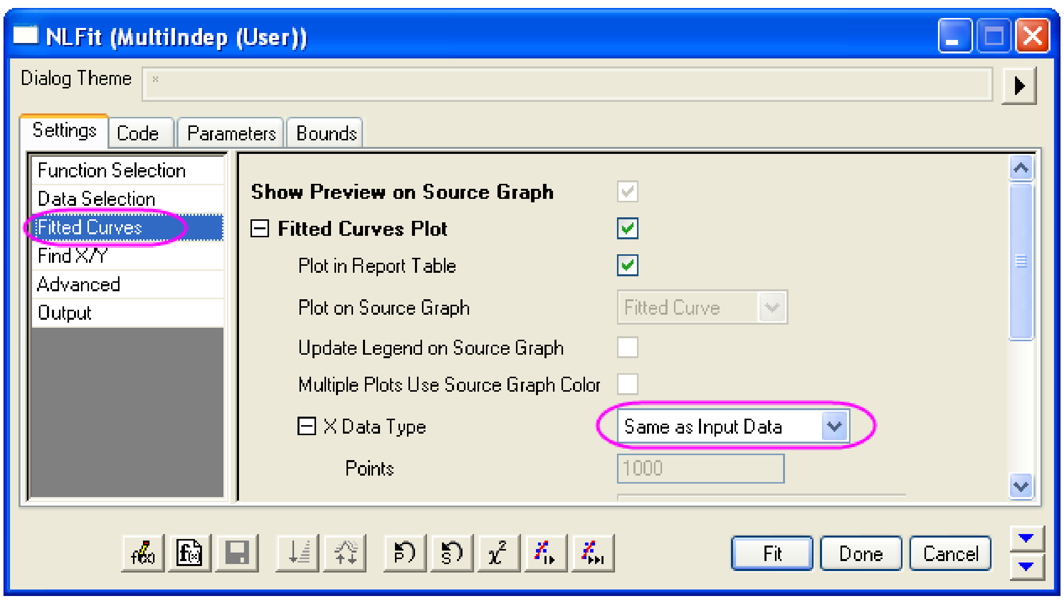
Function Name:	MultiIndep
Function Type:	User-Defined
Independent Variables:	substr,inhib
Dependent Variables:	act
Parameter Names:	ki,km,vm
Function Form:	Origin C
Function:	double mix = inhib / ki; act = vm * substr / (km + (1 + mix) * substr);

3. **NOTE:** Since we are using OriginC, case must match between defined names and their use in the function definition. e.g. Substr does NOT equal substr.
4. Click Save and then OK to save the function and close the Organizer.
5. For more details about User Defined Fitting Function please refer to User Defined Fitting Function using Origin C.
6. Highlight **ONLY** column C and select **Analysis: Fitting: Non-linear Curve Fit** from menu to bring up the NLFit dialog. Select the function **MultiIndep** from **NewCategory** on the **Settings:**

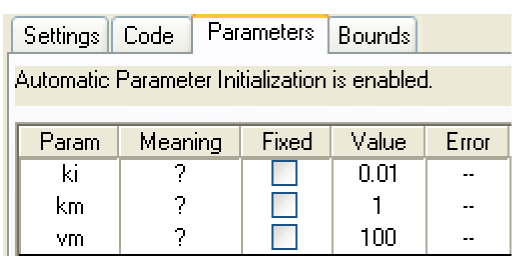
Function Selection page. Set the input datasets in the **Data Selection** page as follow:



7. Select the **Fitted Curves** page and expand the **Fitted Curves Plot** branch. Then select **Sample as Input Data** from the drop-down list next to the **X Data Type** branch.



8. Select **Parameters Tab** and set the initial values as follow:



9. Click **Fit** button to generate the fitting reports. You can see these results from the report worksheet as below:

Parameters

		Value	Standard Error
Activity	ki	0.0373	0.00233
	km	7.30567	0.71748
	vm	653.1116	22.39698

Statistics

	Activity
Number of Points	18
Degrees of Freedom	15
Reduced Chi-Sqr	155.36102
Residual Sum of Squares	2330.41531
Adj. R-Square	0.98357
Fit Status	Succeeded(100)

From the Statistics table we can know that the fitting is fairly successful.

User Defined Fitting Function using GNU Scientific Library

This article demonstrates how to use GSL function as fit function.

Minimum Origin Version Required: Origin 8.0 SR6

1. We will fit the sample Data below by the following model:

$$y = y_0 + a\int_0^x e^{\beta \cdot t}\,dt$$

```
0.1     0.10517
0.2     0.2214
0.3     0.34986
0.4     0.49182
0.5     0.64872
0.6     0.82212
0.7     1.01375
0.8     1.22554
0.9     1.4596
1       1.71828
1.1     2.00417
1.2     2.32012
1.3     2.6693
1.4     3.0552
1.5     3.48169
1.6     3.95303
1.7     4.47395
1.8     5.04965
1.9     5.68589
2       6.38906
2.1     7.16617
2.2     8.02501
2.3     8.97418
2.4     10.02318
2.5     11.18249
```

```
2.6     12.46374
2.7     13.87973
2.8     15.44465
2.9     17.17415
3       19.08554
3.1     21.19795
3.2     23.53253
```

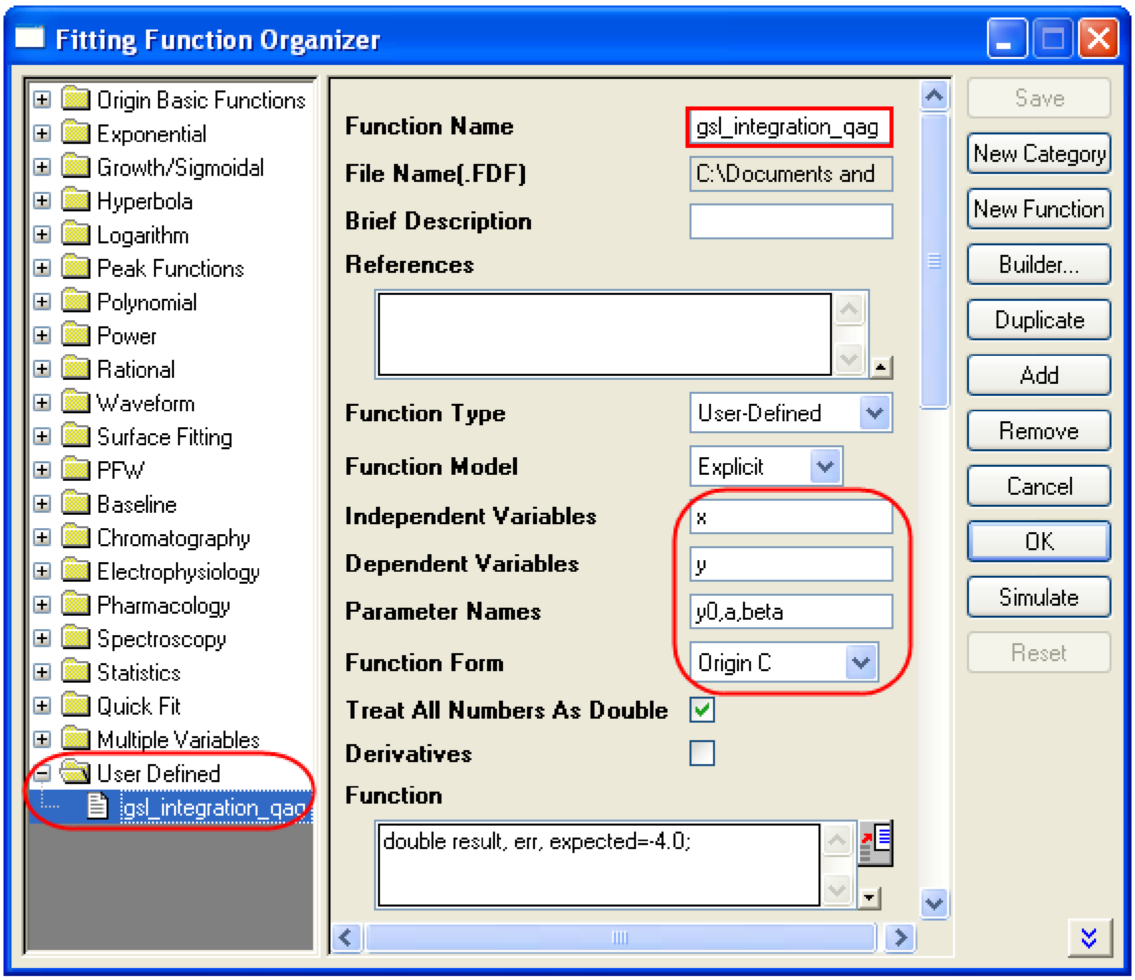
2. Add the file ocgsl.h in **(User Files folder)**, before next step, make sure the gsl dlls are copied to this same location, see Calling GNU Scientific Library.

ocgsl.h

```
#pragma dll(libgsl, header)
// this is OC special pragma,
// header keyword is to indicate libgsl.dll is in same location as this
file

#define GSL_EXPORT     // for OC, this is not needed, so make it empty

// you can directly search and copy gsl function prototypes here

typedef double (* FUNC)(double x, void * params);

struct gsl_function_struct
{
        FUNC function;
        void * params;
};

typedef struct gsl_function_struct gsl_function ;

typedef struct
{
    size_t limit;
    size_t size;
    size_t nrmax;
    size_t i;
    size_t maximum_level;
    double *alist;
    double *blist;
    double *rlist;
    double *elist;
    size_t *order;
    size_t *level;
}
gsl_integration_workspace;

GSL_EXPORT gsl_integration_workspace *gsl_integration_workspace_alloc
(const size_t n);

GSL_EXPORT void gsl_integration_workspace_free
(gsl_integration_workspace * w);

GSL_EXPORT int gsl_integration_qag (const gsl_function * f,
                                    double a, double b,
```

```
                                double epsabs, double epsrel,
size_t limit,
                                int key,
                                gsl_integration_workspace *
workspace,
                                double *result, double *abserr);
```

3. Press F9 to open the **Fitting Function Organizer** and then add a new function as follows:



4. Press the button on the right hand side of the **Function** Field to open the code builder and add the following codes and compile: **_nlfgsl_integration_qag.fit**

```
#include "..\ocgsl.h"

static double f_callback(double x, void * params)
{
        double alpha = *(double *)params;
        return exp(alpha*x);
}

void _nlsfgsl_integration_qag(
// Fit Parameter(s):
double y0, double a, double beta,
// Independent Variable(s):
```

```
double x,
// Dependent Variable(s):
double& y)
{
        // Beginning of editable part
        double result, err, expected = -4.0;

        // Allocates a workspace suffcient to hold 1000 double precision
intervals,
        // their integration results and error estimates
        gsl_integration_workspace *ww =
gsl_integration_workspace_alloc(1000);

        gsl_function F;
        F.function = f_callback;
        F.params = &beta ;

        // integral interval (0, x), within the desired absolute
        // error 0 and relative error 1e-7
        gsl_integration_qag(&F, 0, x, 0, 1e-7, 1000, 0, ww, &result,
&err);

        // frees the memory associated with the workspace w
        gsl_integration_workspace_free (ww);

        y = y0 + a*result;

        // End of editable part
}
```

Furthermore, a more elaborate but efficient version of the fitting function is given as follows

```
//------------------------------------------------------------
//
#include <ONLSF.h>
#include "..\ocgsl.h"

static double f_callback(double x, void * params)
{
        double alpha = *(double *)params;
        return exp(alpha*x);
}

void _nlsfgsl_integration_qag(
// Fit Parameter(s):
double y0, double a, double beta,
// Independent Variable(s):
double x,
// Dependent Variable(s):
double& y)
{
        // Beginning of editable part

        NLFitContext *pCtxt = Project.GetNLFitContext();
        if ( pCtxt )
        {
```

```
                static vector vInteg;
                NLSFCURRINFO    stCurrInfo;
                pCtxt->GetFitCurrInfo(&stCurrInfo);
                int nCurrentIndex = stCurrInfo.nCurrDataIndex;

                BOOL bIsNewParamValues = pCtxt->IsNewParamValues();
                if ( bIsNewParamValues )
                {
                        vector vx;
                        pCtxt->GetIndepData(&vx);
                        int nSize = vx.GetSize();
                        vInteg.SetSize(nSize);

                        // Allocates a workspace suffcient to hold 1000 
double precision intervals,
                        // their integration results and error estimates
                        gsl_integration_workspace *ww = 
gsl_integration_workspace_alloc(1000);

                        gsl_function F;
                        F.function = f_callback;
                        F.params = &beta ;

                        double result, err, expected = -4.0;
                        for(int ii=0; ii<nSize; ++ii)
                        {
                                // integral interval (0, vx[ii]), within 
the desired absolute
                                // error 0 and relative error 1e-7
                                gsl_integration_qag(&F, 0, vx[ii], 0, 1e-
7, 1000, 0, ww, &result, &err);
                                vInteg[ii] = result;
                        }

                        // frees the memory associated with the workspace 
w
                        gsl_integration_workspace_free (ww);

                }

                y = y0 + a*vInteg[nCurrentIndex];
                x;
        }

        // End of editable part
}
```

5. Add the following initilization codes:

Parameter Init

```
//Code to be executed to initialize parameters
sort( x_y_curve );
double coeff[2];
fitpoly( x_y_curve, 1, coeff);
a = coeff[0];
```

```
y0 = coeff[1];
beta=1.0
```

6. Fit using the user-defined function **gsl_integration_qag**, here are the results:

y0 = -1.06363E-6

a = 1

beta =1

Fitting with NAG Special Function

Summary

Origin allows user to define an Origin C fitting function using NAG special functions. You can call NAG routine to evaluate the special function.

Minimum Origin Version Required: Origin 8.0 SR6

What you will learn

This tutorial will show you how to:

- Create fitting function using Fitting Function Organizer
- Create fitting function using NAG special function

Example and Steps

We will fit the following model:

$$inorm = A*exp(-td/2.0/(t-t0))*(I0(td/2.0/(t-t0))+I1(td/2.0/(t-t0)))$$

Here *A*, *td* and *t*0 are the model parameters we want to obtain from the data fitting. *I*0 and *I*1 are the first kind of Modified Bessel function of order 0 and order 1, respectively. For current example, we use the sample data in the end of this tutorial. The fitting procedure can be outlined into the following steps:

Press **F9** to open the **Fitting Function Organizer** and then create a new Category named ***FittingWithNAGSpecialFunc***. Define a new fitting function ***FittingWithBessel*** in the new category as follow:

Function Name:	FittingWithBessel
Function Type:	User-Defined
Independent Variables:	t
Dependent Variables:	inorm
Parameter Names:	A,t0,td
Function Form:	Origin C
Function:	

Click the button (icon) beside the **Function** box to open the code builder and define and compile and save the fitting function as follows:

```
#include <origin.h>
```

```
// Add your special include files here.
// For example, if you want to fit with functions from the NAG library,
// add the header file for the NAG functions here.
#include <OC_nag8.h>

// Add code here for other Origin C functions that you want to define
in this file,
// and access in your fitting function.

// You can access C functions defined in other files, if those files
are loaded and compiled
// in your workspace, and the functions have been prototyped in a
header file that you have
// included above.

// You can access NLSF object methods and properties directly in your
function code.

// You should follow C-language syntax in defining your function.
// For instance, if your parameter name is P1, you cannot use p1 in
your function code.
// When using fractions, remember that integer division such as 1/2 is
equal to 0, and not 0.5
// Use 0.5 or 1/2.0 to get the correct value.

// For more information and examples, please refer to the "User-Defined
Fitting Function"
// section of the Origin Help file.


//----------------------------------------------------------
//
void _nlsfFittingWithBessel(
// Fit Parameter(s):
double A, double t0, double td,
// Independent Variable(s):
double t,
// Dependent Variable(s):
double& inorm)
{
	// Beginning of editable part
	//inorm=  A* exp(-td/2.0/(t-t0)) *   ( s18aec(td/2.0/(t-
t0),NAGERR_DEFAULT)+s18afc(td/2.0/(t-t0),NAGERR_DEFAULT)           );

	static NagError fail1;
	static NagError fail2;
	double dtemp = td/2.0/(t-t0);
	inorm=  A* exp(-dtemp) * (
s18aec(dtemp,&fail1)+s18afc(dtemp,&fail2) );
	if(fail1.code !=NE_NOERROR)
		printf("%s\n",fail1.message);
	if(fail2.code !=NE_NOERROR)
		printf("%s\n",fail2.message);


	// End of editable part
}
```

Simulate the Function

After the function body is defined, you can click the **Compile** button in **Code Builder** to check syntax errors. And then click **Return to Dialog** button to go back **Fitting Function Organizer** dialog box. Now click the **Save** button to generate the .FDF file (Function definition file).

Once you have a .FDF file, you can click the **Simulate** button to simulate a curve, this will be very helpful to evaluate the initial values. In the **simcurve** dialog, enter some proper parameter values and X range, and see what the curve looks like in the **Preview** panel.

Set the Initial Values for the Parameters

As it is a user-defined fitting function, you have to supply the initial guess values for the parameters before performing your fitting task for the data. You may do it by set them manually in the **Parameter** tab in **Nonlinear Curve Fit** dialog. For the sample data shown below, you can just set the initial values for the parameters A = 1, td = 1, t0 = 1. After the parameters are initialized, you can then do the fitting to obtain the fitting result, as shown to the right of the sample data.

Sample Data

Copy the below sample data and use **Import Wizard** to import the data from **Clipboard**, then do the fitting using the given initial values for the parameters: A = 1, td = 1, t0 = 1.

Sample Data

X	Y
2	0.7868954118
2.080808081	0.8133022141
2.161616162	0.8178216765
2.242424242	0.8427866729
2.323232323	0.8315815363
2.404040404	0.8484657180
2.565656566	0.8618233553
2.646464646	0.8745962570
2.727272727	0.8921620316
2.808080808	0.8687399759

Results

Parameters

		Value	Standard Error
"Y"	A	0.96431	0.06562
	t0	1.39545	0.40134
	td	0.53711	0.54076

Reduced Chi-sqr = 1.02442755048E-4
COD(R^2) = 0.92247024814828
Iterations Performed = 11
Total Iterations in Session = 11
Fit converged - tolerance criterion satisfied.

Fitting Integral Function with parametric limit using NAG Library

Summary

Before you start delving into this tutorial, you are recommended to read the relevant tutorial in Fitting with Integral using NAG Library. And as far as programming is concerned, the two tutorials are basically the same, except that here you will learn to define Origin C fitting function with fitting parameters in the integral limit, while in the previous tutorial we in fact define a fitting independent variable in the integral limit. Also note that **a different NAG integrator** is used here.

Minimum Origin Version Required: Origin 8.0 SR6

What you will learn

This tutorial will show you how to:

- Create a fitting function with Definite Integral using the NAG integration routine
- Create a fitting function with a parametric integral limit
- Use a log function to scale a large return value from the fitting function

Example and Steps

For example, we will fit the sample data at the bottom of this page with the following model:

$$y = \int_c^d \frac{\cosh\left((x_i + b^2 \cdot x^2)/(b + x)\right)}{a + (x_i^2 + x^2)}\, dx_i$$

Note that we use x_i to indicate the integral independent variable while x indicates the fitting independent variable. The model parameters *a*, *b*, *c*, and *d* are fitted parameters we want to obtain from the sample data. To prepare the data, you just need to copy the sample data to an Origin Work sheet. The fitting procedure is similar to the previous tutorial:

Define Fitting Function in Fitting Function Organizer

Press **F9** to open the Fitting Function Organizer and add the User-Defined integral fitting function ***nag_integration_fitting_cosh*** to the Category ***FittingWithIntegral***, similar to the first tutorial.

Function Name:	nag_integration_fitting_cosh
Function Type:	User-Defined
Independent Variables:	x
Dependent Variables:	y
Parameter Names:	a, b, c, d
Function Form:	Origin C
Function:	

Click the button (icon) beside the **Function** box to open the code builder and define and compile the fitting function as follows: (Note: Remember to save the Function after compiling it and returning to the Function Organizer Dialog):

```
#include <origin.h>
// Add your special include files here.
// For example, if you want to fit with functions from the NAG
library,
// add the header file for the NAG functions here.
#include <oc_nag8.h>


// Add code here for other Origin C functions that you want to
define in this file,
// and access in your fitting function.
struct user
{
  double a, b, fitX;  // fitX the independent variable of
fitting function
```

```
};
static double NAG_CALL f_callback(double x, Nag_User *comm)
// x is the independent variable of the integrand
{

  struct user *sp = (struct user *)(comm->p);

        double aa, bb, fitX; // temp variable to accept the
parameters in the Nag_User communication struct
        aa = sp->a;
        bb = sp->b;
        fitX = sp->fitX;

        return
cosh((x*x+bb*bb*fitX*fitX)/(bb+fitX))/(aa+(x*x+fitX*fitX));
}

// You can access C functions defined in other files, if those
files are loaded and compiled
// in your workspace, and the functions have been prototyped
in a header file that you have
// included above.

// You can access NLSF object methods and properties directly
in your function code.

// You should follow C-language syntax in defining your
function.
// For instance, if your parameter name is P1, you cannot use
p1 in your function code.
// When using fractions, remember that integer division such
as 1/2 is equal to 0, and not 0.5
// Use 0.5 or 1/2.0 to get the correct value.

// For more information and examples, please refer to the
"User-Defined Fitting Function"
// section of the Origin Help file.

//---------------------------------------------------------
//
void _nlsfnag_integration_fitting_cosh(
// Fit Parameter(s):
double a, double b, double c, double d,
// Independent Variable(s):
double x,
// Dependent Variable(s):
double& y)
{
  // Beginning of editable part
  double epsabs = 0.00001, epsrel = 0.0000001, result, abserr;
  Integer max_num_subint = 500;
        // you may use epsabs and epsrel and this quantity to
enhance your desired precision
        // when not enough precision encountered

  Nag_QuadProgress qp;
```

```
  static NagError fail;

  // the parameters parameterize the integrand can be input to
the call_back function
        // through the Nag_User communication struct
        Nag_User comm;
  struct user s;
  s.a = a;
  s.b = b;
  s.fitX = x;
        comm.p = (Pointer)&s;

  d01sjc(f_callback, c, d, epsabs, epsrel, max_num_subint,
&result, &abserr, &qp, &comm, &fail);


        // you may want to exam the error by printing out
error message, just uncomment the following lines
  // if (fail.code != NE_NOERROR)
        // printf("%s\n", fail.message);


  // For the error other than the following three errors which
are due to bad input parameters
  // or allocation failure  NE_INT_ARG_LT  NE_BAD_PARAM
NE_ALLOC_FAIL
  // You will need to free the memory allocation before calling
the integration routine again to
        // avoid memory leakage
  if (fail.code != NE_INT_ARG_LT && fail.code != NE_BAD_PARAM
&& fail.code != NE_ALLOC_FAIL)
  {
         NAG_FREE(qp.sub_int_beg_pts);
         NAG_FREE(qp.sub_int_end_pts);
         NAG_FREE(qp.sub_int_result);
         NAG_FREE(qp.sub_int_error);
  }


  y = log(result);
        // note use log of the integral result as return as
the integral result is large,
        // you are not necessary to do so

  // End of editable part
}
```

In the above code, we define the integrand as a callback function ***f_callback*** just outside the fitting function body ***_nlsfnag_integration_fitting_cosh***. Note that we parametrize the integrand function with the variables ***a***, ***b*** and ***fitX***, and pass them into the callback funtion through the *Nag_User* struct. After that we perform the integration using NAG integrator ***d01sjc***. Besides, you can also use other Quadrature Routines as you want. In the current example, we also use a log scale for the fitting function. (The sample data are already scaled by a log function)

Compile the code, return to the dialog and then Save the fitting function in the function Organizer and open the **Nonlinear Curve Fit** dialog in the **Analysis**-**Fitting** menu. You can then select this user-defined fitting function in the **Function Selection** page under **Setting** Tab.

Set the Initial Values for the Parameters

Similarly, as it is a user-defined fitting function, you have to supply the initial guess values for the parameters. You may manually set them in the **Parameter** tab in **Nonlinear Curve Fit** dialog. For current example, you can just set the initial values for the parameters *a* = 1, *b* = 10, *c* = 3, *d* = 4. After the parameters are initialized, you can perform the fitting to obtain the fitting result, as shown in the following.

Sample Data

X	Y
-5	498.19046
-4.33333	329.43196
-3.66667	210.28005
-3	126.55799
-2.33333	69.01544
-1.66667	31.3555
-1	9.1393
-0.33333	-0.84496
0.33333	-0.99914
1	6.86736

Results:

	Value	Standard Error
a	0.99303	0.06577
b	10	5.3108E-5
c	3.00083	0.0062
d	4.00022	9.38713E-4

Fitting with Integral using NAG Library

Contents

Summary

Origin allows user to define an Origin C fitting function which involves an integral. You can call NAG functions to perform the integration while defining the fitting function. There are built-in functions in Origin C which perform integration. For the current example, the NAG solution is recommended. It has a better performance compared to the built-in integration algorithm. Note that an **infinite NAG integrator** is used here.

Minimum Origin Version Required: Origin 8.0 SR6

What you will learn

This tutorial will show you how to:

- Create a fitting function using the Fitting Function Organizer
- Create a fitting function with a Definite Integral using a NAG integration routine
- Set up the Initial Code for the fitting function

Example and Steps

We will fit the following model:

$$y = y_0 + \int_{-\infty}^{x} \frac{A}{w\sqrt{\frac{\pi}{2}}} e^{-2\frac{(t-x_c)^2}{w^2}}, dt$$

Here y_0 A, xc and w are the model parameters we want to obtain from the data fitting. The fitting procedure can be outlined into the following steps:

Define the Function

Press **F9** to open the **Fitting Function Organizer** and then create a new Category named ***FittingWithIntegral***. Define a new fitting function ***nag_integration_fitting*** in the new category as follow:

Function Name:	nag_integration_fitting
Function Type:	User-Defined
Independent Variables:	x
Dependent Variables:	y
Parameter Names:	y0, A, xc, w
Function Form:	Origin C
Function:	

Click the button (icon) beside the **Function** box to open the code builder and define and compile and save the fitting function as follows:

```
#include <origin.h>
// Add your special include files here.
// For example, if you want to fit with functions from the NAG library,
// add the header file for the NAG functions here.
#include <oc_nag8.h>


// Add code here for other Origin C functions that you want to define
in this file,
// and access in your fitting function.

struct user   // parameters in the integrand
{
        double amp, center, width;

};
// Function supplied by user, return the value of the integrand at a
given x.
static double NAG_CALL f_callback(double x, Nag_User *comm)
```

```
{
        struct user *sp = (struct user *)(comm->p);

        double amp, center, width;    // temp variable to accept the
parameters in the Nag_User communication struct
        amp = sp->amp;
        center = sp->center;
        width = sp->width;

        return amp * exp( -2*(x - center)*(x - center)/width/width ) /
(width*sqrt(PI/2));
}


// You can access C functions defined in other files, if those files
are loaded and compiled
// in your workspace, and the functions have been prototyped in a
header file that you have
// included above.

// You can access NLSF object methods and properties directly in your
function code.

// You should follow C-language syntax in defining your function.
// For instance, if your parameter name is P1, you cannot use p1 in
your function code.
// When using fractions, remember that integer division such as 1/2 is
equal to 0, and not 0.5
// Use 0.5 or 1/2.0 to get the correct value.

// For more information and examples, please refer to the "User-Defined
Fitting Function"
// section of the Origin Help file.


//----------------------------------------------------------
//
void _nlsfnag_integration_fitting(
// Fit Parameter(s):
double y0, double A, double xc, double w,
// Independent Variable(s):
double x,
// Dependent Variable(s):
double& y)
{
        // Beginning of editable part

        // Through the absolute accuracy epsabs, relative accuracy
epsrel and max_num_subint you can
        // control the precision of the integration you need
        // if epsrel is set negative, the absolute accuracy will be
used.
        // Similarly, you can control only relative accuracy by set the
epsabs negative
        double epsabs = 0.0, epsrel = 0.0001;
```

```
        // The max number of sub-intervals needed to evaluate the
function in the integral
        // The more diffcult the integrand the larger max_num_subint
should be
        // For most problems 200 to 500 is adequate and recommmended
        Integer max_num_subint = 200;

        // Result keeps the approximate integral value returned by the
algorithm
        // abserr is an estimate of the error which should be an upper
bound for the |I - result|
        // where I is the integral value
        double result, abserr;

        // The structure of type Nag_QuadProgress,
        // it contains pointers allocated memory internally with
max_num_subint elements
        Nag_QuadProgress qp;

        // The NAG error parameter (structure)
        static NagError fail;

        // Parameters passed to integrand by Nag_User communication
struct
        Nag_User comm;
        struct user s;
        s.amp = A;
        s.center = xc;
        s.width = w;
        comm.p = (Pointer)&s;

        // Perform integration
        // There are 3 kinds of infinite boundary types you can use in
Nag infinite integrator
        // Nag_LowerSemiInfinite, Nag_UpperSemiInfinite, Nag_Infinite
        d01smc(f_callback, Nag_LowerSemiInfinite, x, epsabs, epsrel,
max_num_subint, &result, &abserr, &qp, &comm, &fail);

        // you may want to exam the error by printing out error
message, just uncomment the following lines
        // if (fail.code != NE_NOERROR)
        // printf("%s\n", fail.message);

        // For the error other than the following three errors which are
due to bad input parameters
        // or allocation failure  NE_INT_ARG_LT  NE_BAD_PARAM
NE_ALLOC_FAIL
        // You will need to free the memory allocation before calling
the integration routine again to avoid memory leakage
        if (fail.code != NE_INT_ARG_LT && fail.code != NE_BAD_PARAM &&
fail.code != NE_ALLOC_FAIL)
        {
                NAG_FREE(qp.sub_int_beg_pts);
                NAG_FREE(qp.sub_int_end_pts);
                NAG_FREE(qp.sub_int_result);
                NAG_FREE(qp.sub_int_error);
        }
```

```
        // Calculate the fitted value
        y = y0 + result;

        // End of editable part
}
```

In the above code, we firstly define the integrand as a callback function ***f_callback*** just outside the fitting function body ***_nlsfnag_integration_fitting***. Note that we parametrize the integrand function with the variables ***amp***, ***center*** and ***width***, and pass them into the callback funtion through the *Nag_User* struct. Inside the fitting function, we perform the integration using NAG integrator ***d01smc***.

Calling NAG functions should be more efficient than writing your own routines. Using an analogous method, you can perform finite, infinite, one-dimension and multi-dimension quadrature in your fitting function. Please read the NAG Quadrature page and select a proper routine.

Simulate the Function

After entering the function body codes, you can click the **Compile** button in **Code Builder** to check syntax errors. And then click **Return to Dialog** button to go back **Fitting Function Organizer** dialog box. Now click the **Save** button to generate the .FDF file (Function definition file).

Once you have a .FDF file, you can click the **Simulate** button to simulate a curve, this will be very helpful to evaluate the initial values. In the **simcurve** dialog, enter some proper parameter values and X range, and see what the curve looks like in the **Preview** panel.

Fit the Curve

Before you start to fit the curve, it is very helpful to simulate the function first. Performing integration may take some time, if there is any mistake, you may see Origin "freeze" after you click the **Fit** button. So in the **Fitting Function Organizer** dialog, select the function we defined and click the **Simulate** button. This will bring up the **simcurve** X-Function. Enter some "guess" values and click the **Apply** button. If the simulated curve looks like your source data, you can go further to fit.

To test the fitting function, import *\Samples\Curve Fitting\Replicate Response Data.dat* to Origin. Set *Col*(*A*) = *log*(*Col*(*A*)) in the **Set Column Values** dialog. This will make a sigmoid curve. Highlight column A and B and create a scatter plot. Then bring up the **NLFit** dialog from **Analysis: Fitting: Nonlinear Curve Fit** menu item. Select the fitting function we just defined and go to the **Parameters** tab, initialize all parameters by 1 and fit. You are supposed to see these results:

	Value	Standard Error
y0	-0.00806	0.18319
A	3.16479	0.39624
xc	-0.19393	0.10108
w	1.77252	0.33878

Fitting with Integral using LabTalk Function

Contents

Summary

Since version **Origin 8.6**, Origin introduces a new LabTalk function, ***integral()***, to do one-dimensional integration. This function returns the integral value of:

$$\int_{Lower\,Limit}^{Upper\,Limit} f(t, arg1, arg2, ...), dt$$

And the interface of the *integral()* function is defined as below:

```
integral(integrandName, LowerLimit, UpperLimit [, arg1, arg2,
...])
```

Where *integrandName* here is the function name of the integrand:

$$f(t, arg1, arg2, ...)$$

In other word, the *integral()* function do the following things:

- Accept another function (the first argument) as the integrand.
- Perform integration on specified lower and upper limit and return the integral value.
- If need, pass the subsequent arguments (*Arg1*, *Arg2*, ...) into the integrand function.

Using this feature, we can define a fitting function with *integral()* function, and pass proper fitting parameters to the integrand, to do integration in curve fitting.

In this tutorial, we will modify the other tutorial, calling NAG functions to do integration during fitting, into LabTalk form, and show you how simple and straightforward to fit an integration function.

Minimum Origin Version Required: Origin 8.6

What you will learn

This tutorial will show you how to:

- Create a fitting function using the Fitting Function Builder
- Create a fitting function with a Definite Integral using Labtalk function
- Set up the Initial Code for the fitting function

Example and Steps

The Fitting Model

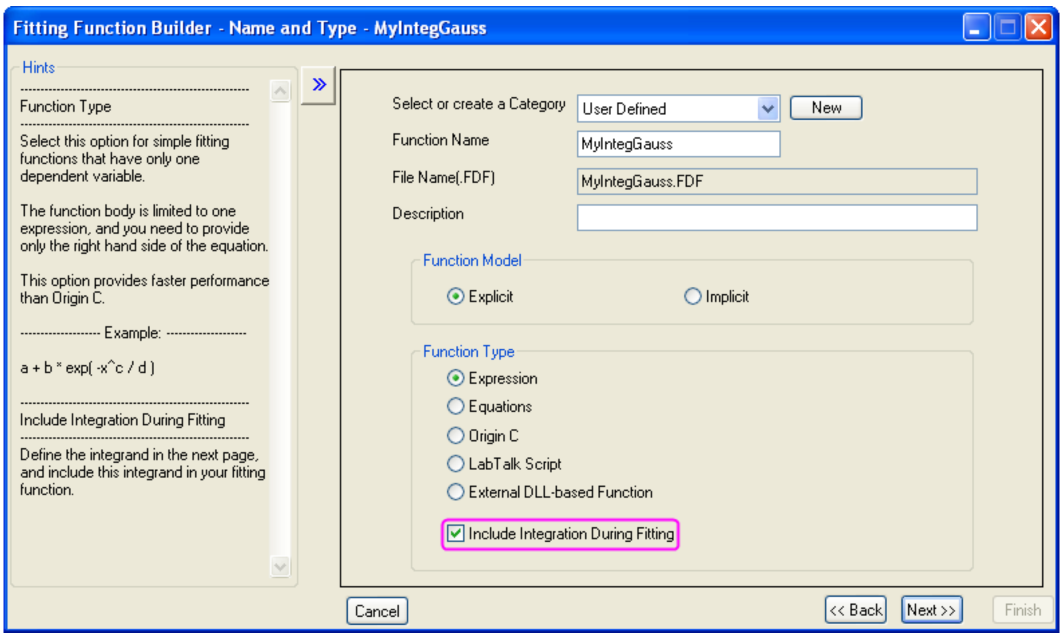
The fitting model is described as below:

$$y = y_0 + \int_{-\infty}^{x} \frac{A}{w\sqrt{\frac{\pi}{2}}} e^{-2\frac{(t-x_c)^2}{w^2}}, dt$$

There are four parameters in the fitting function, and we need to pass three of them into the integrand, and use the independent variable as upper limit, to do integration. So you should define the integrand first, and then use the *integral()* function to perform integration inside your fitting function body.

Define the Function

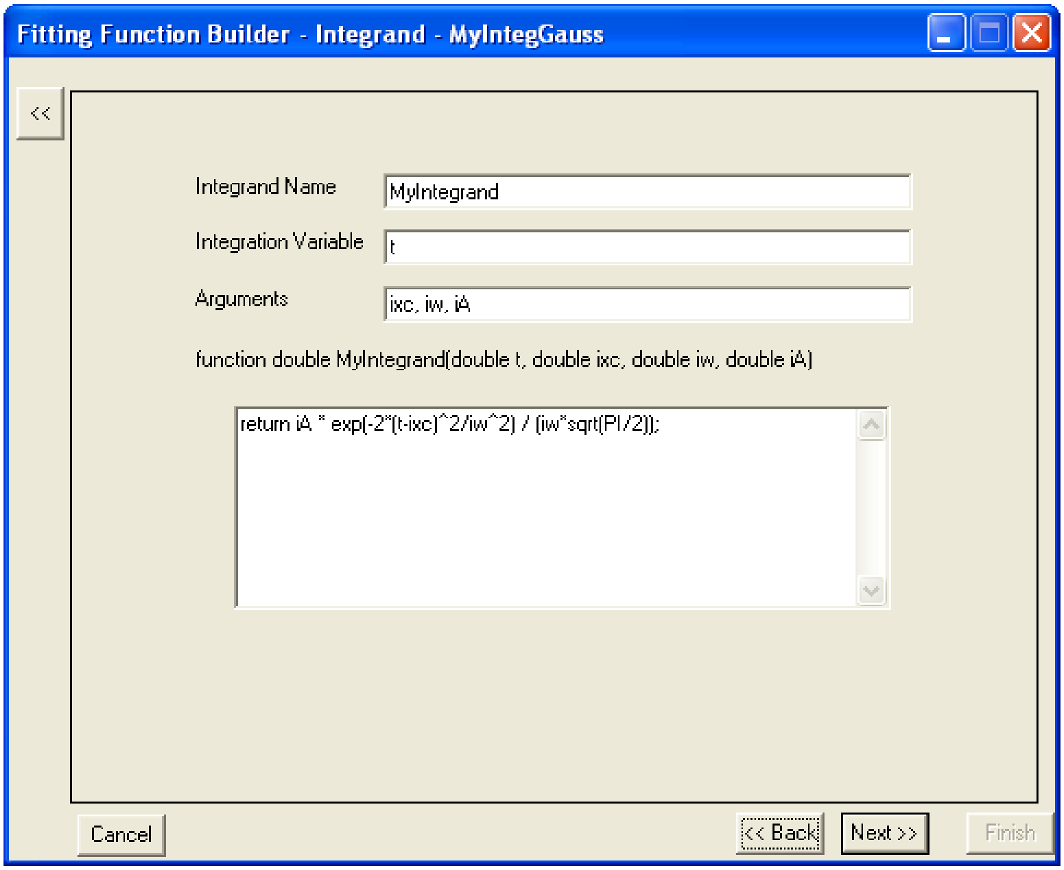
1. Press **F8** to open ***Fitting Function Builder*** dialog. Make sure choosing ***Create a New Function*** option, and click ***Next*** to navigate to the next page.
2. In the ***Name and Type*** page, enter a function name, say ***MyIntegGauss***. Leave the default function type as ***Expression***, and then check the ***Include Integration During Fitting*** checkbox. This will lead you to a new page in the next step.



3. In the ***Integrand*** page, you can define the expression of the integrand. Currently, Origin supports one-dimensional integral only, so the integrand should have ONE integration variable. In this example, the expression of the integrand is:

$$\frac{A}{w\sqrt{\frac{\pi}{2}}}e^{-2\frac{(t-x_c)^2}{w^2}}$$

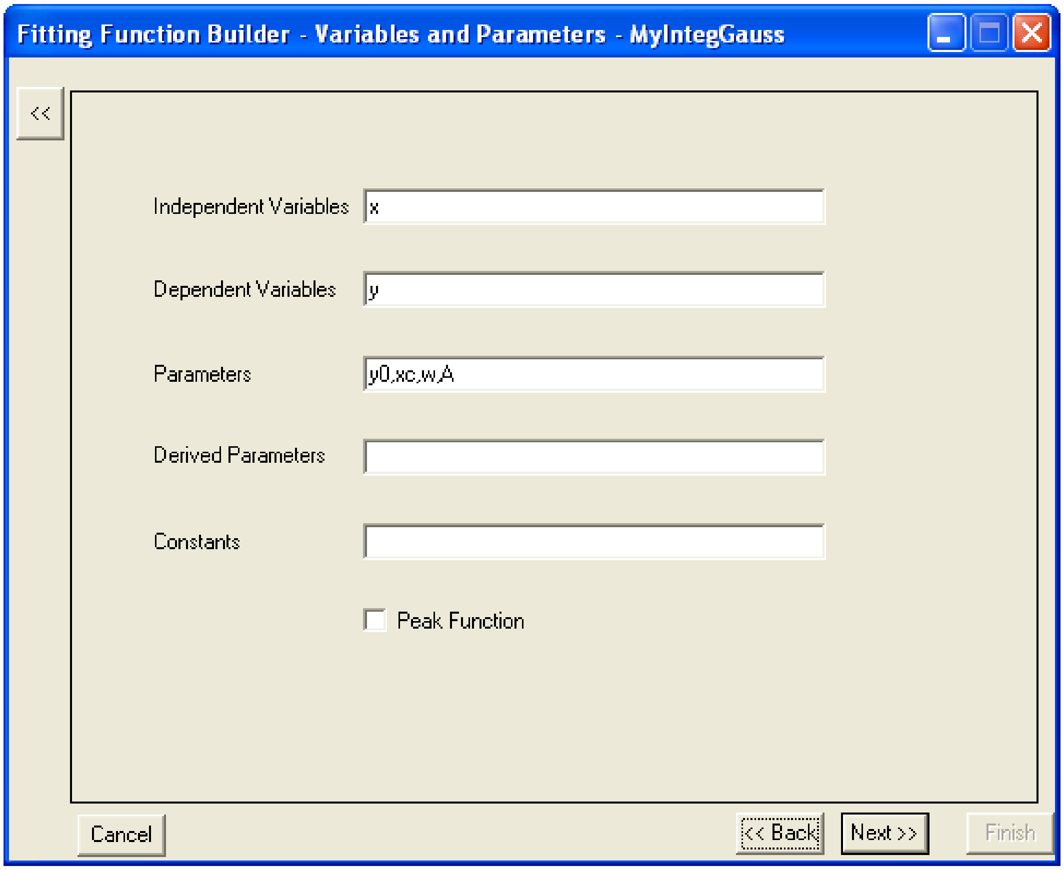
The other variables, like *xc*, *w*, and *A*, are parameters of the integrand. To distinguish from fitting parameters, we named them *Arguments* here, and use the arguments name *ixc*, *iw*, and *iA* instead. Later, we can pass fitting parameters into these arguments. So, the integrand definition should looks like:



Note that this is a LabTalk function. To get the integration value, you must have a **RETURN** statement in the function body. And the integrand expression in this example should be:

```
return iA * exp(-2*(t-ixc)^2/iw^2) / (iw*sqrt(PI/2));
```

4. When all set, click ***Next*** to go to the ***Variables and Parameters*** page to define the variables and parameters for the fitting function as below:



5. The next ***Function*** page is where you define the fitting function body. Once you choose to include integration in your fitting function at the beginning of the fitting function builder wizard, there is an extra tab, ***Integrand***, shown on this page. In this tab, you can map the fitting variables and parameters with elements of the integrand, including lower limit, upper limit, and integrand arguments. And in this example, we will map the variables as below:

Integrand elements	Values pass into integrand
Lower Limit	-inf
Upper Limit	x
ixc	xc
iw	w
iA	A

6. Once set up all the mappings as above table, click the ***Insert*** button, then well prepared *integral()* function is inserted into the ***Function Body*** box as:

```
integral(MyIntegrand, -inf ,x ,xc ,w ,A)
```

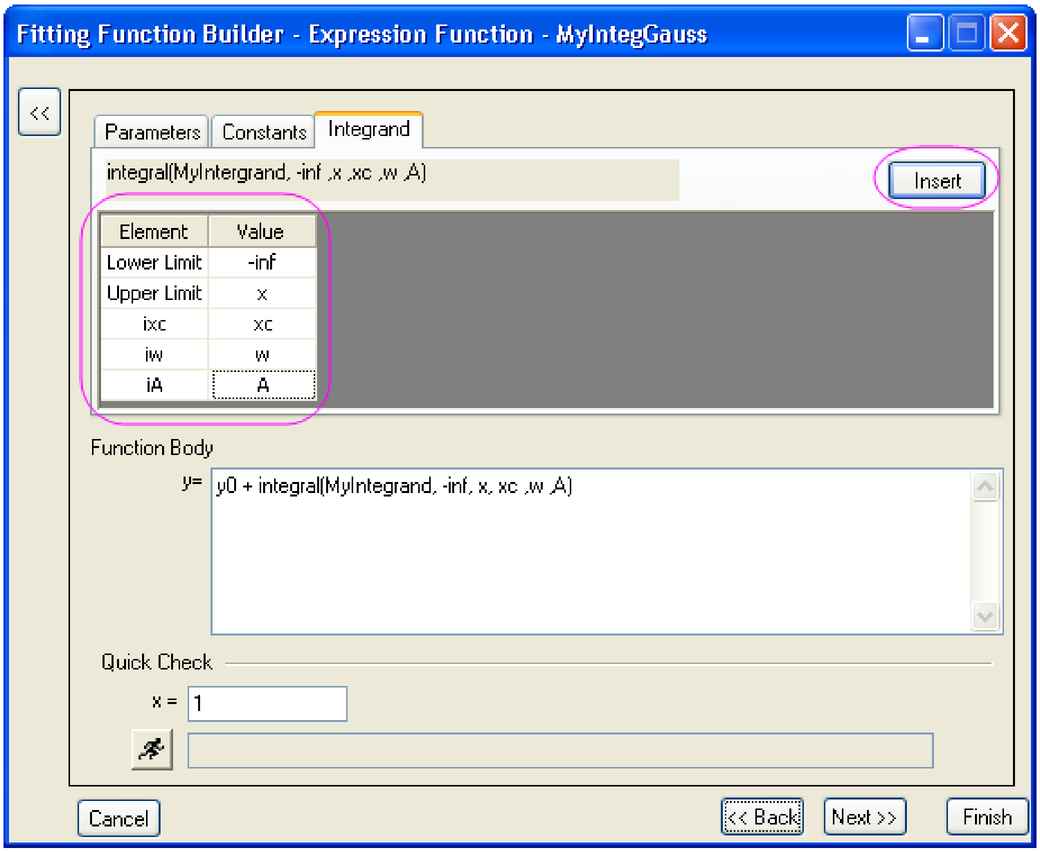
7. This expression means perform integration on the function, whose name is *MyIntegrand*, from negative infinite to *x*, and pass the three fitting parameters, *xc*, *w*, and *A* to the integrand.

8. By adding the constant parameter, *y0*, into the expression, the whole fitting function body should be:

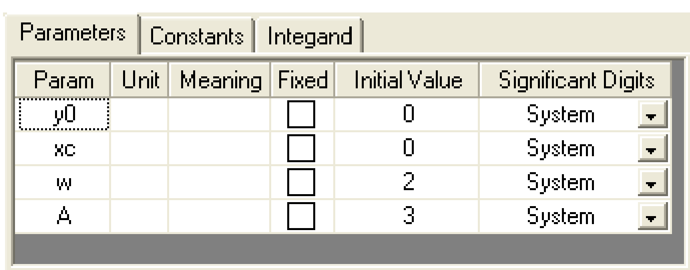
```
y0 + integral(MyIntegrand, -inf, x, xc ,w ,A);
```

9. And the page may looks as below:

10.



11. Then active the ***Parameters*** tab, and give some proper initial values for each fitting parameters as:



Now, you can click ***Finish*** button to save this fitting function.

Fit the Curve

Copy and paste the following data into Origin worksheet:

X	Y
-1.69897	0.13136
-1.22185	0.34384
-0.92082	0.6554
-0.82391	0.73699
-0.69897	1.00157
0	1.70785
0.30103	2.31437
0.69897	2.77326
1	2.79321

Highlight the *Y* column, and press ***Ctrl + Y*** to open the ***NLFit*** dialog. Select the function you just defined, and click ***Fit*** button to perform fitting. The fitting result should be the same like using NAG function directly:

Parameters

		Value	Standard Error
B	y0	-0.01749	0.20203
	xc	-0.21058	0.11852
	w	1.76925	0.36484
	A	3.15913	0.42476

Reduced Chi-sqr = 0.0114837622841
COD(R^2) = 0.99334903530799
Iterations Performed = 6
Total Iterations in Session = 6
Fit converged. Chi-Sqr tolerance value of 1E-9 was reached.

Fitting with Two Integrals using LabTalk Function

Contents

Summary

In some circumstance, one may want to create fitting function with multiple integrals:

$$\int_{LL1}^{UL1} f(t, arg1, arg2, ...)\ dt + \int_{LL2}^{UL2} g(x, arg3, arg4, ...)\ dx$$

We refer to Fitting with Integral using LabTalk Function for detailed description of parameters in the expression.

In version Origin 8.6, however, **Fitting Function Builder** just supports one integral in fitting function. Bypassing complex Origin C code, we can use **Fitting Function Organizer** to reach the goal.

In this tutorial, we will show you how to create a fitting function comprised of two integrals using function organizer. Of course, one can include more integrals as desired.

Minimum Origin Version Required: Origin 8.6

What you will learn

This tutorial will show you how to:

- Create a fitting function using the Fitting Function Organizer
- Create a fitting function with two integrals using LabTalk function

Example and Steps

The Fitting Model

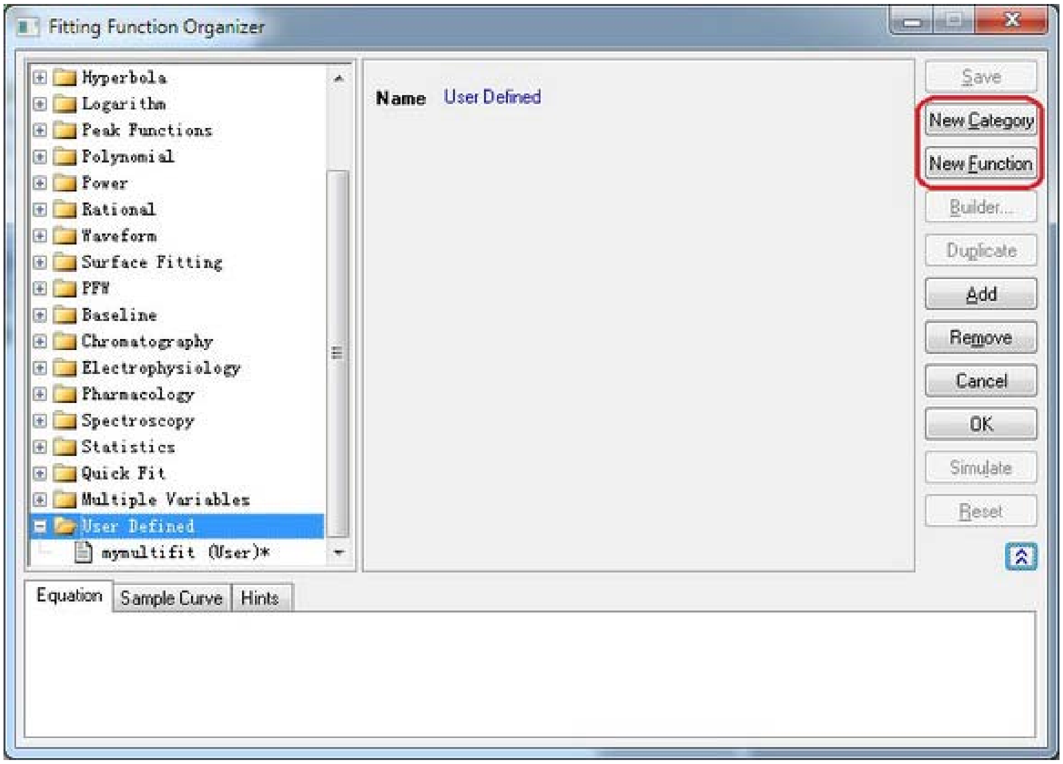
The fitting model is described as below:

$$y = y_0 + \int_{-5}^{x} A\ t\ dt - \int_{-\infty}^{x} B\ t\ \exp(-t^2/w^2)\ dt$$

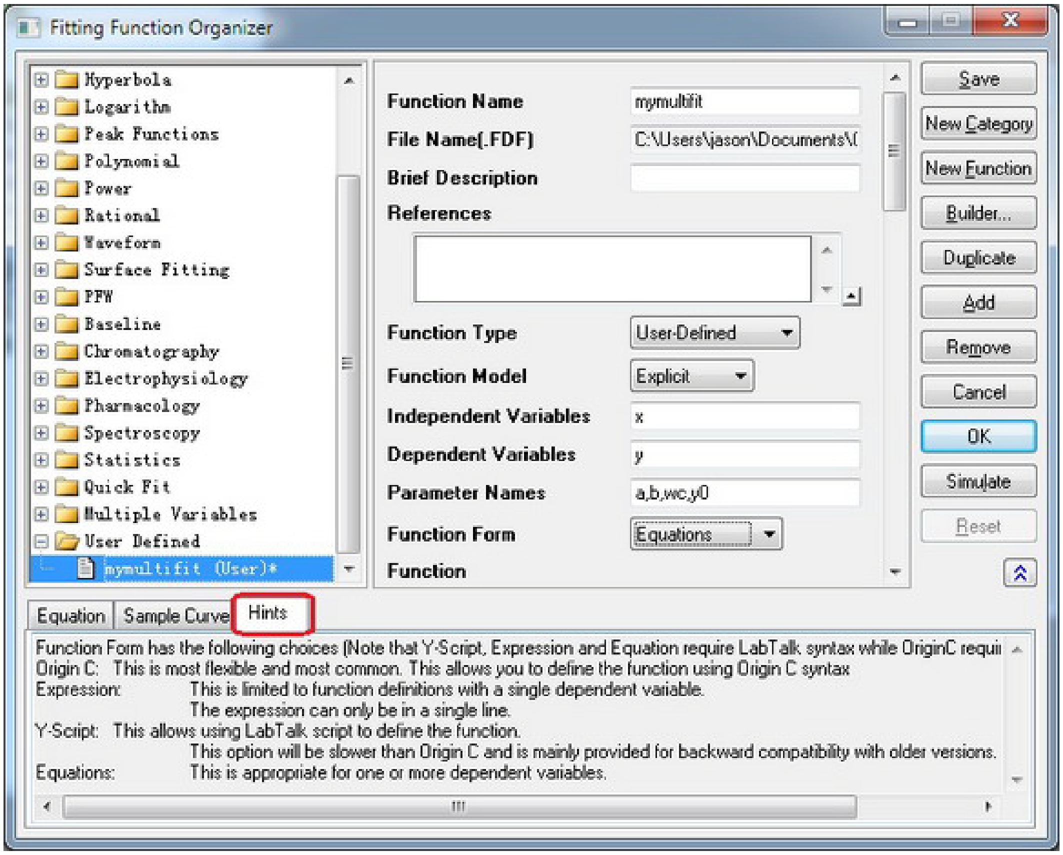
There are four parameters in the fitting function, and we need to pass three of them into the integrand, and use the independent variable as upper limit, to do integration.

Define the Function

1. Press F9 to open **Fitting Function Organizer** dialog. Add a new function by pressing **New Function** after you select a category in which you want to put your function. One can also add new category by pressing the **New Category** button.



2. Specify the function name in the **Function Name** edit box as you like. Define **Independent Variables**, **Dependent Variables** and **Parameter Names** in corresponding edit box.
3. Select the **Function Form** in the drop box. One can find explanations in the **Hints** tab at the bottom of the dialog.

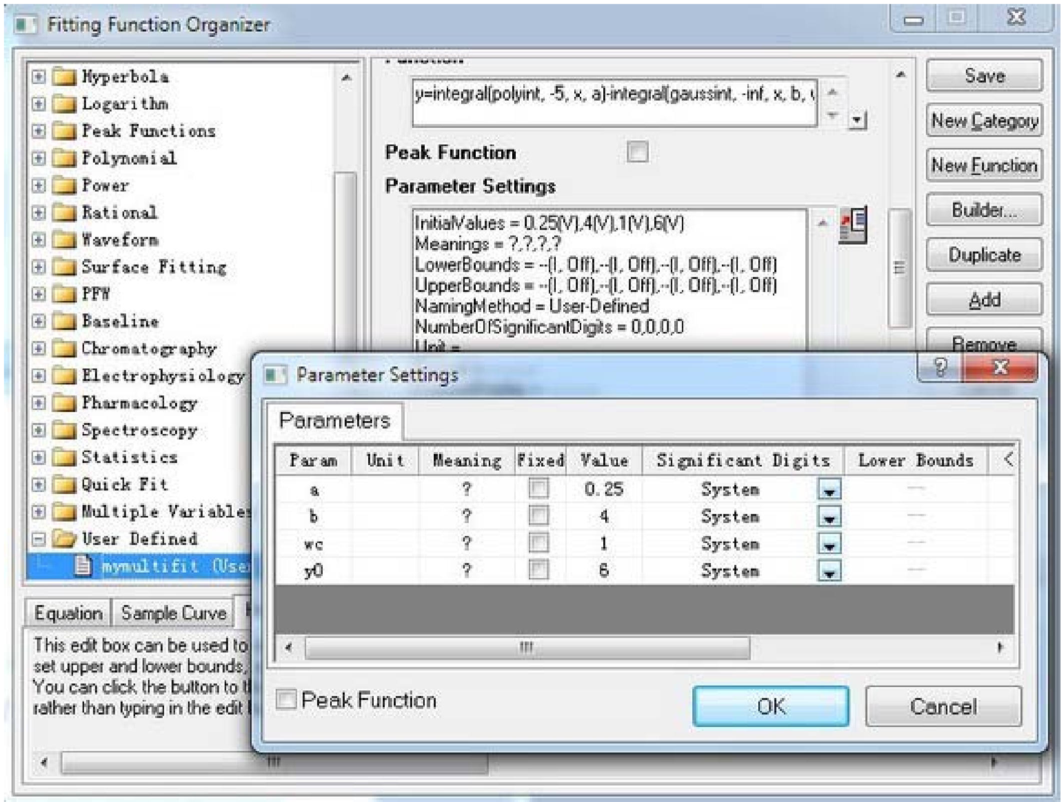


4. In the **Function** edit box, define your fitting function. The integrals are written in the form of LabTalk **integral** function.

```
y=integral(polyint, -5, x, a)-integral(gaussint, -inf, x, b,
wc)+y0
```

5. As described in Fitting with Integral using LabTalk Function, *x*, *a*, *b* and *wc* are parameters passed into the integrand functions.

6. Press the button at the right corner of **Parameter Settings** box to active **Parameter Settings** dialog. Set initial values as well as other constraints such as lower bounds, upper bounds for each parameter.



7. Define the two integrals in the **LabTalk Functions Definition and Initializations** box. In this case, the functions should be:

```
function double polyint(double t, double ia)
{
        return ia*t ;

}


function double gaussint(t, ib, iwc)
{
        return ib *t* exp(-(t)^2/iwc^2) ;
}
```

8. We have successfully set our two integrals fitting function. You can set other information in corresponding box. Do not forget to **Save** your fitting function after you finish.

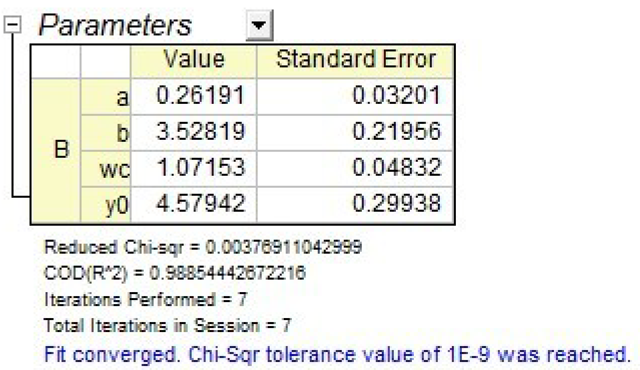
Fit the Curve

Copy and paste the following data into Origin worksheet:

X	Y
-3	2.47613

-2.6	2.24016
-2.2	2.01543
-1.8	1.83094
-1.5	1.85038
-1.1	2.17725
-0.9	2.44967
-0.7	2.61423
-0.5	3.02305
-0.3	3.23057
-0.1	3.37822
0.1	3.2827
0.3	3.18775
0.5	2.86194
0.7	2.69104
0.9	2.39315
1.4	2.04046
1.8	1.85287
2.2	1.85325
2.6	2.20569

Highlight the *Y* column, and press CTRL + Y to open the **NLFit** dialog. Select the function you just defined, and click fit button  to perform fitting.


Parameters

		Value	Standard Error
B	a	0.26191	0.03201
	b	3.52819	0.21956
	wc	1.07153	0.04832
	y0	4.57942	0.29938

Reduced Chi-sqr = 0.00376911042999
COD(R^2) = 0.98854442672216
Iterations Performed = 7
Total Iterations in Session = 7
Fit converged. Chi-Sqr tolerance value of 1E-9 was reached.

Fitting with Summation

Contents

Summary

We have showed you how to perform fitting with an integral using the NAG Library, and now you'll learn how to do that without calling NAG functions. In this tutorial, we will show you how to do integration by the trapezoidal rule and include the summation procedure in the fitting function.

Minimum Origin Version Required: Origin 8.0 SR6

What you will learn

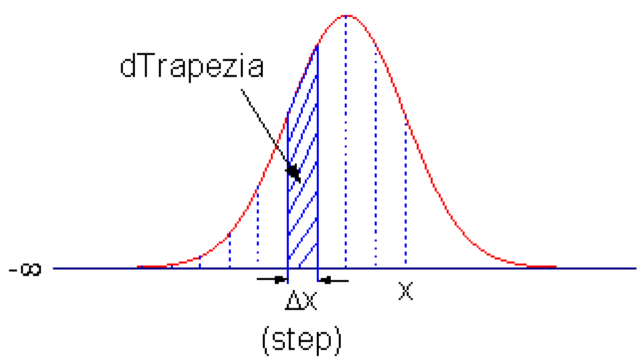
- How to include summation in your fitting function.
- Trapezoidal rule for integration.

Example and Steps

We will fit the same model as the integral fit using NAG:

$$y = y_0 + \int_{-\infty}^{x} \frac{A}{w\sqrt{\frac{\pi}{2}}} e^{-2\frac{(x-x_c)^2}{w^2}}, dx$$

The difference is that we will perform the integration within the fitting function. Using the trapezoidal rule, we will first divide the curve into pieces and then approximate the integral area by multiple trapezoids. The precision of the result then depends on how many trapezoids will be used. Since this is a semi-infinite integration, we will set an increment (steps) and construct trapezoids from the upper integral limit, x, to the lower integral limit, negative infinity and then accumulate the area of these trapezoids. When the increment of the area is significantly small, we will stop the summation. Before doing the summation, you should guarantee that the function is **CONVERGENT**, or you should include a convergence check in your code.



Define the Function

Select **Tools:Fitting Function Organizer** or alternatively press the **F9** key to open the **Fitting Function Organizer** and then define the function as follows:

Function Name:	summation
Function Type:	User-Defined

Independent Variables: x

Dependent Variables: y

Parameter Names: y0, A, xc, w

Function Form: Origin C

Function:

Click the button (icon) beside the **Function** box to open Code Builder. Define, compile and save the fitting function as follows:

```
#pragma warning(error : 15618)
#include <origin.h>

// Subroutine for integrand
double f(double x, double A, double xc, double w)
{
	return A * exp(-2*(x-xc)*(x-xc)/w/w) / w / sqrt(PI/2);
}

//----------------------------------------------------------
//
void _nlsfsummation(
// Fit Parameter(s):
double y0, double A, double xc, double w,
// Independent Variable(s):
double x,
// Dependent Variable(s):
double& y)
{
	// Beginning of editable part
	// Set the tolerance for stop integration.
	double dPrecision = 1e-12;
	// Initialization
	double dIntegral = 0.0;
	double dTrapezia = 0.0;
	// Steps, or Precision.
	double dStep = 0.01;
	// Perform integrate by trapezoidal rule.
	// Note that you should guarantee that the function is
CONVERGENT.
	do
	{
		// Trapezia area.
		dTrapezia = 0.5 * ( f(x, A, xc, w) + f((x-dStep), A, xc,
w) ) * dStep;
		// Accumulate area.
		dIntegral += dTrapezia;
		x -= dStep;
	}while( (dTrapezia/dIntegral) > dPrecision );
	// Set y value.
	y = y0 + dIntegral;
	// End of editable part
}
```

Fit the Curve

We can use the same data to test the result.

1. Import *\Samples\Curve Fitting\Replicate Response Data.dat*.
2. Highlight the first column, right-click on it, and select Set Column Values from the context menu.
3. Set *Col*(*A*) = *log*(*Col*(*A*)) in the **Set Column Values** dialog. This will make a sigmoidal curve.
4. Highlight columns A and B and create a scatter plot.
5. Then bring up the **NLFit** dialog by pressing **Ctrl** + **Y**. Select the fitting function we just defined and go to the **Parameters** tab, initialize all parameters to 1 and fit. You should see these results:

	Value	Standard Error
y0	-0.00806	0.18319
A	3.16479	0.39624
xc	-0.19393	0.10108
w	1.7725	0.33878

Fitting Complex Function

Summary

When fitting with a complex function, we can easily separate the complex function to two functions: one corresponding to its real part and the other corresponding to its imaginary part. With these two functions, we can define the complex fitting function with two dependent variables by **Fitting Function Organizer** and can access it in **NLFit** dialog. We will illustrate how to fit with complex function below. More details about fitting with multiple dependent or independent variable please refer to Fitting with Multiple Independent Variables.

Minimum Origin Version Required: Origin 8.0 SR6

What you will learn

This tutorial will show you how to:

- Create a user-defined complex fitting function with two dependent variables and one independent variable
- Fit with such function in NLFit

Steps

1. Select whole form below (including header line) and right click to choose **Copy** to put the data in clipboard.

Omega	Y1	Y2
0	3	0
0.01	2.88462	-0.28846
0.02	2.58621	-0.51724
0.03	2.20588	-0.66176
0.04	1.82927	-0.73171
0.05	1.5	-0.75
0.06	1.22951	-0.7377

0.07	1.01351	-0.70946
0.08	0.8427	-0.67416
0.09	0.70755	-0.63679
0.1	0.6	-0.6
0.11	0.5137	-0.56507

2. Select **Import/ Import Wizard** to open Import Wizard dialog. Then choose **Clipboard** in **Data Source** group and click **Finish** to import the data.

3. Select **Tools: Fitting Function Organizer** from menu (or press **F9**) to bring up the **Fitting Function Organizer** and define a new fitting function named **ComplexFitting** in **NewCategory** (create the category if not exist) as follow:

Function Name:	ComplexFitting
Function Type:	User-Defined
Independent Variables:	omega
Dependent Variables:	y1,y2
Parameter Names:	A,tau
Function Form:	Origin C

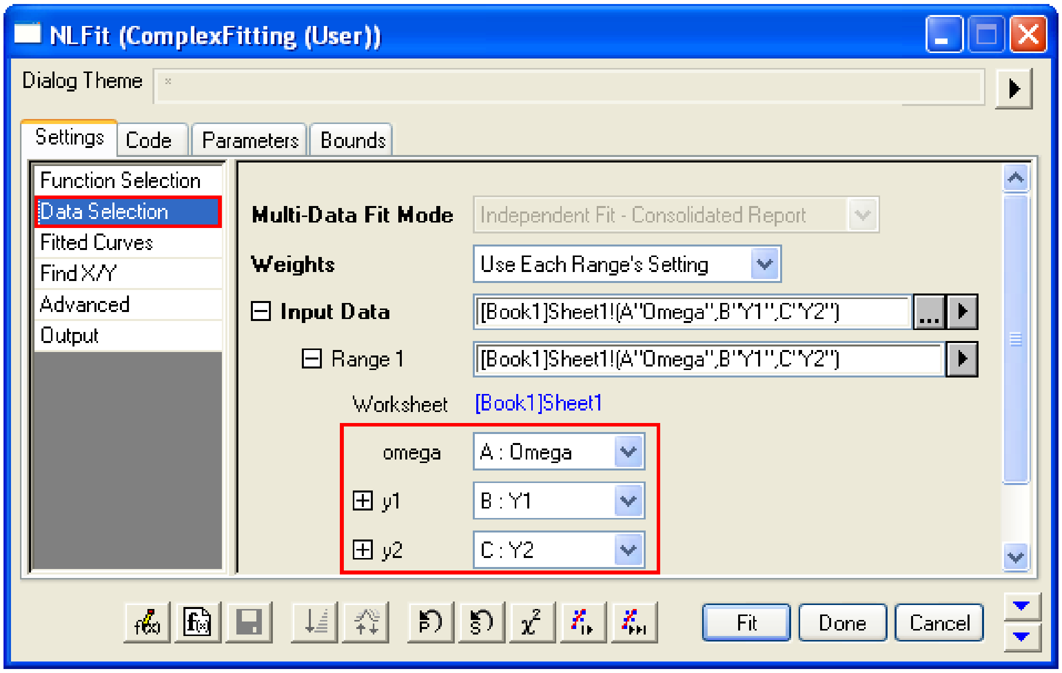
Function:	complex cc = A/(1+1i*omega*tau);
	y1 = cc.m_re;
	y2 = cc.m_im;

4. Note: To use the imaginary unit "i" for creating complex numbers, you need to write it as "1i" in Origin C, as in the above **Function** row. And **complex** is a class that implements a complex number data type. It contains both a Real and an Imaginary component.

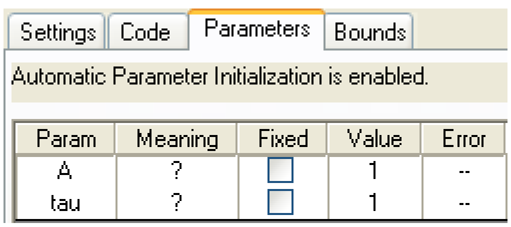
5. For more details about creating user-defined fitting function, please refer to User Defined Fitting Function using Origin C.

6. Highlight all the columns and select **Analysis: Fitting: Non-linear Curve Fit** from menu to bring up the NLFit dialog. Select the function **ComplexFitting** from **NewCategory** on the **Settings: Function Selection** page. Set the input datasets in the **Data Selection** page as

follow:



7. Select **Parameters Tab** and set the initial values as follows:



Param	Meaning	Fixed	Value	Error
A	?		1	--
tau	?		1	--

8. Click **Fit** to generate the fitting report sheet. You can see the results from the report worksheet as below:

Parameters

		Value	Standard Error
Y1,Y2	A	2.36712	0.15413
	tau	15.84746	1.94844

Statistics

	Y1,Y2
Number of Points	24
Degrees of Freedom	22
Reduced Chi-Sqr	0.12339
Residual Sum of Squares	2.71451
Adj. R-Square	0.92387
Fit Status	Succeeded(100)

From the Statistics table, we can see that the fitting is fairly successful.

Fitting with Convolution

Contents

Summary

When performing curve fitting to experimental data, one may encounter the need to account for instrument response in the data. One way to do this is to first perform deconvolution on the data to remove the instrument response, and then perform curve fitting as a second step. However, deconvolution is not always reliable as the results can be very sensitive to any noise present in the data. A more reliable way is to perform convolution of the fitting function with the instrument response while performing the fitting. This tutorial will demonstrate how to perform convolution while fitting.

Minimum Origin Version Required: Origin 8 SR6.

What you will learn

This tutorial will show you how to:

- Access fitting information during iterations.
- Perform convolution while fitting.

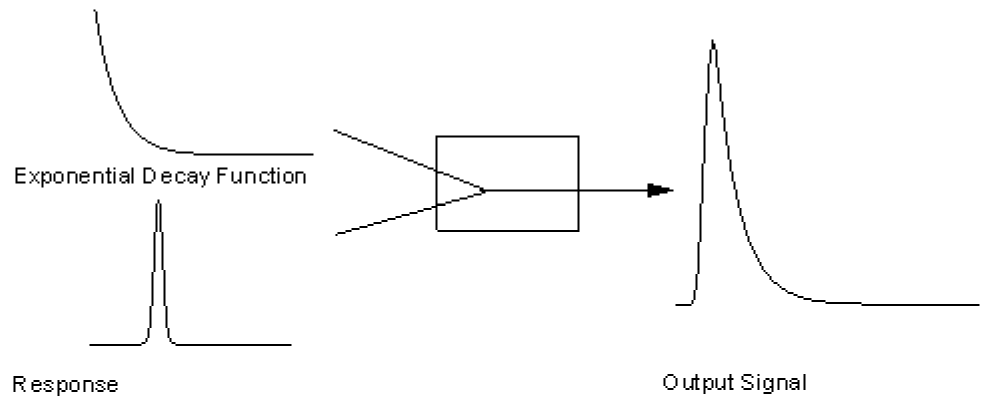
Example and Steps

Background

Let's start this example by importing *Samples**Curve Fitting**FitConv.dat*.

A(X)	B(Y)	C(Y)
Sampling	Signal	Impulse
0	-0.19775	0
0.1	-0.32893	0
0.2	0.10055	0
0.3	0.09394	0
0.4	-0.1292	0
0.5	0.06346	1.48672E-6
0.6	0.19453	1.3383E-4

The source data includes sampling points, output signal, and the impulse response. This experiment assumes that the output signal was the convolution of an exponential decay function with a Gaussian response:



Now that we already have the output signal and response data, we can get the exponential decay function by fitting the signal with the below model:

$$y = y_0 + \int_{-\infty}^{+\infty} Ae^{-tx} \otimes Response, dx$$

Define the Function

Obviously, column 1 and column 2 are x and y respectively in the function. How about column 3, the impulse response? We will access this column within the fitting function, and compute the theoretical exponential curve from the sampling points. Then we can use fast Fourier transform to perform the convolution.

Press **F9** to open the **Fitting Function Organizer** and define a function like:

Function Name:	FitConv
Function Type:	User-Defined
Independent Variables:	x
Dependent Variables:	y
Parameter Names:	y0, A, t
Function Form:	Origin C
Function:	

Click the button (icon) beside the **Function** box and write the function body in **Code Builder**:

```
#pragma warning(error : 15618)
#include <origin.h>
// Header files need to be included
#include <ONLSF.H>
#include <fft_utils.h>
//
//
void _nlsfTestConv(
// Fit Parameter(s):
double y0, double A, double t,
// Independent Variable(s):
double x,
// Dependent Variable(s):
double& y)
{
        // Beginning of editable part
```

```
        Worksheet wks = Project.ActiveLayer();
        NLFitContext *pCtxt = Project.GetNLFitContext();
        if ( pCtxt )
        {
                // Vector for the output signal in each iteration.
                static vector vSignal;
                // If parameters were updated, we will recalculate the
convolution result.
                BOOL bIsNewParamValues = pCtxt->IsNewParamValues();
                if ( bIsNewParamValues )
                {
                        // Read sampling and response data from
worksheet.
                        Dataset dsSampling(wks, 0);
                        Dataset dsResponse(wks, 2);
                        int iSize = dsSampling.GetSize();

                        vector vResponse, vSample;

                        vResponse = dsResponse;
                        vSample = dsSampling;

                        vSignal.SetSize(iSize);
                        vResponse.SetSize(iSize);
                        vSample.SetSize(iSize);

                        // Compute the exponential decay curve
                        vSignal = A * exp( -t*vSample );
                        // Perform convolution
                        int iRet = fft_fft_convolution(iSize, vSignal,
vResponse);

                }

                NLSFCURRINFO    stCurrInfo;
                pCtxt->GetFitCurrInfo(&stCurrInfo);
                // Get the data index for the iteration
                int nCurrentIndex = stCurrInfo.nCurrDataIndex;
                // Get the evaluated y value
                y = vSignal[nCurrentIndex] + y0;
                // For compile the function, since we haven't use x
here.
                x;
        }
        // End of editable part
}
```

Traditionally, for a particular x, the function will return the corresponding y value. However, when convolution is involved, we need to perform the operation on the entire curve, not only for a particular data point. So, from Origin 8 SR2, we introduced the NLFitContext class to achieve some key information within the fitter. In each iteration, we use NLFitContext to monitor the fitted parameters; once they are updated, we will compute the convolution using the fast Fourier transform by the fft_fft_convolution method. The results are saved in the vSignal vector. Then for each x, we can get the evaluated y from vSignal with the current data index in NLSFCURRINFO.

Fit the Curve

In the fitting function body, we read the response data directly from the active worksheet. So, you should perform the fit from the worksheet.

1. Highlight column B and press **Ctrl + Y** to bring up the **Nonlinear Fitting** dialog.
2. Choose *X Data Type* from *Fitted Curves* page as ***Same as Input Data***.
3. Go back to the Function Selection page to select the *FitConv* function you just defined.
4. Go to the Parameter tab to initialize the parameters as y0=0, A=10, t=1.
5. Click the **Fit** button to generate the results.

Quoting Built-in Functions in Your New Function

Contents

Summary

This tutorial will show you how to reference a built-in function when creating a user-defined fitting function.

Minimum Origin Version Required: Origin 8.0 SR6

What you will learn

This tutorial will show you how to:

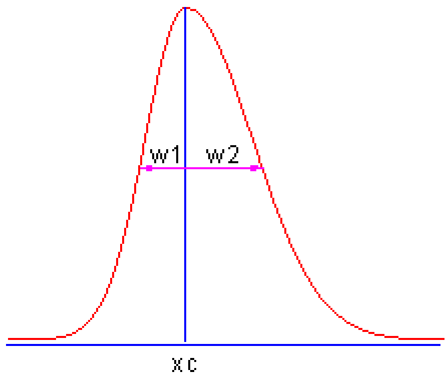
- Define a piecewise fitting function
- Access built-in functions in you new function
- Auto initialize the parameters

Steps

Data

Start by importing the file *\Samples\Curve Fitting\Asymmetric Gaussian.dat* into a new workbook.

Highlight column B and create a graph. The peak in the data is slightly skewed to the right. How to fit such a curve? One idea is to divide the curve into two parts - We can consider this curve to be composed of two Gaussian function as below. These two Gaussian curves share the same baseline and peak center, but differ in peak width and amplitude.



Define the Function

Press **F9** to open the **Fitting Function Organizer** and define the function as below:

Function Name:	AsymmetricGauss
Function Type:	User-Defined
Independent Variables:	x
Dependent Variables:	y
Parameter Names:	y0, xc, w1, w2, A1, A2
Function Form:	Origin C
Function:	y = x<xc? nlf_Gauss(x, y0, xc, w1, A1) : nlf_Gauss(x, y0, xc, w2, A2);

Note:

For versions before **Origin 8.1**, the function body should be defined as:

```
y = x<xc? nlfxGauss(x, y0, xc, w1, A1) : nlfxGauss(x, y0, xc, w2,
A2);
x; y0; xc; w1; w2; A1; A2;
```

Listing the parameters at the end is used to avoid the "parameter not used inside the function body" error, although you already use these parameters. This is required to compile the function successfully.

When calling **nlf_*FuncName*** to reuse built-in functions, the syntax is:

nlf_FuncName(independent variable, parameter list ...)

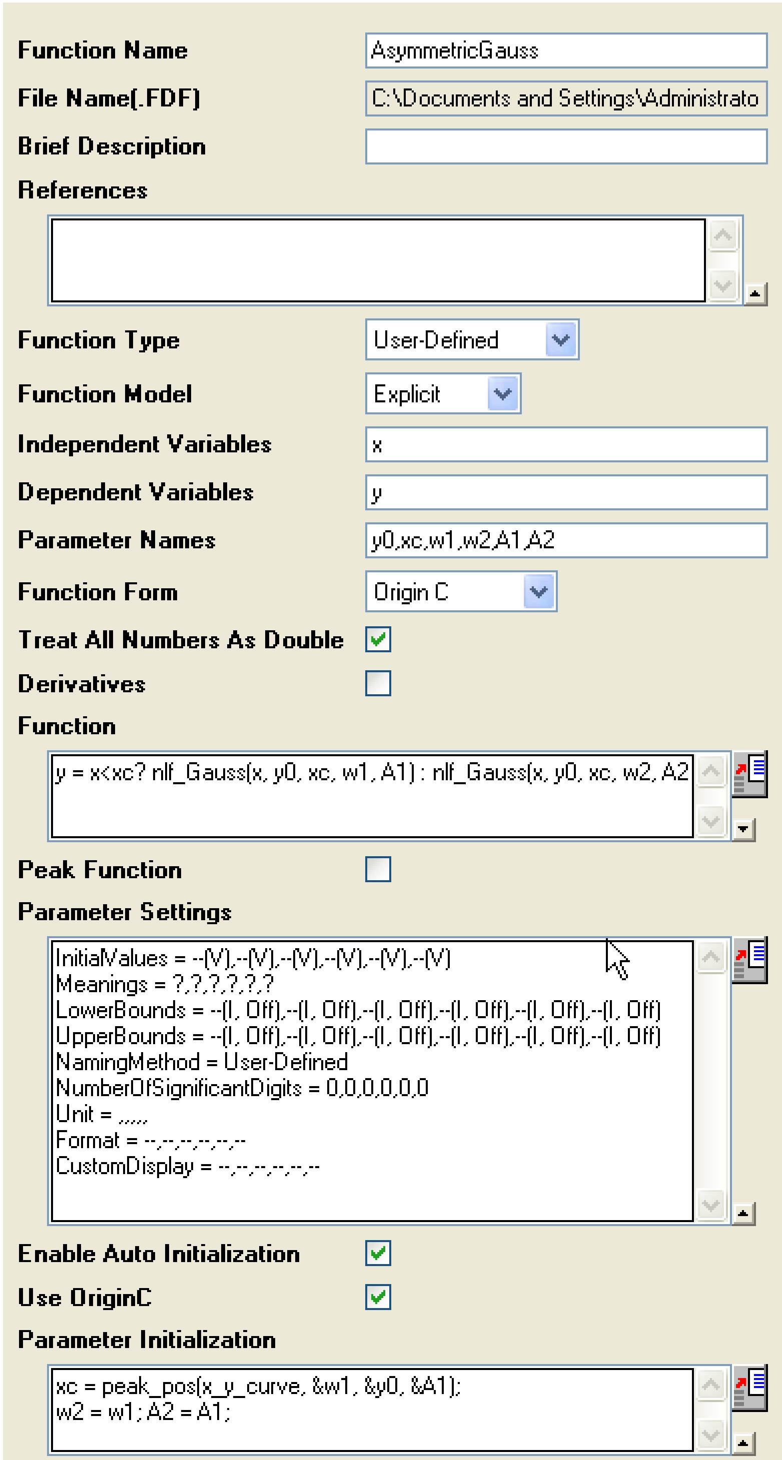
where ***FuncName*** is the fitting function name. Besides, the old notation, ***nlfxFuncName*** also supported.

The *Parameter List* follows the parameter order in function definition file for the built-in function (the FDF file. You can open the FDF file in Notepad. The files are located in the \\Origin EXE Folder\FitFunc\). Note that, the function name we use is the DLL interface name. The actual name in the [General Information] section of the FDF file. Look at the *Function Source* item and the value is **fgroup.*FuncName***, and use the *FuncName*. In most cases, this function name is consistent with the function name visible in the NLFit dialog. For a few few functions such as Voigt, these names are different.

For parameter initialization of this skewed gaussian function, we can simply copy the initialization code of the built-in gauss function, and make a few minor modifications:

```
xc = peak_pos(x_y_curve, &w1, &y0, &A1);
w2 = w1;
A2 = A1;
```

The final function body should be as below:



Once compiled successfully, save the function and fit the curve. The results should be as below:

		Value	Standard Error
Amplitude 1	y0	1.8	4.79E-5
	xc	4.5	3.45E-5
	w1	1.8	4.5E-5
	w2	3	4.88E-5
	A1	30	0
	A2	50	0

Fit Function with Non-constant Background

Contents

Summary

Many of the Origin built-in functions are defined as:

$$y = y_0 + \ldots\ldots$$

Where y0 can be treated as the "constant background". How about fitting a curve with a non-constant background? One option is to use the Peak Analyzer we provide. The Peak Analyzer includes multiple methods to subtract the baseline, including exponential or polynomial backgrounds. In this tutorial, we will show you how to fit such curves without using the Peak Analyzer.

Minimum Origin Version Required: Origin 8.0 SR6

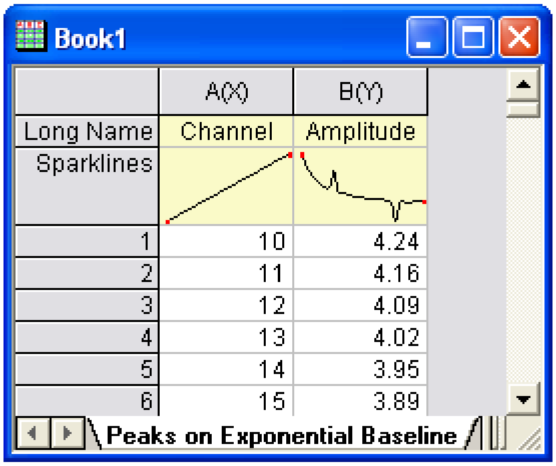
What you will learn

- Review Worksheet Query.
- Quote a built-in function by nlfx*FuncName* method.
- Auto Initialize the parameters.

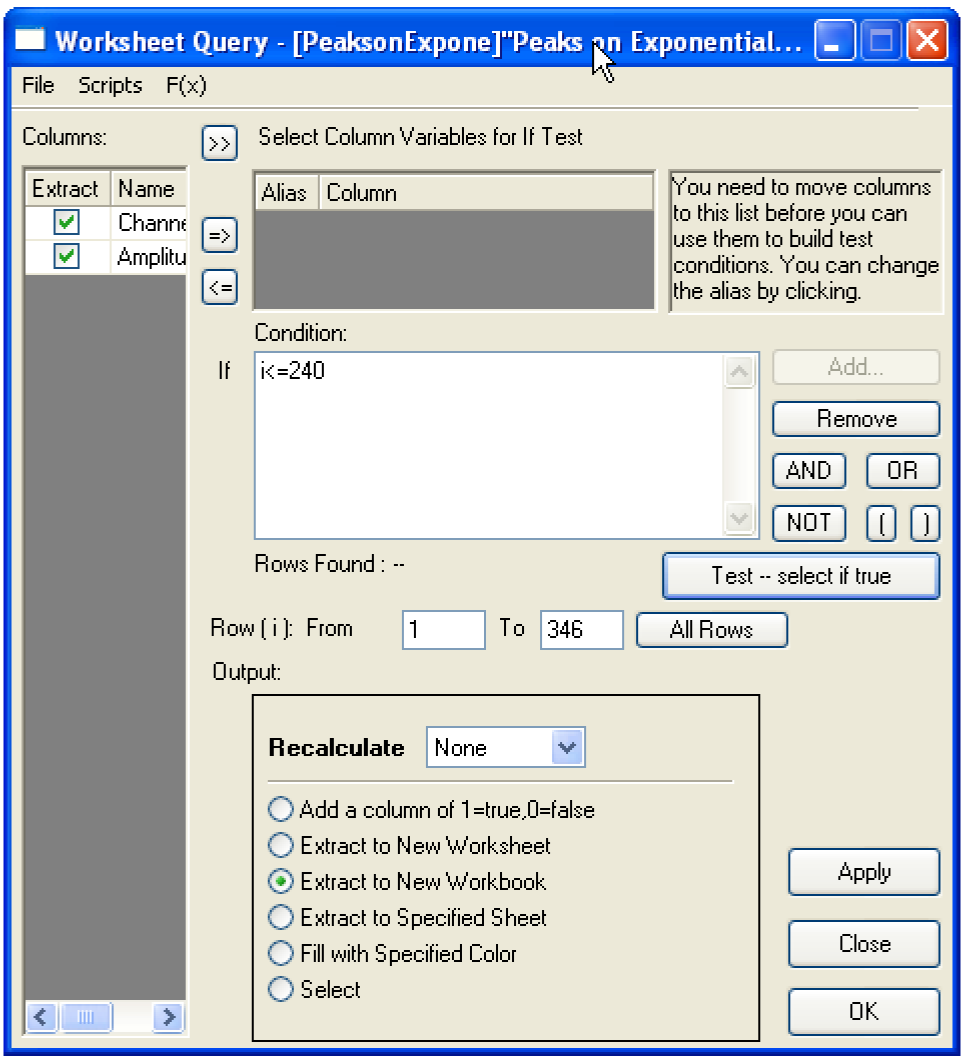
Example and Steps

Prepare the Data

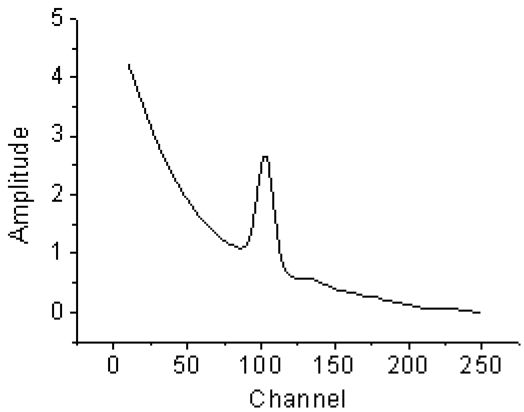
Let's start this tutorial by importing *\Samples\Spectroscopy\Peaks on Exponential Baseline.dat*. From the worksheet sparkline, we can see that there are two peaks in the curve. To simplify the problem, we will fit just one peak in this example.



Now bring up the Worksheet Query dialog from **Worksheet : Worksheet Query**. And we will extract data from row 1 to row 240:

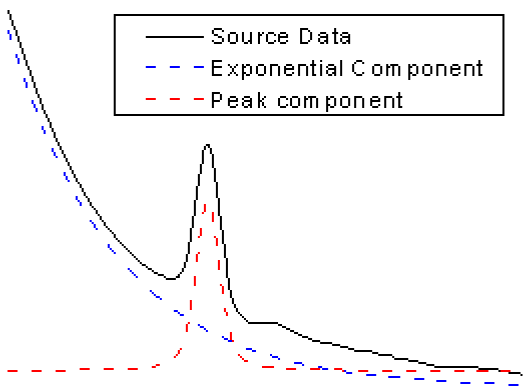


So the curve we will fit should look like this:



Define the Function

As illustrated below, we can consider the source curve is the combination of an exponential decay component (the background) with a Voigt peak:



So should we write down the whole equation to define the function? Like:

$$y = y_0 + A_1 e^{-x/t1} + A_2 \frac{2w_L \ln 2}{\pi^{\frac{3}{2}} w_G^2} \int_{-\infty}^{\infty} \frac{e^{-t^2}}{\left(\sqrt{\ln 2}\frac{w_L}{w_G}\right)^2 + \left(\sqrt{4\ln 2}\frac{x-x_c}{w_G} - t\right)^2} dt$$

Well, this is a complicated equation and it includes infinite integration. Writing such an equation directly is painful. Now that we already have these two built-in functions:

ExpDec1:

$$y = y_0 + Ae^{-x/t}$$

Voigt:

$$y = y_0 + A\frac{2w_L \ln 2}{\pi^{\frac{3}{2}} w_G^2} \int_{-\infty}^{\infty} \frac{e^{-t^2}}{\left(\sqrt{\ln 2}\frac{w_L}{w_G}\right)^2 + \left(\sqrt{4\ln 2}\frac{x-x_c}{w_G} - t\right)^2} dt$$

we can simply use the **nlfx*FuncName*** method to quote these two built-in functions and create a new one. Press **F9** to open the **Fitting Function Organizer** and define a function as below:

Function Name:	ExpVoigt

Function Type: User-Defined
Independent Variables: x
Dependent Variables: y
Parameter Names: y0, A1, t1, xc, A2, wG, wL
Function Form: Origin C
Function: y = nlf_ExpDec1(x, y0, A1, t1) + nlf_Voigt(x, y0, xc, A2, wG, wL) - y0;

Note:

Some of the built-in function names do not consistent with the actual DLL function name. Just like this Voigt function, it's defined in Voigt5.FDF, and if you open the FDF file by Notepad, you can see a line under [GENERAL INFORMATION] section says:

Function Source=fgroup.Voigt5

The name after "fgroup" is the actual name we should put into **nlf_*FuncName***.

Besides, for versions before **Origin 8.1 SR2**, the function body should use old *nlfxFuncName* notation and define as:

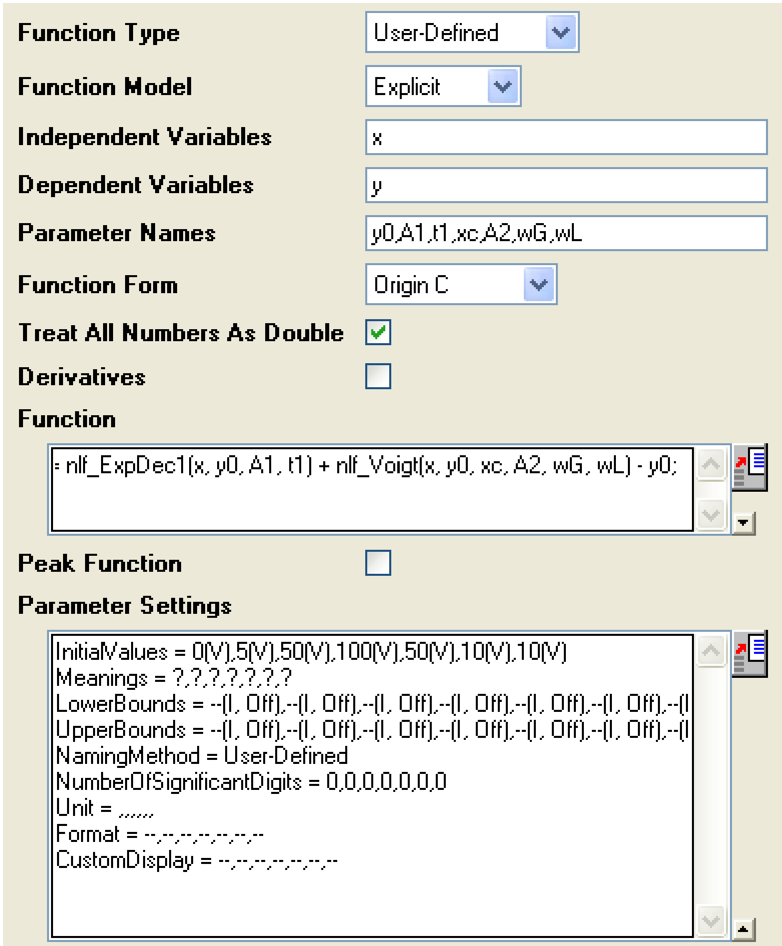
```
y = nlfxExpDec1(x, y0, A1, t1) + nlfxVoigt(x, y0, xc, A2,
wG, wL) - y0;
x; xc; A1; t1; A2; wG; wL;
```

Listing the parameters at the end is done to avoid the "parameter not used inside the function body" error, although you already use these parameters. If not, you will not compile the function successfully.

Click the button on the right of the **Parameter Settings** and enter these parameter initial values:

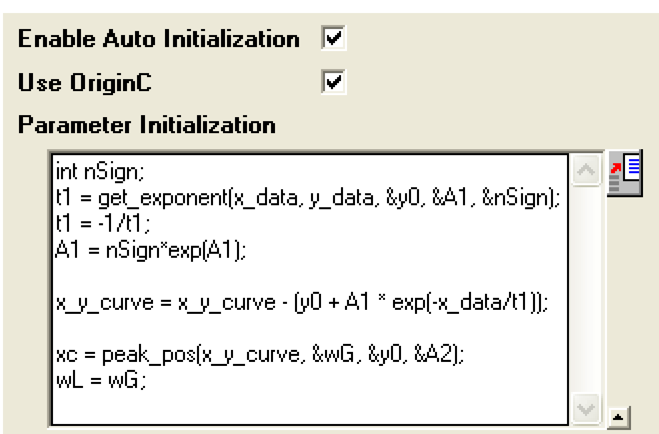
y0: 0
A1: 5
t1: 50
xc: 100
A2: 50
wG: 10
wL: 10

So the final function definition part should look like:



Auto Parameter Initialization

In the above section, we set fixed parameter initial values. If you know the possible fitted results, you can set the initial values in this way. But how about when the data is changed? Origin provides an Origin C interface to "guess" the initial values. To use the parameter initialization code, make sure to check the **Enable Auto Initialization** and **Use OriginC** checkboxes, and edit the code in Code Builder by clicking the icon.
(P.S: If you know the initial values very well, or you don't like coding, please skip this section.)



Now that the curve is composed by two components, we can guess the parameter values by separating these two parts, the initialization code includes:

1. Use the get_exponent function to fit the curve and get the parameter values for exponential component.
2. Remove the background -- exponential component -- from source data.
3. Approaching the peak by Gaussian peak using peak_pos function and set the initial values for peak component

So, the initialization code in Code Builder should look like this:

```
void _nlsfParamExpVoigt(
// Fit Parameter(s):
double& y0, double& A1, double& t1, double& xc, double& A2, double& wG,
double& wL,
// Independent Dataset(s):
vector& x_data,
// Dependent Dataset(s):
vector& y_data,
// Curve(s):
Curve x_y_curve,
// Auxilary error code:
int& nErr)
{
        // Beginning of editable part
        int nSign;
        // Evaluates the parameters' value, y0, ln(A) and R for y =
y0+A*exp(R*x).
        t1 = get_exponent(x_data, y_data, &y0, &A1, &nSign);
        // Set the exponential component values for the fitting
function.
        t1 = -1/t1;
        A1 = nSign*exp(A1);
        // Remove the exponential component from the curve;
        x_y_curve = x_y_curve - (y0 + A1 * exp(-x_data/t1));
        // Fit to get peak values.
        xc = peak_pos(x_y_curve, &wG, &y0, &A2);
        wL = wG;
        // End of editable part
}
```

> **Note:**
>
> When you check the **Enable Auto Initialization** and enter the initialization code, this code will cover the initial values in **Parameter Settings**.

Fit the Curve

No matter what kind of parameter initialization method you used, highlight column B and press **Ctrl + Y** to bring up the NLFit dialog, select the ExpVoigt function and fit. The result should be:

		Value	Standard Error
Amplitude	y0	0.04862	0.00724
	A1	5.08842	0.02599
	t1	50.67096	0.51939
	xc	102.81043	0.07241
	A2	32.91106	0.92012
	wG	9.65255	0.67731
	wL	5.7529	0.81022

Fitting with Piecewise Functions

Contents

Summary

We will show you how to define piecewise fitting function in this tutorial.

Minimum Origin Version Required: Origin 8.0 SR6

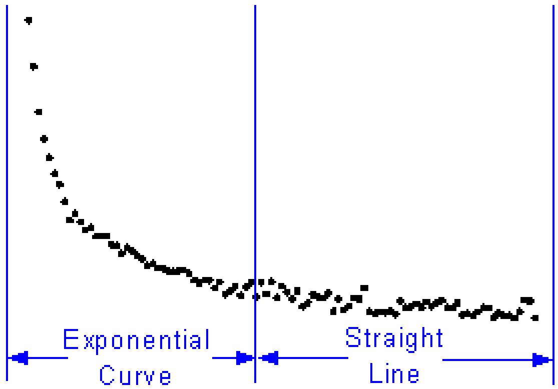
What you will learn

This tutorial will show you how to:

- Define piecewise (conditional) fitting functions.

Example and Steps

We can start this tutorial by importing the sample *\Samples\Curve Fitting\Exponential Decay.dat* data file. Highlight column D and plot a Scatter Graph. You can fit this curve using built-in functions under Growth/Sigmoidal category, however, in this tutorial, we will separate the curve into two parts by a piecewise function.



So the equation will be:

$$y = \begin{cases} a + bx + e^{-\frac{x - x_c}{t1}}, & \text{if } x < x_c \\ a + bx, & \text{if } x \geq x_c \end{cases}$$

Define the Function

Press **F9** to open the **Fitting Function Organizer** and define a function like:

Function Name:	piecewise
Function Type:	User-Defined
Independent Variables:	x
Dependent Variables:	y
Parameter Names:	xc, a, b, t1
Function Form:	Origin C
Function:	

Click the button on the right of the **Function** edit box and define the fitting function in Code Builder using:

```
void _nlsfpiecewise(
// Fit Parameter(s):
double xc, double a, double b, double t1,
// Independent Variable(s):
double x,
// Dependent Variable(s):
double& y)
{
        // Beginning of editable part
        // Divide the curve by if condition.
        if(x<xc) {
                y = a+b*x+exp(-(x-xc)/t1);
        } else {
                y = a+b*x;
        }
        // End of editable part
}
```

Fit the Curve

Press **Ctrl + Y** to bring up NLFit dialog with the graph window active. Select the *piecewise* function we defined and initialize the parameter values:

xc: 1

a: 1

b: -1

t1: 0.1

Click **Fit** button to generate the results:

xc: 0.24

a: 36.76585

b: -24.62876

t1: 0.04961

Note that this function is sensitive to *xc* and *t1*, different initial values could generate different results.

Fit Curve Through Certain Points

Contents

Summary

This tutorial shows you three methods to force a fit curve to go thru a particular point. Choice of method depends on function expression and the data point you want the fit curve to go through.

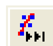
What you will learn

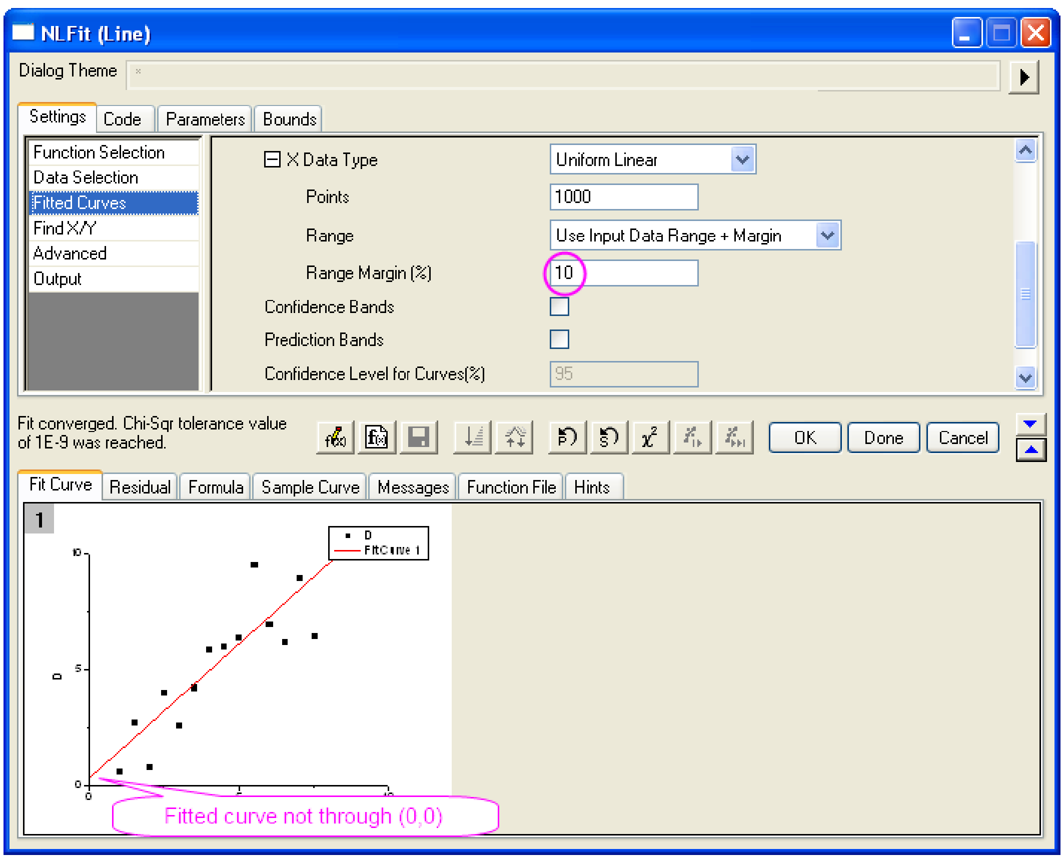
- Learn different methods to force a curve to go through a point.
- Fix fitting parameters during nonlinear fitting.
- Use general linear constraints in nonlinear fitting.
- Fit with weights.

Example and Steps

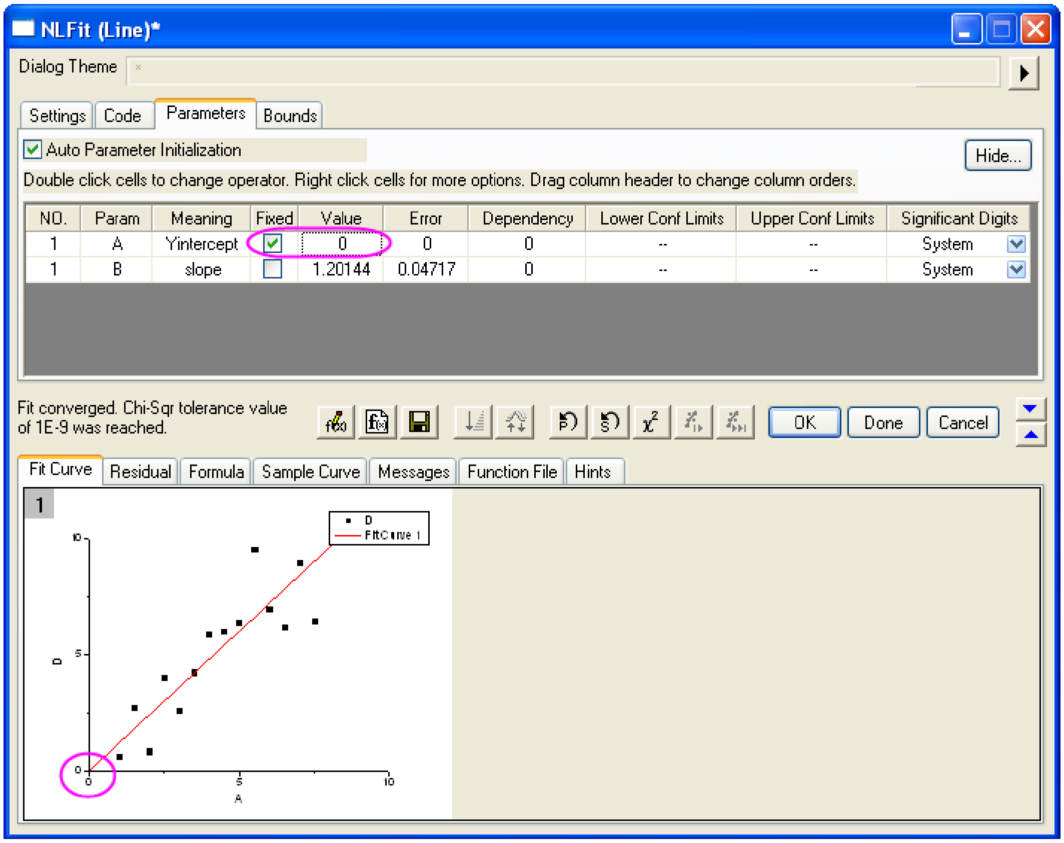
Fixing Function Parameters

This method works only when the point you want to fit through is related to a function parameter. One typical example: force the fitted line to go through the origin point, (0, 0), when fitting a straight line, *y* = *a* + *b* * *x*. In this particular case we know that if we let *a* = 0, the line will go through (0,0).

1. Import the data "*\Samples\Curve Fitting\Linear Fit.dat*" into an Origin worksheet.
2. Highlight one of the Y column, column D for example, and select **Analysis: Fitting: Nonlinear Curve Fit** to bring up the **NLFit** dialog.
3. Choose the ***Line*** function after selecting the ***Polynomial*** category.
4. Click the **Fitted Curves** page on **Settings** tab. Under the **X Data Type** branch, make sure the **Range** option is ***Use Input Data Range + Margin***, and then enter **10** in the **Range Margin(%)** edit box. This option will lengthen the fitted curve.
5. Click the **Fit until converged** button . You can see from the **Fit Curve** tab that the curve does not go through the origin point.



6. Now go to the **Parameters** tab, check the **Fixed** checkbox for parameter **A** and fix the value to **0**. Click the **Fit until converged** to fit the curve again. Now you can see the curve go through zero.



> **Note**: You can also use the **Fix Intercept** option in the **Linear Fit** dialog to force the linear fitted line to go through the origin point.

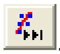
Use Linear Constraint

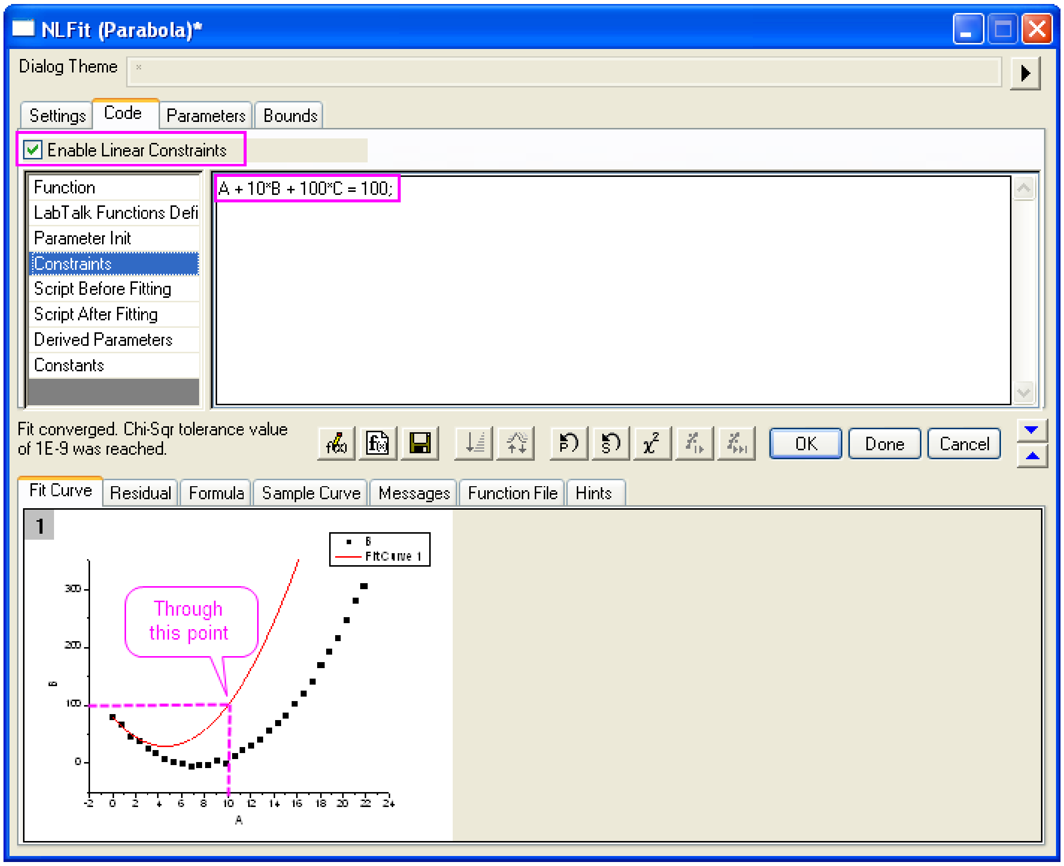
This method works when the fitting function is based on a LINEAR model, such as ***Line***, ***Parabola***, or ***Cubic***, etc.

We will show you how to force the fitted curve to go through a particular point by using linear constraint:

1. Import the data "*\Samples\Curve Fitting\Polynomial Fit.dat*" into Origin worksheet.
2. Highlight column B and press **Ctrl + Y** to bring up the **NLFit** dialog.
3. Select **Parabola** ($y = A + B * x + C * x^2$) from the **Polynomial** category. From the **Fit Curve** tab, we can see the initial value already fits the data very well.
4. Suppose we want to force the curve through (10,100). Substitute (10,100) to the fitting function ($y = A + B * x + C * x^2$). we then have 100 = A + 10 * B + 100 * C. We can use this equation as a general linear constraint condition. Select the **Constraints** page on **Code** tab. Check the **Enable Linear Constraints** checkbox, and enter the following expression into the edit box.

```
A + 10*B + 100*C = 100
```

5. Click the **Fit until converged** button . We can see the fitted curve deviates from the data points, but it goes through the specified point.



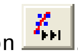
Use weighting

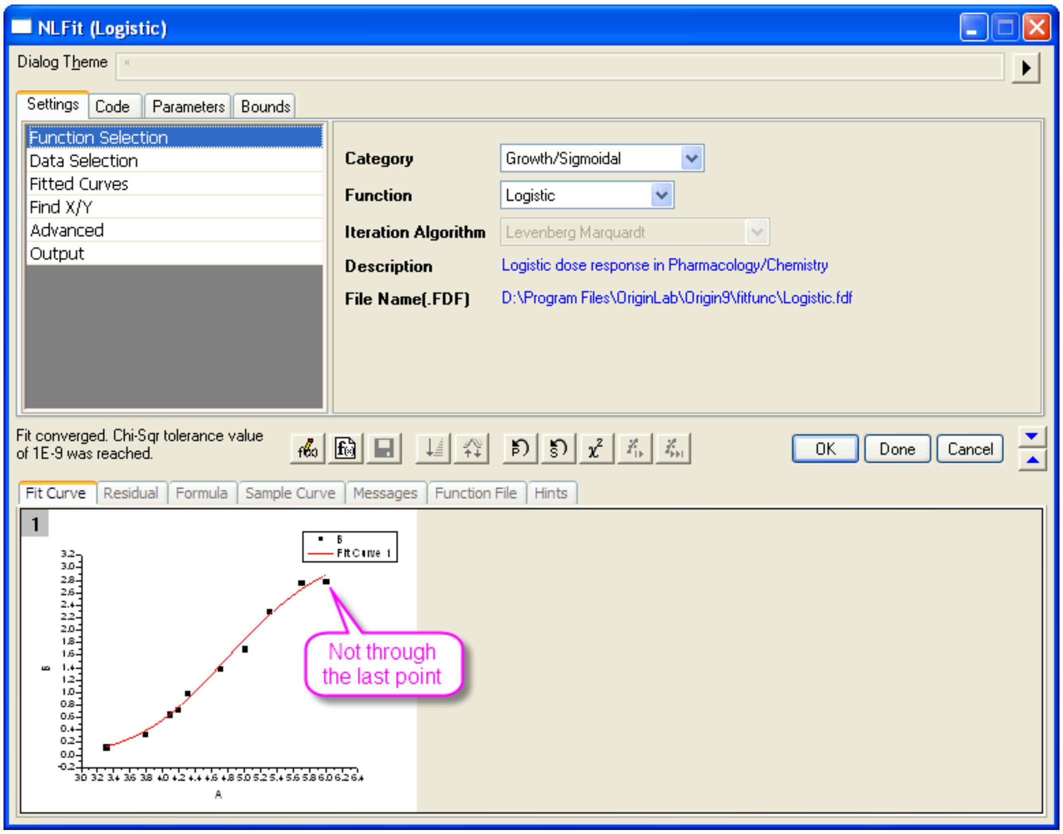
If the parameters are some eigenvalues such as upper or lower asymptotes, and your raw data includes the points you want to fit through, you can fit the curve by assigning larger weights to these particular points. This is not an analytical solution, but you can assign larger weights to reduce the error:

1. Prepare data by running the following script:

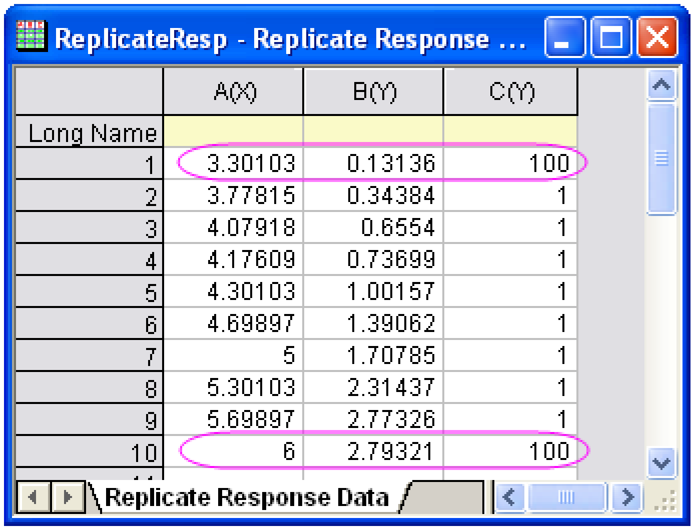
```
newbook;
string fname$ = system.path.program$;
fname$ += "Samples\Curve Fitting\Replicate Response Data.dat";
impasc fname$ options.PartImp.Partial:=1
options.PartImp.LastCol:=2;
wks.addcol();
col(a) = log(col(a)) + 5;
col(c)[1] = 100;
for(int ii = 2; ii < wks.maxrows; ii++)
{
    col(c)[ii] = 1;
}
```

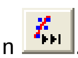
```
col(c)[wks.maxrows] = 100;
```

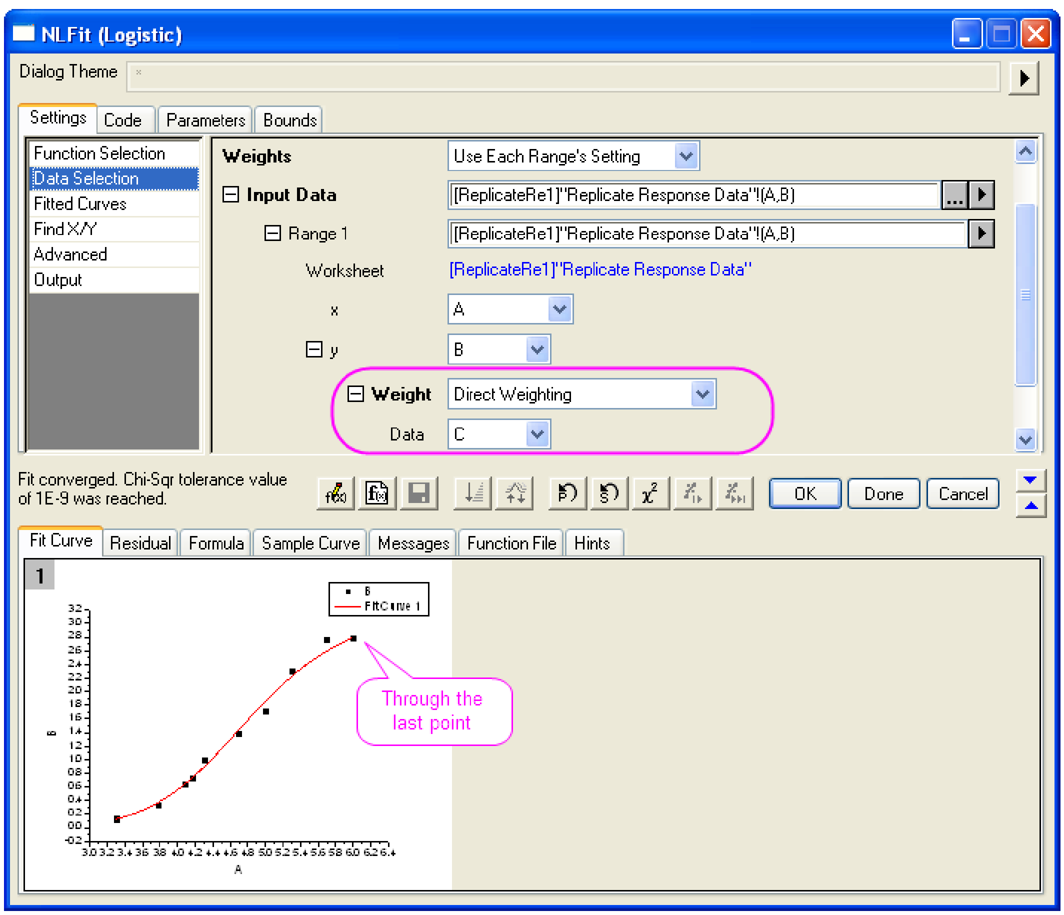
2. First, let's see how the fitted curve looks when there is no weighting. Highlight column B and bring up NLFit dialog from **Analysis: Fitting: Nonlinear Curve Fit**. Select the **Logistic** function from **Growth/Sigmoidal** category. Then click the **Fit until converged** button . From the **Fit Curve** tab, we can see the curve does not go through any points near the top.



3. Note that in the raw data worksheet, we have prepared column C and assigned large values for the first and last data points. If we use this column as weights, these two points will contribute more impact on the fitted curve and hence force the curve to go through these two points.



Now, activate the **Data Selection** page on the **Settings** tab. Expand the **Input Data** branch as below to expose the weighting option. Choose the **Direct Weighting** method and assign column **C** as the weighting dataset. Then click the **Fit until converged** button .



From the preview result, we can see that the fitted curve goes through the first and last data points.

Peak Fitting on Frequency Count Result

Summary

To know the location or scale parameters of a sample distribution, one can perform statistical tests on the data. However, you can also fit a probability density function on the binned data to get these values. This tutorial shows you how to estimate these parameters by curve fitting.

What you will learn

- Perform simple descriptive statistics.
- Perform frequency counts on dataset.
- Curve fitting on binned data.

Example and Steps

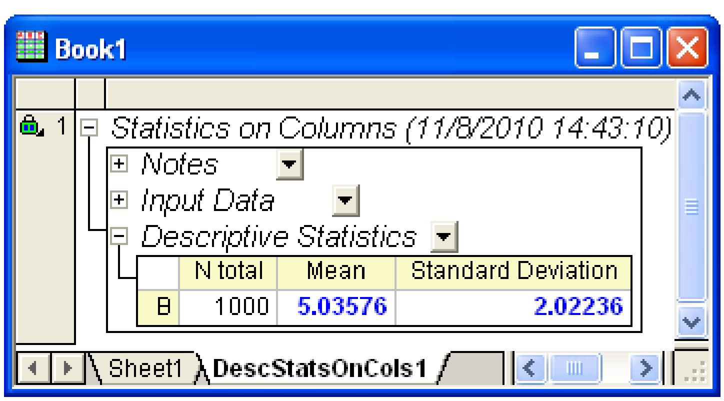
1. Run the following script to create sample dataset

```
newbook;
col(2) = normal(1000) * 2 + 5;
```

2. This script generates 1000 normally distributed points where *mean* ≈ *5* and *σ* ≈ *2*.

3. We can first perform simple descriptive statistics on this column to see the corresponding *Moments* output.

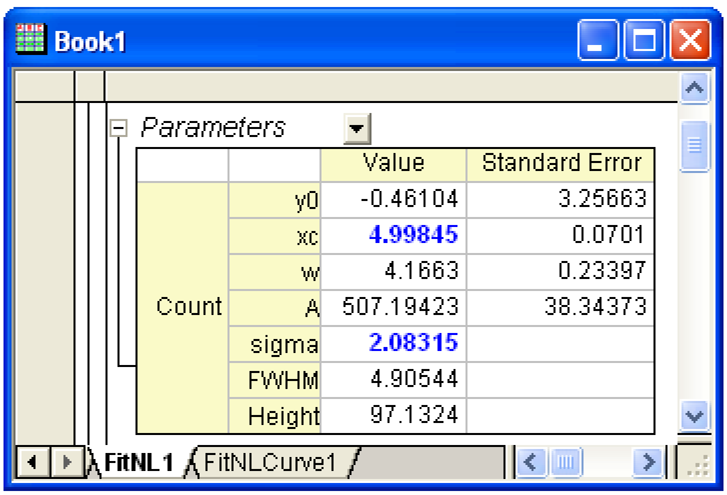
   Highlight the data column and select **Statistics: Descriptive Statistics: Statistics on Column** to open the dialog. Make sure the ***Mean*** and ***Standard Deviation*** checkboxes are selected. And the click **OK** to generate report.



From the report worksheet, we can see the Mean and Standard Deviation are very close to the value we just set.

4. Besides, you can also estimate the moments by fitting a proper probability density function on binned dataset. For example, highlight the source data column and select **Statistics: Description Statistics: Frequency Counts** from menu. This dialog will count the number of data points on specified bins.
   - o Expand the **Computation Control** branch, and make sure **Bin Size** radio button is check in **Step by** group. And then set the **Bin Size** to *0.5* .
   - o Make sure the **Bin Center** and ***Count*** check boxes under **Quantities to Compute** branch are selected. Then click **OK** to count the data.
5. In the frequency counts result sheet, the bin center value is set to X, while the bin counts is set to Y. You can fit the density function using these data.

   Highlight the *Counts* column on the Frequency Counts result worksheet, and press **Ctrl + Y** to open the **NLFit** dialog. Then select the ***Gauss*** function from the *Origin Basic Function* category. Leave other options as defaults and click the **Fit** button directly to output fitting report.



From the fitting report, we can see that the fitted *xc* and *sigma* are close to *5* and *2*.

Surface Fitting with Multiple Peaks

Origin provides several built-in surface fitting functions which can be used to perform fitting on 3D data. The surface fitting function is similar to the nonlinear fitting function.

Surface Fitting is only available in OriginPro.

Minimum Origin Version Required: OriginPro 9.0 SR0

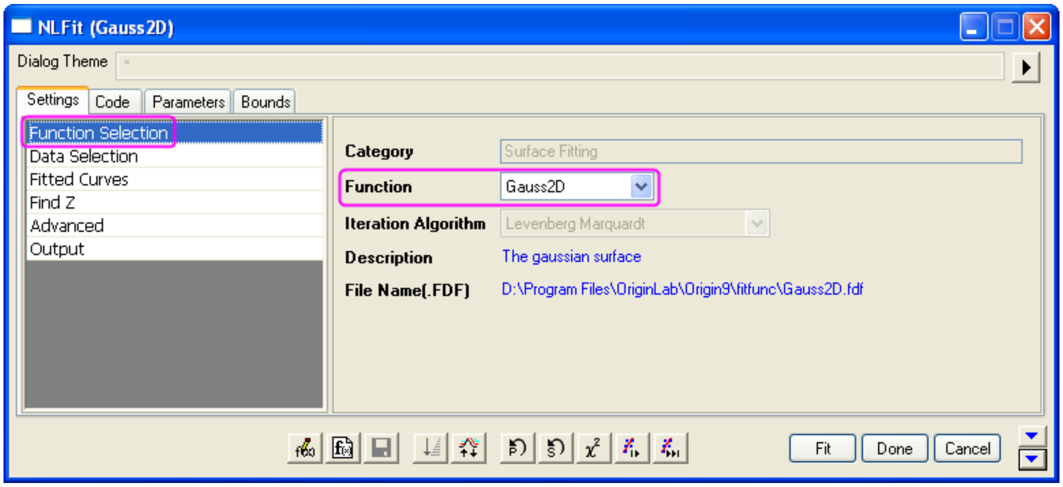
What you will learn

- How to do surface fitting on matrix data.
- How to fit the surface with multiple peaks.

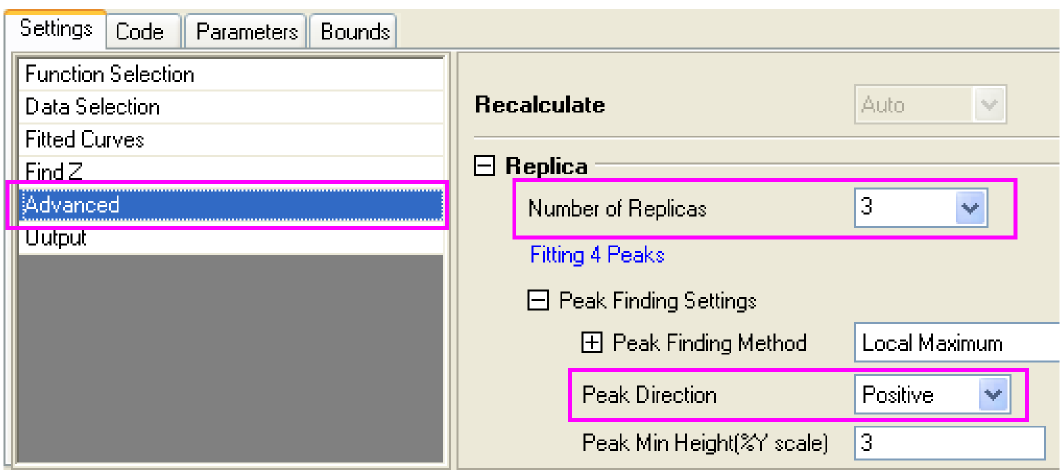
Steps

This tutorial is associated with the **Analysis - OriginPro: Surface Fitting (Pro Only)** folder in the Sample project (\Samples\Analysis.opj).

1. With the matrix sheet active, click **Analysis** on the main menu, and then click **Nonlinear Matrix Fit...** to open the NLFit dialog. (Alternatively, you can plot the matrix as a 3D surface or contour, and then select **Nonlinear Surface Fit...** to open the same dialog.)
2. Click **Function Selection**, select **Gauss2D** from the **Function** dropdown menu.



3. Click **Advanced**, set **Number of Replicas** to **3**, and set **Peak Direction** as **Positive**.



4. Click **Fit** to perform a multiple peak fit and generate a report worksheet with fitting results.

Fitting with a Piecewise Linear Function

Contents

Summary

In this tutorial we will show you how to define a piecewise fitting function consisting of two linear segments, perform a fit of the data using this fitting function, and calculate the intersection location for two linear segments from the fitting result.

Minimum Origin Version Required: Origin 8.6 SR0

What you will learn

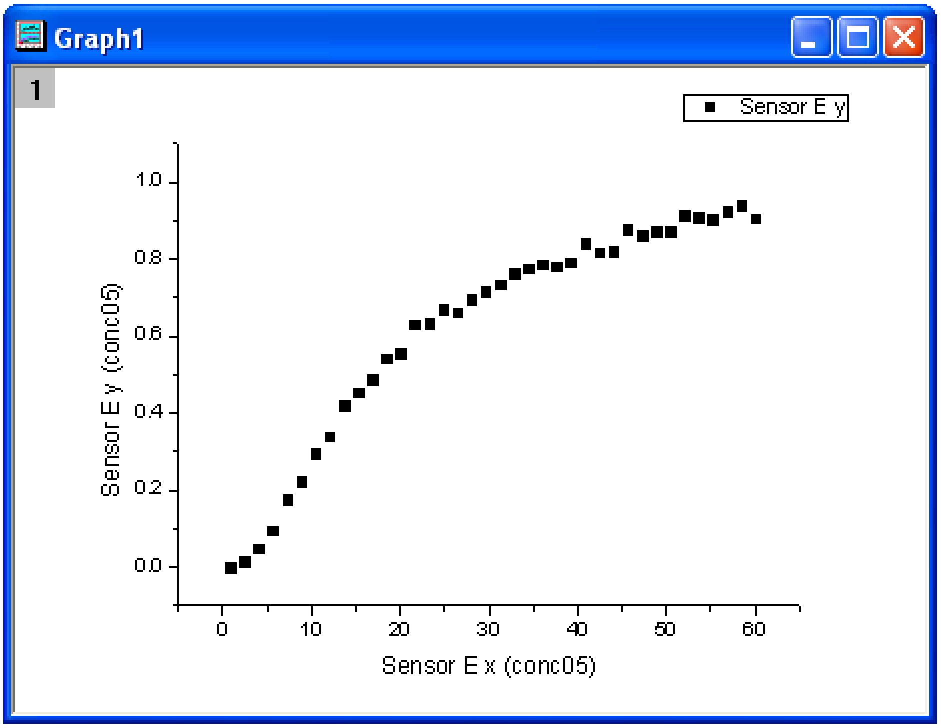
This tutorial will show you how to:

- Define a piecewise (conditional) fitting function.
- Auto initialize parameters.
- Calculate the intersection location of the piecewise fit lines.

Example and Steps

Import Data

1. Open a new workbook.
2. Click the **Import Single ASCII** button  to bring up the Open dialog. Browse to **\Samples\Curve Fitting** folder and select the file **Step01.dat**.
3. Right click on the **Sensor E x** column (column J), and select **Set As: X** from the context menu. Highlight **Sensor E y** column, and select **Plot: Symbol: Scatter** from Origin menu. The graph should look like:



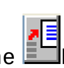
Define Fitting Function

From the above graph, the curve consists of two segments of lines. It can be fitted with a piecewise linear function. The function can be expressed as:

$$y = \begin{cases} \frac{y_1(x_3-x)+y_3(x-x_1)}{x_3-x_1}, & \text{if } x < x_3 \\ \frac{y_3(x_2-x)+y_2(x-x_3)}{x_2-x_3}, & \text{if } x \geq x_3 \end{cases}$$

where **x1** and **x2** are x values of the curve's endpoints and they are fixed during fitting, **x3** is the x value at the intersection of two segments, and **y1**, **y2**, **y3** are y values at $x_i, \ i = 1, 2, 3$ respectively.

The fitting function can be defined using the **Fitting Function Builder** tool.

1. Select **Tools: Fitting Function Builder** from Origin menu.
2. In the **Fitting Function Builder** dialog's **Goal** page, click **Next** button.
3. In the **Name and Type** page, select **User Defined** from **Select or create a Category** drop-down list, type **pwl2s** in the **Function Name** field, and select **Origin C** in **Function Type** group. And click **Next** button.
4. In the **Variables and Parameters** page, type **x1,y1,x2,y2,x3,y3** in the **Parameters** field. Click **Next** button.
5. In the **Origin C Fitting Function** page, click the button on the right of the **Function Body** edit box and define the fitting function in **Code Builder** as follows.

```
if( x < x3 )
    y = (y1*(x3-x)+y3*(x-x1))/(x3-x1);
```

```
    else
       y = (y3*(x2-x)+y2*(x-x3))/(x2-x3);
```

6. Click **Compile** button to compile the function body. Then click **Return to Dialog** button. Click **Next** button.

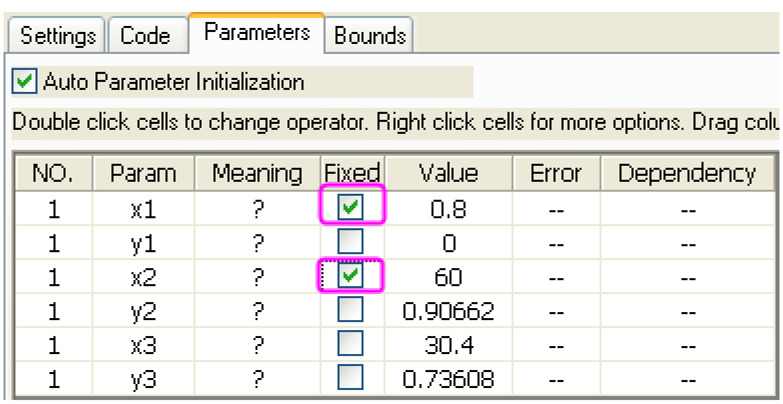
7. In the **Parameter Initialization Code** page, click the button on the right of the **Initialization Code** edit box and initialize the fitting parameters in **Code Builder** as follows.

```
    int n1, n2, n3;

    x_data.GetMinMax( x1, x2, &n1, &n2 );
    x3 = x1 + (x2 - x1)/2;

    y1 = y_data[n1];
    y2 = y_data[n2];

    vector vd;
    vd = abs( x_data - x3 );
    double xta, xtb;
    vd.GetMinMax( xta, xtb, &n3 );
    y3 = y_data[n3];
```

8. Click **Compile** button to compile it. Then click **Return to Dialog** button. Click **Finish** button.

Fit the Curve

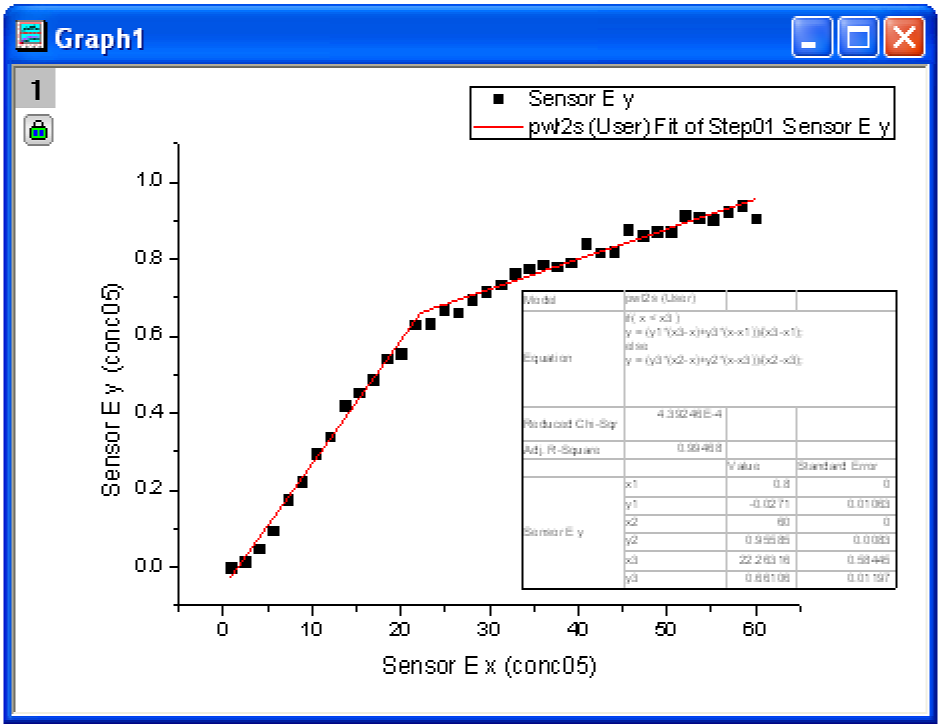
1. Select **Analysis: Fitting: Nonlinear Curve Fit** from Origin menu. In the **NLFit** dialog, select **Settings: Function Selection**, in the page select **User Defined** from the **Category** drop-down list and **pwl2s** function from the **Function** drop-down list.
2. In the **NLFit** dialog, select **Parameters** tab, and fix parameters **x1**, **x2** as shown in the dialog.



3. Click **Fit** button to fit the curve.

Fitting Results

The fitted curve should look like:



Fitted Parameters are shown as follows.

Parameter	Value	Standard Error
x1	0.8	0
y1	-0.0271	0.01063
x2	60	0
y2	0.95585	0.0083
x3	22.26316	0.58445
y3	0.66106	0.01197

Thus the intersection point for the two segments is (22.26316, 0.66106).

Note that fitting with a piecewise linear function for more than two segments can be done in a similar way.

Fitting Integral Function with a Sharp Peak

Contents

Summary

In this tutorial, we will show you how to define an integral fitting function with a sharp peak in the integral function, and perform a fit of the data using this fitting function.

Because the integral function contains a sharp peak, the integral should be performed in three segments so that the sharp peak can be integrated in a narrow interval.

Minimum Origin Version Required: Origin 9.0 SR0

What you will learn

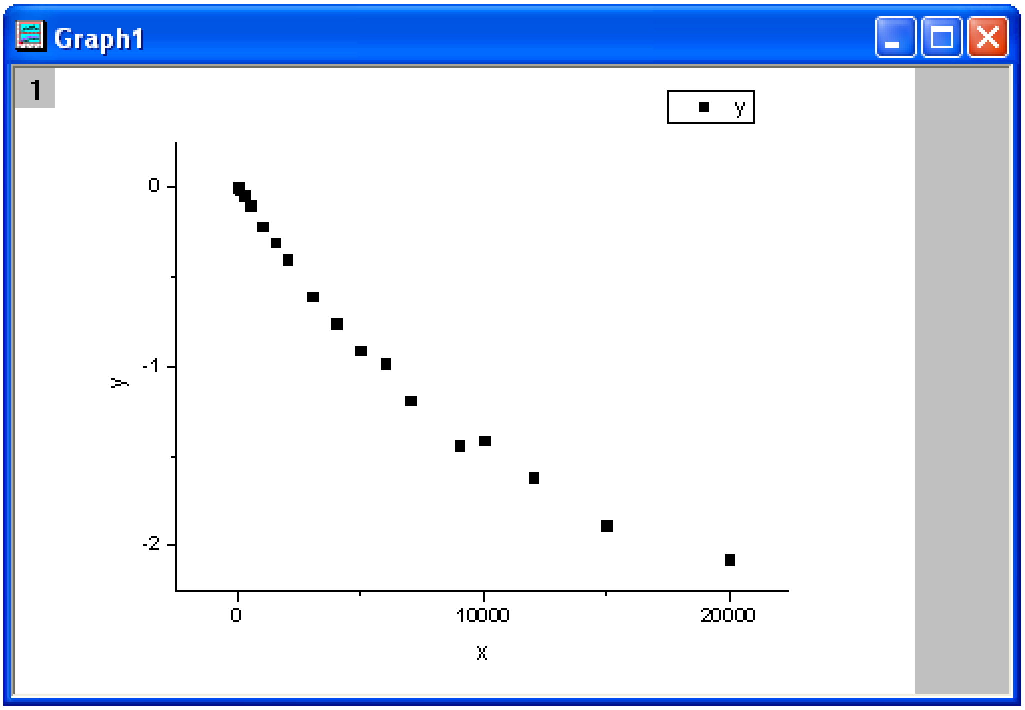
This tutorial will show you how to:

- Define an integral fitting function.
- Integrate a function with a sharp peak.
- Divide the integral interval into several segments.

Example and Steps

Import Data

1. Open a new workbook.
2. Copy data in Sample Data to the workbook.
3. Highlight column B, and select **Plot: Symbol: Scatter** from Origin menu. The graph should look like:



Define Fitting Function

The fitting integral function is described as follows:

$$y = \log\left(\int_0^1 \frac{1}{\sqrt{2\pi}b} e^{-\frac{(t-a)^2}{2b^2} - xt}\, dt\right)$$

where a and b are parameters in the fitting function.

Initial parameters are: a=1e-4, b=1e-4. Note that the integral function contains a peak whose center is about a and width is 2b. And the peak's width (2e-4) is very narrow compared with the integral interval [0,1]. To make sure it is integrated correctly at the neighborhood of the peak center, the integral interval [0,1] is divided into three segments: [0,a-5*b], [a-5*b,a+5*b], [a+5*b,1]. It is integrated in each segment, and then the three integrals are summed up.

The fitting function can be defined using the **Fitting Function Builder** tool.

1. Select **Tools: Fitting Function Builder** from Origin menu.
2. In the **Fitting Function Builder** dialog's **Goal** page, click **Next** button.
3. In the **Name and Type** page, select **User Defined** from **Select or create a Category** drop-down list, type **fintpeak** in the **Function Name** field, and select **Expression** in **Function Type** group, check **Include Integration During Fitting** check box. And click **Next** button.
4. In the **Integrand** page, type **myint** in **Integrand Name** edit box, **t** in **Integration Variable** edit box and **a, b, x** in **Arguments** edit box. Type the following script in **Integrand Function** box.

```
return 1/(sqrt(2*pi)*b)*exp(-(t-a)^2/(2*b^2)-x*t);
```

5. And click **Next** button.
6. In the **Variables and Parameters** page, type **a, b** in the **Parameters** field. Click **Next** button.

7. In the **Expression** page, click **Parameters** tab, and set **Initial Value** for parameters a and b to 1e-4, click **Integrand** tab, and set **Value** for **Lower Limit** and **Upper Limit** to 0 and 1, **Value** for **a**, **b**, **x** to a, b, x respectively.
8. In the **Expression** page, click **Insert** button. In the **Quick Check** group, type 0 in **x=** edit box, click **Evaluate** button, and it shows y=9.3e-21. This implies that the peak is not integrated correctly because y should approach 1 for x=0. Divide the integral into three segments, and type following script in **Function Body** box.

```
integral(myint, 0, a-5*b, a ,b ,x)+integral(myint, a-5*b,
a+5*b, a ,b ,x)+
integral(myint, a+5*b, 1, a ,b ,x)
```

9. Click **Evaluate** button again, and it shows y=0.84, hence it is clear that the peak is integrated correctly this time.

10. In the **Expression** page, update the script in **Function Body** box as follows.

```
log(integral(myint, 0, a-5*b, a ,b ,x)+integral(myint, a-5*b,
a+5*b, a ,b ,x)
+integral(myint, a+5*b, 1, a ,b ,x))
```

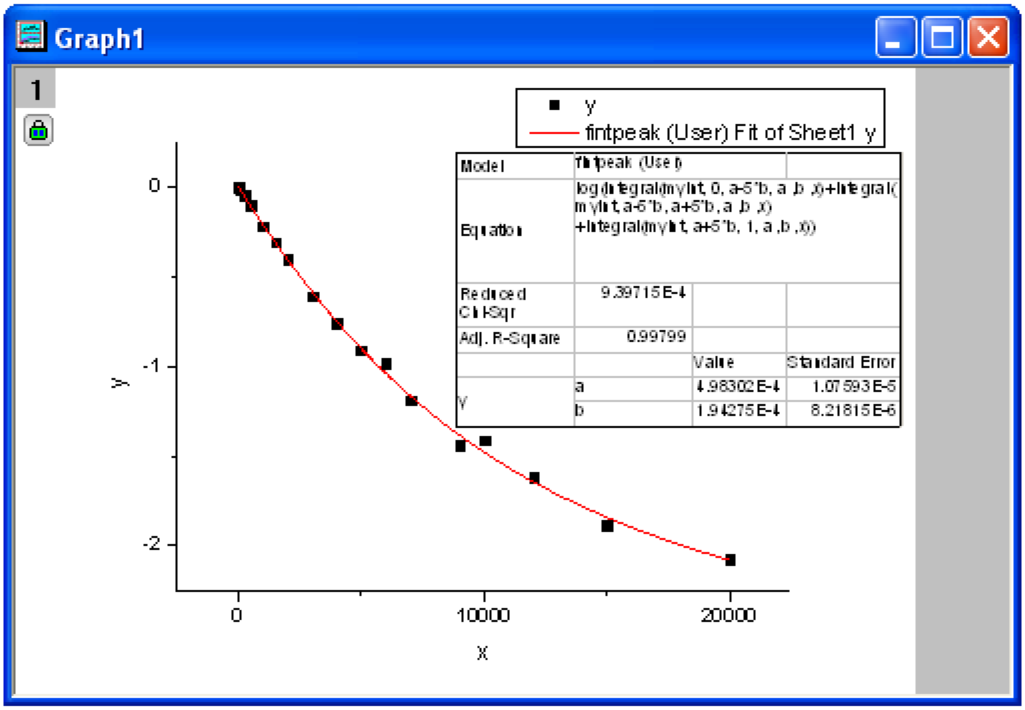
11. Click **Finish** button.

Fit the Curve

1. Select **Analysis: Fitting: Nonlinear Curve Fit** from Origin menu. In the **NLFit** dialog, select **Settings: Function Selection**, in the page select **User Defined** from the **Category** drop-down list and **fintpeak** function from the **Function** drop-down list. **Note** that initial parameters have been set during defining the fitting function.
2. Click **Fit** button to fit the curve.

Fitting Results

The fitted curve should look like:



Fitted Parameters are shown as follows:

Parameter	Value	Standard Error
a	4.98302E-4	1.07593E-5
b	1.94275E-4	8.21815E-6

The **Adj. R-Square** is 0.99799. Thus the fitting result is very good.

Sample Data

x	y
0	-0.00267
60	-0.01561
240	-0.05268
500	-0.10462
1000	-0.22092
1500	-0.31004
2000	-0.40695
3000	-0.61328
4000	-0.75884
5000	-0.9127
6000	-0.98605
7000	-1.18957
9000	-1.43831
10000	-1.41393
12000	-1.61458
15000	-1.88098

20000	-2.07792

Fitting with Convolution of Two Functions

Contents

Summary

In this tutorial, we will show you how to define a convolution of two functions, and perform a fit of the data with non-evenly spaced X using this fitting function.

Minimum Origin Version Required: Origin 9.0 SR0

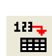
What you will learn

This tutorial will show you how to:

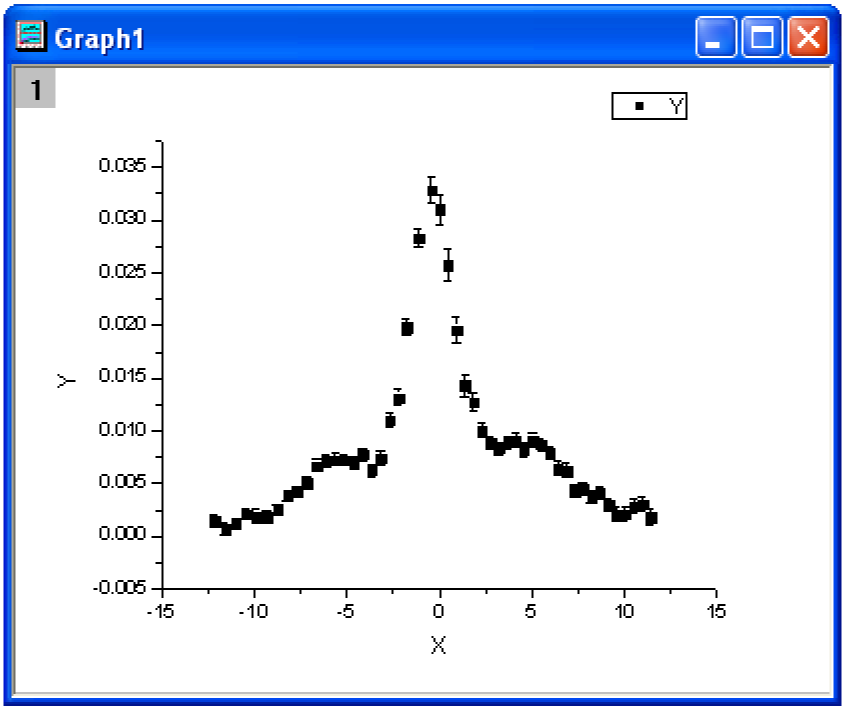
- Sample a function.
- Calculate a convolution of two functions.
- Define constants in the fitting function.
- Pad zeroes before the convolution.
- Interpolate the convolution result for non-evenly spaced X.
- Use a parameter to balance the speed and the precision.
- Use Y Error Bar as weight.

Example and Steps

Import Data

1. Open a new workbook.
2. Click the **Import Single ASCII** button  to bring up the Open dialog. Browse to **\Samples\Curve Fitting** folder and select the file **ConvData.dat**. Note that column A is not evenly spaced. We can use LabTalk **diff** function to verify it.

3. Right click on column C, and select **Set As: Y Error** from the short-cut menu. Highlight column B and C, and select **Plot: Symbol: Scatter** from the Origin menu. The graph should look like:



Define Fitting Function

The fitting function is a convolution of two functions. It can be described as follows:

$$y = y_0 + b_1 x + \frac{b_2 A_2}{w_2\sqrt{\pi/2}} e^{-\frac{2(x-x_{c2})^2}{w_2^2}} + (f \, * \, g)(x)$$

where

$$f(x) = \frac{s}{\pi} \cdot \frac{\tau_L x_0^2 (x_L^2 - x_0^2)}{(x-x_{c1})\tau_L((x-x_{c1})^2 - x_L^2)^2 + ((x-x_{c1})^2 - x_0^2)^2},$$

$$g(x) = \frac{1}{w_1\sqrt{\pi/2}} e^{-\frac{2x^2}{w_1^2}} .$$

And x_0, x_L, τ_L, s, y_0, b_1 and b_2 are fitting parameters. w_1, x_{c1}, w_2, x_{c2} and A_2 are constants in the fitting function.

The fitting function can be defined using the **Fitting Function Builder** tool.

1. Select **Tools: Fitting Function Builder** from Origin menu.
2. In the **Fitting Function Builder** dialog's **Goal** page, click **Next**.
3. In the **Name and Type** page, select **User Defined** from **Select or create a Category** drop-down list, type **convfunc** in the **Function Name** field, and select **Origin C** in **Function Type** group. And click **Next**.

4. In the **Variables and Parameters** page, type **x0,xL,tL,s,y0,b1,b2** in the **Parameters** field, **w1,xc1,w2,xc2,A2** in the **Constants** field. Click **Next**.
5. In the **Origin C Fitting Function** page, set initial parameters as follows:

```
x0 = 3.1
xL = 6.3
tL = 0.4
s = 0.14
y0 = 1.95e-3
b1 = 2.28e-5
b2 = 0.2
```

6. Click **Constants** tab, set constants as follows:

```
w1 = 1.98005
xc1 = -0.30372
w2 = 5.76967
xc2 = 3.57111
A2 = 9.47765e-2
```

7. Click the button on the right of the **Function Body** edit box and define the fitting function in **Code Builder** as follows:

8. Include header files,

```
#include <ONLSF.H>
#include <fft_utils.h>
```

9. Define the function body

```
  NLFitContext *pCtxt = Project.GetNLFitContext();
  if ( pCtxt )
  {
    // Vector for the output in each iteration.
    static vector vX, vY;

    static int nSize;

    BOOL bIsNewParamValues = pCtxt->IsNewParamValues();

    // If parameters were updated, we will recalculate the
convolution result.
    if ( bIsNewParamValues )
    {
      //Sampling Interval
      double dx = 0.05;
      vX.Data(-16.0, 16.0, dx);
      nSize = vX.GetSize();

      vector vF, vG, vTerm1, vTerm2, vDenominator, vBase,
vAddBase;

      double Numerator = tL * x0^2 * (xL^2 - x0^2);
      vTerm1 = ( (vX - xc1) * tL * ( (vX - xc1)^2 - xL^2 ) )^2;
```

```
      vTerm2 = ( (vX - xc1)^2 - x0^2 )^2;
      vDenominator = vTerm1 + vTerm2;

      //Function f(x)
      vF = (s/pi) * Numerator / vDenominator;

      //Function g(x)
      vG =  1/(w1*sqrt(pi/2))*exp(-2*vX^2/w1^2);

      //Pad zeroes at the end of f and g before convolution
      vector vA(2*nSize-1), vB(2*nSize-1);
      vA.SetSubVector( vF );
      vB.SetSubVector( vG );

      //Perform circular convolution
      int iRet = fft_fft_convolution(2*nSize-1, vA, vB);

      //Truncate the beginning and the end
      vY.SetSize(nSize);
      vA.GetSubVector( vY, floor(nSize/2), nSize +
floor(nSize/2)-1 );

      //Baseline
      vBase = (b1*vX + y0);
      vAddBase =  b2 * A2/(w2*sqrt(pi/2))*exp( -2*(vX-
xc2)^2/w2^2 );

      //Fitted Y
      vY = dx*vY + vBase + vAddBase;
    }

    //Interpolate y from x for the fitting data on the
convolution result.
    ocmath_interpolate( &x, &y, 1, vX, vY, nSize );
  }
```

10. Click **Compile** button to compile the function body. And click the **Return to Dialog** button.

11. Click **Evaluate** button, and it shows y=0.02165 at x =1. And this indicates the defined fitting function is correct. Click **Next**.

12. Click **Next**. In the **Bounds and General Linear Constraints** page, set the following bounds:

```
0 < x0 < 7
0 < xL < 10
0 < tL < 1
0 <= s <= 5
0 < b2 <= 3
```

13. Click **Finish**.

Fit the Curve

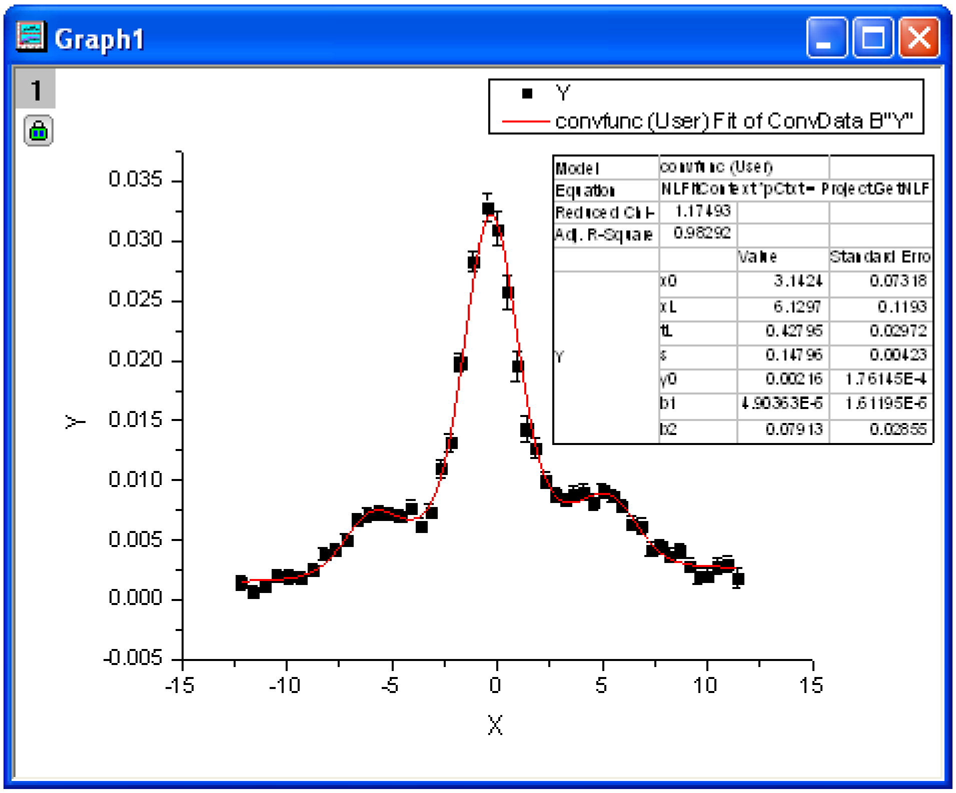
1. Select **Analysis: Fitting: Nonlinear Curve Fit** from the Origin menu. In the **NLFit** dialog, select **Settings: Function Selection**, in the page select **User Defined** from the **Category** drop-down

list and **convfunc** function from the **Function** drop-down list. Note that **Y Error Bar** is shown in the active graph, so column C is used as Y weight, and **Instrument** weighting method is chosen by default.

2. Click the **Fit** button to fit the curve.

Fitting Results

The fitted curve should look like:



Fitted Parameters are shown as follows:

Parameter	Value	Standard Error
x0	3.1424	0.07318
xL	6.1297	0.1193
tL	0.42795	0.02972
s	0.14796	0.00423
y0	0.00216	1.76145E-4
b1	4.90363E-5	1.61195E-5
b2	0.07913	0.02855

Note that you can set a smaller value for **dx** in the fitting function body, the result may be more accurate, but at the same time it may take a longer time for fitting.

Parameter Initialization for Rational Functions

Contents

Summary

In this tutorial, we will show you how to calculate initial parameters for rational fitting functions using the multiple linear regression method, and perform a fit of the data using calculated initial parameters.

Minimum Origin Version Required: Origin 9.0 SR0

What you will learn

This tutorial will show you how to:

- Calculate initial parameters for rational fitting functions.
- Perform multiple linear regression using Origin C code.

Example and Steps

Algorithm

In this tutorial, we will use the following rational function as an example:

$$y = \frac{a + bx + cx^2}{1 + dx + ex^2}$$

where **x** is the independent variable, **y** is the dependent variable, and **a**, **b**, **c**, **d**, **e** are fitting parameters.

Multiplying both sides by the denominator on the right side yields:

$$y + dxy + ex^2y \;=\; a + bx + cx^2$$

and the equation can be expressed as:

$$y \;=\; a + bx + cx^2 - dxy - ex^2y$$

Substituting fitting data $(x_i, y_i) \;\; i = 1 \ldots N$ into the equation gives:

$$\begin{cases} a + bx_1 + cx_1^2 - dx_1y_1 - ex_1^2y_1 = y_1 \\ a + bx_2 + cx_2^2 - dx_2y_2 - ex_2^2y_2 = y_2 \\ \vdots \\ a + bx_N + cx_N^2 - dx_Ny_N - ex_N^2y_N = y_N \end{cases}$$

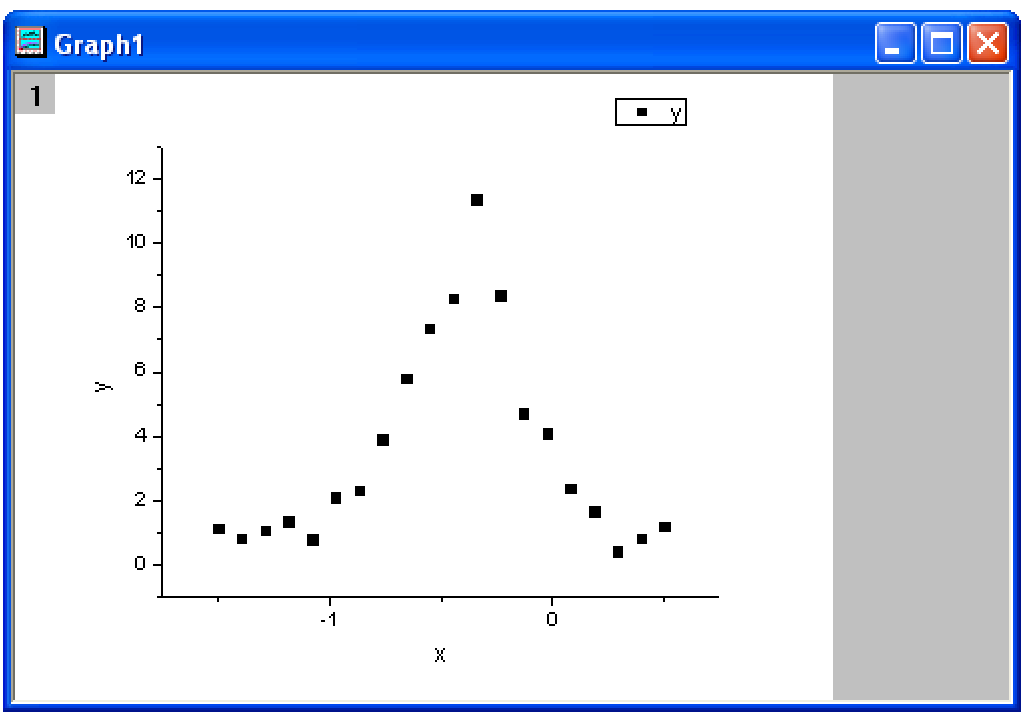
Hence estimating initial parameters for a rational polynomial fitting function becomes a multiple linear regression problem with linear coefficients **a**, **b**, **c**, **d**, **e**.

$$\begin{bmatrix} 1 & x_1 & x_1^2 & -x_1y_1 & -x_1^2y_1 \\ 1 & x_2 & x_2^2 & -x_2y_2 & -x_2^2y_2 \\ \vdots \\ 1 & x_N & x_N^2 & -x_Ny_N & -x_N^2y_N \end{bmatrix} \begin{bmatrix} a \\ b \\ c \\ d \\ e \end{bmatrix} = \begin{bmatrix} y_1 \\ y_2 \\ \vdots \\ y_N \end{bmatrix}$$

Origin provides a function ocmath_multiple_linear_regression in Origin C for multiple linear regression, which can be called in initialization code.

Import Data

1. Open a new workbook.
2. Copy data in Sample Data to the workbook.
3. Highlight column B, and select **Plot: Symbol: Scatter** from Origin menu. The graph should look like:



Define Fitting Function and Initialize Parameters

The fitting function can be defined using the **Fitting Function Builder** tool.

1. Select **Tools: Fitting Function Builder** from Origin menu.
2. In the **Fitting Function Builder** dialog's **Goal** page, click the **Next** button.
3. In the **Name and Type** page, select **User Defined** from the **Select or create a Category** drop-down list, type **rationalfunc** in the **Function Name** field, and select **Expression** in the **Function Type** group. Click the **Next** button.
4. In the **Variables and Parameters** page, type **a, b, c, d, e** in the **Parameters** field. Click the **Next** button.
5. In the **Expression** page, type the following script in the **Function Body** box.

```
(a+b*x+c*x^2)/(1+d*x+e*x^2)
```

6. Click the **Evaluate** button, and it shows y=1 at x=1, hence this implies the expression is correct. Click the **Next** button.

7. In the **Parameter Initialization Code** page, click the **Open Code Builder** button  to the right of the **Initialization Code** box, and initialize fitting parameters as follows, in terms of the algorithm.

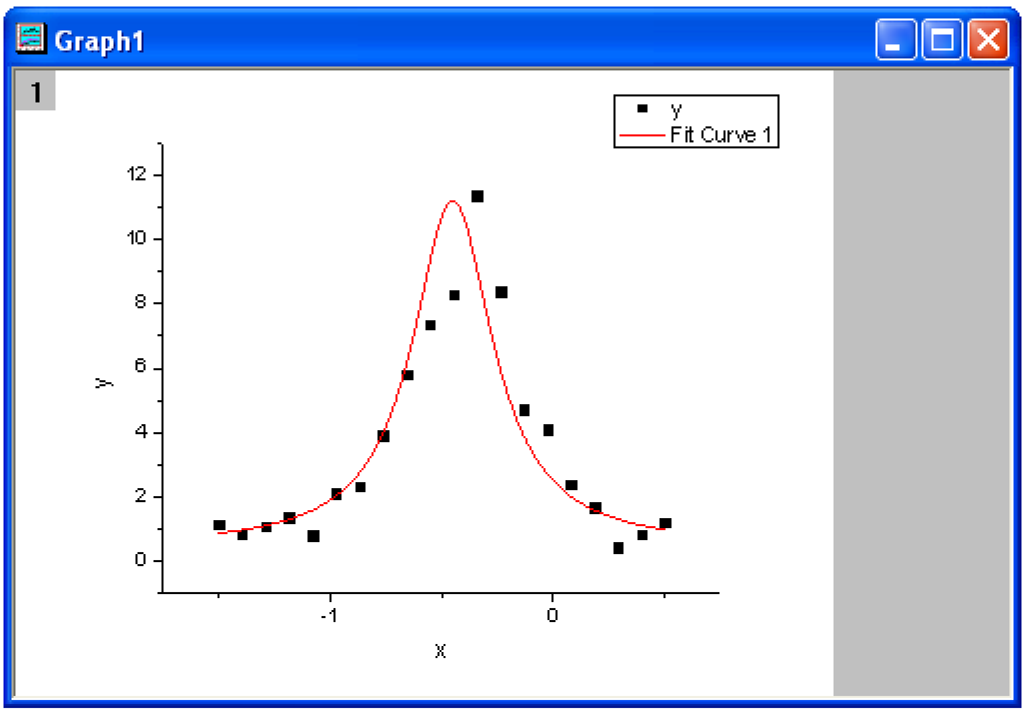
```
  UINT nOSizeN = x_data.GetSize(); //Number of points
  UINT nVSizeM = 5; //Number of parameters

  matrix mX(nOSizeN, 5);

  //Construct matrix for data points of independent variables
  vector vCa(nOSizeN), vCb, vCc, vCd, vCe;
  vCa = 1;
  mX.SetColumn( vCa, 0 );
  vCb = x_data;
  mX.SetColumn( vCb, 1 );
  vCc = x_data^2;
  mX.SetColumn( vCc, 2 );
  vCd = -x_data*y_data;
  mX.SetColumn( vCd, 3 );
  vCe = -x_data^2*y_data;
  mX.SetColumn( vCe, 4 );

  //Options for multiple linear regression
  LROptions stLROptions;
  stLROptions.UseReducedChiSq = 1;
  stLROptions.FixIntercept = 1; //Fix the intercept at 0.

  FitParameter stFitParameters[ 6 ]; // should be nVSizeM+1
  UINT nFitSize = nVSizeM + 1;

  int nRet = ocmath_multiple_linear_regression(mX, nOSizeN,
nVSizeM, y_data,
    NULL, 0, &stLROptions, stFitParameters, nFitSize );

  if( nRet == STATS_NO_ERROR )
```

```
{
  a = stFitParameters[1].Value;
  b = stFitParameters[2].Value;
  c = stFitParameters[3].Value;
  d = stFitParameters[4].Value;
  e = stFitParameters[5].Value;
}
```

8. Click the **Compile** button to compile the code. And click the **Return to Dialog** button. Click **Finish** to close the **Fitting Function Builder** dialog.

Fit the Curve

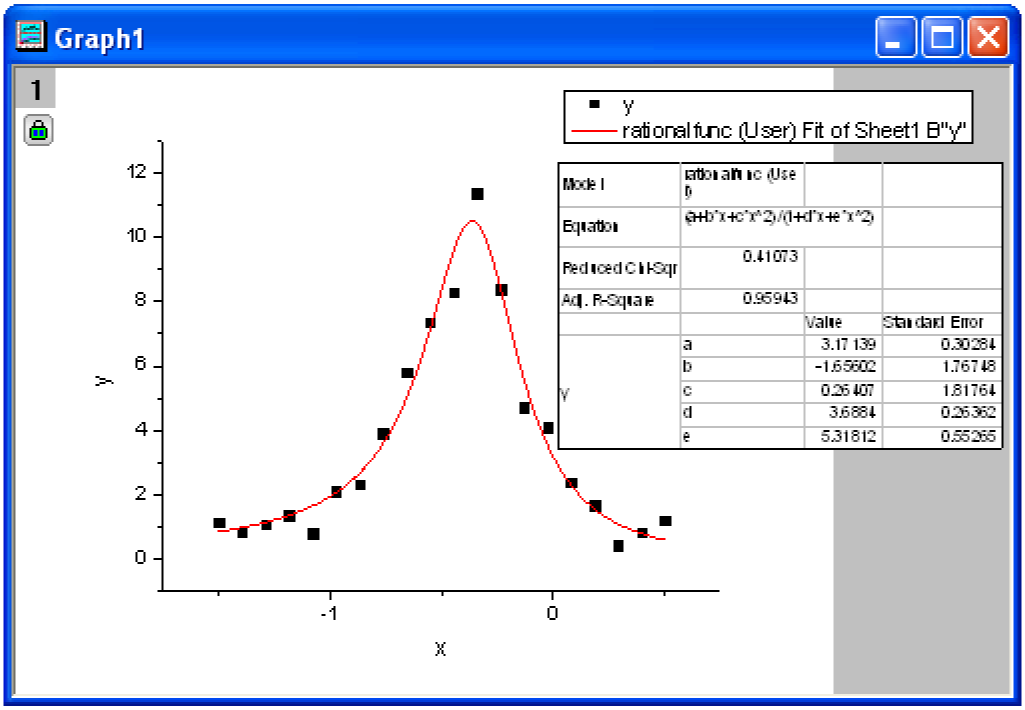
1. Select **Analysis: Fitting: Nonlinear Curve Fit** from Origin menu. In the **NLFit** dialog, select **Settings: Function Selection**, in the page select **User Defined** from the **Category** drop-down list and **rationalfunc** function from the **Function** drop-down list.
2. Click the **Parameters** tab. Initial parameters calculated from initialization code are listed in the dialog, and the fitting function curve for initial parameters is shown as follows. It seems that initial parameters from initialization code are very good.



3. Click the **Fit** button to fit the curve.

Fitting Results

The fitted curve should look like:



Fitted Parameters are shown as follows:

Parameter	Value	Standard Error
a	3.17139	0.30284
b	-1.65602	1.76748
c	0.26407	1.81764
d	3.6884	0.26362
e	5.31812	0.55265

Sample Data

x	y
-1.5	1.13173
-1.39474	0.8262
-1.28947	1.06999
-1.18421	1.37155
-1.07895	0.79569
-0.97368	2.11346
-0.86842	2.32006
-0.76316	3.9205
-0.65789	5.81904
-0.55263	7.38037
-0.44737	8.31272
-0.34211	11.39718
-0.23684	8.39808
-0.13158	4.7305

-0.02632	4.11105
0.07895	2.39105
0.18421	1.65394
0.28947	0.42953
0.39474	0.83337
0.5	1.18758

Note: You can also use this method to initialize parameters for other rational polynomial fitting functions.

Nonlinear Multiple Variables Fitting

Summary

Origin supports fitting functions with multiple dependent or independent variables. With the nonlinear fitting function, you can define multiple variables and separate them with semicolons. Since global fitting allows you to fit only one function at a time, this is a good way to defeat that limitation.

Origin ships with three built-in functions with multiple dependent and independent variables. These functions, available in the Multiple Variables category, are actually composites consisting of two ordinary functions. The GaussianLorentz function, for example, is a combination of the Gaussian and Lorentz functions, sharing *y0* and *xc*:

$$y_1 = y_0 + \frac{A_1}{w_1\sqrt{\pi/2}} e^{-2\frac{(x-x_c)^2}{w_1^2}}$$

$$y_2 = y_0 + \frac{2A_2}{\pi} \frac{w_2}{4(x-x_c)^2 + w_2^2}$$

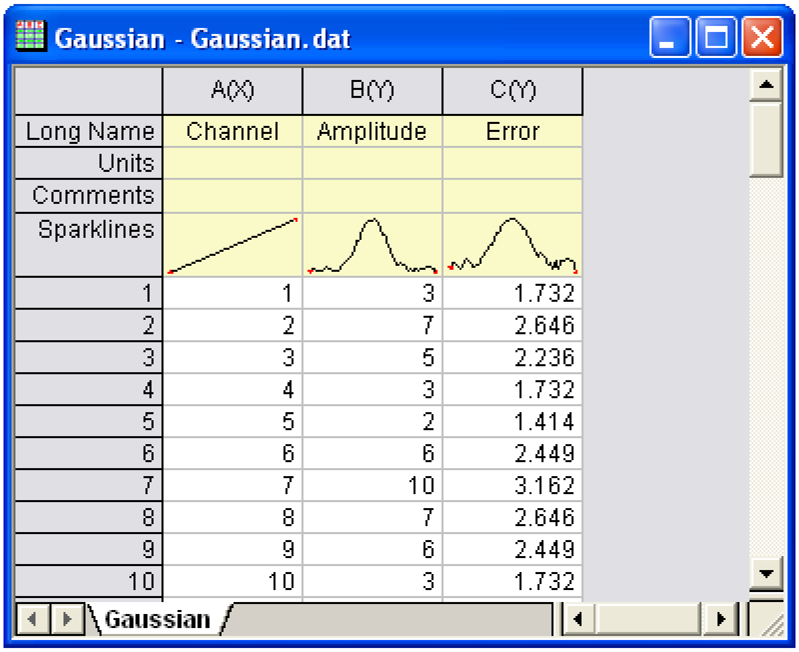
This tutorial will demonstrate how to fit such multi-variable functions.

What you will learn

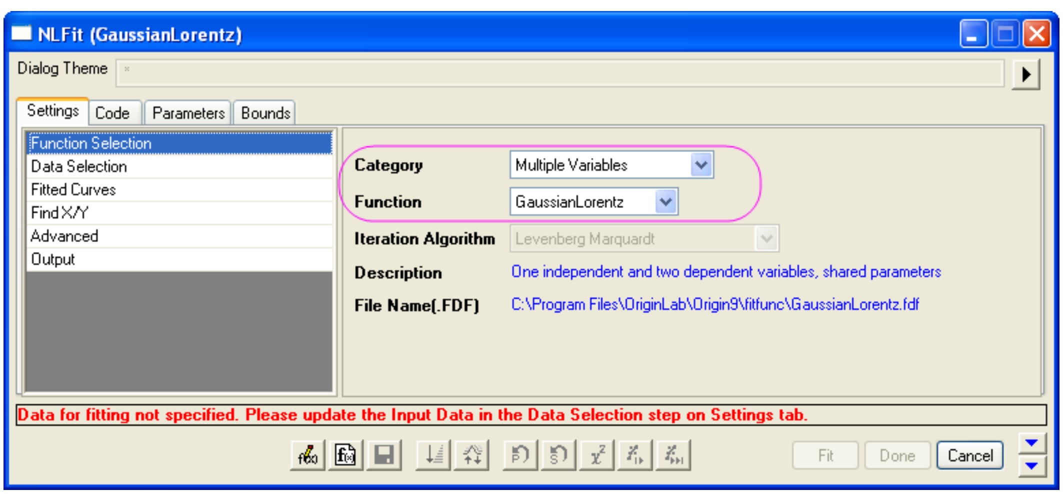
- Use Nonlinear Multiple Variables Fitting to fit a curve with two different functions.
- Assign data to fitting variables.

Steps

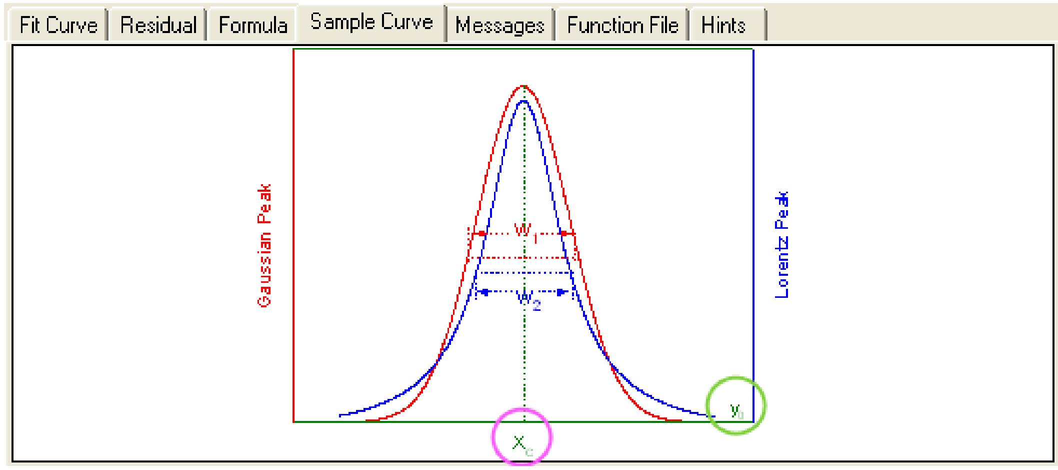
1. Start with a new project or create a new workbook and import the data file **\samples\curve fitting\Gaussian.dat**.



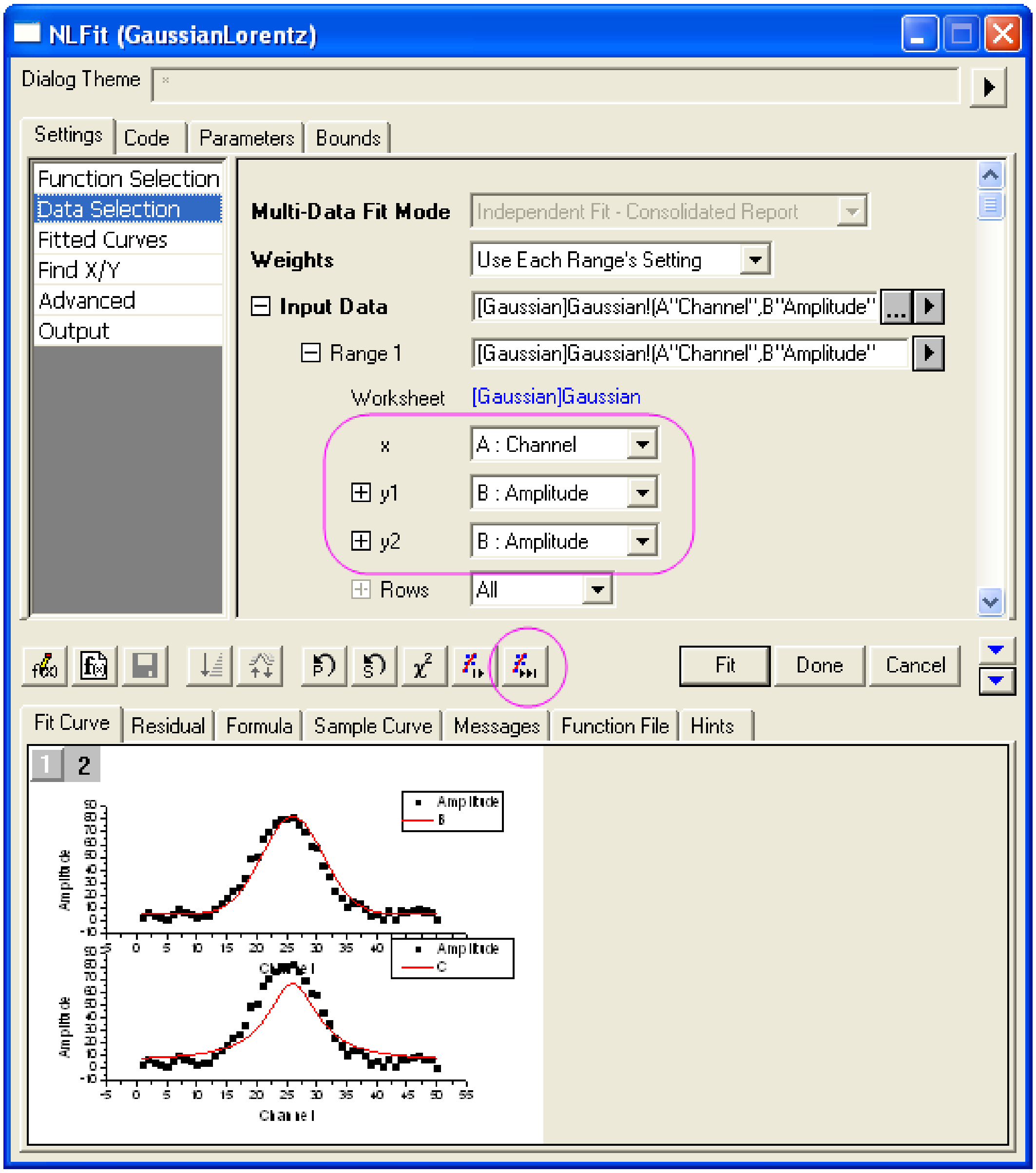
2. Highlight Column(A) and Column(B). In the main menu, click **Analysis**, then point to **Fitting**, and then click **Nonlinear Curve Fit**.
3. In the NLFit dialog?s left panel, select **Function Selection**. In the right panel, select **Multiple Variables** in the Category dropdown menu. In the Function dropdown menu, select **GaussianLorentz**.



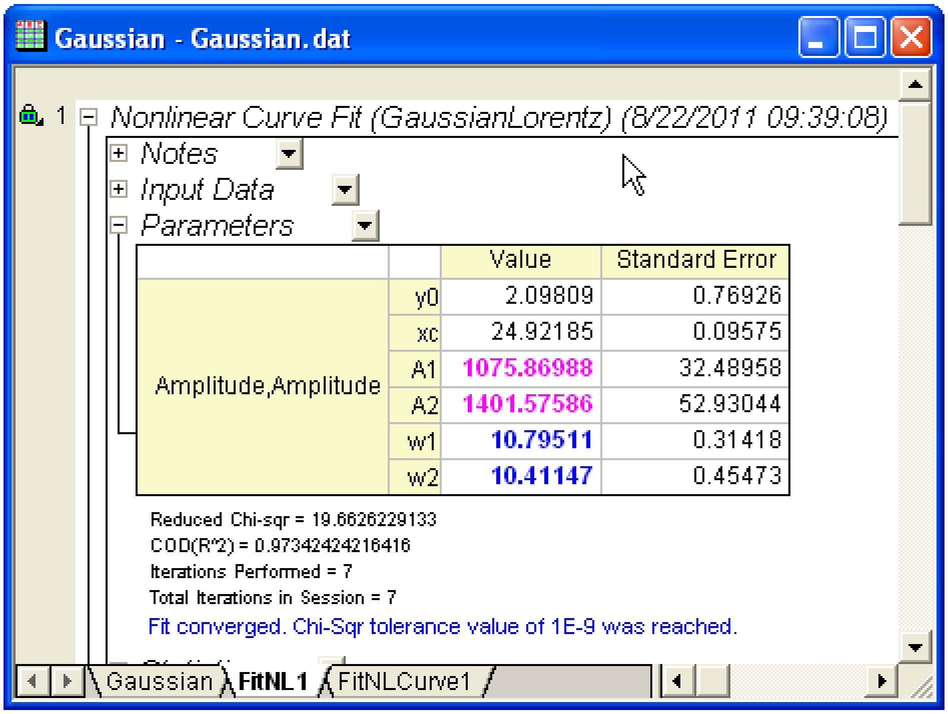
As you can see on the Sample Curve tab, the equations in this fitting function share the same parameters, *y0* and *xc*.



4. In the NLFit dialog?s left panel, select **Data Selection**. In the right panel, expand the **Range** node and assign data to the fitting variables. In this example, we have assigned column B to both *y1* and *y2*, which means that both expressions will fit the same dataset.



5. Click **Fit until converged** to fit, then **OK**. In the results sheet, compare parameters A and w, with the Gaussian and Lorentz functions sharing the same offset and peak center.



Fit Multiple Datasets by Fitting One and then Using Those Fit Parameters for Other Datasets

Summary

In some cases, you might have multiple datasets, and want to fit them using the user-defined function without parameter initial code. For increased efficiency, you can fit one and then apply the fit parameters for other datasets.

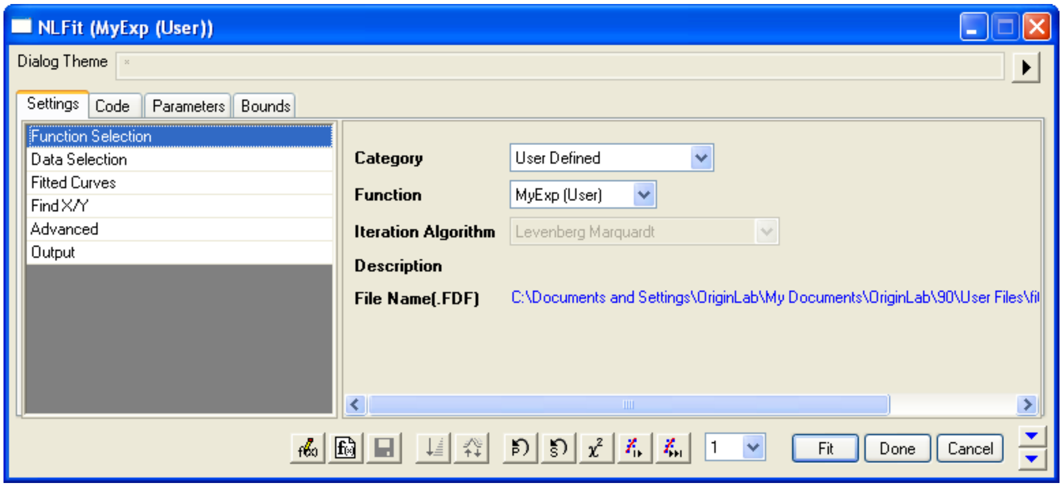
Minimum Origin Version Required: Origin 8.6

What you will learn

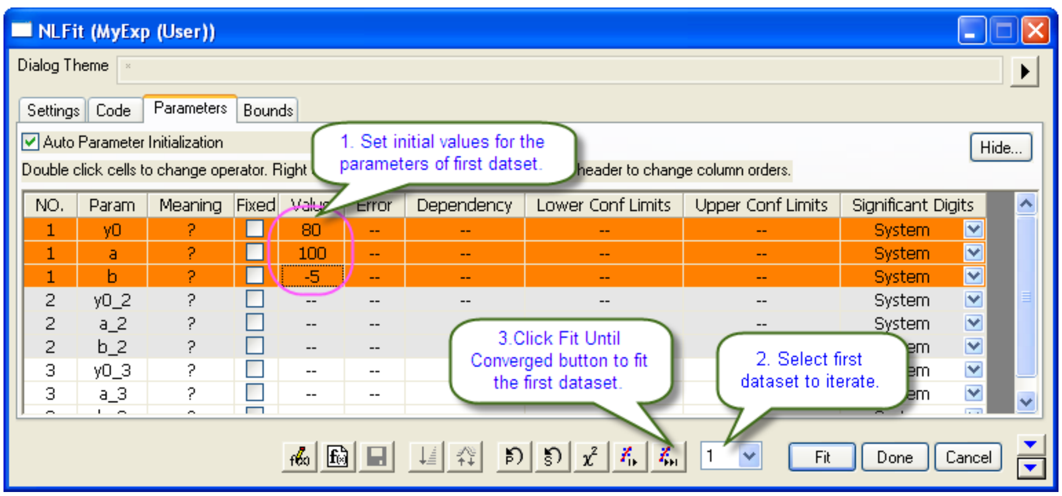
- How to perform an independent fit.
- How to fit one of multiple datasets by doing 1-iteration, full-iteration.
- How to apply the parameter values of one dataset to other datasets.

Example

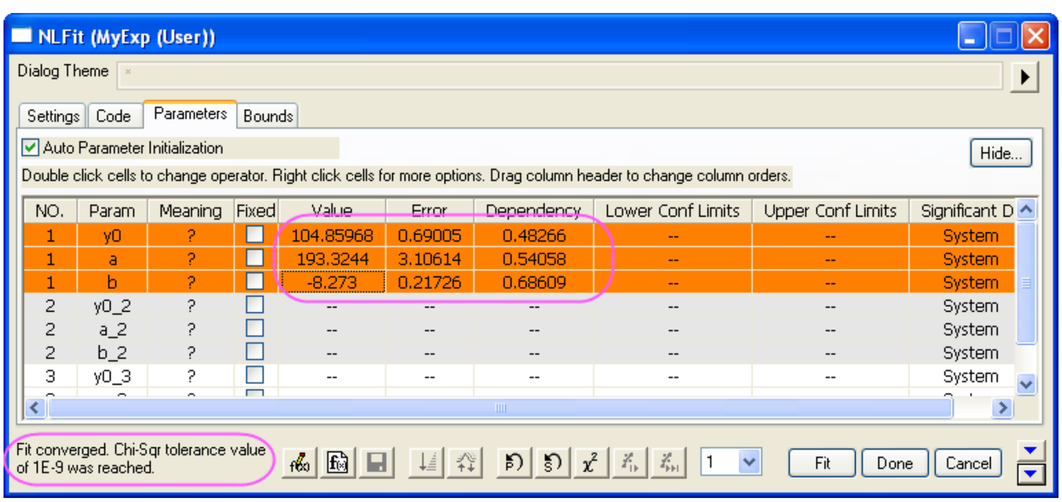
1. Create a user-defined function *MyExp* by following the steps in this tutorial.
2. Import *\Samples\Curve Fitting\Exponential Decay.dat* to Origin worksheet.
3. Highlight all columns and then select **Analysis: Fitting: Non-linear Curve Fit** from the menu to open the **NLFit** dialog.
4. Select the function just defined in **Settings** tab, **Function Selection** page:



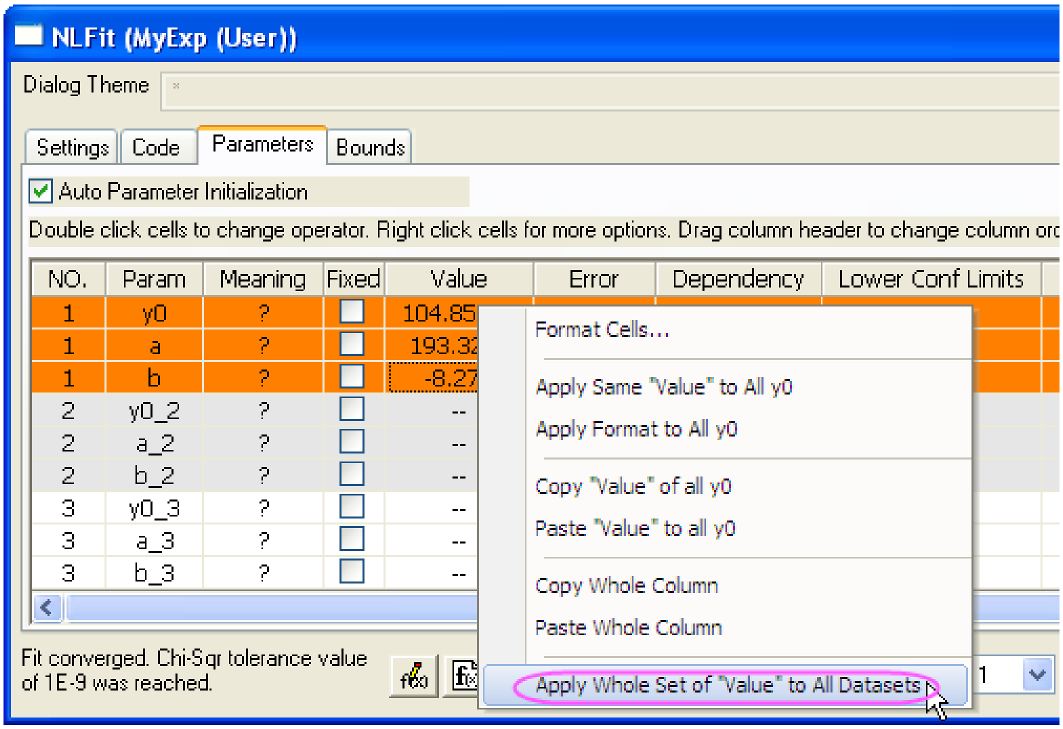
5. Switch to **Parameters** tab, enter *80*, *100*, *-5* on the **Value** column as initial values for *y0*, *a*, *b* of the first dataset. Then select *1* from the independent fit drop-down list and iterate the first dataset until converged.



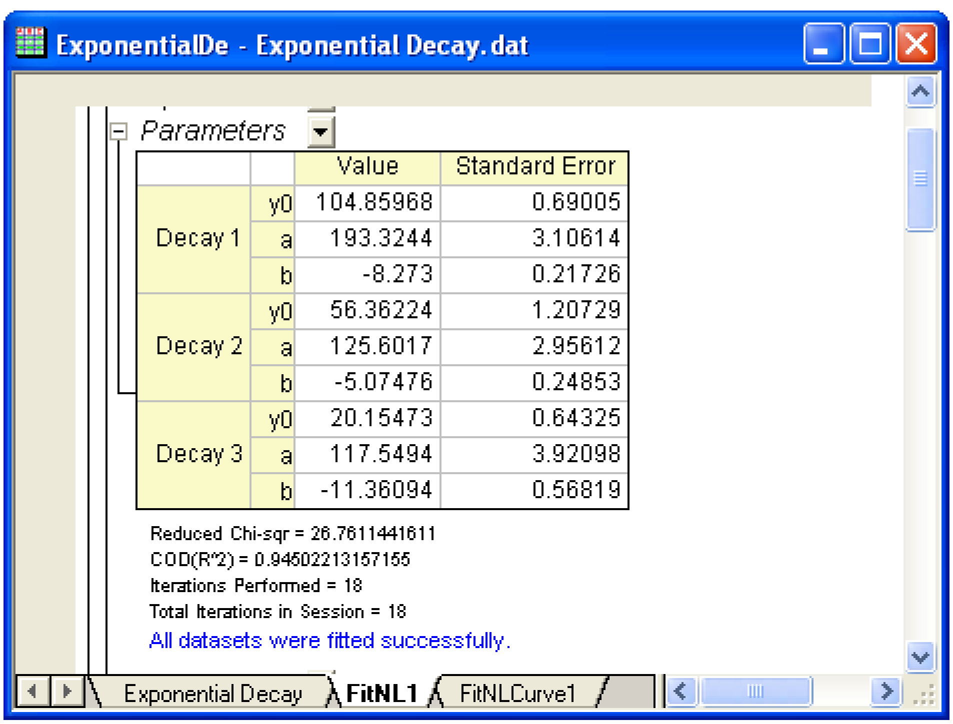
6. The parameter values are calculated and the fitting result is hinted in the left corner of this dialog.



7. To initialize the parameters of other datasets, apply the parameter values of the first dataset to other datasets by right-clicking the values, and then selecting **Apply Whole Set of "Value" to All Datasets** from the fly-out menu.



8. Click **Fit** button to finish the fitting for all datasets.



Adding Derived Parameters

Summary

Derived parameters are additional parameters computed using the fitted parameter values. You can define your own derived parameters for any built-in or user-defined fitting functions.
They are not involved in the fitting process and are computed only at the end of the fitting session.

What you will learn

- How to add derived parameters in a built-in fitting function by Fitting Function Organizer.

Steps

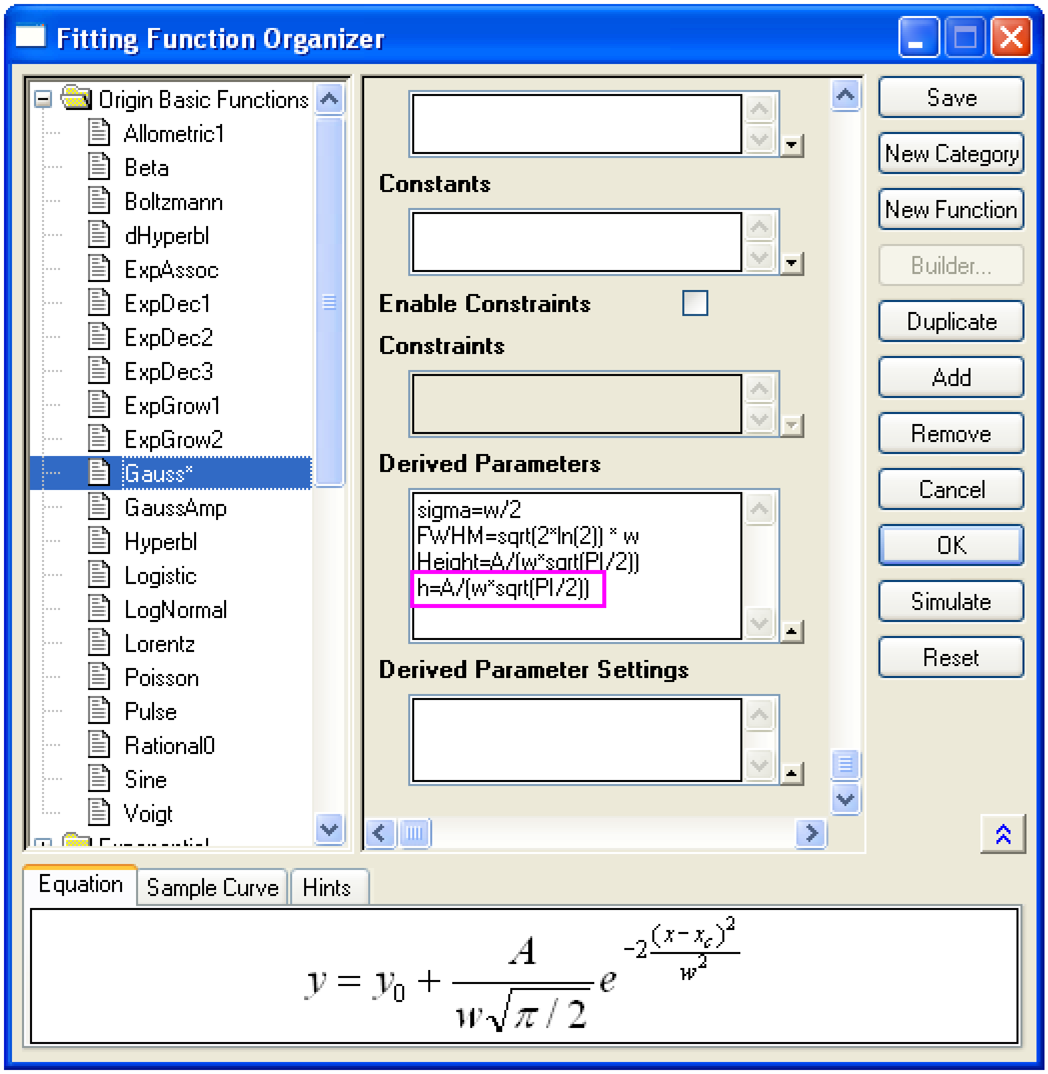
The built-in Gauss function fits the peak area with the following equation:

$$y = y_0 + \frac{A}{w\sqrt{\pi/2}} e^{-2\frac{(x-x_c)^2}{w^2}}$$

Suppose that you also want to know the peak height, which can be computated as follows:

$$h = \frac{A}{w\sqrt{\pi/2}}$$

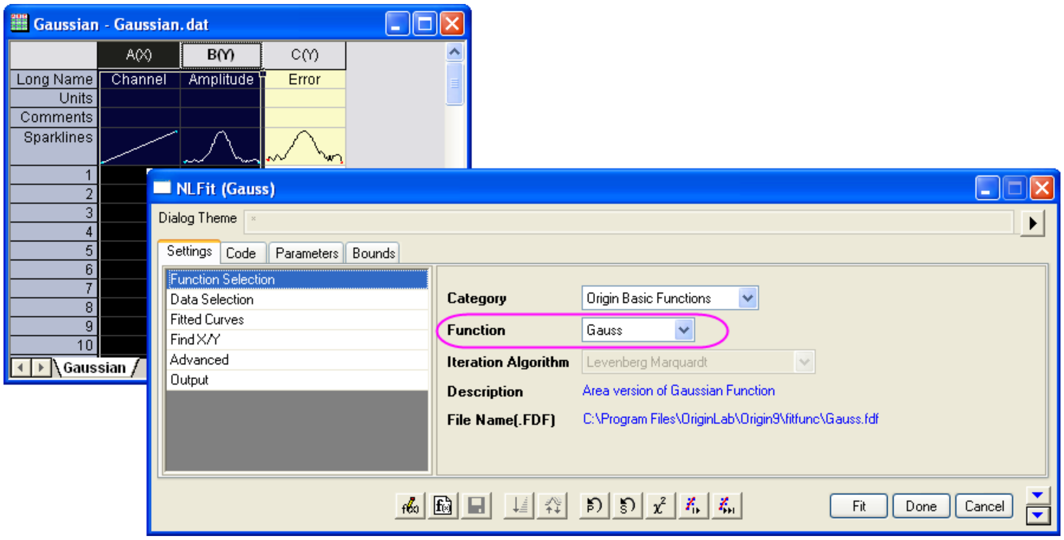
1. Select **Tool: Fitting Function Organizer** in the menu. In the left panel of the dialog, select **Origin Basic Funcions: Gauss**.
2. Go to **Derived Parameters** section and input the *h=A/(w*sqrt(PI/2))*



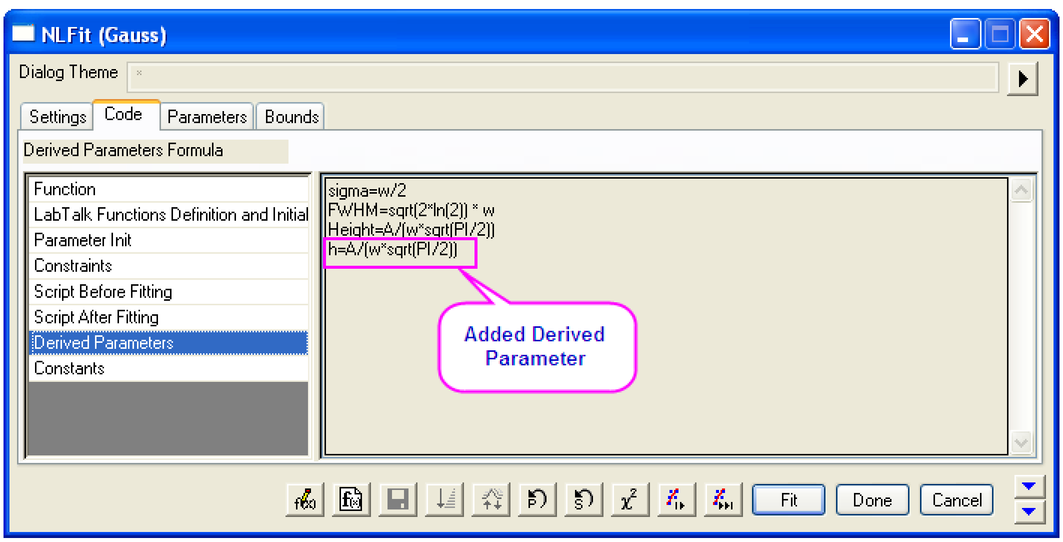
Note:

- Each derived parameter should be specified in a separate row.
- Derived parameters cannot be computed from other derived parameters.

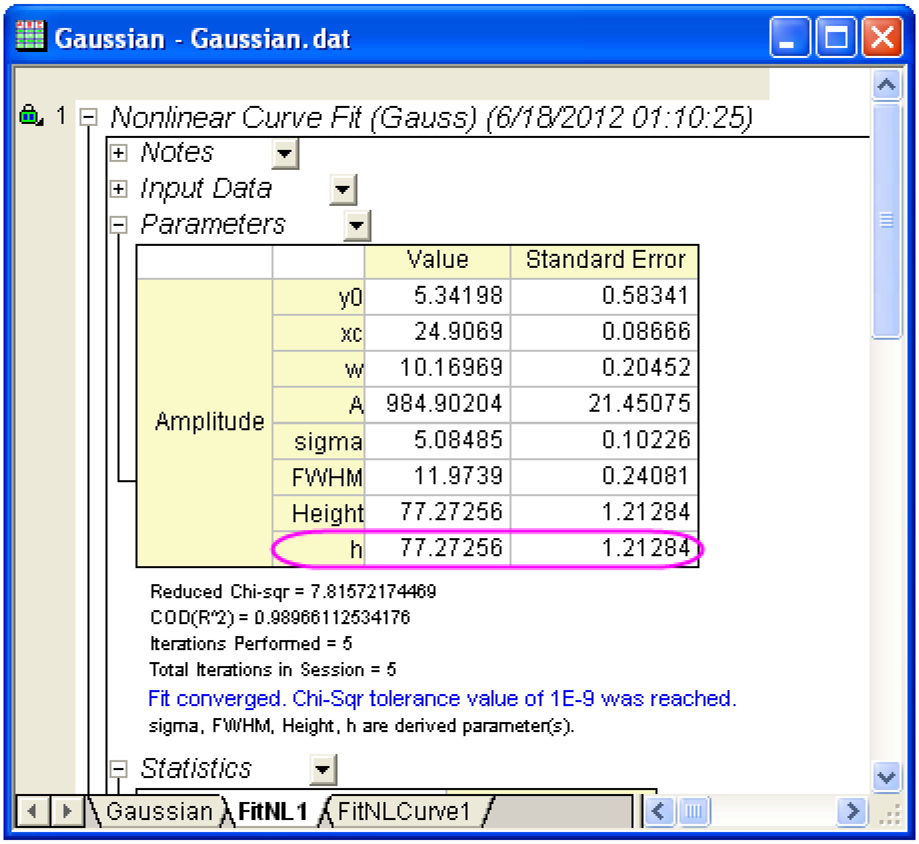
3. Click on **Save** and the **OK** button.
4. Create a new workbook and import the data file **\samples\curve fitting\Gaussian.dat**, using **File: Import: Single ASCII**.
5. Highlight column A and B, then select **Analysis: Fitting: Nonlinear Curve Fit** to open the **Nonlinear Curve Fitting** dialog, and select **Gauss** from the Function drop-down list.



6. Go to **Code** tab and click **Derived parameters** on the left panel. You can see the Derived Parameters listed on the right panel.



7. Click **Fit** button to generate report sheets. After fitting, Origin will compute the height and include the result in the Parameters table on the fitting results worksheet.



3.3 Signal Processing

3.3.1 FFT Filter

Summary

Origin offers an FFT filter, which performs filtering by using Fourier transforms to analyze the frequency components in the input dataset.

There are five types of filters available in the FFT filter function: low-pass, high-pass, band-pass, band-block, and threshold. Low-pass filters block all frequency components above the cutoff frequency, allowing only the low frequency components to pass. High-pass filters are just the opposite, they block frequency components that are below the cutoff frequency.

This tutorial will show you how to perform the high-pass, low-pass, and band-pass filtering using the FFT filter.

Minimum Origin Version Required: Origin 8.0 SR6

What you will learn

This tutorial will show you how to:

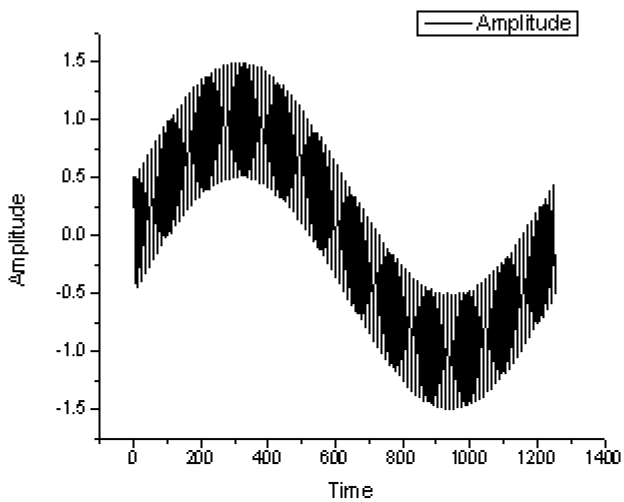
- Perform low pass filtering.
- Perform high pass filtering.
- Perform band pass filtering.

Steps

Low-pass Filter

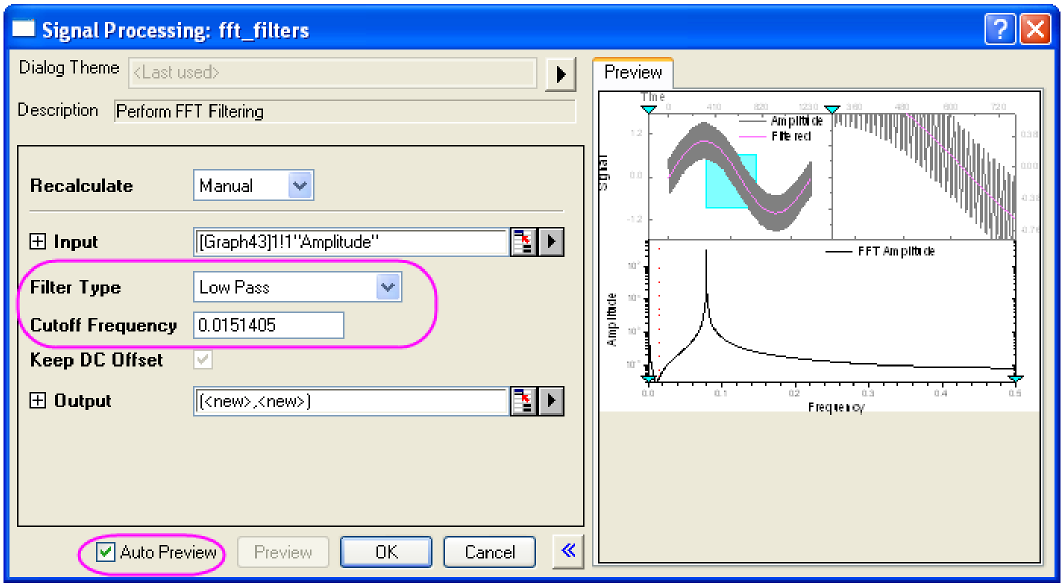
This tutorial is associated with the **Analysis: FFT Filter** folder in the **Analysis** project (\Samples\Analysis.opj) which can be opened by selecting **File: Open Sample Projects: Analysis** from the main menu.

1. Highlight the col(A) and col(B) in the worksheet to plot a line graph.

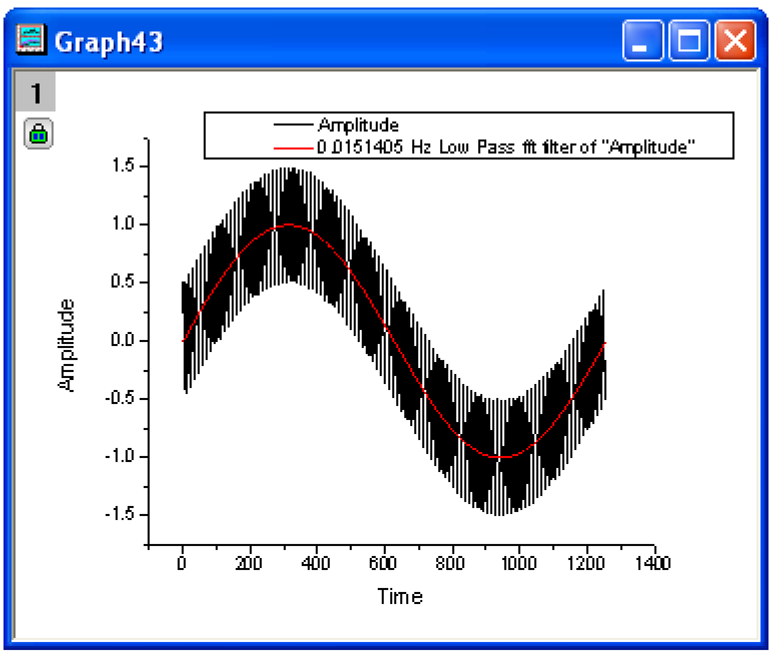


In this graph, we are going to block the high frequency components to get the low frequency component, which shows the overall trend of this curve.

2. Select **Analysis: Signal Processing: FFT Filters** to open the **fft_filters** dialog. Select **Low Pass** for the **Filter Type** drop-down list and set the **Cutoff Frequency** as *0.0151405*. Check the **Auto Preview** check box to preview the result.

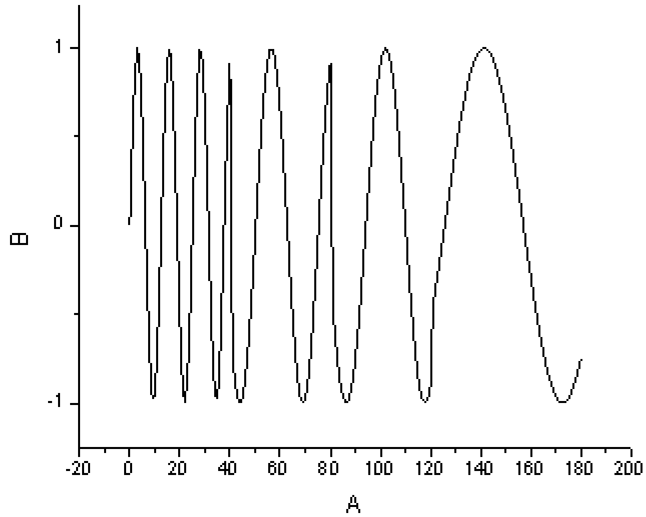


3. Click **OK** button, the high frequency component will be exported to the source worksheet and source graph.



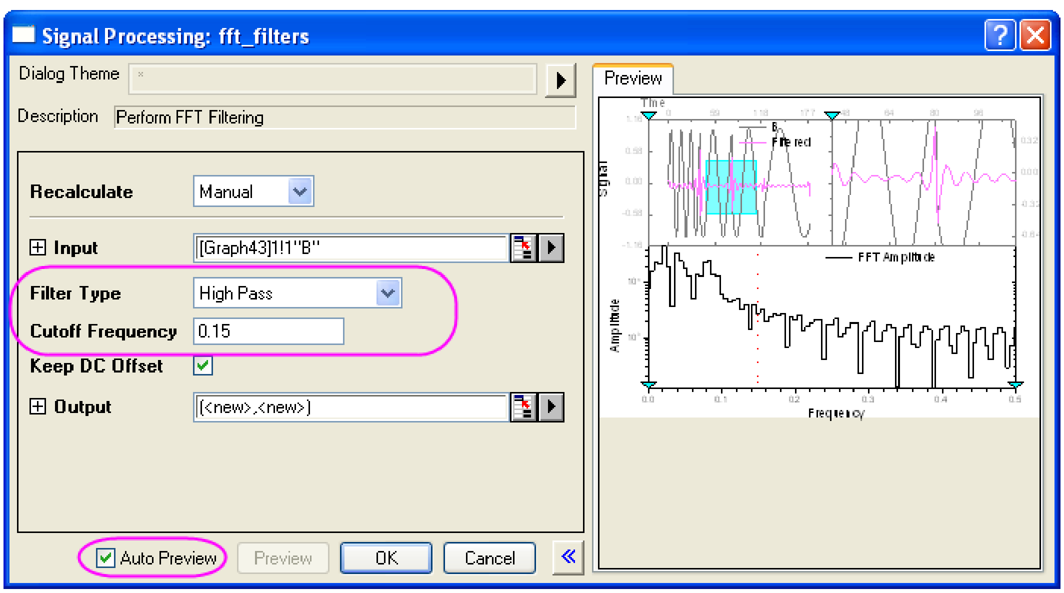
High-pass Filter

1. Start with a new workbook and import the file *\Samples\Signal Processing\fftfilter2.dat*.
2. Highlight col(A) and col(B) to select **Plot: Line: Line** from the main menu to plot a line graph.

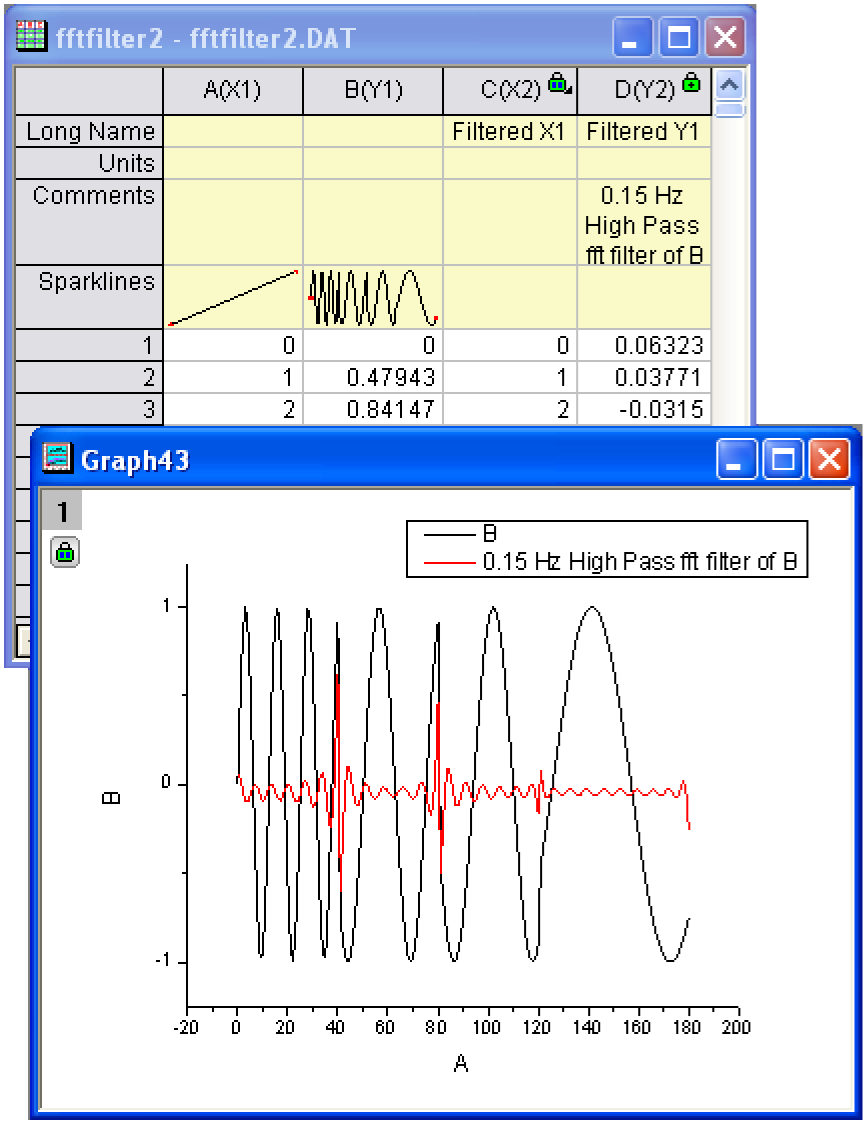


In this graph, we are going to remove the low frequency component (less than 0.15HZ).

3. Select **Analysis: Signal Processing: FFT Filters** to open the **fft_filters** dialog. Select **High Pass** for the **Filter Type** drop-down list and set the **Cutoff Frequency** as *0.15*. Check the **Auto Preview** check box to preview the result.

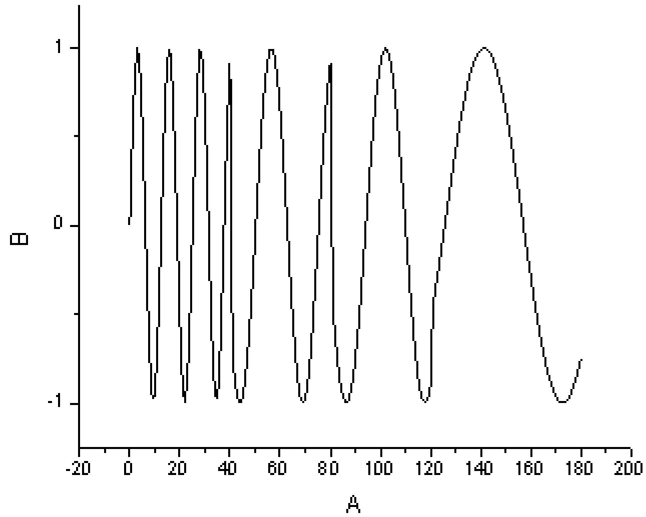


4. Click **OK** button, the high frequency component will be exported to the source worksheet.



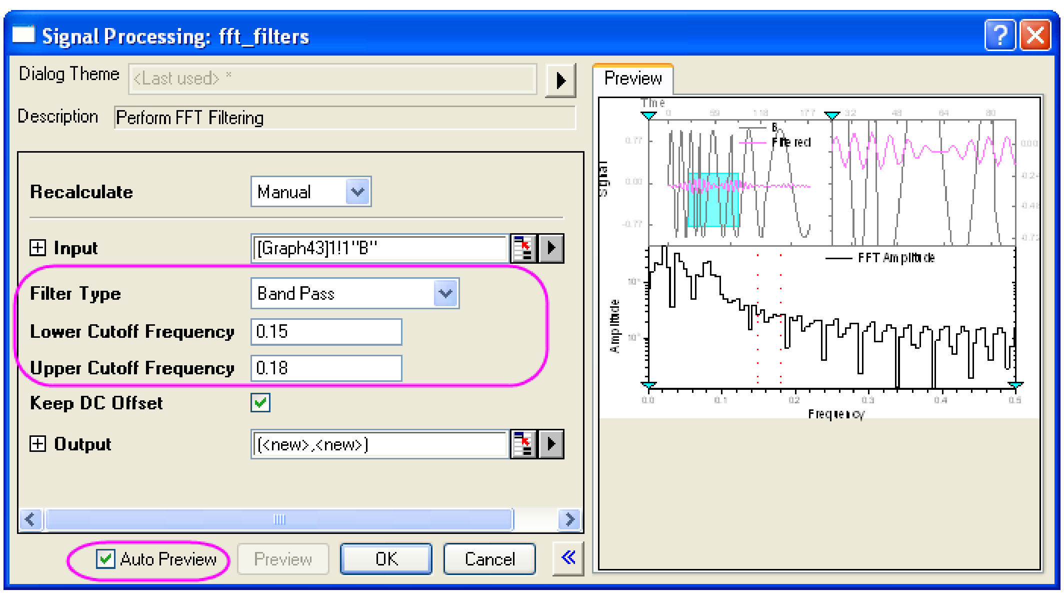
Band-pass Filter

1. Start with a new workbook and import the file *\Samples\Signal Processing\fftfilter2.dat*.
2. Highlight col(A) and col(B) to select **Plot: Line: Line** from the main menu to plot a line graph.

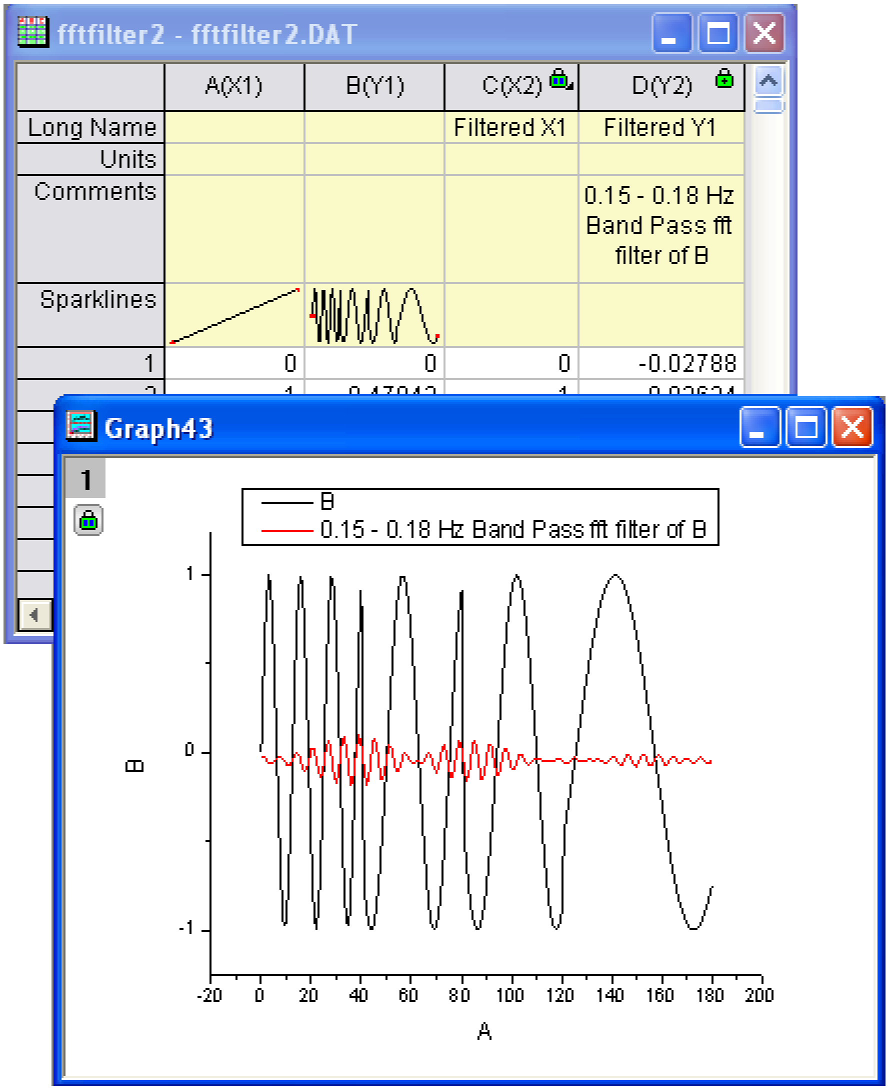


In this graph, we are going to remove the low frequency components (less than 0.15HZ) and high frequency components (more than 0.18HZ).

3. Select **Analysis: Signal Processing: FFT Filters** to open the **fft_filters** dialog. Select **Band Pass** for the **Filter Type** drop-down list and set the **Lower Cutoff Frequency** and **Upper Cutoff Frequency** as *0.15* and *0.18* respectively. Check the **Auto Preview** check box to see preview the result.



4. Click **OK** button, the high frequency component will be exported to the source worksheet.



3.3.2 IIR Filter

Summary

In OriginPro, it is possible to design, analyze, and implement IIR (Infinite Impulse Response) digital filters. The IIR filter supports four methods, including **Butterworth**, **Chebyshev Type I**, **Chebyshev Type II**, and **Elliptic**.

This provides users more choices in signal processing.

Minimum Origin Version Required: 9.0 SR0

What You Will Learn

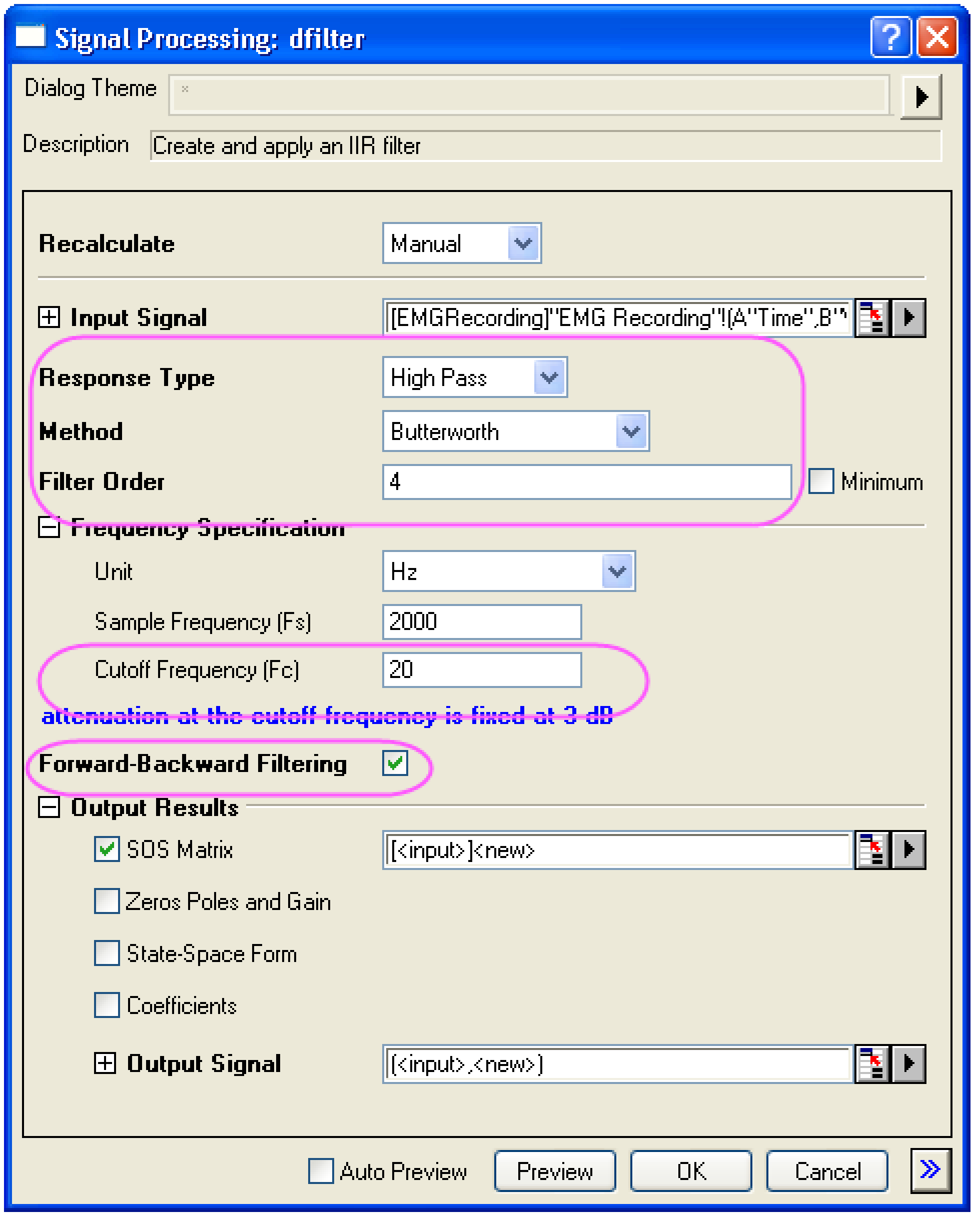
This tutorial will show you:

- How to design and apply an IIR filter
- A comparison between IIR filter and FFT filter

Steps

Design and Apply IIR Filter

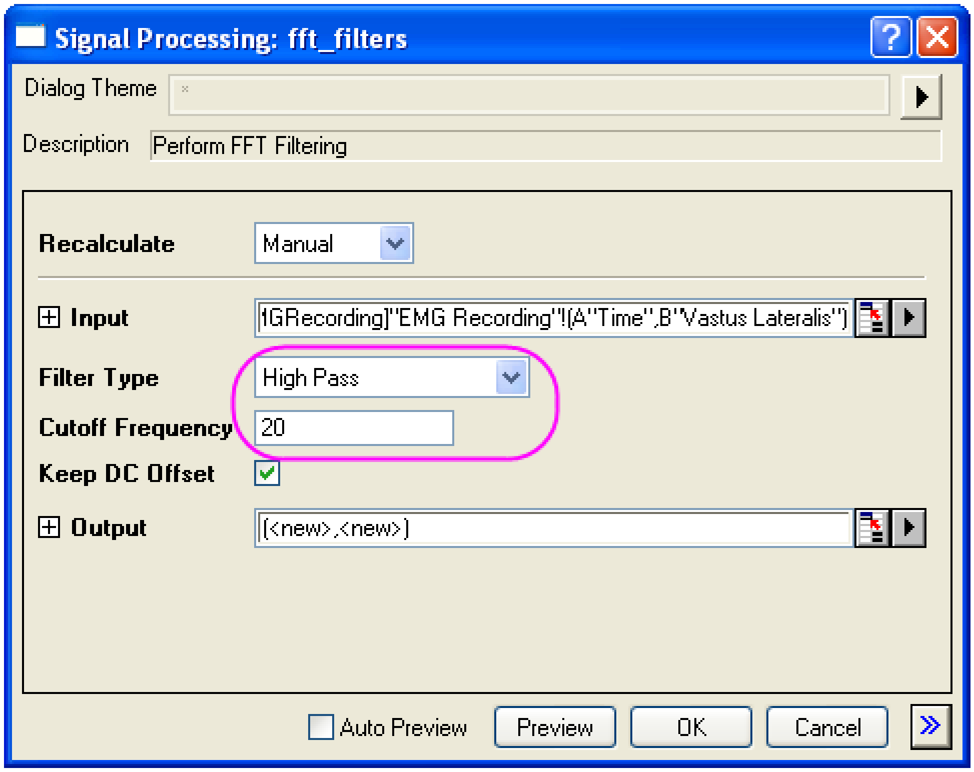
1. Start with a new worksheet and import the **EMG Recording.dat** file from *\Samples\Signal Processing*.
2. Highlight column B and Select **Analysis:Signal Processing:IIR Filter** from the top menu to open the dialog.
3. Change the Response type as **High Pass**, keep the Method as **Butterworth**, uncheck the **Minimum** for Filter Order and set it as **4**. In the Frequency Specification branch, set the Cutoff Frequency(Fc) as **20**, then check the **Forward-Backward Filtering**. The dialog settings should look like the following figure, and the IIR filter is designed.



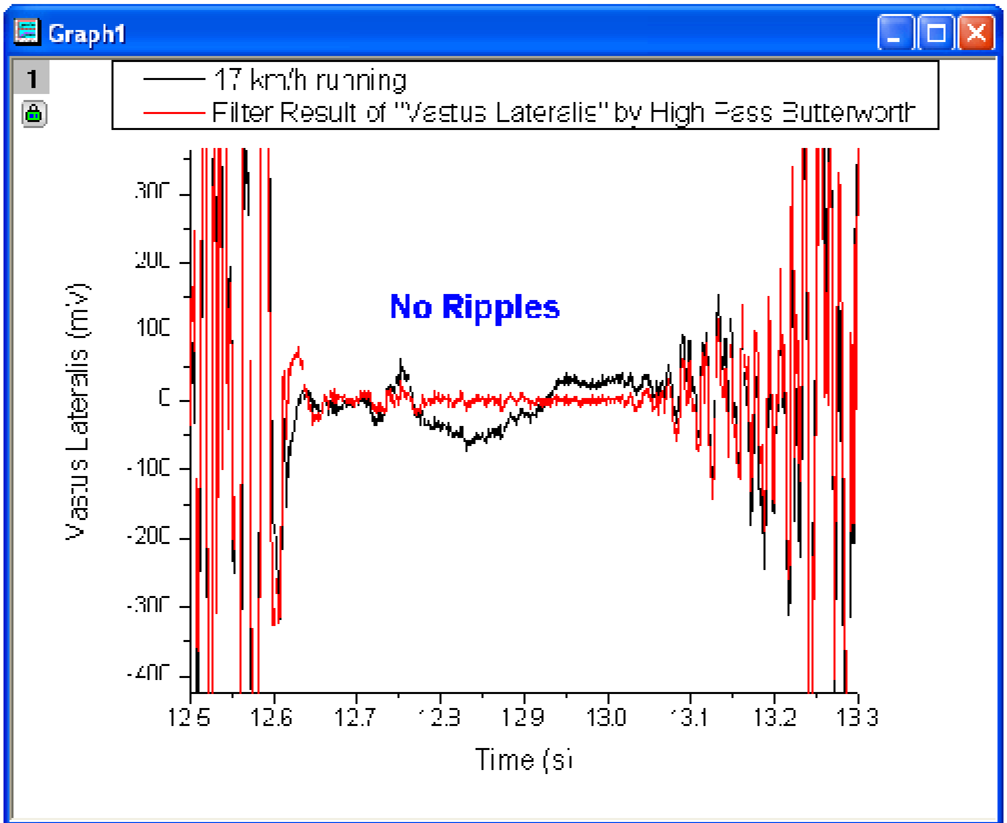
4. Click **OK** to apply the created IIR filter to the input dataset.
5. A new column will be added to the original data as a new column of filtered data and a new SOS Matrix worksheet.

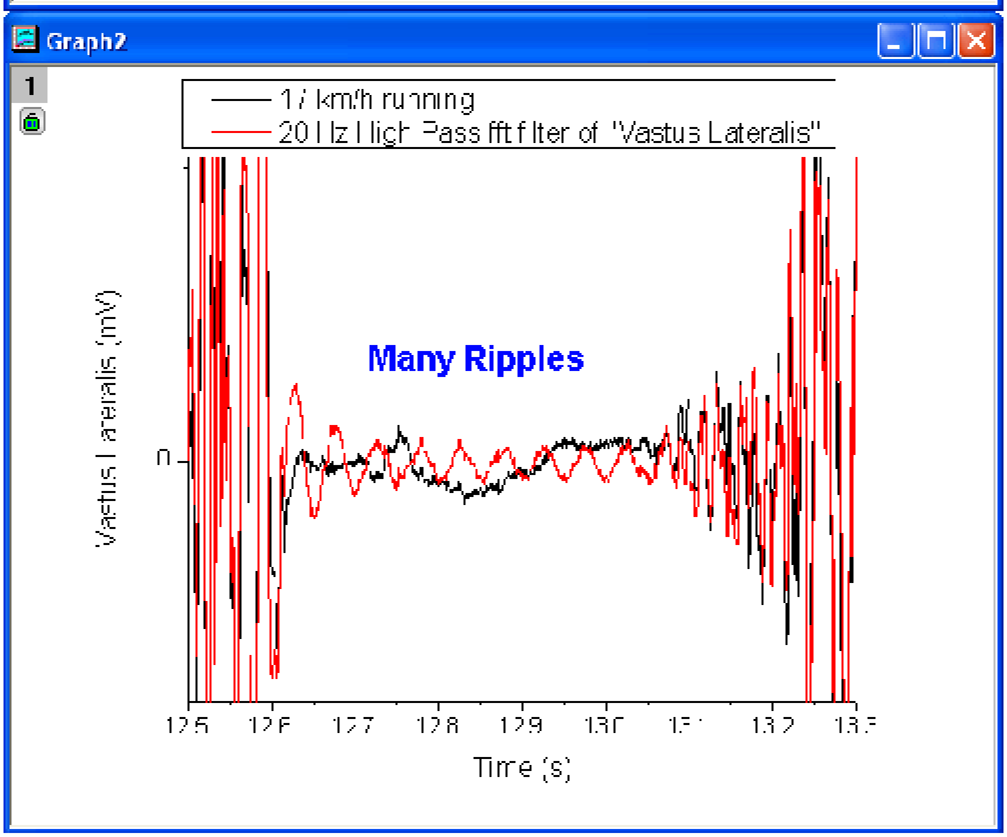
Compare Results with FFT Filter

1. Highlight column B in the original worksheet, perform FFT filter by **Analysis:Signal Processing:FFT Filters**.
2. In the opened dialog, choose **High Pass** for Filter Type and set **20** as Cutoff Frequency.



3. Column C in the EMGRecording worksheet is the filtered result of the previously designed IIR filter, highlight column B and column C to generate a line plot with the button (Graph 1).
4. Use the scale in button to zoom the area between 12.5s and 13.3s.
5. Column E in the EMGRecording worksheet is the filtered result of the FFT filter, highlight column B and column E to generate a line plot with the button (Graph 2).
6. Also use the scale in button to zoom the area between 12.5s and 13.3s, the two graphs could be used for visualized comparison.

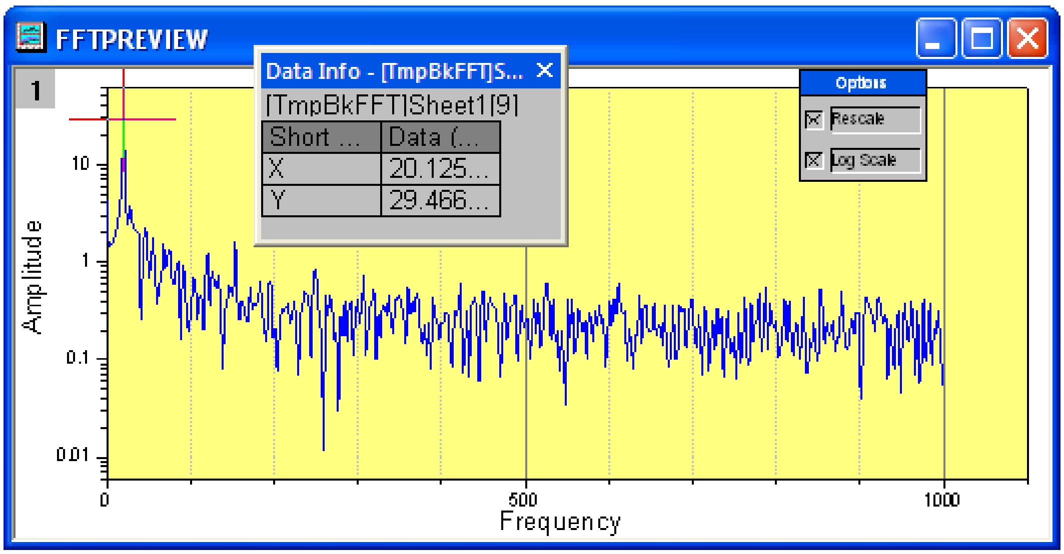

Graph1
17 km/h running
Filter Result of "Vastus Lateralis" by High Pass Butterworth
No Ripples
Vastus Lateralis (mV)
Time (s)


Graph2
17 km/h running
20 Hz High Pass fft filter of "Vastus Lateralis"
Many Ripples
Vastus Lateralis (mV)
Time (s)

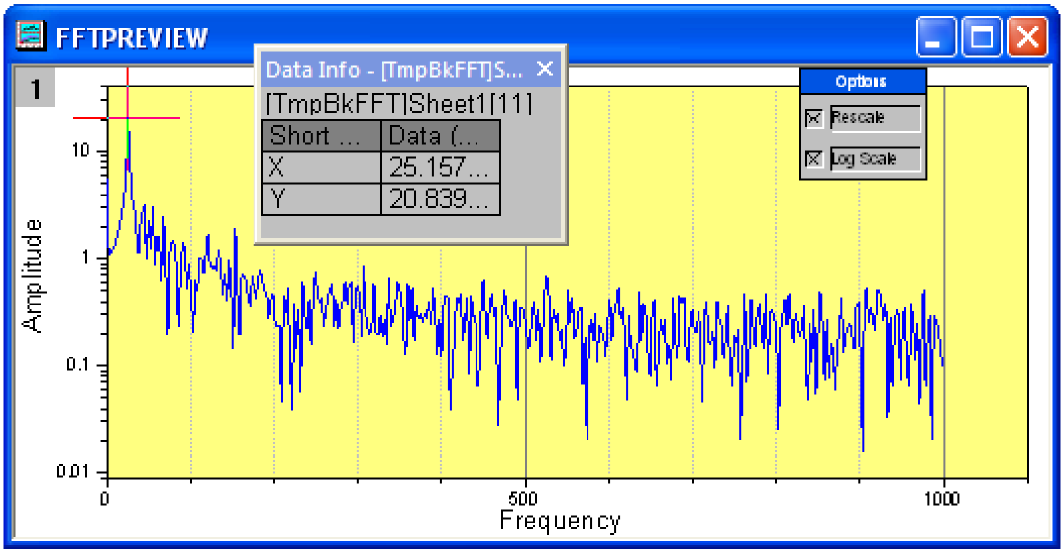
- Note that there are many ripples in the FFT filter result, but almost no ripples in the IIR filter result.

Ripples in FFT Filter

1. Highlight column E and click on the  button to create a line plot (Graph 3).
2. Activate Graph 3, choose **Gadget:FFT** and set the X Scale as From **12.664** To **13.052**.
3. Click OK to bring up the preview window, in which ripples are almost pure 20.125 Hz sine.



4. Now we would try to remove the ripples at 20.125 Hz by applying another high pass filter at 25 Hz, keep highlighting column E and select **Analysis:Signal Processing:FFT Filters**.
5. Select **High Pass** for Filter Type and set Cutoff Frequency as **25**.
6. The result is listed in Column G, highlight column G and click on the  button to create a line plot(Graph 4).
7. Activate Graph 4, choose **Gadget:FFT** and set the X Scale as From **12.664** To **13.052**, in the preview window, there are still ripples, and they are shifted from 20.125 Hz to 25.157 Hz.



- Note that the ripples could not be removed by FFT filter for this dataset.

3.4 Peak Analysis

Topics covered in this section:

1. Peak Finding (Tutorials)
2. Peak Integration (Tutorials)
3. Peak Fitting (Tutorials)

3.4.1 Peak Finding

- Picking and Marking Peaks

Picking and Marking Peaks

Summary

The Peak Analyzer provides several methods to pick peaks automatically. Also, user can opt to add/delete/modify the peaks manually.

Labels are added to the peak centers after they are found or added, to show user the positions of the current peaks.

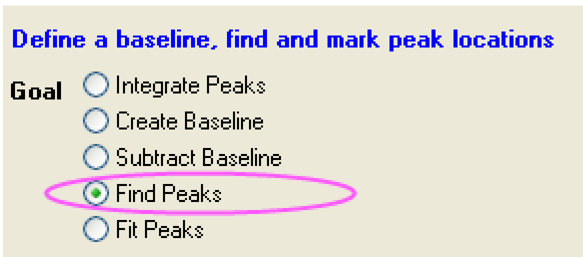
Minimum Origin Version Required: Origin 8.0 SR6

What you will learn

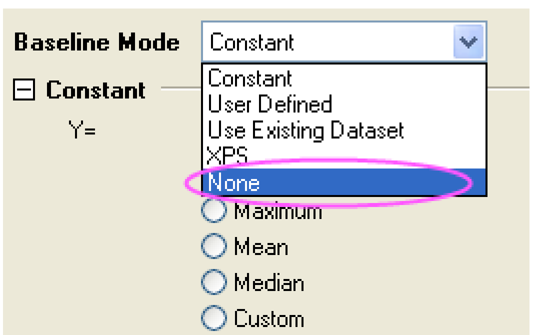
- How to use and customize the auto peak finding in the Peak analyzer
- How to customize the labels for the peak centers

Steps

1. Start a new workbook and import the file <Origin Program Folder>\Samples\Spectroscopy\HiddenPeaks.dat.
2. Highlight the second column.
3. Create a line plot by selecting **Plot: Line: Line**.
4. With the graph active, select **Analysis: Peaks and Baseline: Peak Analyzer** to open the dialog of the Peak Analyzer.
5. In the first page (the **Start** page), select the **Find Peaks** radio button in the **Goal** group. Then click the **Next** button to go to the next page.

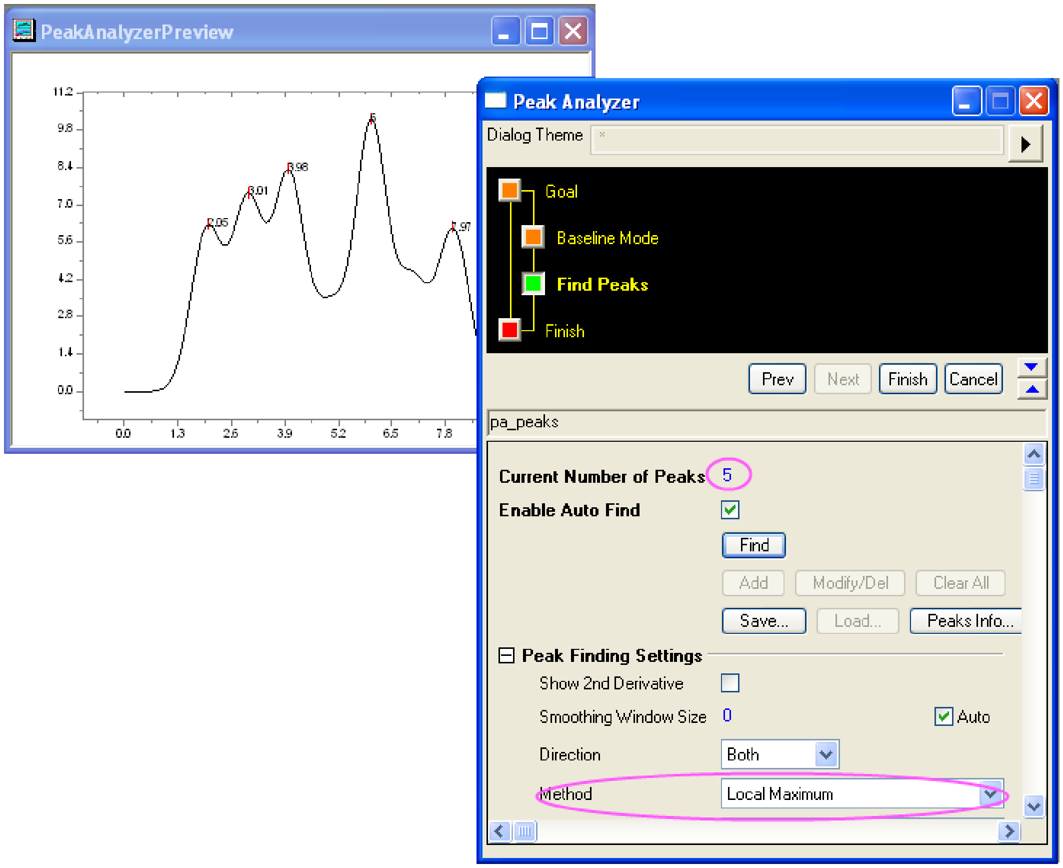


6. In the **Baseline Mode** page, select **None** for **Baseline Mode**.

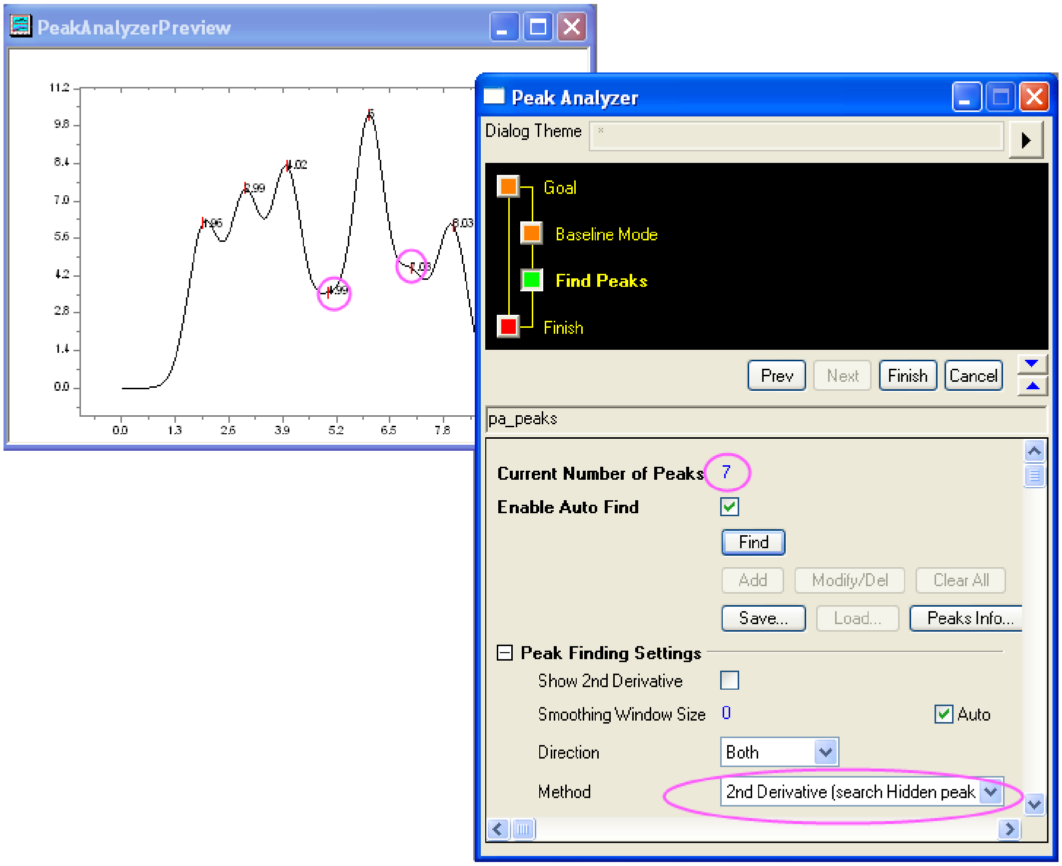


Click the **Next** button to go to the **Find Peaks** page.

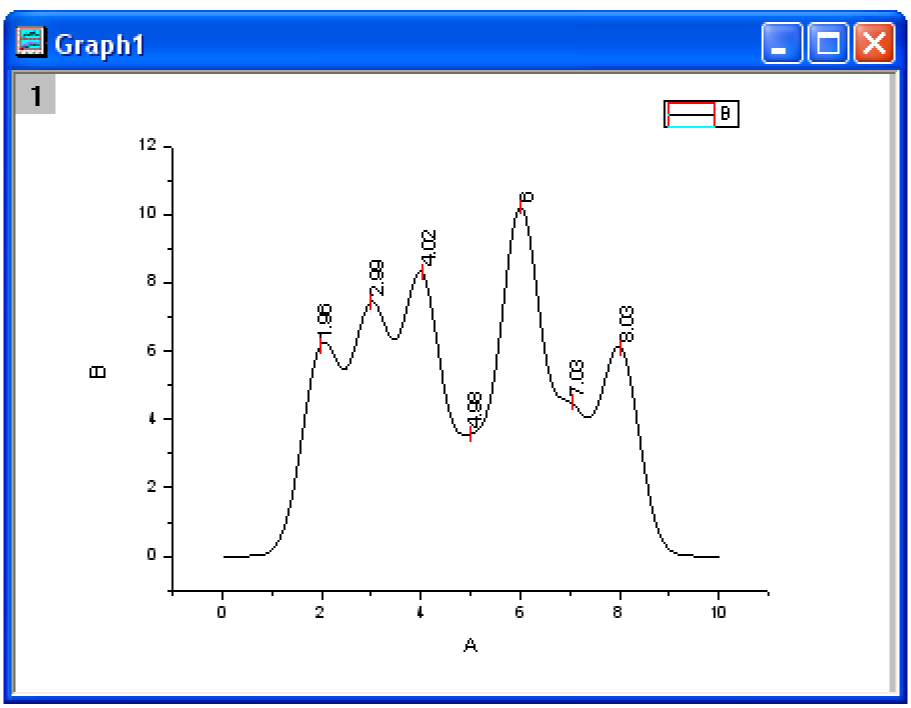
7. In the **find Peaks** page:
    1. Expand the **Peak Finding Settings** branch. Make sure that **Local Maximum** is selected for **Method**. Then click the **Find** button. Only five peaks are detected.



2. Change **Method** to **2nd Derivative (Search Hidden Peaks)**. Click the **Find** button again. This time, seven peaks are detected.



3. Click **Finish** to complete the analysis. We will get this final graph:



3.4.2 Peak Integration

Peak Integration

- Integrating Peaks

Integrating Peaks

Summary

Use the Peak Analyzer to integrate peaks and find their areas.

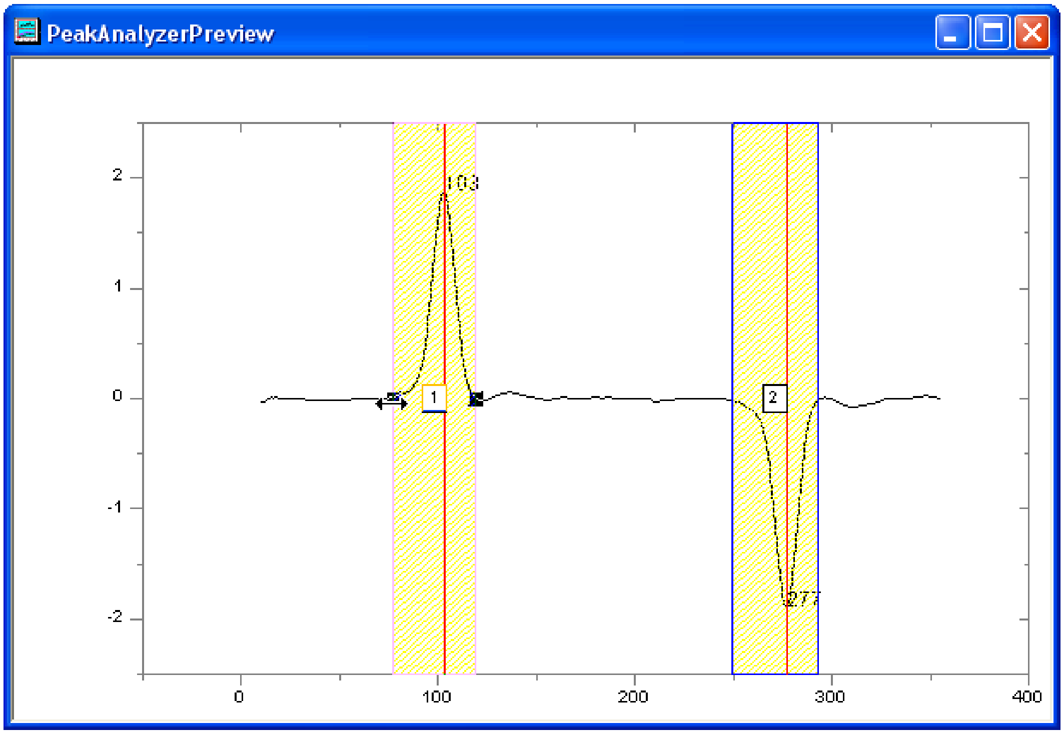
Minimum Origin Version Required: Origin 8.0 SR6

What you will learn

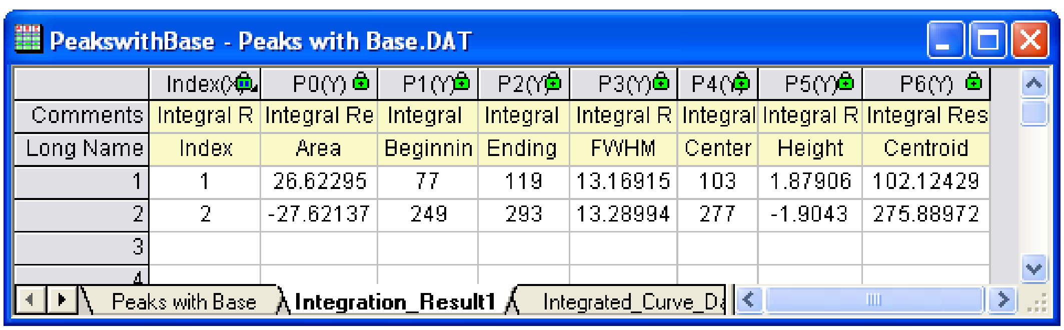
- How to pick an existing dataset as baseline
- How to subtract a baseline from the spectrum data
- How to calculate peak areas with the Peak Analyzer

Steps

1. Start a new workbook and import the file *\Samples\Spectroscopy\Peaks with Base.DAT*.
2. Highlight the second column.
3. In the main menu, click **Analysis**, then point to **Peaks and Baseline**, and then click **Peak Analyzer**.
4. In the first page (the **Goal** page) of the **Peak Analyzer**, select **Integrate Peaks** in the **Goal** group. Click **Next** to go to the **Baseline Mode** page.
5. On the **Baseline Mode** page, select **Use Existing Dataset** from the **Baseline Mode** dropdown list. Click the **triangular button** to the right of the Dataset dropdown menu, and then select **[PeakswithBase]"Peaks with Base"!C"Base"** on the fly-out menu. Click **Next** to go to the **Baseline Treatment** page of the Peak Analyzer.
6. Select the **Auto Subtract Baseline** check box. Click the **Integrate Peaks** page icon in the upper panel (or click **Next** twice to go to the Integrate Peaks page). In the preview graph, you will see two numbered yellow rectangles representing two peaks found with the default settings.
7. For **Integration Window Width**, select **Adjust on Preview Graph**. On the graph, click inside the rectangle marked with **1**. A pair of "handles" appears on either side of the yellow integration window. Drag these handles to adjust the range for which integration will be performed. Click inside the rectangle marked with **2** and adjust the integration range for the second peak.



8. In the Integrate Peaks page of the Peak Analyzer, make sure all the desired quantities to compute have been selected in the **Quantities** group. For example, if you want to calculate the peak centroid for each peak, select the **Peak Centroid** check box. If you don't want to output the percent areas, clear the **Percent Area** check box. When you are done, click **Finish** to perform the analysis. The result is in a worksheet named *Integration_Result1*.


PeakswithBase - Peaks with Base.DAT

	Index(X)	P0(Y)	P1(Y)	P2(Y)	P3(Y)	P4(Y)	P5(Y)	P6(Y)
Comments	Integral R	Integral Re	Integral	Integral	Integral R	Integral	Integral R	Integral Res
Long Name	Index	Area	Beginnin	Ending	FWHM	Center	Height	Centroid
1	1	26.62295	77	119	13.16915	103	1.87906	102.12429
2	2	-27.62137	249	293	13.28994	277	-1.9043	275.88972
3								
4								

Peaks with Base / Integration_Result1 / Integrated_Curve_D

If the Area check box was selected in the Quantities group, peak area data appears in the Area column of the result worksheet.

3.4.3 Peak Fitting

Topics covered in this section:

1. Peak Fitting with Baseline
2. Peak Fitting with Preset Peak Parameters

3. Setting the Fix Share Status or Bounds for Multiple Peak Parameters Simultaneously

Peak Fitting with Baseline

Summary

In OriginPro, the **Peak Analyzer** is capable of performing multiple peak fitting with several baseline subtraction options.

There are various ways to create a baseline for your spectrum data. You can select a few anchor points and then fit them with a function. The fitting of the baseline can be done along with the peak fitting.

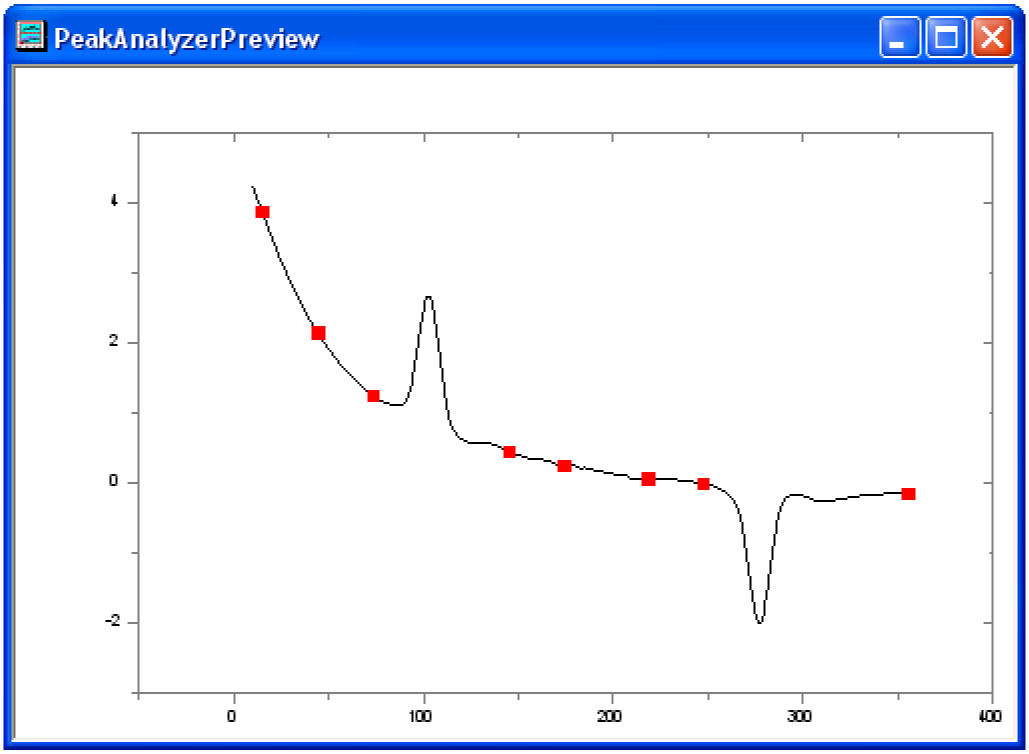
Minimum Origin Version Required: OriginPro 8.0 SR6

What You Will Learn

- How to perform fitting of peaks
- How to fit the baseline

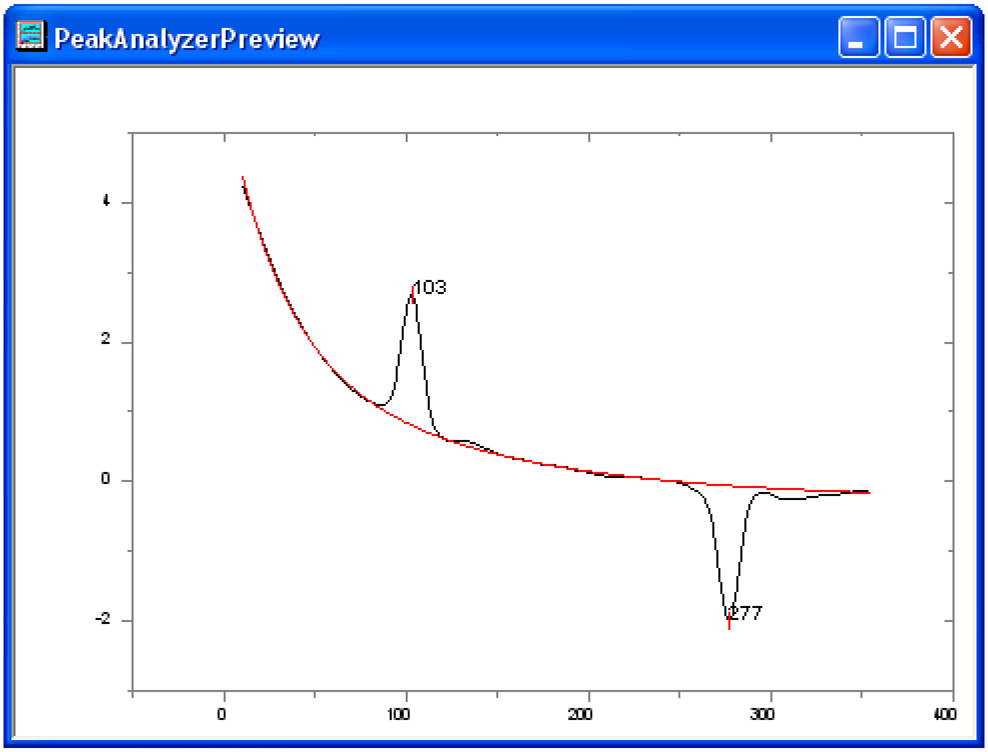
Steps

1. Start a new worksheet and import the file <Origin Program Folder>\Samples\Spectroscopy\Peaks on Exponential Baseline.dat.
2. Highlight the second column in the worksheet.
3. Select **Analysis: Peaks and Baseline: Peak Analyzer** from the main menu to open the **Peak Analyzer**.
4. Select the **Fit Peaks** radio button in the **Goal** group on the first page. Click **Next** to go to the **Baseline Mode** page.
5. In the **Baseline Mode** page, select **User Defined** from the **Baseline Mode** drop-down list. Click the **Find** button in the **Baseline Anchor Points** group. Eight anchor points should be found.



Click **Next** to go to the **Create Baseline** page.

6. In the **Create Baseline** page, select **Fitting** with the **Connect By** drop-down list. In the **Fitting** group, select **ExpDec2** from the **Function** drop-down list. Click **Next** to go to the **Baseline Treatment** page.
7. In the **Baseline Treatment** page, select the **Fit Baseline with Peaks** check box. Click **Next** to go to the **Find Peaks** page.
8. In the **Find Peaks** page, click the **Find** button to search peaks. Two peaks should be found.



Click **Next** to go to the **Fit Peaks** page.

9. In the **Fit Peaks** page, click the **Fit Control** button to open the **Peak Fit Parameters** dialog.
10. In the **Peak Fit Parameters** dialog, make sure that both peak types are Gaussian. Click the **Fit Until Converge** button. When the fitting is done, click **OK** to close the dialog.
11. Back in the **Fit Peaks** page, click **Finish** to complete the analysis. See the results in the source workbook and the graph report.

Peak Fitting with Preset Peak Parameters

Summary

In some cases, you may want to perform peak fitting with preset peak parameters. For example, you may have many datasets with fixed numbers of peaks and the centers of these peaks do not vary from dataset to dataset. What you are interested in is mainly other parameters of the peaks, for example, heights. Using the theme feature of the Peak Analyzer, you may carry out peak fitting with fixed peak parameters easily.

Minimum Origin Version Required: OriginPro 8.0 SR6

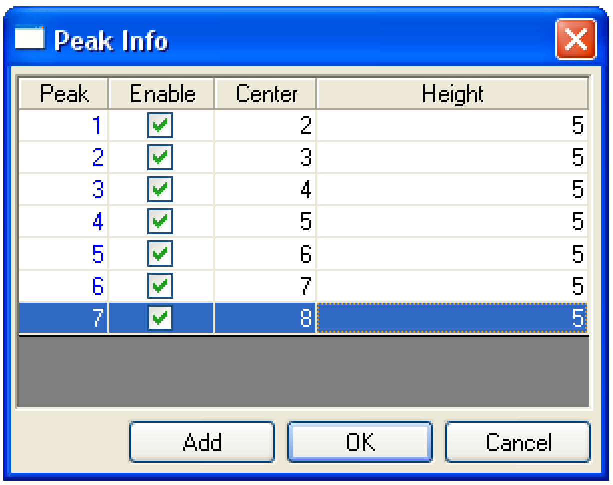
What You Will Learn

1. How to save Peak Analyzer settings in a theme and reuse them
2. How to fix peak parameters

Steps

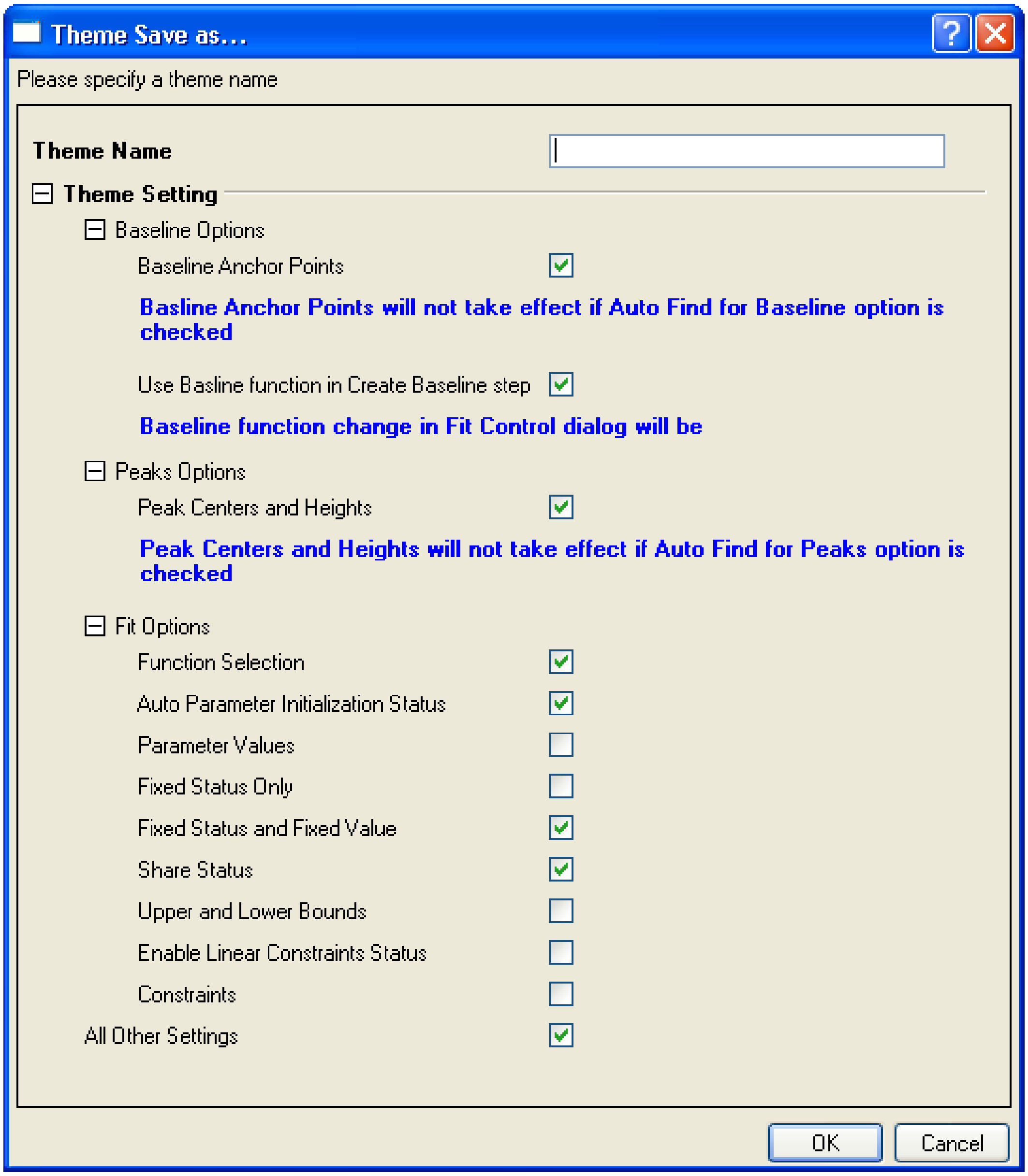
Save a theme with peak positions and peak parameters

1. Start a new worksheet and import the file <Origin Program Folder>\Samples\Spectroscopy\HiddenPeaks.dat.
2. Highlight the second column and select **Analysis: Peaks and Baseline: Peak Analyzer** from the Origin menu to open the **Peak Analyzer**.
3. On the first page, select the **Fit Peaks** radio button in the **Goal** group. Click **Next** to go to the **Baseline Mode** page.
4. In the **Baseline Mode** page, select **None** with the **Baseline Mode** drop-down list. Click **Next** to go to the **Find Peaks** page.
5. In the **Find Peaks** page:
    a. Clear the **Enable Auto Find** check box, because we want to find the peaks manually. Click the **Peaks Info** button to open the **Peak Info** dialog.
    b. In the **Peak Info** dialog, click the **Add** button seven times to add 7 peaks. Enter the peak centers and heights as follows:



Click **OK** to return to the Peak Analyzer.

    c. Click **Next** to go to the **Fit Peaks** page.
6. In the **Fit Peaks** page:
    a. Click **Fit Control** to open the **Peak Fit Parameters** dialog.
    b. In the **Peak Fit Parameters** dialog, click the **Fix or release all peak centers** button. Then click the **Fit Until Converge** button. When the fitting is done, click **OK** to return to the **Peak Analyzer** dialog.
    c. Click the right-sided triangle button to the right of **Dialog Theme** in the upper panel. Select **Save As** from the short-cut menu. The **Theme Save as** dialog opens.
    d. In the **Theme Save as** dialog, enter **MyFitting** after **Theme Name**. Clear and select the check boxes as the screenshot below:



Click **OK** to save the theme. This should bring you back to the **Peak Analyzer** dialog.

e. Click the **Finish** button in the **Peak Analyzer** to complete the analysis.

Reuse the theme

1. Start another new workbook and import the file <Origin Program Folder>\Samples\Spectroscopy\HiddenPeaks.dat.
2. Highlight the second column
3. Select **Analysis: Peaks and Baseline: Peak Analyzer** from the Origin menu to open the **Peak Analyzer** dialog.

4. On the first page of the **Peak Analyzer**, click the right-sided triangle button to the right of **Dialog Theme**. From the short-cut menu, pick **MyFitting**.
5. Click **Next** to check if the settings in every step are correct. Note that in the **Find Peaks** page, you can see the peak centers and heights are same as last time.
6. When you reach the last page, click the **Fit Control** button to open the **Peak Fit Parameters** dialog. Make sure that all peak centers are fixed and the values are the same as last time. Click **OK** to return to the **Peak Analyzer**.
7. Click **Finish** to complete the analysis. Check the results to see whether they are the same as the results we got last time.

Setting the Fix, Share Status or Bounds for Multiple Peak Parameters Simultaneously

Summary

When performing peak analysis, one often wants to fix parameter values, or share parameters between multiple peaks, or specify bounds. If your data has a few peaks, you can simply perform these settings for each peak. But if your data has many, it may be time consuming to set individually. To make the process more efficient, the **Peak Analyzer** offers context menus which can allow you to set the fix, share status or bounds for multiple peak parameters simultaneously. For more details about these settings, please refer to the Origin Help File.

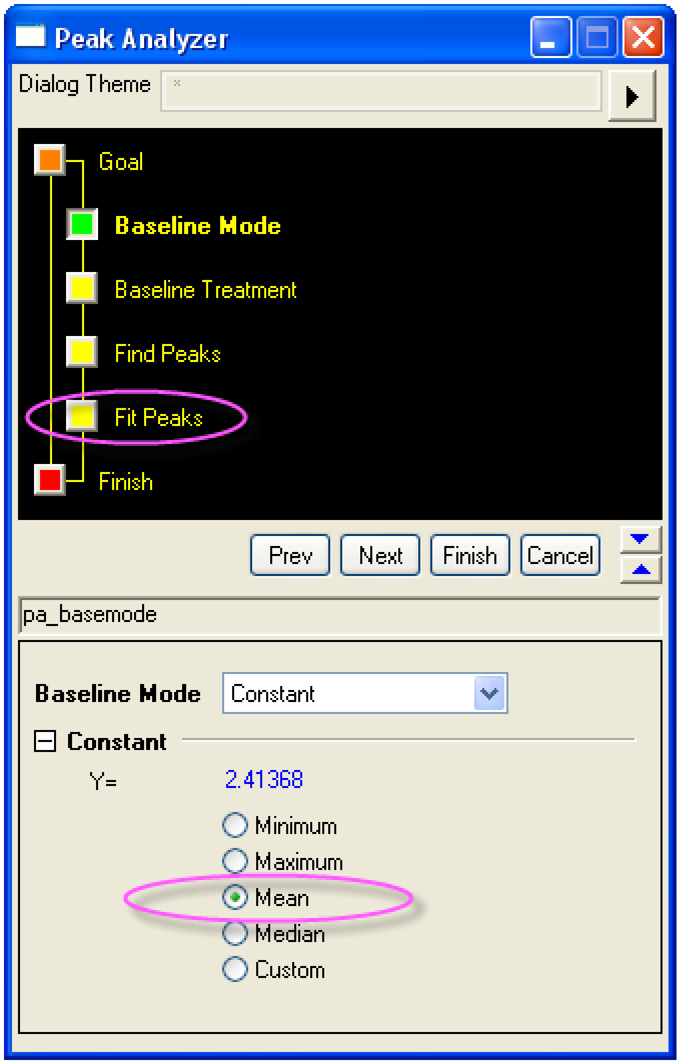
Minimum Origin Version Required: OriginPro 8.0 SR6

What You Will Learn

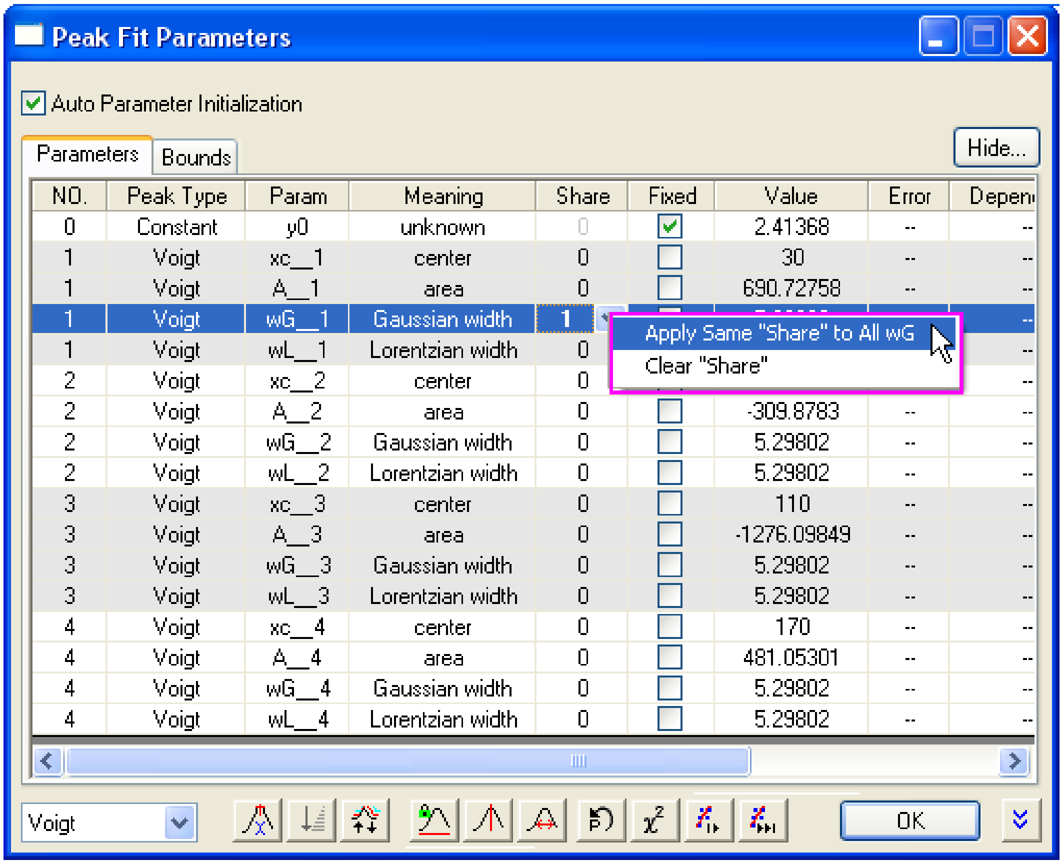
- How to set share status of multiple peak parameters simultaneously.
- How to set upper bounds and upper bound values to multiple peak parameters simultaneously.

Steps

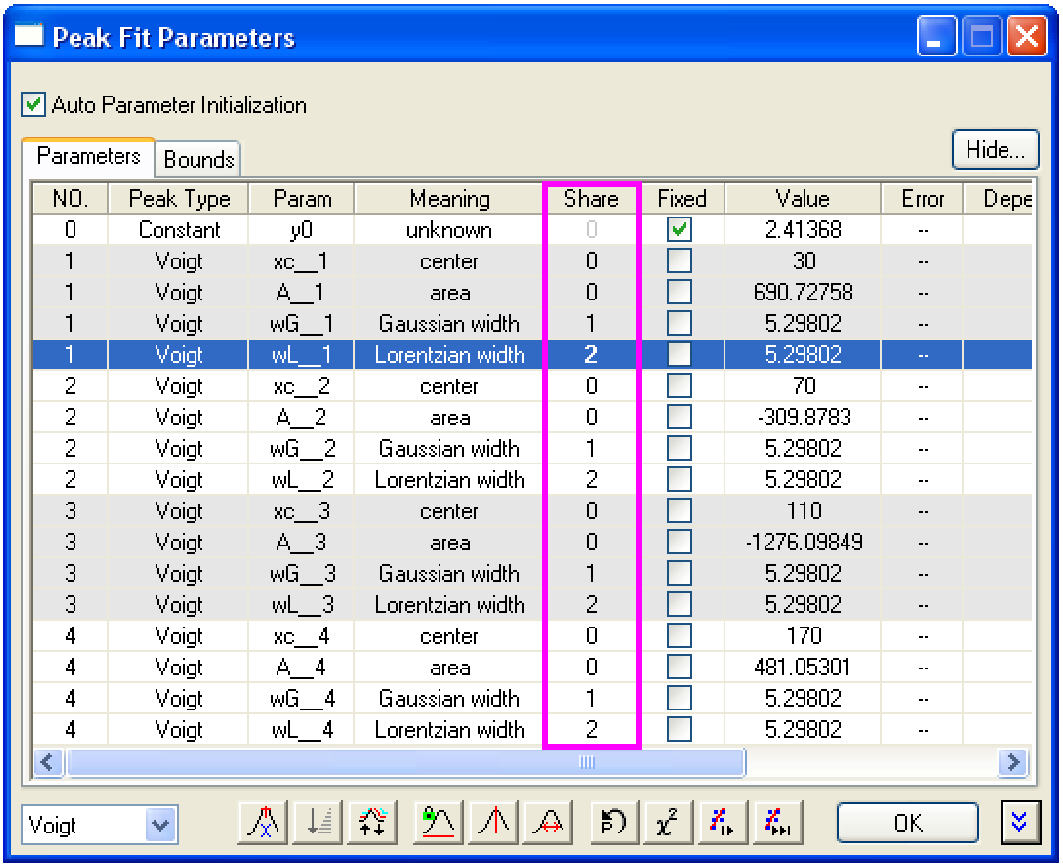
1. Start a new workbook and import the file *<Origin Program Folder>\Samples\Spectroscopy\Positive & Negative Peaks.dat*.
2. Highlight the second column and select **Analysis: Peaks and Baseline: Peak Analyzer** to open the **Peak Analyzer** dialog. In the first page (the **Start** page), select the **Fit Peaks** radio button in the **Goal** group. Then press the **Next** button to go to the next page.
3. In the **Baseline Mode** page, select **Constant** with the **Baseline Mode** drop-down list and choose **Mean** in the **Constant** group. Then click **Fit Peaks** in the wizard map to directly go to the **Fit Peaks** page.



4. In the **Fit Peaks** page:
    1. click **Fit Control** button to open the **Peak Fit Parameters** dialog. In the lower left corner of the dialog, set the fitting function to **Voigt**.
    2. Make sure the **Parameters** tab is active and then select **1** in the **Share** column of the **wG_1** row. Then right click on it and select **Apply Same "Share" to All wG**. Then you will find that all the parameters with the **wG** prefix are shared in the same group.

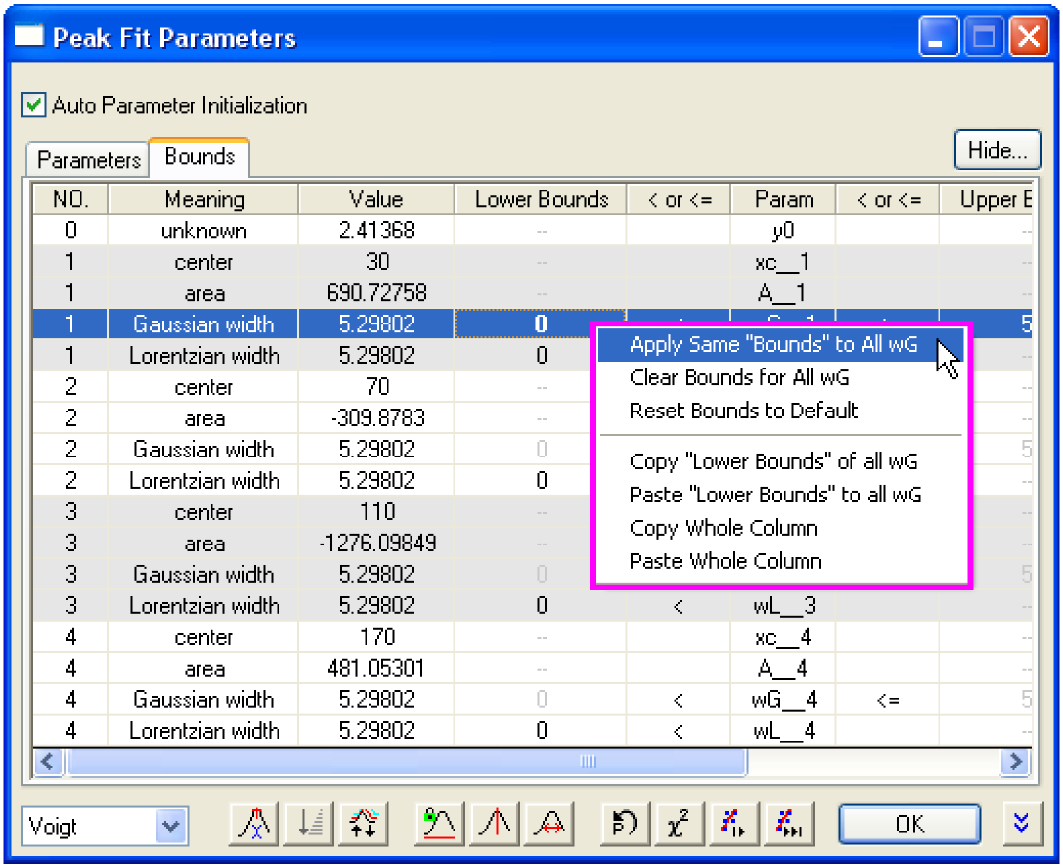


3. Then select **2** in the **Share** column of the **wL_1** row. Then right click on it and select **Apply Same "Share" to All wL**. Then you will find that all the parameters with the **wL** prefix are shared in the same group. After this, the **Parameters** tab should look like below:

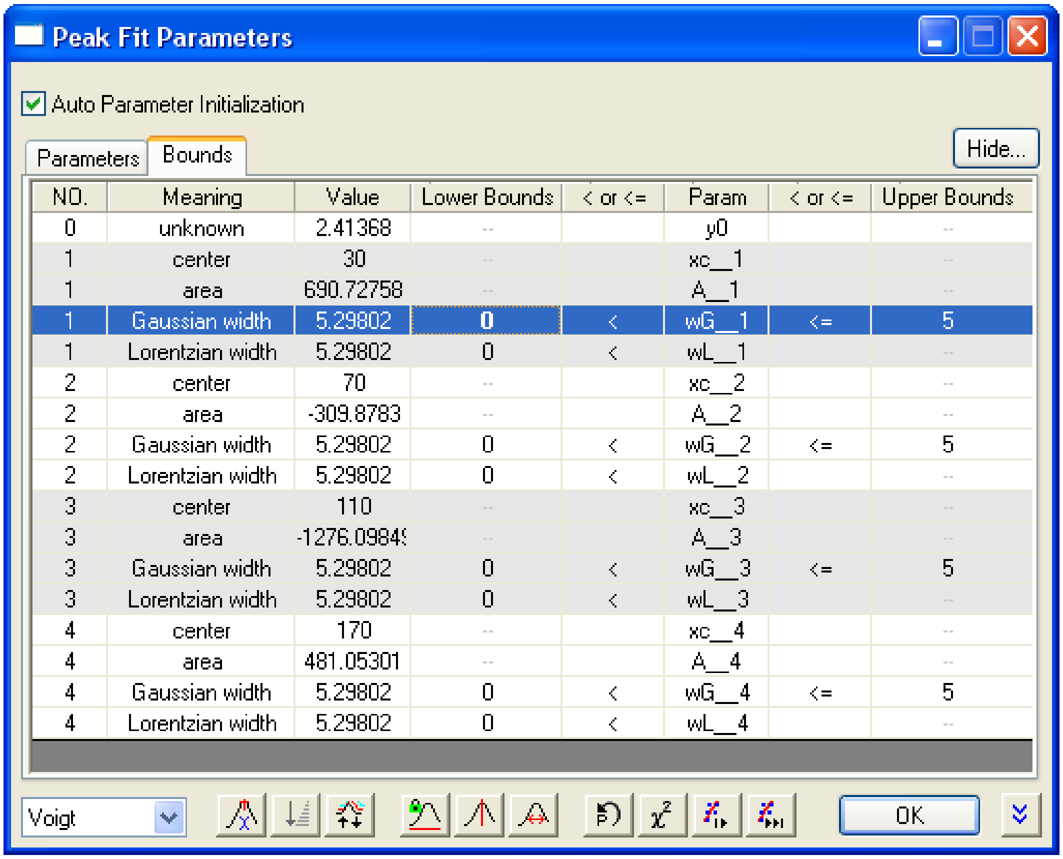


4. Activate the **Bounds** tab. Double-click in the cell in the first **Gaussian width** row and the second **< or <=** column. And you will find **<=** is shown in this cell (the **<** will be shown if you double-click in the cell for one more time). Then type **5** into the **Upper Bounds** column in the same row.

5. Then right click on it and select **Apply Same "Bounds" to All wG**.



After this, the **Bounds** tab should look like:



5. Click the **Fit Until Converged** button. When the fitting is done, click **OK** to close the dialog.
6. Back in the **Fit Peaks** page, click **Finish** to complete the analysis. See the results in the source workbook and the graph report.

3.5 Data Manipulation

Topics covered in this section:

1. Setting Column Values (Tutorials)
2. Worksheet Data Operations (Tutorials)
3. Pivot Table

3.5.1 Setting Column Values

- Setting Column Values

Setting Column Values

Summary

Origin provides several ways to fill a worksheet column with values. Use **Auto Fill** or script commands to fill a series of values. Use the **Set Values** dialog box to define a mathematical formula to generate or transform a data set. Refer to values in other columns from the same sheet or from other sheets and books. Select from a large collection of built-in functions to compute values. Create variables from metadata stored in worksheets or column headers, and use these variables in your column formula.

This tutorial will show you how to compute column values by:

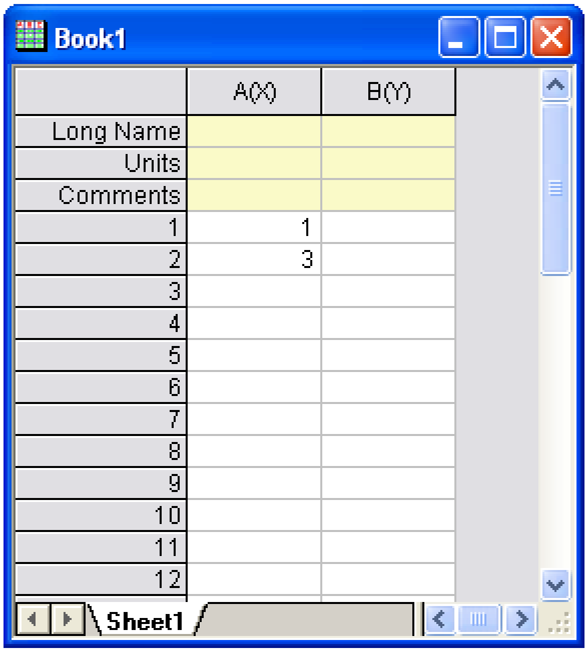
- Filling a Column with an Arithmetic Series
- Using Built-in Functions
- Using Other Columns
- Using Cell Values
- Using Variables from Workbook Metadata

Filling a Column with Arithmetic Series

Origin provides multiple methods to fill a column with arithmetic series.

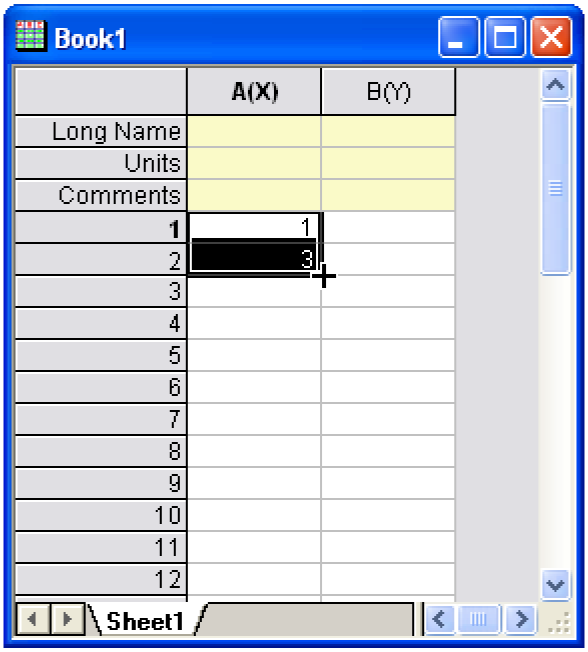
Using Auto Fill

Enter a few starting values in cells.

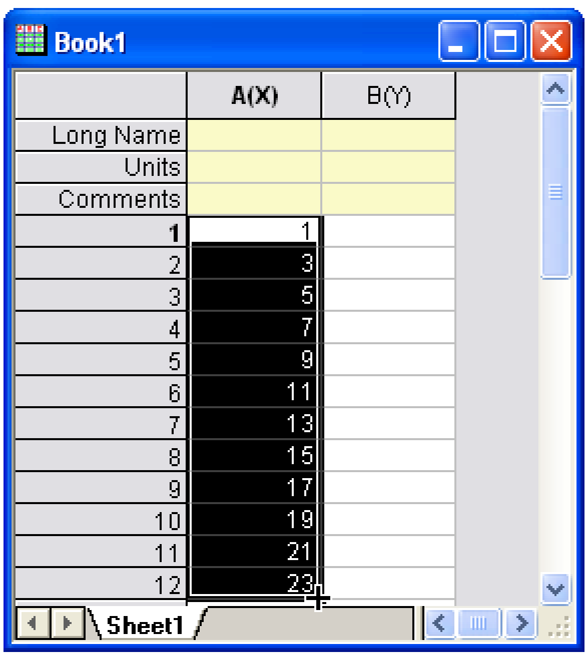


1. Select the two cells.

2. Move the mouse to the bottom right-hand corner of the second cell. The cursor will change to display "+".



3. Drag the mouse toward the bottom of the column. The column will be filled with 1, 3, 5, 7, ...



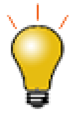
Note that a row can also be auto filled by dragging towards the right. To repeatedly copy values instead of generating new values, hold down the CTRL key and drag the mouse toward the bottom of the column.

Using Data List

Type the following script in the Command window.

```
col(B) = {1:2:23};
```

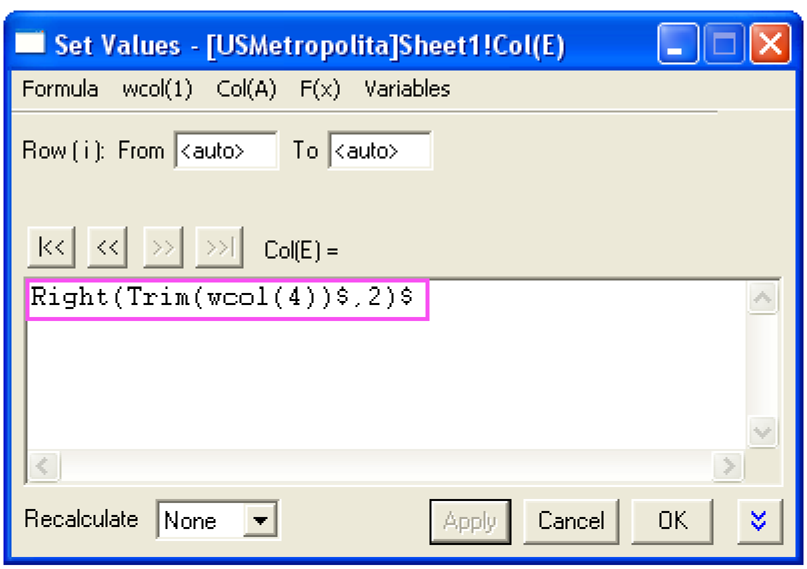
Column B will be filled with values: 1, 3, 5, 7, ..., 23



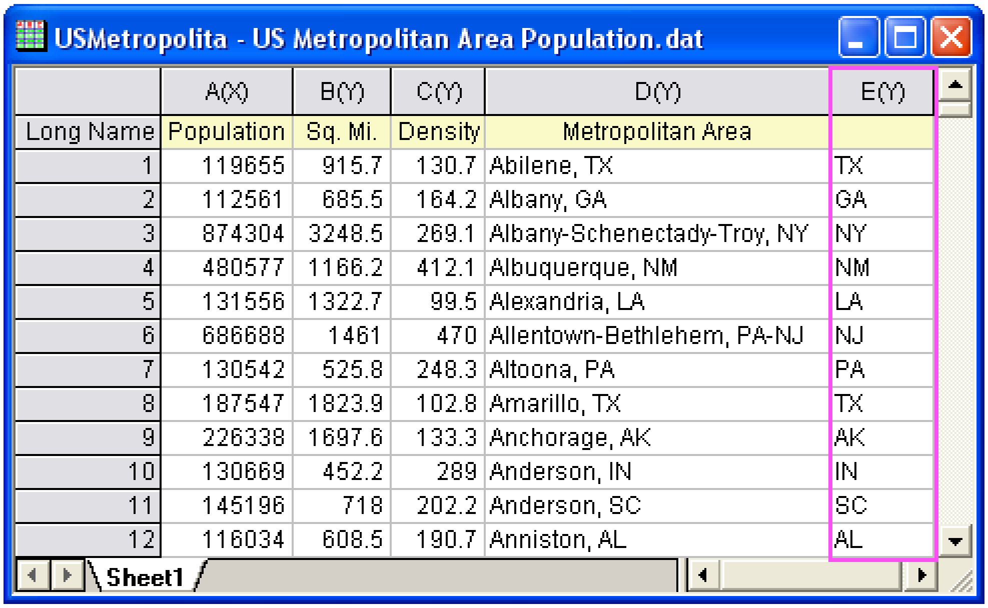
{v1:vstep:vn} produces the same result as the function data(v1,vn,vstep).

Using Built-in Functions

1. Create a new workbook. Import US Metropolitan Area Population.dat from the **\Samples\Data Manipulation** folder.
2. Click the **Add New Columns** button on the Standard toolbar to add a new column E. Highlight this column and right-click, and then click **Set Column Values...** to open the **Set Values** dialog.
3. In the **F(x)** menu, point to **String**, and then click **Right(str$,n)$** to insert this formula into the Column Formula panel.
4. Highlight the characters **str$**. In the **F(x)** menu, point to **String** and then click **Trim(str$[,n])$**. Your formula should look like this: **Right(Trim(str$)$,n)$**.
5. Highlight the characters **str$**. In the **wcol(1)** menu, point to **wcol(4)**. Your formula should look like this: **Right(Trim(wcol(4))$,n)$**.
6. Replace **n** with **2**. Your formula should look like this:



7. Click **OK**. The last column will fill with the state abbreviations from column 4.



	A(X)	B(Y)	C(Y)	D(Y)	E(Y)
Long Name	Population	Sq. Mi.	Density	Metropolitan Area	
1	119655	915.7	130.7	Abilene, TX	TX
2	112561	685.5	164.2	Albany, GA	GA
3	874304	3248.5	269.1	Albany-Schenectady-Troy, NY	NY
4	480577	1166.2	412.1	Albuquerque, NM	NM
5	131556	1322.7	99.5	Alexandria, LA	LA
6	686688	1461	470	Allentown-Bethlehem, PA-NJ	NJ
7	130542	525.8	248.3	Altoona, PA	PA
8	187547	1823.9	102.8	Amarillo, TX	TX
9	226338	1697.6	133.3	Anchorage, AK	AK
10	130669	452.2	289	Anderson, IN	IN
11	145196	718	202.2	Anderson, SC	SC
12	116034	608.5	190.7	Anniston, AL	AL

Note that some columns had two states at the end of the Metropolitan Area name, so to get both names change the formula to:

Right(Col(Metropolitan Area),Len(Col(Metropolitan Area))-Find(Col(Metropolitan Area),",")-1)$

When referring to another column in the same worksheet, you can use index, short name, or long name to identify the column.

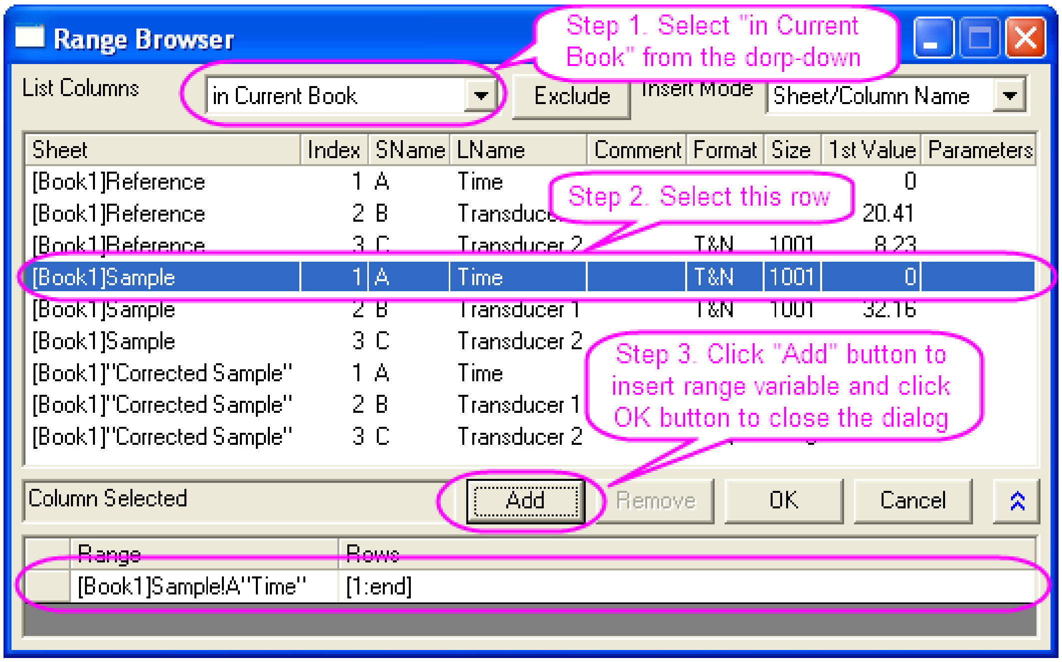
Using Other Columns

1. We will continue with the steps from above to show you how to use other columns in the Set Values dialog. Add a new column to the worksheet (right-click to the right of the last column in the worksheet and select **Add New Column** from the context menu). Change the Long Name of the column to "Population/Sq. Mi."
2. Highlight this column and right-click on it. Select **Set Column Values** to bring up the dialog. Click the Col(A) menu and choose **Col("Population"):A** and then enter the **/** character. Click the Col(A) menu again and choose **Col("Sq. Mi."):B**. The formula should look like: **Col("Population")/Col("Sq. Mi.")**
3. Click OK and the column will get computed using data from the other two columns.

Using Columns from Other Sheets

The **Set Values** dialog provides an **Insert** menu to easily insert range variables that point to columns in other books/sheets, which can then be used to compute column values for the current column.

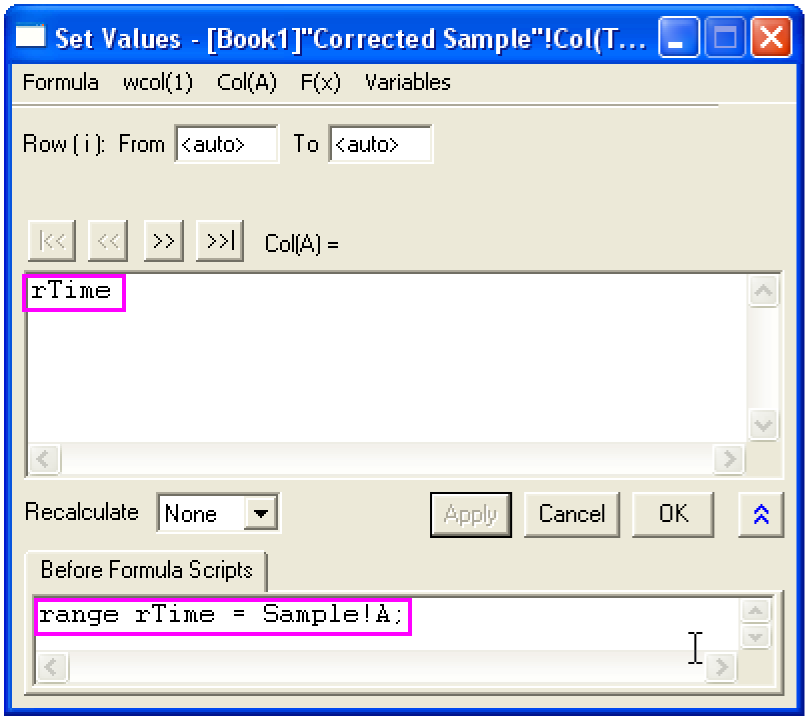
1. Open the project **Samples\Data Manipulation\Setting Column Values.OPJ** and switch to the **Columns from Other Sheets** subfolder.
2. Right-click on the **Sample** sheet and select **Duplicate Without Data**. Rename(by double-clicking on the current name) the new sheet as: **Corrected Sample**.
3. Now you will fill these three columns with data based on formulas that reference columns in the other sheets. Highlight the first column and right-click on it to select **Set Columns Values** to open the dialog. Select **Variables: Insert Range Variables** to open the **Range Browser** dialog. You will use this dialog to add a range variable to the **Before Formula Scripts** panel, according to the instructions in the image below:

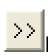


Click **OK** to close the dialog. **range r1 = Sample!A** will be automatically inserted into the **Before Formula Scripts** panel. Please rename it as:

```
range rTime = Sample!A;
```

4. Then enter **rTime** in the **Column Formula** and click the **Apply** button to generate data for the first column.



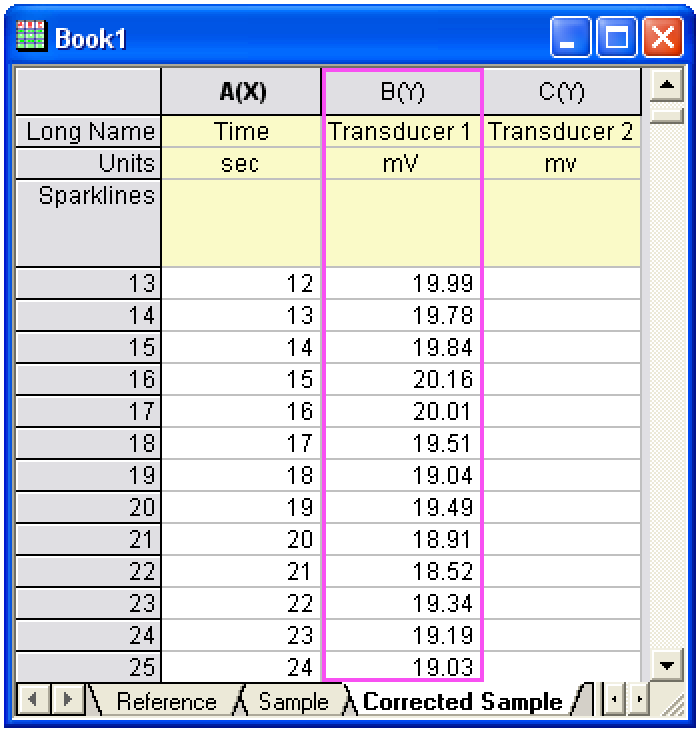
5. Click the  button to go to the next column. Then select **Variables: Insert Range Variables** to open the **Range Browser** dialog. You will use this dialog to insert two range variables to the **Before Formula Script** panel. Sort the data sets by long name (Click the **LName** heading to sort it). Insert two range variables that refer to **Transducer1** columns in both the **Reference** worksheet and the **Sample** worksheet. Rename them as:

```
range rRef = Reference!B;
range rSample = Sample!B;
```

6. Then input the following expression into the **Column Formula**:

```
rSample - (rSample[1] - rRef[1])
```

7. Click the **Apply** button to generate data for the second column of the **Corrected Sample** worksheet. Don't click the **OK** button yet.

8. 

Book1

	A(X)	B(Y)	C(Y)
Long Name	Time	Transducer 1	Transducer 2
Units	sec	mV	mv
Sparklines			
13	12	19.99	
14	13	19.78	
15	14	19.84	
16	15	20.16	
17	16	20.01	
18	17	19.51	
19	18	19.04	
20	19	19.49	
21	20	18.91	
22	21	18.52	
23	22	19.34	
24	23	19.19	
25	24	19.03	

Reference / Sample / Corrected Sample

You reference a particular cell value with square brackets, so [1] in the formula above means the first element.

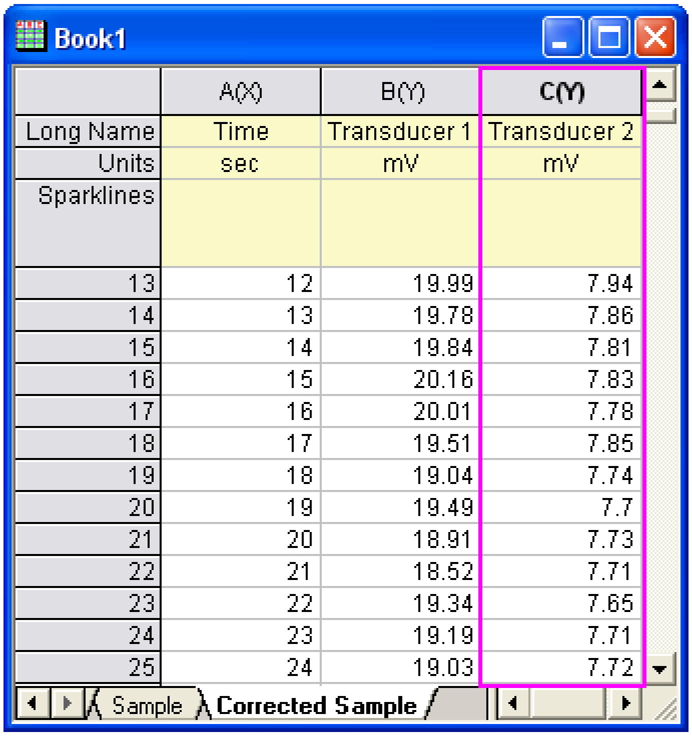
Your formulas can be saved and reloaded into other columns to generate new data.

1. Now we will edit the range variables in the **Before Formula Scripts** panel and use another expression to get the same results. Remove the column names **B"Transducer 1"** of the two range variables and select **F(x): Variables and Constants: wcol(_ThisNumCol)** in both lines so it looks as follows:

```
range rRef = Reference!WCol(_ThisColNum);
range rSample = Sample!WCol(_ThisColNum);
```

2. Leave the expressions in the **Column Formula** panel unchanged and click **Apply** to generate data. You will find that it gives you the same results, but the formula can now be applied to any column in the Corrected Sample worksheet, and the range variables will point to the same column, by index, in the Reference and Sample worksheets.
3. Select **Formula: Save** to open the **Save** dialog and name it "My Correction". Click the **OK** button to save it.

4. Click the >> button to go to the next column. Select **Formula: Load: My Correction** and click the **Apply** button to generate data for the third column.



Book1

	A(X)	B(Y)	C(Y)
Long Name	Time	Transducer 1	Transducer 2
Units	sec	mV	mV
Sparklines			
13	12	19.99	7.94
14	13	19.78	7.86
15	14	19.84	7.81
16	15	20.16	7.83
17	16	20.01	7.78
18	17	19.51	7.85
19	18	19.04	7.74
20	19	19.49	7.7
21	20	18.91	7.73
22	21	18.52	7.71
23	22	19.34	7.65
24	23	19.19	7.71
25	24	19.03	7.72

Sample | Corrected Sample

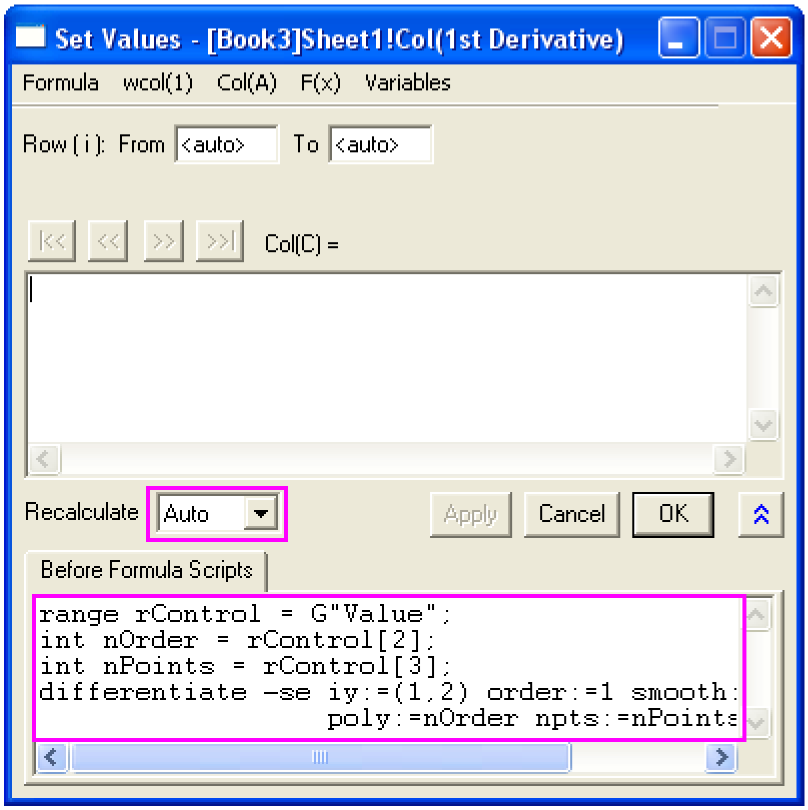
Using Cell Values

Values contained in specific worksheet cells can be referenced and used to compute the formula for setting column values. This provides an easy way to use worksheet cells as control cells for updating values in a column.

1. Open the project **\Samples\Data Manipulation\Setting Column Values.opj** and switch to the **Cells in a Worksheet** subfolder in Project Explorer.
2. Right-click on column C and select the **Set Column Values...** context menu to bring up the Set Values dialog.
3. Use the **Variables: Insert Range Variable...** menu item to open the Range Browser. Then select the column with the long name (LName) **Value**. Press the **Add** button to insert a variable. Press the **OK** button to close the dialog.
4. In the **Before Formula Scripts** panel, change the name of the range variable to be **rControl** and add these additional lines so that the script looks like below

```
range rControl = G"Value";
int nOrder = rControl[2];
int nPoints = rControl[3];
differentiate -se iy:=(1,2) order:=1 smooth:=1 poly:=nOrder
npts:=nPoints
oy:=(1,3);
```

5. The script calls the **differentiate** X-Function and passes the cell values from column G as arguments for polynomial order and number of points, which controls the Savitzky-Golay smoothing performed during the differentiation.
6. Set the **Recalculate** drop-down to **Auto** and press OK to close the dialog.



7. Now you can try to change the values in column G, to change the output.

Note: Allowed values of polynomial order are 1 to 9.

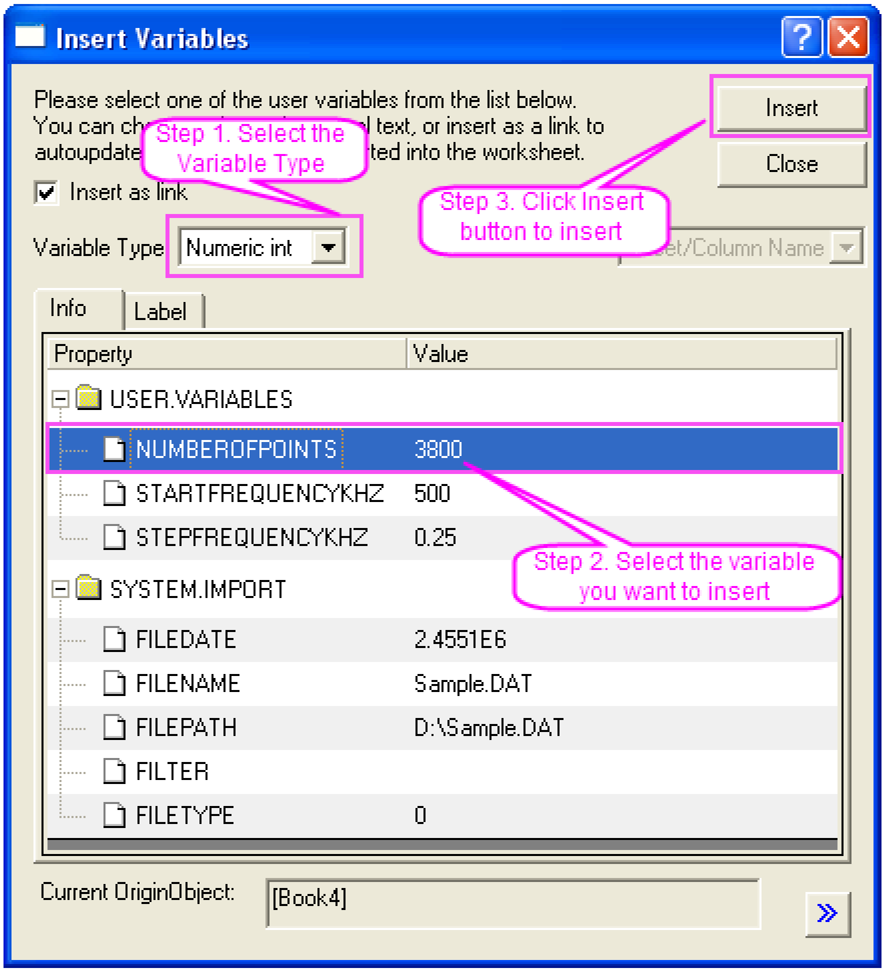
The graph shown in the worksheet was first created and then embedded into the worksheet by merging a group of cells.

Using Variables from Workbook Metadata

Metadata stored in the workbook, such as variables saved when importing data using the Import Wizard, can be referenced and used for computing column values.

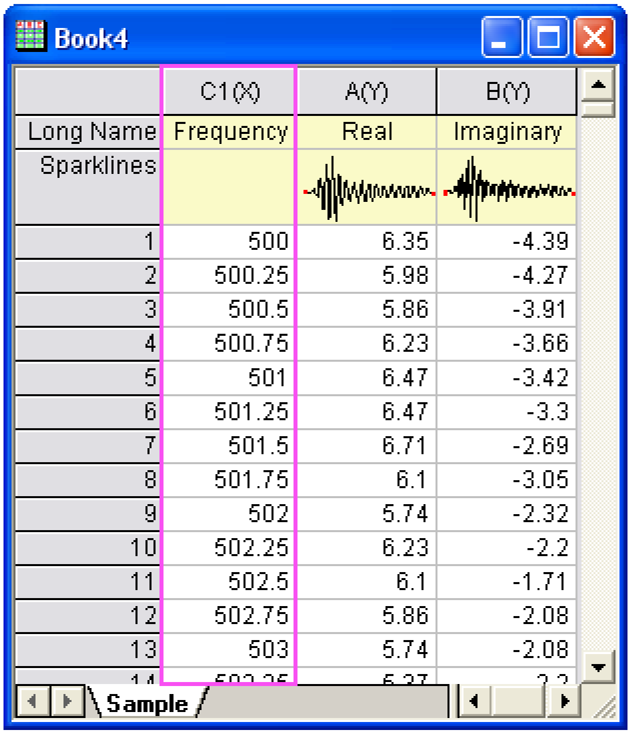
1. Open or continue working with **\Samples\Data Manipulation\Setting Column Values.OPJ**, and switch to the **Worksheet Metadata** subfolder from the Project Explorer window.
2. Select column A and right-click to select the **Insert** menu option. A new column is inserted to the left of column A.
3. Select the first column (this newly inserted column) and right-click on it. Then select the **Set Column Values** menu item to open the Set Values dialog.

4. Select the **Variables: Insert Info Variable** menu item to open the **Insert Variables** dialog. Select **Numeric int** from the **Variable Type** drop-down list. Then select **NumberOfPoints** and press the **Insert** button to insert this variable into the **Before Formula Scripts** panel.



5. Next, set **Variable Type** to **Numeric double**. Hold the **Shift** key down to select both **StartFrequencyKHz** and **StepFrequencyKHz**, and then press **Insert** to insert these two variables. Press the **Close** button to close the dialog.
6. In the upper **Column Formula** panel, input **{d1:d2:d1+(n1-1)*d2}** and then press the OK button to generate data and close the dialog. The column will be filled with frequency values.
7. Highlight the first and second columns, right-click on them and select **Set As: XYY** to change the plotting designations to X and Y. After you change the long name of the first column to

Frequency, the worksheet should look like:



3.5.2 Worksheet Data Operations

Topics covered in this section:
1. Extract Worksheet Data

<u>Worksheet Query</u>

Summary

This tutorial will show you how to use the **Worksheet Query** dialog.

Minimum Origin Version Required: Origin 8.5.1 SR0

What you will learn

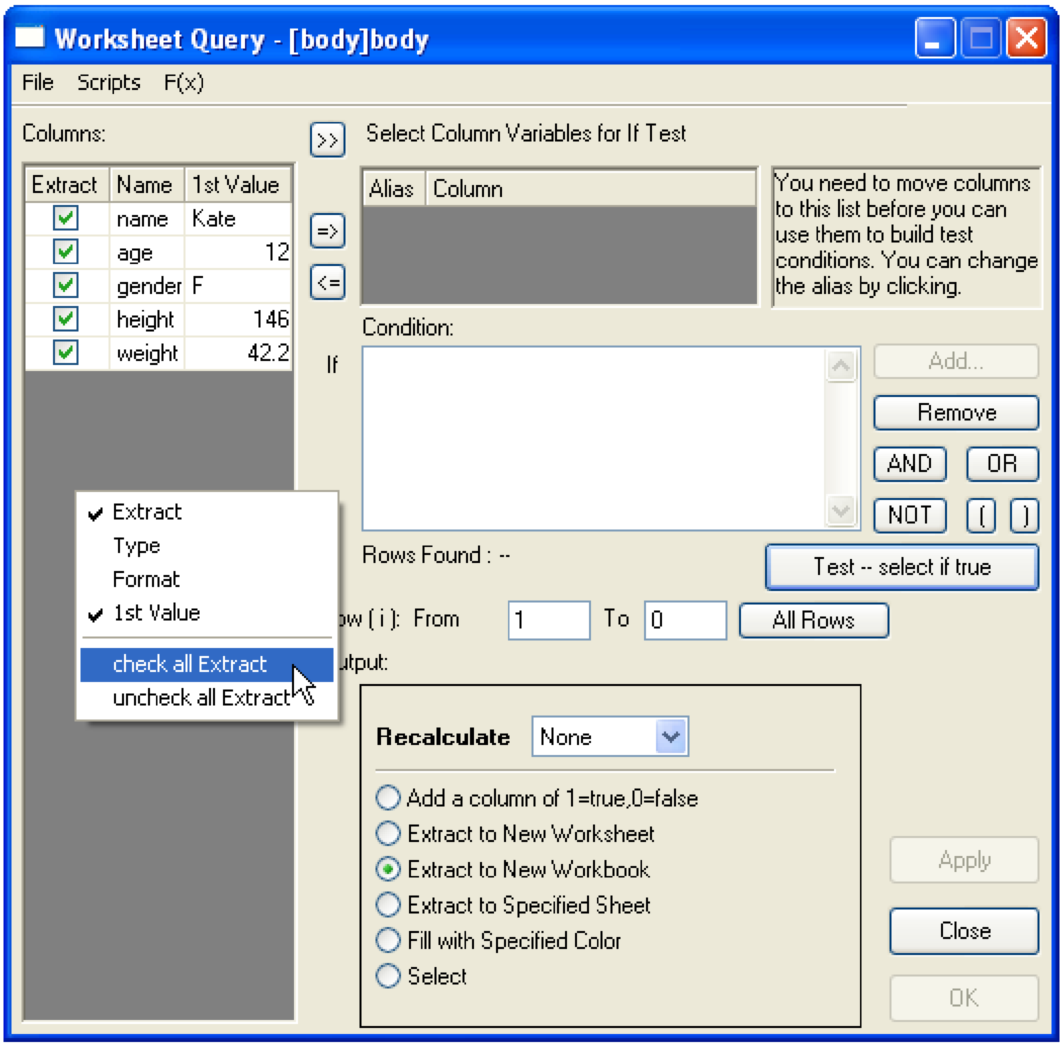
This tutorial will show you how to:

- Extract numeric and time data
- Use an alias in an extraction condition

- Use LabTalk functions in an extraction condition

Dialog overview

Start with a new workbook and import the file *\Samples\Statistics\body.dat*, click the menu item **Worksheet: Worksheet Query** to open the dialog as follows:



Basically, there are two main panels in the **Worksheet Query** dialog. The left panel lists all the columns in the active worksheet, you can right-click and select some column properties you want to see, such as *Format*, *1st Value*, etc.

Note the **Extract** column in this panel, only data selected in the Extract checkbox will be extracted.

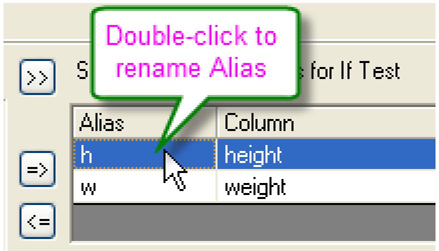
The right panel is where you set and test extract conditions. For example, you can select the column you want to use in the extract condition, and then click the 

button to move it into the **Select Column Variable for If Test** group.

Set the conditions

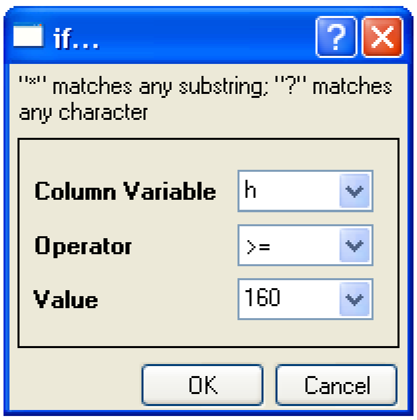
Extract Numerical Data

When there are available columns in the **Select Column Variable for If Test** group, the **Condition** edit box becomes editable for you to set conditions. For example, select *height* and *weight* to the group, Origin will automatically set an alias for each column. You can click into the **Alias** cell and rename the alias:

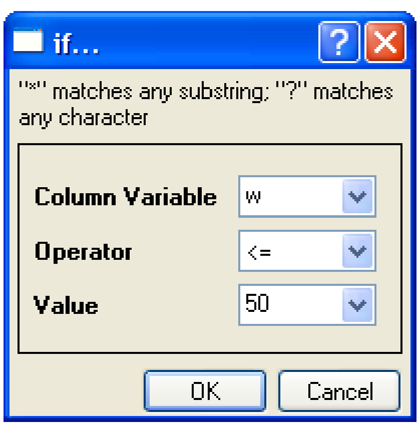


These alias can be used directly in the extract condition. Let's keep the default alias, *h* and *w* in this example.

The buttons on the right side of **Condition** edit box can be help to establish extract conditions. For example, to extract data that height is greater and equal to 160cm, highlight the column on **Select Column Variable for If Test** and click **Add** and build the first condition as follow:



When there are multiple conditions, you can also combine these conditions by logical operation. Click **AND** button to add one more condition. Then highlight w on **Select Column Variable for If Test** group and click **ADD** again, this time, we are looking for weight less than or equal to 50kg:



When the condition is done, click **All Rows** button to select all rows, then click the **Test -- select if true** button and Origin will return 5 found records. Of course, if you familiar to logical operation syntax, you can type the condition on the edit box directly:

```
h>=160 AND w<=50
```

Accept other default settings and click the **OK** button. A new workbook is created with these 5 records.

Extract Strings

When extracting strings, you need to enclose the string by double quotation marks ". For example, select the *gender* column into the **Select Column Variables for If Test** group. Using the alias *g*, you can extract all female data by:

```
g == "F"
```

Extract Time Data

Date and Time data are internally saved as numeric values in Origin. Date is the integer part of the numeric value, while Time is the fractional part. In Origin, you can use the *int()* and *frac()* functions to return the integer and fractional part of a number, and use the *Date(MM/DD/YY)* and *Time(HH:mm:ss)* functions to transfer string to time data. We can combine these functions to extract time data.

For example, using data from Import Time Data tutorial, you can extract data within time period 10:00 ~ 11:00 by:

```
frac(B) > Time(10:00:00) AND frac(B) < Time(11:00:00)
```

You can see Origin found 120 records. Similarly, if you want to extract *Date* data, you can try some condition like:

```
int(A) > Date(01/24/2004)
```

3.5.3 Pivot Table

Summary

The Pivot Table provides a quick way to summarize your data, and to analyze, compare, and detect relationships in your data. This tool can sort, count, sum, or compute minimum, maximum, or mean of data stored in a worksheet.

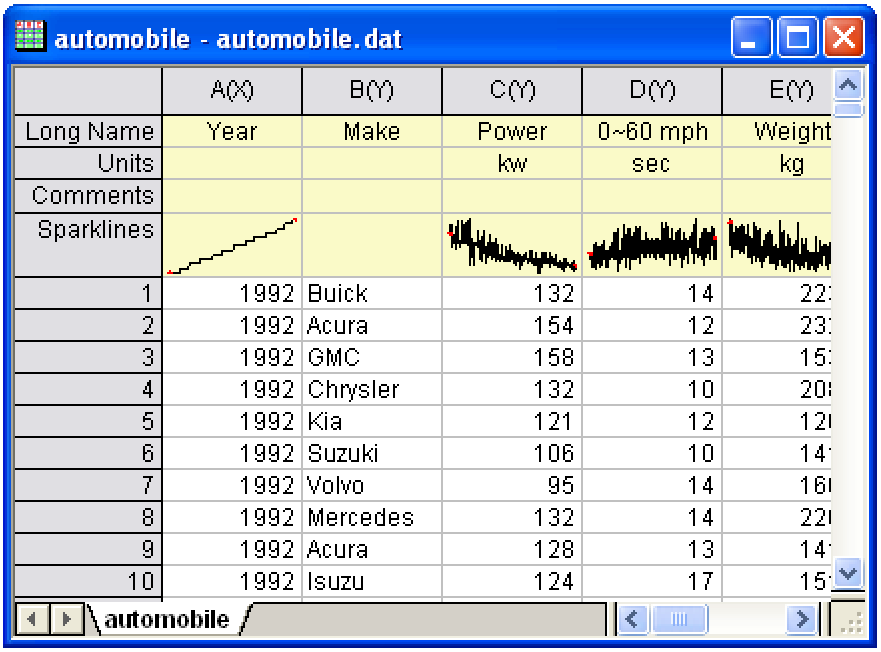
Minimum Origin Version Required: Origin 8.1 SR0

What you will learn

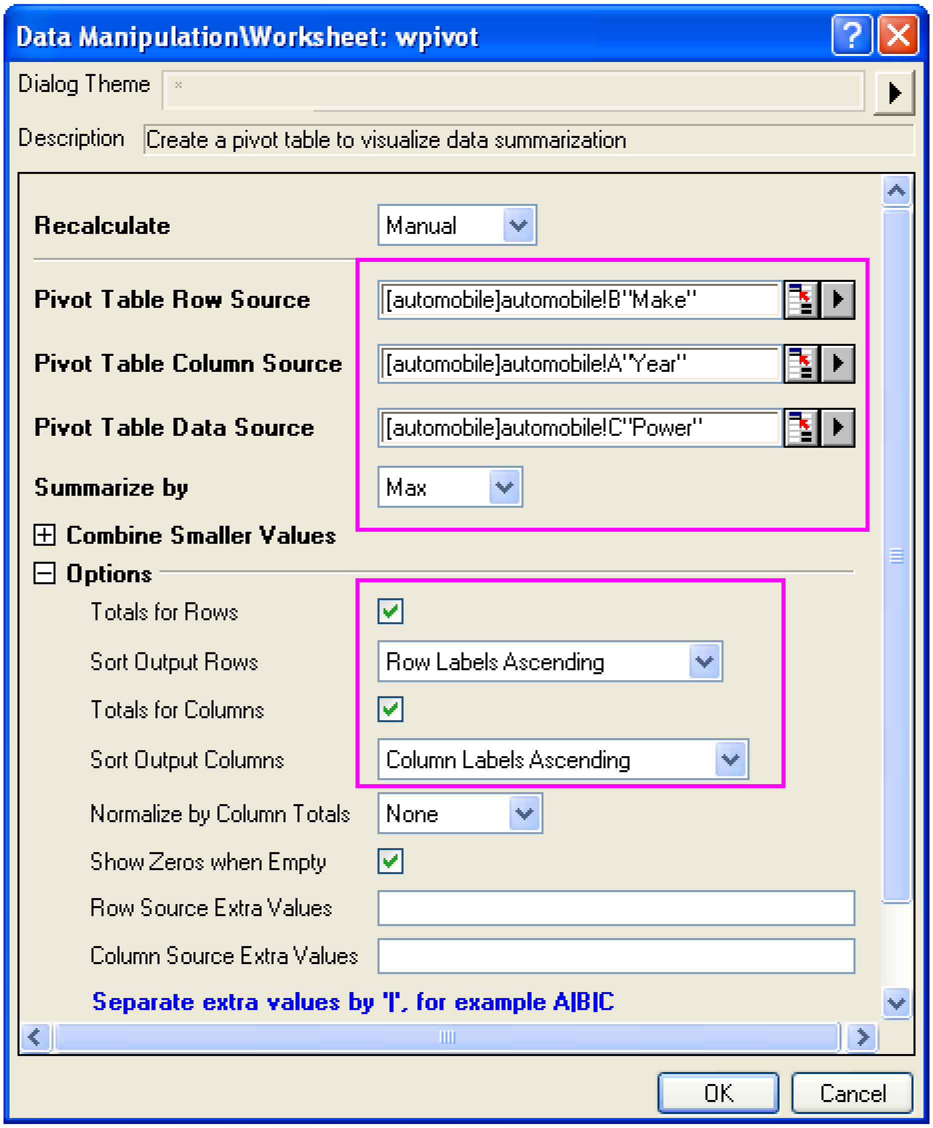
- How to summarize data by a Pivot Table.
- How to sort output by row or column totals in Pivot Table.
- How to combine small values in columns or rows, and custom extra value.

Create a Pivot Table

1. Import the Origin sample data *automobile.dat* which is located in <Origin Program Folder>\Samples\Statistics.

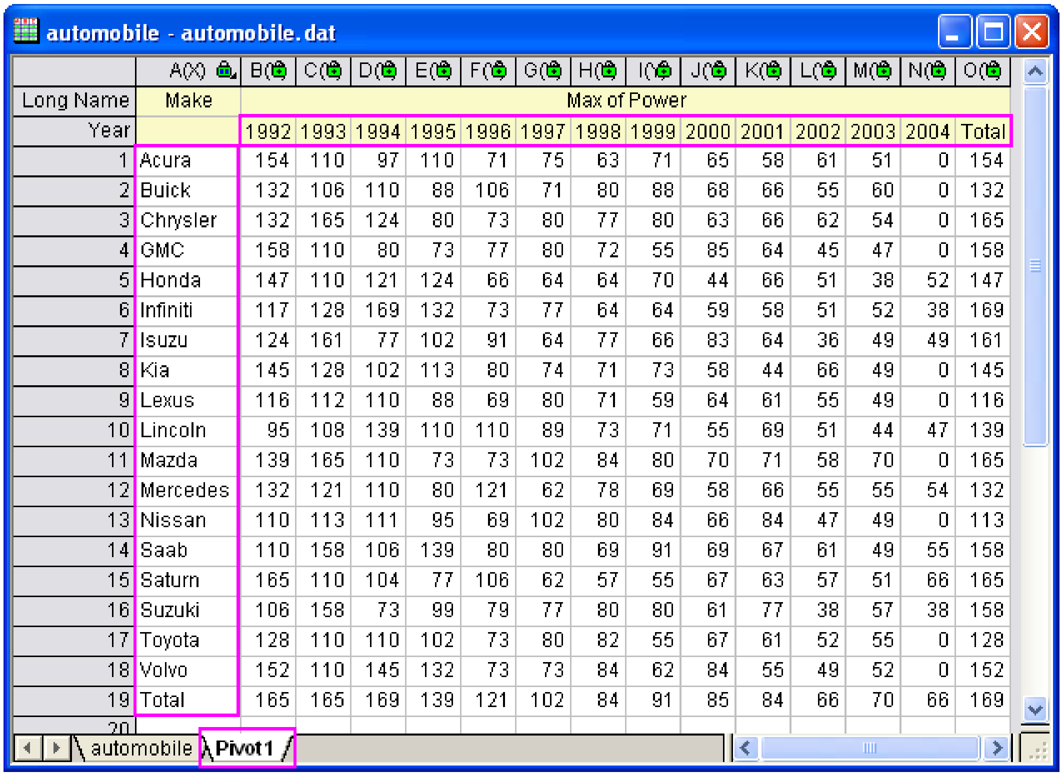


2. Highlight Column B and select **Worksheet: Pivot Table** from the main menu to open the dialog.
3. Specify the following settings in the dialog:
   - Select column A for **Pivot Table Column Sources**.
   - Select **Max** from the **Summarize by** drop-down list, and then select column C for **Pivot Table Data Source**.
   - Expanding **Options** branch, check **Total for Rows** and **Total for Columns** check boxes, and select **Row Label Ascending** from the **Sort Output Rows** drop-down list.



Data Manipulation\Worksheet: wpivot
Dialog Theme
Description
Create a pivot table to visualize data summarization
Recalculate
Manual
Pivot Table Row Source
[automobile]automobile!B"Make"
Pivot Table Column Source
[automobile]automobile!A"Year"
Pivot Table Data Source
[automobile]automobile!C"Power"
Summarize by
Max
Combine Smaller Values
Options
Totals for Rows
Sort Output Rows
Row Labels Ascending
Totals for Columns
Sort Output Columns
Column Labels Ascending
Normalize by Column Totals
None
Show Zeros when Empty
Row Source Extra Values
Column Source Extra Values
Separate extra values by '|', for example A|B|C
OK
Cancel

4. Click the **OK** button to create the **Pivot1** worksheet which will display as shown below:

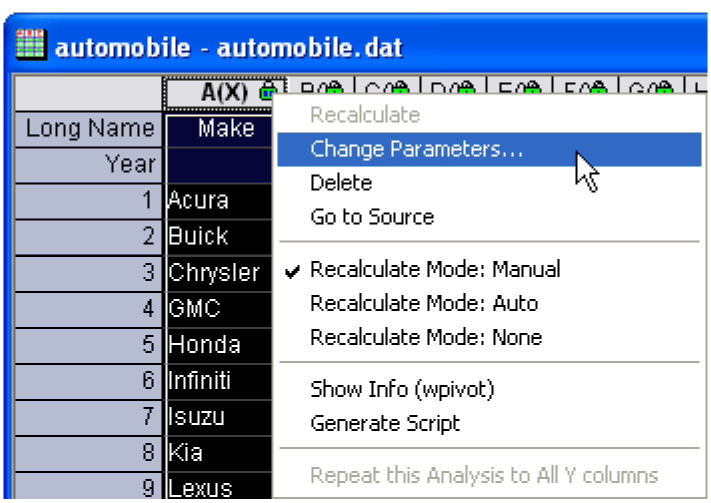


	A(X)	B	C	D	E	F	G	H	I	J	K	L	M	N	O
Long Name	Make	Max of Power													
Year		1992	1993	1994	1995	1996	1997	1998	1999	2000	2001	2002	2003	2004	Total
1	Acura	154	110	97	110	71	75	63	71	65	58	61	51	0	154
2	Buick	132	106	110	88	106	71	80	88	68	66	55	60	0	132
3	Chrysler	132	165	124	80	73	80	77	80	63	66	62	54	0	165
4	GMC	158	110	80	73	77	80	72	55	85	64	45	47	0	158
5	Honda	147	110	121	124	66	64	64	70	44	66	51	38	52	147
6	Infiniti	117	128	169	132	73	77	64	64	59	58	51	52	38	169
7	Isuzu	124	161	77	102	91	64	77	66	83	64	36	49	49	161
8	Kia	145	128	102	113	80	74	71	73	58	44	66	49	0	145
9	Lexus	116	112	110	88	69	80	71	59	64	61	55	49	0	116
10	Lincoln	95	108	139	110	110	89	73	71	55	69	51	44	47	139
11	Mazda	139	165	110	73	73	102	84	80	70	71	58	70	0	165
12	Mercedes	132	121	110	80	121	62	78	69	58	66	55	55	54	132
13	Nissan	110	113	111	95	69	102	80	84	66	84	47	49	0	113
14	Saab	110	158	106	139	80	80	69	91	69	67	61	49	55	158
15	Saturn	165	110	104	77	106	62	57	55	67	63	57	51	66	165
16	Suzuki	106	158	73	99	79	77	80	80	61	77	38	57	38	158
17	Toyota	128	110	110	102	73	80	82	55	67	61	52	55	0	128
18	Volvo	152	110	145	132	73	73	84	62	84	55	49	52	0	152
19	Total	165	165	169	139	121	102	84	91	85	84	66	70	66	169

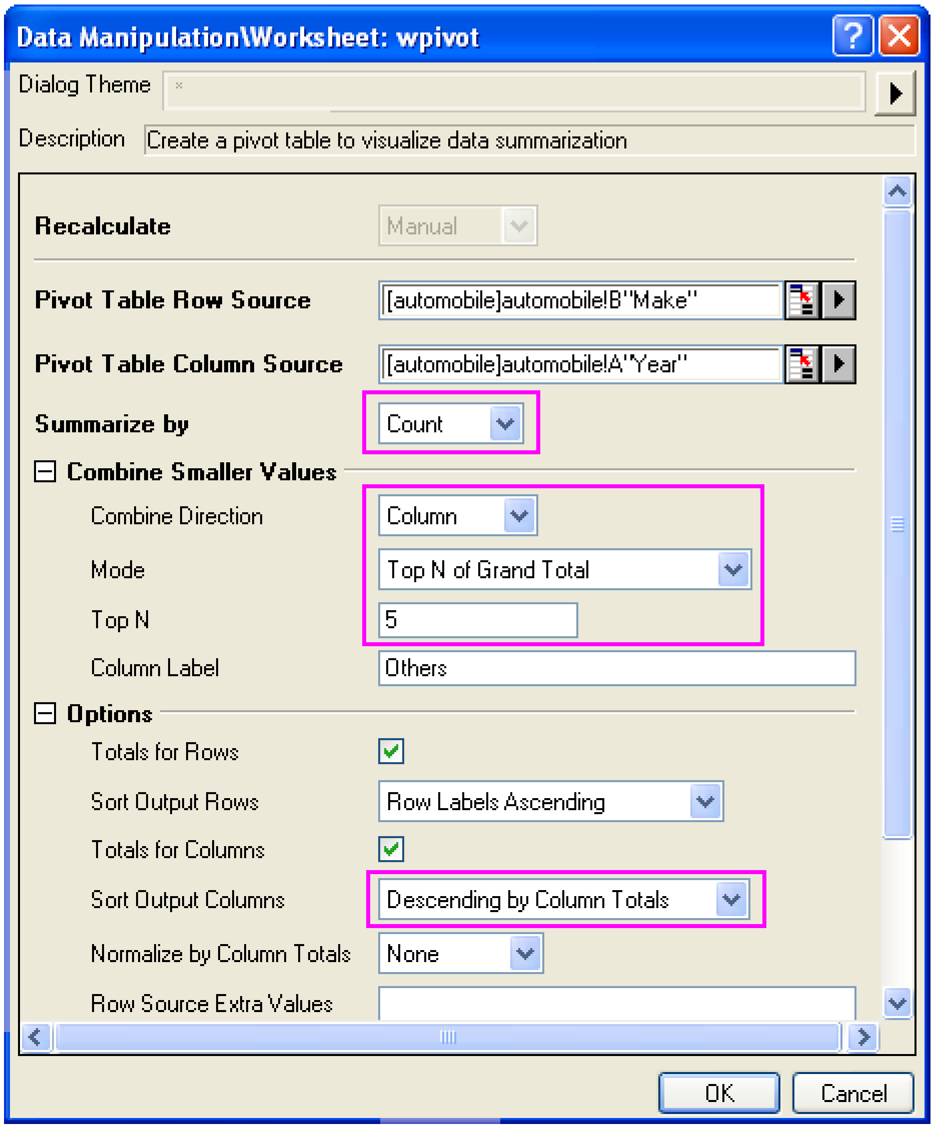
Combine Small Values

In this section, we will show you how to show just the top 5 columns of **Total** value, and combine small value columns into a column.

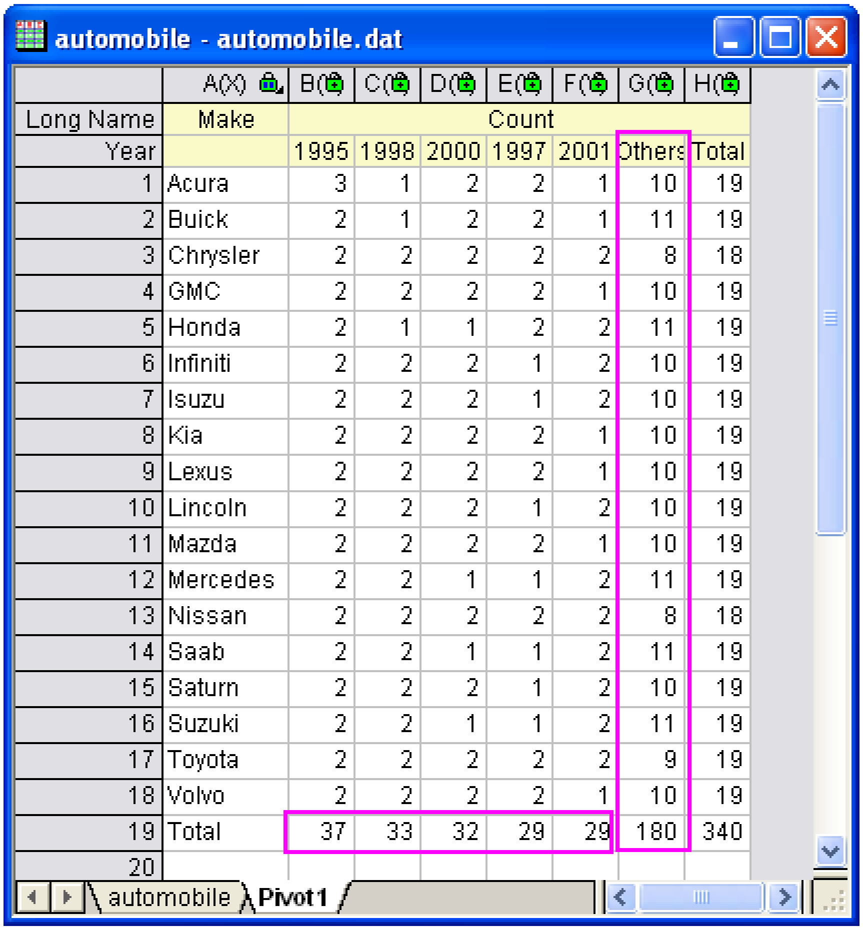
1. Based on the above example, click on the lock icon in the **Pivot1** worksheet, and select **Change Parameters** to open the dialog again.



2. Specify the following settings in the dialog:
   - Select **Count** in **Summarize by** drop-down list.
   - Expanding **Combine Smaller Values** branch, select **Column** in the **Combine Direction** drop-down list, **Top N of Grand Total** in **Mode** drop-down list, and enter **5** in **Top N** textbox.
   - Select **Descending by Column Totals** in **Sort Output Columns** drop-down list.

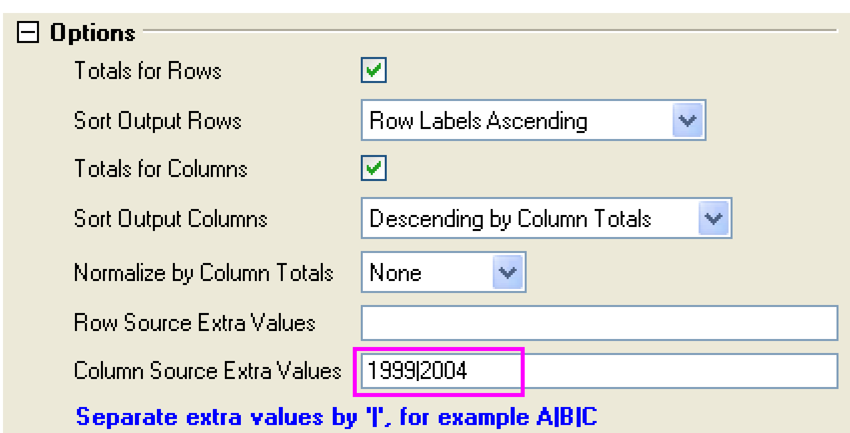


3. Click **OK** button. The pivot table shows the summarization of data by **Count**. And only the columns that are top 5 total values have been kept, and other columns are merged into a single column **Others**.

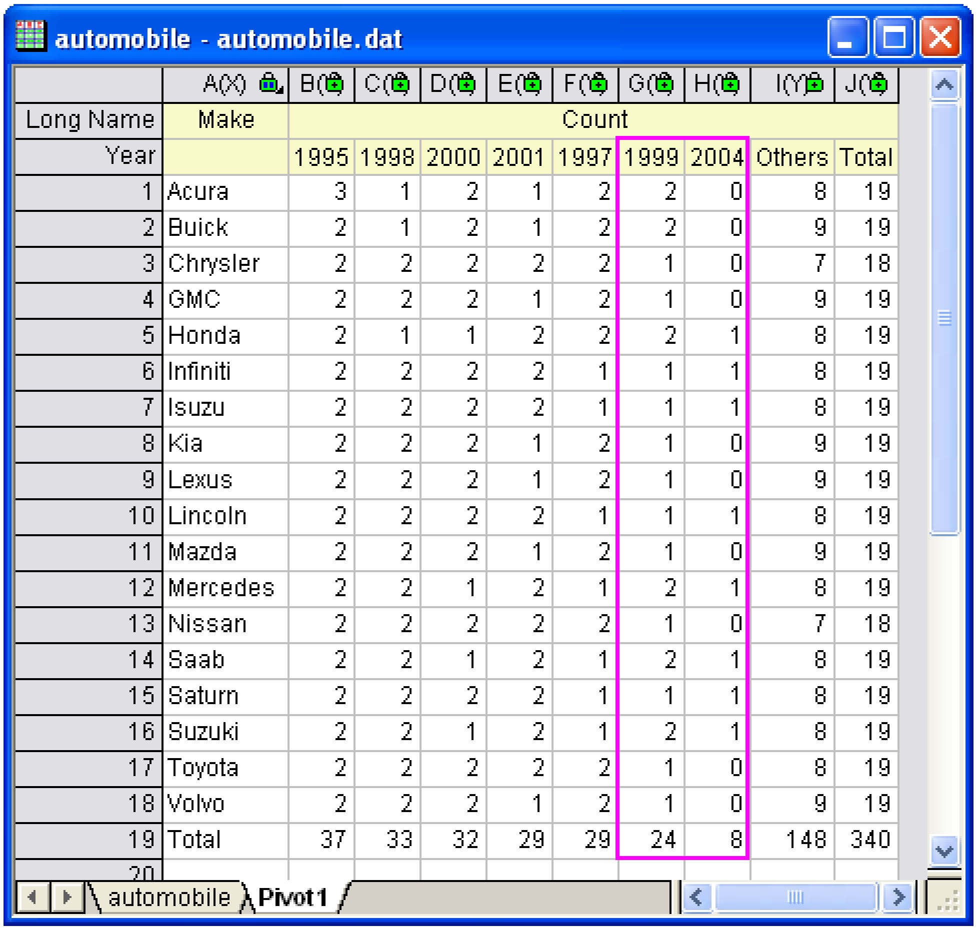


	A(X)	B	C	D	E	F	G	H
Long Name	Make	Count						
Year		1995	1998	2000	1997	2001	Others	Total
1	Acura	3	1	2	2	1	10	19
2	Buick	2	1	2	2	1	11	19
3	Chrysler	2	2	2	2	2	8	18
4	GMC	2	2	2	2	1	10	19
5	Honda	2	1	1	2	2	11	19
6	Infiniti	2	2	2	1	2	10	19
7	Isuzu	2	2	2	1	2	10	19
8	Kia	2	2	2	2	1	10	19
9	Lexus	2	2	2	2	1	10	19
10	Lincoln	2	2	2	1	2	10	19
11	Mazda	2	2	2	2	1	10	19
12	Mercedes	2	2	1	1	2	11	19
13	Nissan	2	2	2	2	2	8	18
14	Saab	2	2	1	1	2	11	19
15	Saturn	2	2	2	1	2	10	19
16	Suzuki	2	2	1	1	2	11	19
17	Toyota	2	2	2	2	2	9	19
18	Volvo	2	2	2	2	1	10	19
19	Total	37	33	32	29	29	180	340
20								

4. To show additional values for **1999** and **2004** columns in the pivot table, reopen the **wpivot** dialog. Then enter *1999|2004* in the **Column Source Extra Values** textbox.



5. Click **OK** button. **1999** column and **2004** column are displayed in the pivot table.



	A(X)	B(Y)	C(Y)	D(Y)	E(Y)	F(Y)	G(Y)	H(Y)	I(Y)	J(Y)
Long Name	Make	Count								
Year		1995	1998	2000	2001	1997	1999	2004	Others	Total
1	Acura	3	1	2	1	2	2	0	8	19
2	Buick	2	1	2	1	2	2	0	9	19
3	Chrysler	2	2	2	2	2	1	0	7	18
4	GMC	2	2	2	1	2	1	0	9	19
5	Honda	2	1	1	2	2	2	1	8	19
6	Infiniti	2	2	2	2	1	1	1	8	19
7	Isuzu	2	2	2	2	1	1	1	8	19
8	Kia	2	2	2	1	2	1	0	9	19
9	Lexus	2	2	2	1	2	1	0	9	19
10	Lincoln	2	2	2	2	1	1	1	8	19
11	Mazda	2	2	2	1	2	1	0	9	19
12	Mercedes	2	2	1	2	1	2	1	8	19
13	Nissan	2	2	2	2	2	1	0	7	18
14	Saab	2	2	1	2	1	2	1	8	19
15	Saturn	2	2	2	2	1	1	1	8	19
16	Suzuki	2	2	1	2	1	2	1	8	19
17	Toyota	2	2	2	2	2	1	0	8	19
18	Volvo	2	2	2	1	2	1	0	9	19
19	Total	37	33	32	29	29	24	8	148	340
20										

3.6 Analysis Templates

Topics covered in this section:

1. Creating and Using Analysis Templates
2. Creating Analysis Templates using Set Column Value
3. Creating a Custom Report Sheet

3.6.1 Creating and Using Analysis Templates

Summary

Routine tasks can be simplified by creating an Analysis Template. Such templates can contain multiple analysis results and also custom report sheets. A new instance of the template can then be opened any time and source data can be changed to update all analysis results and custom reports.

Minimum Origin Version Required: Origin 8.0 SR6

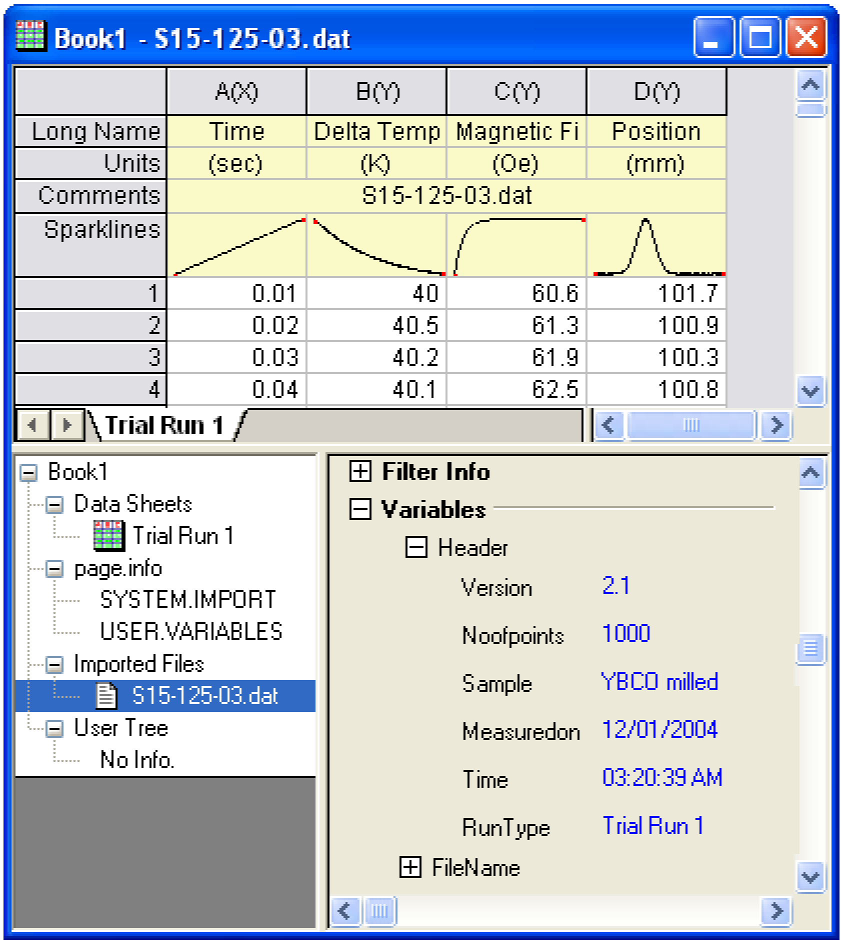
What you will learn

- How to create analysis template (OGW)
- How to re-use analysis template with new data

Steps

Importing Data

1. Start with a new workbook.
2. Click the **Import Wizard** button on the Standard Toolbar, or invoke the **Import: Import Wizard...** menu item. the Import Wizard dialog will open.
3. Click the **...** button next to the **File** edit box and select the file **<Origin EXE folder>\Samples\Import and Export\S15-125-03.dat**.
4. Verify that the **Import Filters for Current Data Type** drop-down shows **Data Folder: VarFromFileNameAndHeader**.
5. Change the **Import Mode** drop-down to **Replace Existing Data**.
6. This is an import filter shipped with the sample file, that specifies how to import the file and what header and file name strings to parse to create import variables. Walk through the wizard pages to view the settings (Optional) and then click Finish button to import the file.
7. Right-click on workbook title bar and select **Show Organizer** to turn on organizer panel. Expand branches and verify that variables have been created and saved, as in the picture below:



Performing Analysis

1. Highlight column D and use the **Analysis: Fitting: Nonlinear Curve Fit...** menu item to open the **NLFit** dialog.
2. Fit the data with **Gauss** function. This will add a hierarchical report sheet to the book, with result tables and embedded graph with data and fit curve.
3. Go to the **FitNL1** report sheet and double-click to open the graph containing data and fit curve. Perform some customization of the graph such as adding grid lines, changing font size etc. Click the **X** button on the graph window to put the modified graph back into the report.

Saving the Analysis Template

1. Go to the source data sheet of the workbook, which should be the first sheet. Select the **Worksheet: Clear Worksheet...** menu item and press **OK** in the dialog that opens. This will clear all the data from the sheet. The analysis report sheet will now be empty. Clearing the data is optional, and it makes the size of the analysis template file to be smaller.
2. Use the **File: Save Window As...** menu item and save the book as an **OGW** file under your **User Files Folder** with a suitable name such as **Analysis Template**. This OGW file can now serve as an Analysis Template for future analysis of similar data.

Re-using the Analysis Template

1. Start a new project and then select the menu item **File: Recent Books** and from the fly-out options select the Analysis Template saved earlier.
2. Make the data sheet active, and select **File: Import: Import Wizard...** and select the file **<Origin EXE path>\Samples\Import and Export\S21-235-07.dat**.
3. Make sure the filter drop-down shows **VarsFromFileNameAndHeader** and change the **Import Mode** drop-down to **Replace Existing Data** and click **Finish**.
4. Press the **Recalculate** button on the Standard toolbar. Origin will recalculate the analysis results and update the custom report sheet links, and at this point you can view and print the custom report sheet.

3.6.2 Creating Analysis Templates using Set Column Value

Summary

This tutorial will demonstrate how to add a column, set up **Before Formula Script** and have that script run whenever data changes in other columns. This technique can be used to create an **Analysis Template** for repeated analysis of similar data.

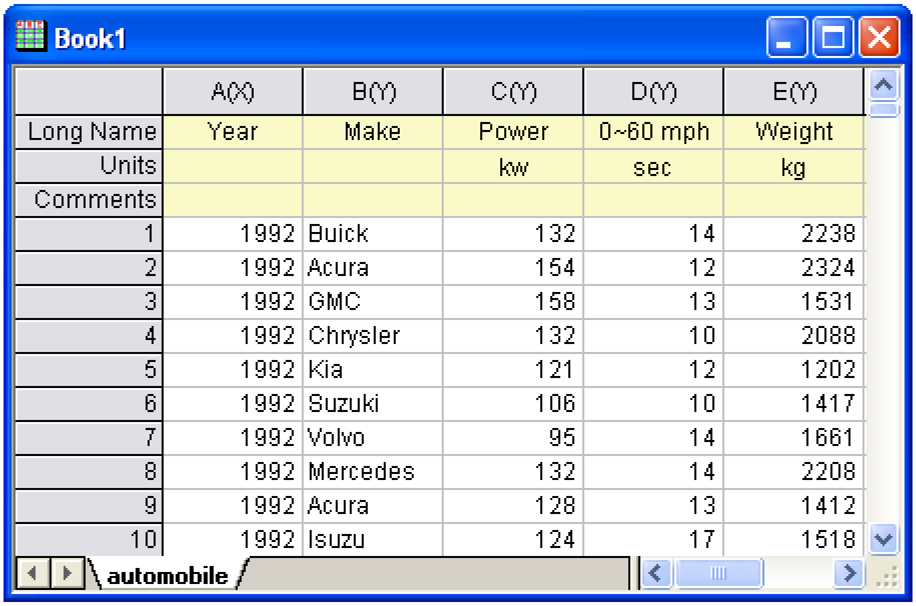
Minimum Origin Version Required: Origin 8.0 SR6

What you will learn

- How to use Set Column Values to create an analysis template
- How to select rows via the *Go to* function

Steps

1. Import the data from *\Samples\Statistics\automobile.dat* into a newly created workbook, as below. In this example, we will extract data, according to the *Make* column, into different worksheets.



Book1

	A(X)	B(Y)	C(Y)	D(Y)	E(Y)
Long Name	Year	Make	Power	0~60 mph	Weight
Units			kw	sec	kg
Comments					
1	1992	Buick	132	14	2238
2	1992	Acura	154	12	2324
3	1992	GMC	158	13	1531
4	1992	Chrysler	132	10	2088
5	1992	Kia	121	12	1202
6	1992	Suzuki	106	10	1417
7	1992	Volvo	95	14	1661
8	1992	Mercedes	132	14	2208
9	1992	Acura	128	13	1412
10	1992	Isuzu	124	17	1518

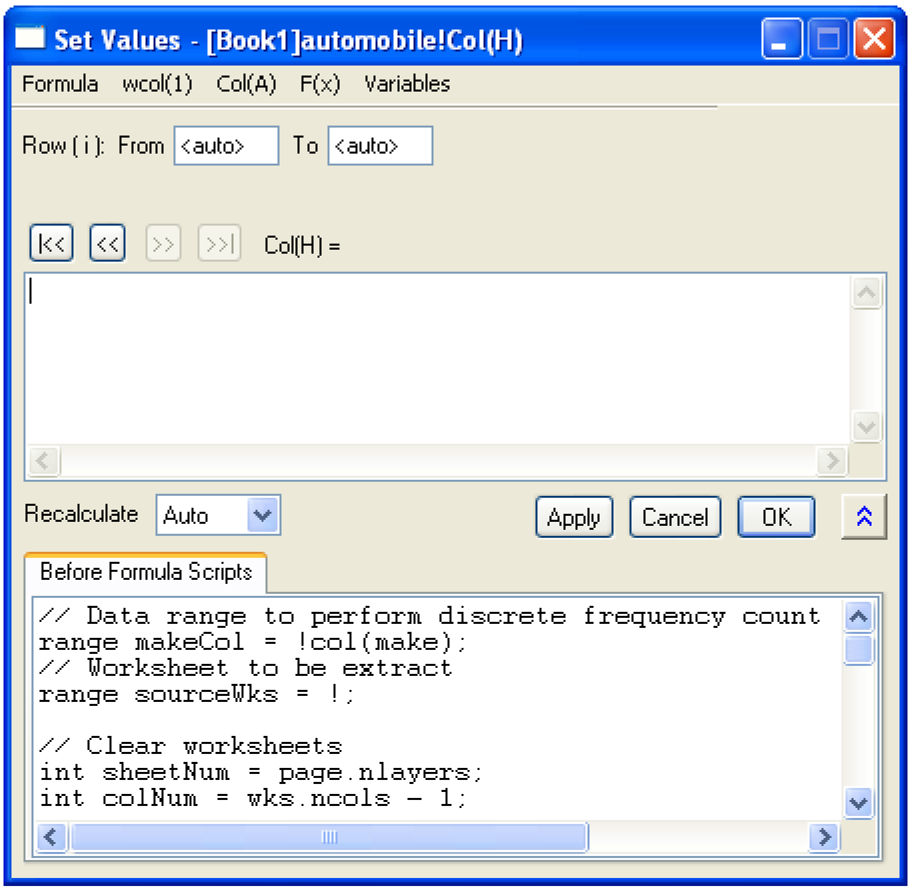
automobile

2. Add an empty column to the worksheet and bring up the **Set Column Values** dialog of the column. In the **Before Formula Script** box, enter the script below.

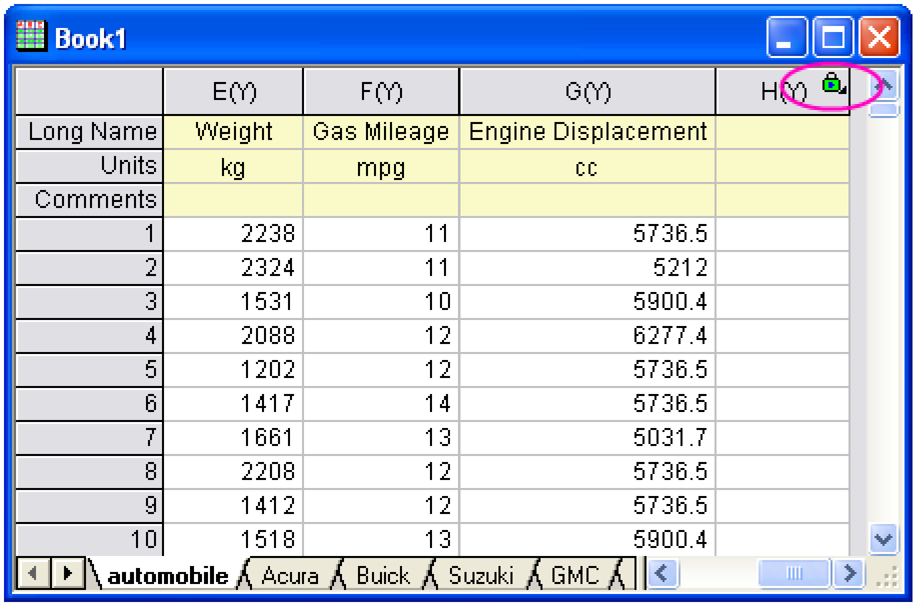
```
// Data range on which to perform discrete frequency count
range makeCol = !col(make);
// Worksheet to be extracted
range sourceWks = !;

// Clear worksheets
int sheetNum = page.nlayers;
int colNum = wks.ncols - 1;
if (sheetNum>1)
{
        for (jj=2; jj<=sheetNum; jj++)
        {
                layer -d 2;
        }
}

// Tree variable to hold discfreqs outputs
tree tr;
// Perform discrete frequency count
discfreqs irng:=makeCol rd:=tr;
// String array to get result from tree
StringArray sa;
sa.append(tr.FreqCount1.Data1);

if( sa.GetSize() != NANUM )
{
        // Loop to extract data
        for (ii=1; ii<=sa.GetSize(); ii++)
        {
                string sn$ = sa.GetAt(ii)$;
                // Extract condition string
```

```
                string cond$ = "makeCol$ = " + sn$;
                // Create worksheet with different Make name
                newsheet name:=sn$ cols:=colNum outname:=on$
active:=0;
                // Extract data
                wxt test:=cond$ iw:=sourceWks c2:=colNum
ow:=on$;
        }
}
```

3. This script will first perform a discrete frequency count on the *Make* column to get distinct values for *Make*. It will then create a new worksheet for each brand and extract data into these sheets.

4. 

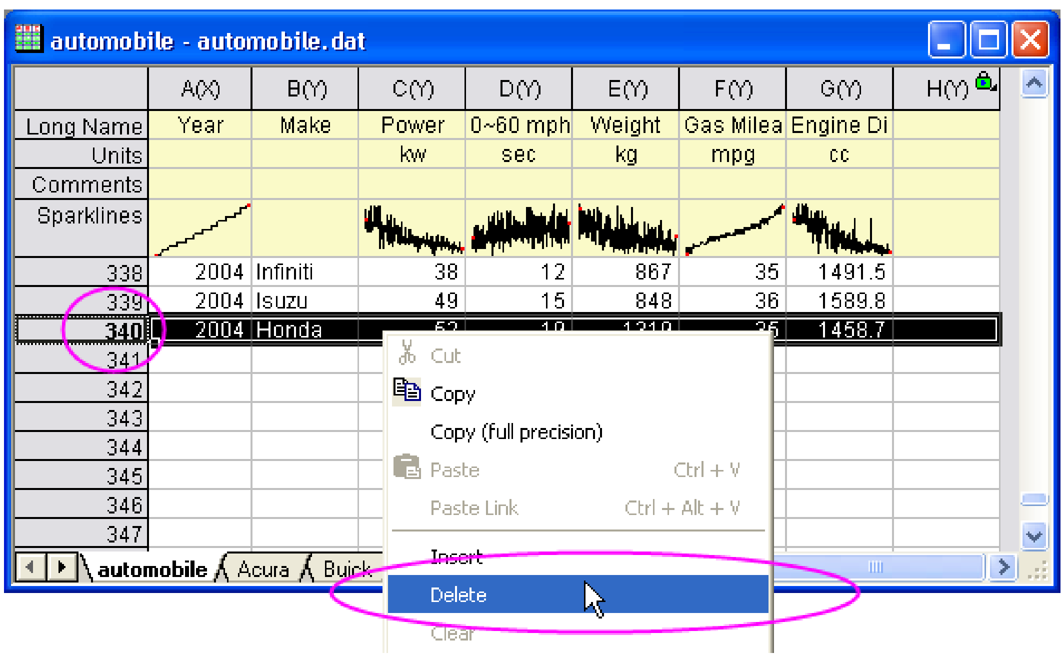
5. Make sure the recalculate mode is set to **Auto** and click **OK**. The data will be separated into different worksheets. Then the empty column (H) with a green lock icon indicates that this procedure can be updated automatically.



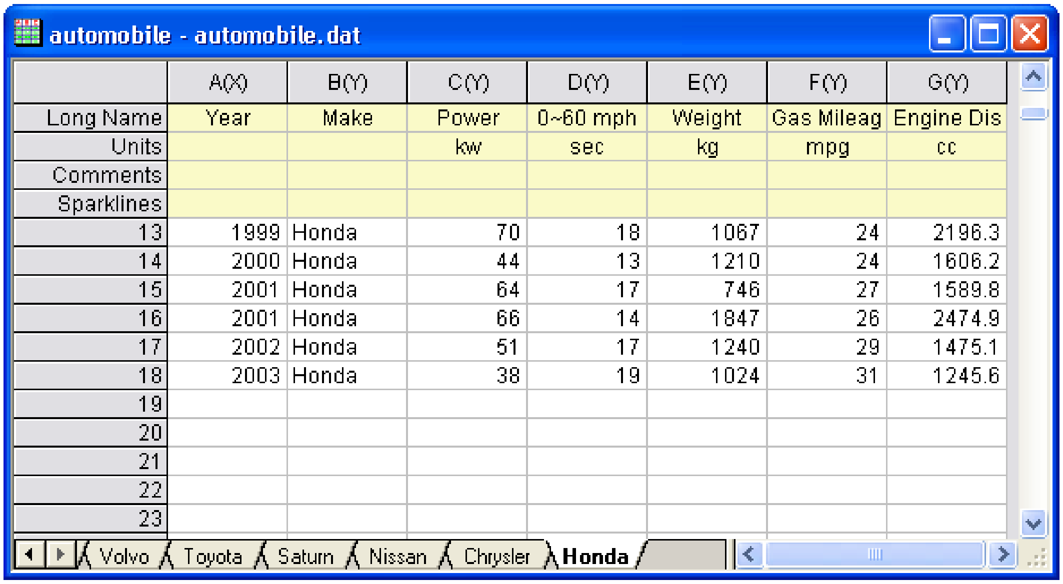
6. There are 18 makes of cars in the source data, so 18 new worksheets were created. Now we can check whether auto-update works.

   Go to the **Honda** tab. Notice that there are 19 rows.

7. Switch to the first sheet. Scroll down and delete the last row.



   Then auto-update will be triggered. Go to the **Honda** tab. You will see there are only 18 rows.


automobile - automobile.dat

	A(X)	B(Y)	C(Y)	D(Y)	E(Y)	F(Y)	G(Y)
Long Name	Year	Make	Power	0~60 mph	Weight	Gas Mileag	Engine Dis
Units			kw	sec	kg	mpg	cc
Comments							
Sparklines							
13	1999	Honda	70	18	1067	24	2196.3
14	2000	Honda	44	13	1210	24	1606.2
15	2001	Honda	64	17	746	27	1589.8
16	2001	Honda	66	14	1847	26	2474.9
17	2002	Honda	51	17	1240	29	1475.1
18	2003	Honda	38	19	1024	31	1245.6
19							
20							
21							
22							
23							

Volvo / Toyota / Saturn / Nissan / Chrysler / **Honda**

3.6.3 Creating a Custom Report Sheet

Summary

Worksheets in Origin can be customized by merging cells and placing various objects such as graphs, external images, links to variables and tables/cells in other sheets, in order to create custom reports. Such custom reports can be part of an analysis template, thus allowing user to open the analysis template, change data, and simply print their updated custom report.

Minimum Origin Version Required: Origin 8.0 SR6

What you will learn

- How to create a custom report sheet
- How to save custom report as part of Analysis Template (OGW) and re-use with new data

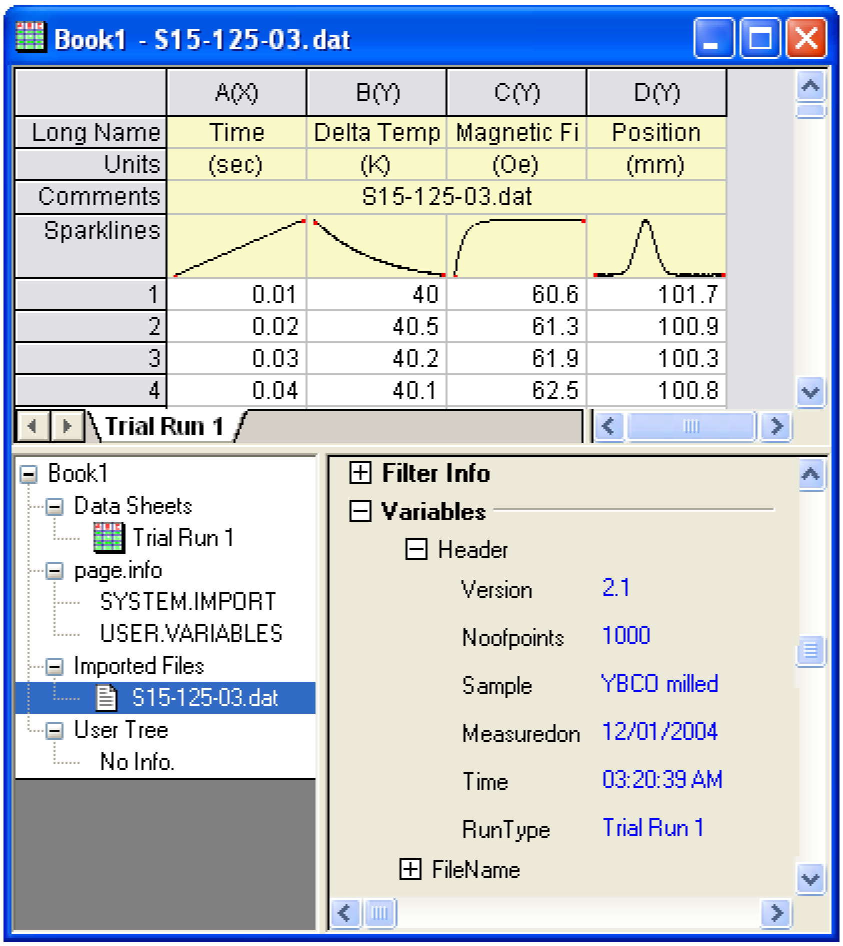
Steps

Note: First finish the previous tutorial named "Creating and Using Analysis Templates" where an analysis template named **Analysis Template.OGW** is created.

Importing Data

1. Use the **File :Open** menu item and open the Analysis Template **Analysis Template.OGW**. This analysis template already has a nonlinear fitting analysis operation set up for data from column D of the first sheet.
2. Click the **Import Wizard** button on the Standard Toolbar, or invoke the **Import: Import Wizard...** menu item. the Import Wizard dialog will open.
3. Click the **...** button next to the **File** edit box and select the file **<Origin EXE folder>\Samples\Import and Export\S15-125-03.dat**.

4. Verify that the **Import Filters for Current Data Type** drop-down shows "Data Folder: VarFromFileNameAndHeader".
5. Verify that the **Import Mode** drop-down is set to **Replace Existing Data**.
6. This is an import filter shipped with the sample file, that specifies how to import the file and what header and file name strings to parse to create import variables. Walk thru the wizard pages to view the settings (Optional) and then click Finish button to import the file.
7. Right-click on workbook title bar and select **Show Organizer** to turn on organizer panel. Expand branches and verify that variables have been created and saved, as in the picture below:

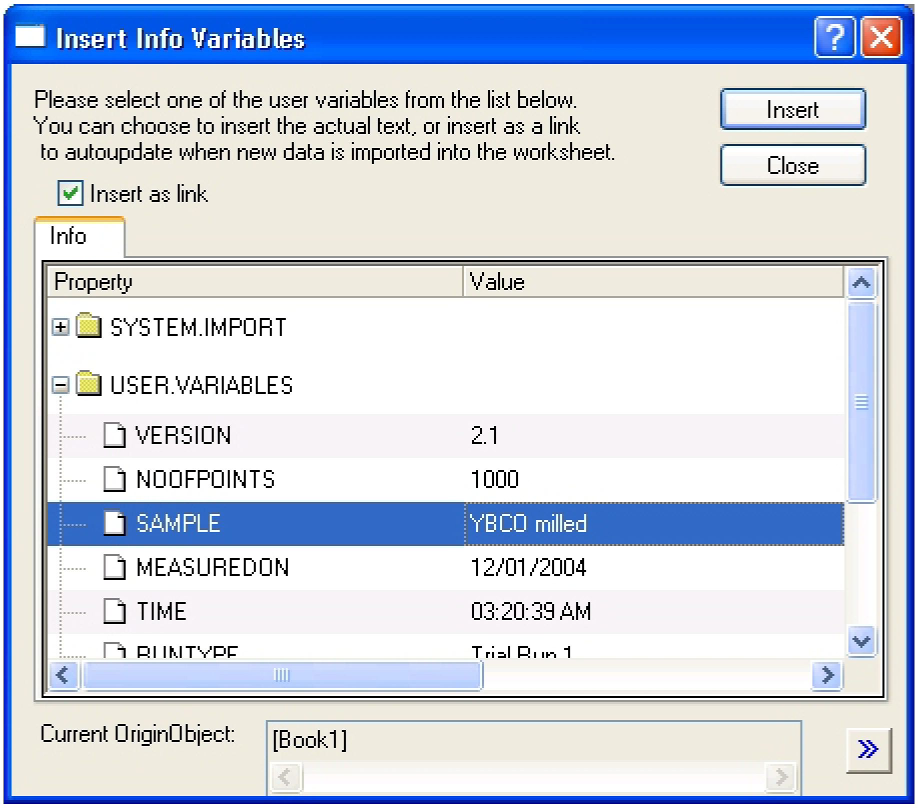


8. Press the **Recalculate** button on the **Standard** toolbar to update the analysis result sheet. Verify that the analysis was updated and the embedded graph shows the new raw data and fit curve.

Creating Custom Report Sheet

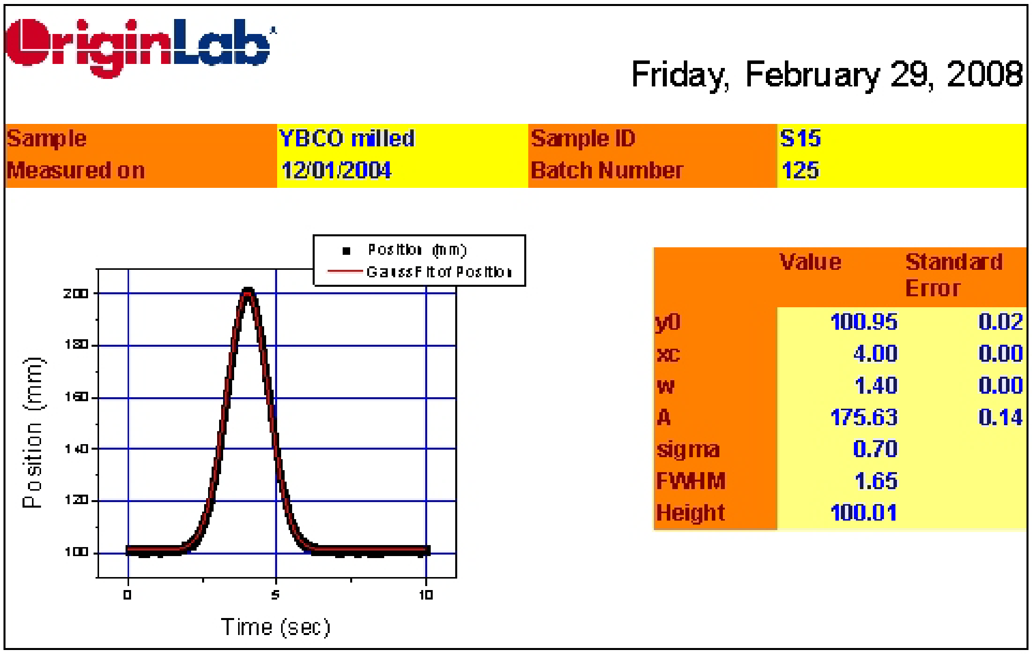
1. Right click on one of the worksheet tabs and select **Add** to add a new worksheet. Rename this worksheet as **Custom Report**.
2. Make the **Custom Report** sheet active and add multiple blank columns.
3. Go to **FitNL1** worksheet and right click on the graph with data and fit curve and select **Copy** from the context menu.

4. Go to **Custom Report** and right click in 1st column in a middle row and select **Paste Link**. This will paste a link to the embedded graph. Click and select a group of cells with this pasted cell at the top-left. Then click the **Merge Cells** button  on the **Styles** toolbar. This will merge the group of cells and the graph image will be shown larger in size.
5. Go to **FitNL1** report sheet and right click on the **Parameters** node and select **Copy Table** from the context menu.
6. Go to the **Custom Report** sheet and right click in a cell to the right of the graph and select **Paste Link** from the context menu. This will place links to all values of the parameter table entries in the custom report.
7. Select the numeric value cells and right-click and select **Format Cells...** to bring up format dialog. Change the **Digits** drop-down to **Set Decimal Places=** and enter **2** in the **Decimal Number** edit box and click **OK** to format the numbers.
8. Select various cells in the table and use the **Style** toolbar controls to change foreground and background color, and use the **Standard** toolbar to change font size etc.
9. Right-click on top-left cell in the custom report sheet and select **Insert Images from Files...** context menu and select some image such as a company logo image. Click and drag to cover more cells and then click the **Merge Cells** button to increase the size of the logo display.
10. Click inside a cell on top-right and type in the string **var://@D** and press **Enter**. Right click on cell and select **Format Cells...**, set the **Format** as **Date**, and then elect a suitable format from the **Display** drop-down. This will place the current date, pointed to by @D LabTalk variable, into the worksheet cell. Click and expand the selection to multiple cells and press the **Merge Cells** button to show the date with larger font size.
11. Right-click on a cell below the logo and date, and select **Insert Variables** context menu. In the dialog that opens, select **User.Variables** branch and select **Sample**. Check the **Insert as Link** check box on top and press **Insert** to insert variable as link into the report sheet.



Click on a cell to the left of the inserted variable, and enter the static text **Sample**.

12. Insert more variables and format the cells for color and font.
13. Invoke the **Format: Worksheet...** menu item to open the **Worksheet Properties** dialog. Under the **View** tab, expand **Show Headers** and uncheck the column and row header check boxes. Expand the **show Grid Lines** branch and uncheck the column and row grid check boxes. Select the **Format** tab and check the **Show Missing as Blank** check box. Click **OK** to close this dialog.
14. Right-click on worksheet title bar and select **View: Long Name** to turn off long name row. Also turn off **Units** and **Comments**.
15. Select the **File: Print** menu item to open the print dialog, and press **Options** button, and uncheck the Horizontal/Vertical grid lines. Select **File: Print Preview**. Your custom report sheet should look like the image below:



Saving the Analysis Template

1. Go to the source data sheet of the workbook, which should be the first sheet. Select the **Worksheet: Clear Worksheet...** menu item and press **OK** in the dialog that opens. This will clear all the data from the sheet. The analysis report sheet and the custom report sheet will now be empty. Clearing the data is optional, and it makes the size of the analysis template file to be smaller.
2. Use the **File: Save Window As...** menu item and save the book as an **OGW** file under your **User Files Folder** with a suitable name such as **My Custom Analysis**. This OGW file can now serve as an Analysis Template for future analysis of similar data.

Re-using the Analysis Template

1. Start a new project and then select the menu item **File: Recent Books** and from the fly-out options select the Analysis Template saved earlier.
2. Make the data sheet active, and select **File: Import: Import Wizard...** and select the file **<Origin EXE path>\Samples\Import and Export\S21-235-07.dat**.
3. Make sure the filter drop-down shows **VarsFromFileNameAndHeader** and change the **Import Mode** drop-down to **Replace Existing Data** and click **Finish**.
4. Press the **Recalculate** button on the **Standard** toolbar. Origin will recalculate the analysis results and update the custom report sheet links, and at this point you can view and print the custom report sheet.

3.7 Analysis Themes

3.7.1 Summary

In Origin 8, analysis procedures can be controlled by Themes. Themes are actually XML files which save settings in the analysis dialog.For example, after performing the analysis, there will now be a <Last Used> theme for this dialog which has saved the most recently used settings. You can assign a proper name for the theme and use it in the future.

For this tutorial, the Statistics on Columns dialog will be used to demonstrate how to create and use an analysis theme. This analysis provides descriptive statistics about the data such as mean, standard deviation, minimum, maximum, and more. For visualization, a histogram or box chart can also be created in the Analysis Result Sheet.

Minimum Origin Version Required: Origin 8.0 SR6

3.7.2 What you will learn

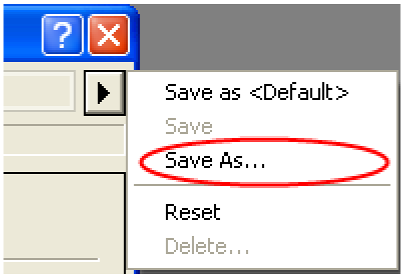
This tutorial will show you how to:

- Perform simple descriptive statistics
- Create an Analysis Theme
- Use the theme

3.7.3 Steps

Save the analysis procedure as Theme

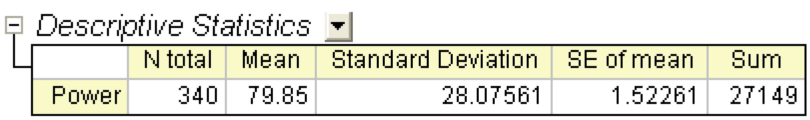
1. Start with a new workbook and import the file *\Samples\Statistics\automobile.dat*.
2. Highlight column C and use the menu item **Statistics: Descriptive Statistics: Statistics on Columns** to open the dialog.
3. Expand the **Moments** tree node, and check the *N Total*, *Mean*, *Standard Deviation*, *SE of Mean* and *Sum* box.
4. Expand the **Plots** tree node, and check the *Histograms* and *Box Charts* check boxes. You will then get the corresponding histogram as well as box chart graphs.
5. Your selections in this analysis dialog can be saved as your theme, so that you may easily repeat the procedure. Click the "Save Theme as..." button:



to bring up this dialog:

6. Type a proper theme name, such as "*MyTheme*" and click **OK** button.

7. Click the **OK** button in the **Statistics on Columns** dialog. You will see the result in a new worksheet named DescStatsOnCols1.

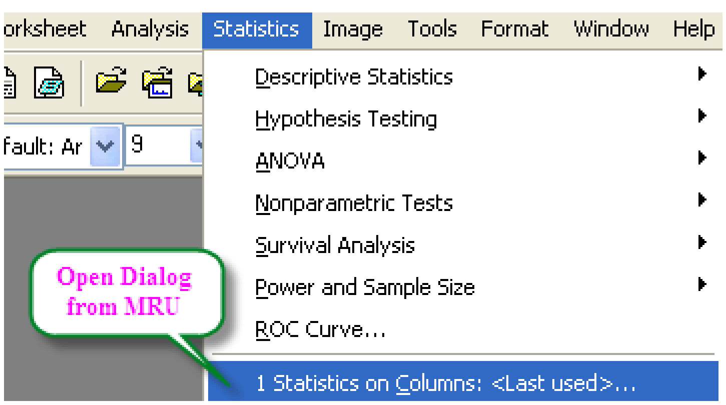

Descriptive Statistics

	N total	Mean	Standard Deviation	SE of mean	Sum
Power	340	79.85	28.07561	1.52261	27149

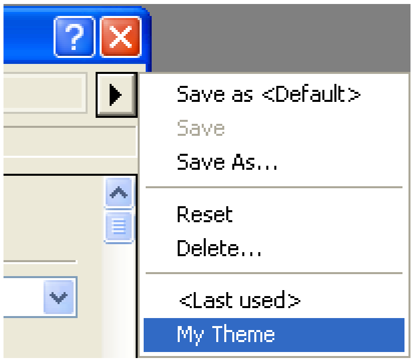
Repeat the analysis procedure by Theme

Once you save a theme, there are many ways to use it. For example, you can highlight column E and perform the same statistics on it.

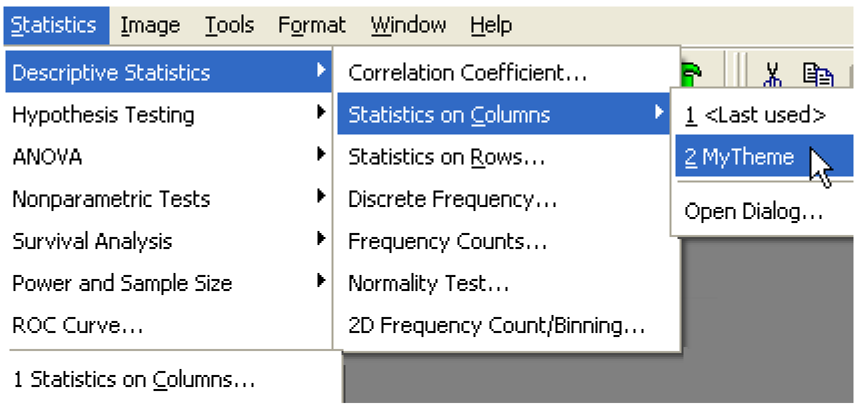
- Open the **Statistics on Columns** dialog from the **Most Recently Used** menu. Most of the menu accessible dialogs can be found from MRU.



When you open the dialog from MRU, the default theme is <Last Used>. To use the **MyTheme** theme, select *MyTheme* from the **Dialog Theme** drop-down list. The settings from that theme will then be displayed in the dialog. Click the **OK** button to do the analysis.



- Another way to apply the analysis theme is to use the cascaded menu item. Once you use an analysis dialog, or save a theme for a dialog, there will be one more menu level added. You can choose the *MyTheme* menu.



When selecting, *Open Dialog...*, the dialog will open to the <Factory Default> theme. To change the settings of your theme, you can select your theme name from the **Dialog Theme** drop-down list inside the dialog, make changes, and resave the theme. Alternatively, to open a dialog with a saved theme without performing the analysis, hold down the *shift* key while selecting the theme from the menu. This will bring up the dialog with your theme applied so you can make changes as needed.

3.8 Batch Processing

Topics covered in this section:

1. Analysis Template and Batch Processing
2. Batch Processing with Summary Report in Excel

3.8.1 Analysis Template and Batch Processing

Summary

The idea of batch processing is that the same analysis process could be repeated easily and quickly. Origin allows users to perform batch processing of multiple files or data sets in columns, using an analysis template.

What You Will Learn

This tutorial will show you how to:

- Create an analysis template
- Perform batch processing with multiple columns
- Perform batch processing with multiple data files

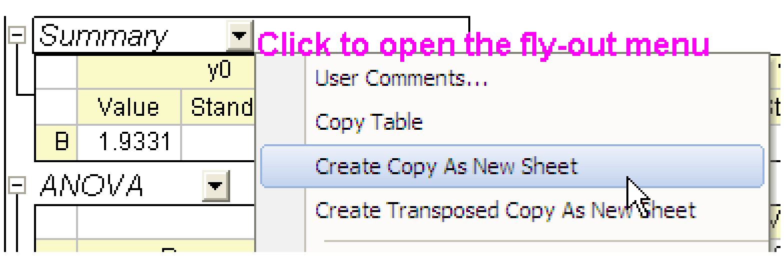
Steps

Batch processing could be applied to multiple columns on the same worksheet. In this example, the input worksheet contains multiple columns, each stands for a curve to be fitted. We will first create an analysis template fitting the first curve, and then use batch processing for other columns. Later, the analysis template could also be used for other files.

To carry out the example, follow the steps below:

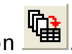
Creating an Analysis Template

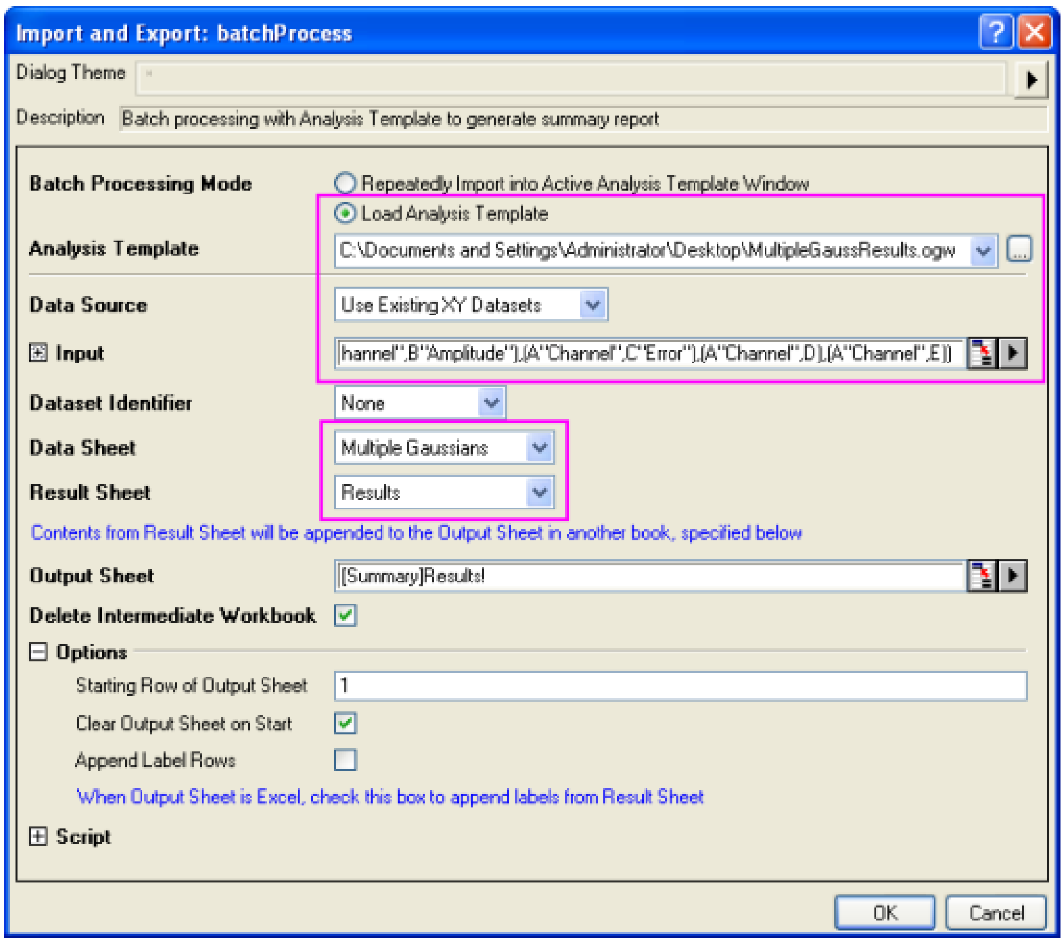
1. Start with a new project and import the file **Multiple Gaussians.dat** under the path *<Origin Folder>/Samples/Curve Fitting* by selecting from top menu **File:Import:Single ASCII**;
2. Highlight **column B(Y)** and choose **Analysis: Fitting: Nonlinear Curve Fit** from the top menu (or press Ctrl + Y) to open the **NLFit** dialog;
3. Select **Gauss** in the **Function** drop-down list, and click **Fit** to do the fitting;
4. Click **Yes** when asked if you want to view the report sheet;
5. On the **FitNL1** report sheet, right-click on the word **Summary** above the summary peak fit results table and choose **Create Copy as New Sheet**.



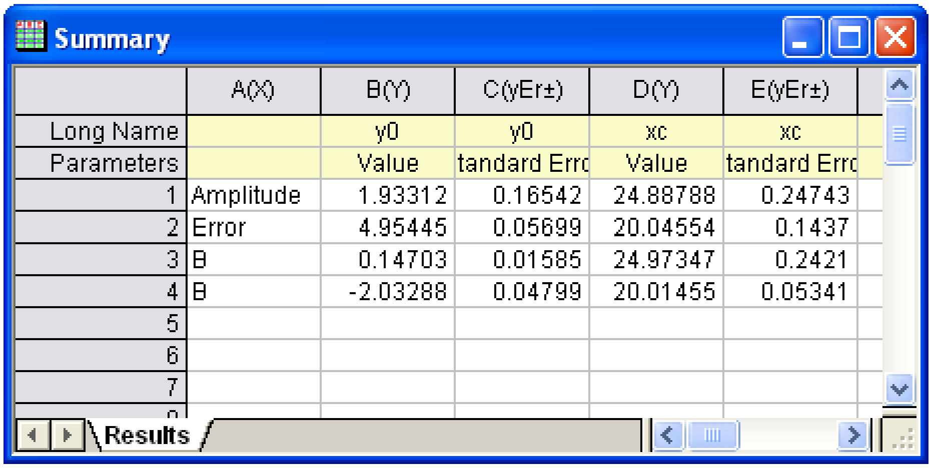
6. Double-click on the sheet name, **Sheet 2** and rename it as **Results**. Here, a custom report sheet is created, and will be included in the analysis template
7. Select **File: Save Workbook as Analysis Template** and give it a descriptive name such as **My Gauss Results**.

Performing Batch Processing with Multiple Columns

1. Make the original worksheet, **Multiple Gaussians** active.
2. Highlight columns **B(Y)** through **E(Y)**.
3. Select **File: Batch Processing** from the menu or click on the **Batch Processing** button .
4. Select the **Load Analysis Template** checkbox for the **Batch Processing Mode**.
5. In the **Analysis Template** drop-down, select the **My Gauss Results.ogw** file we saved earlier.
6. Select **Use Existing XY Datasets** from the **Data Source** drop-down list.
7. Make sure the **Data Sheet** drop-down is set as the original sheet, **Multiple Gaussians**.
8. Make sure the **Result Sheet** is set as **Results**. The settings should look like the following figure:

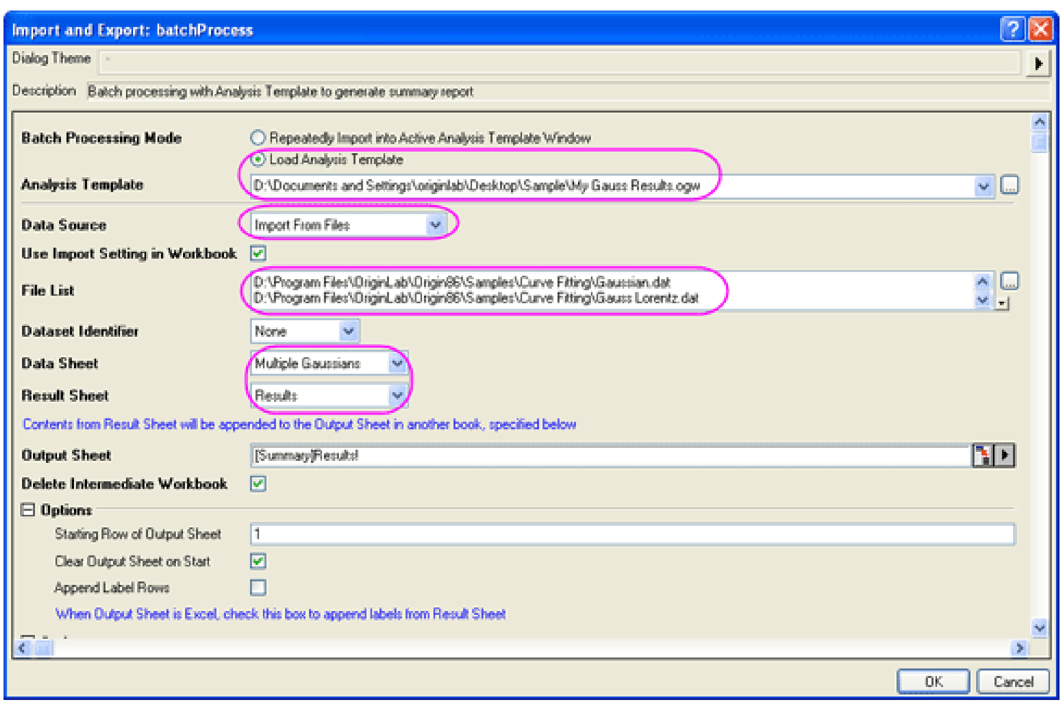


9. Click OK to perform the batch processing, and generate your **Summary: Results Sheet**.

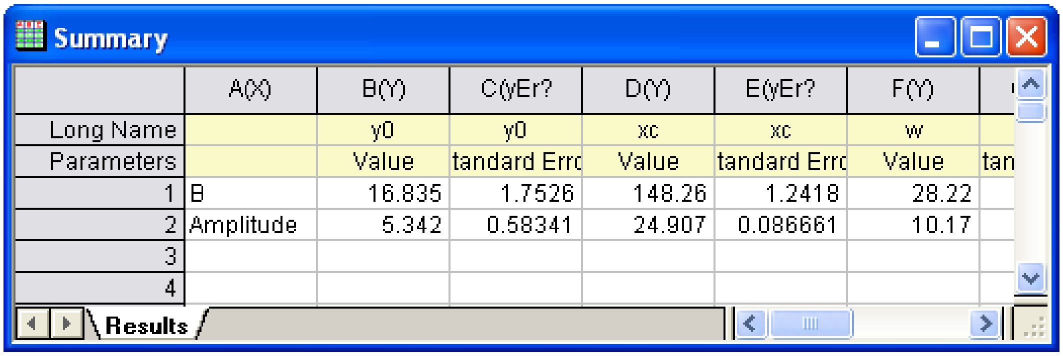


Performing Batch Processing with Multiple Data Files

1. Start with a new project.
2. Click on the **Batch Processing** button.
3. Select **Load Analysis Template** radio button for the **Batch Processing Mode**.
4. In the **Analysis Template** drop-down, select the **My Gauss Results.ogw** file we saved earlier.
5. Select **Import From Files** in the **Data Source** drop-down list.
6. Click on the **Browse** button in the **File List** section and select the **Gauss Lorentz.dat** and **Gaussian.dat** from the *Samples\Curve Fitting* folder.
7. Click **Add File(s)**, then click **OK**.
8. Make sure **Data Sheet** is set as **Multiple Gaussians**.
9. Make sure **Result Sheet** is set as **Results**.



10. Click **OK**.



	A(X)	B(Y)	C(yEr?	D(Y)	E(yEr?	F(Y)	
Long Name		y0	y0	xc	xc	w	
Parameters		Value	tandard Errc	Value	tandard Errc	Value	tan
1	B	16.835	1.7526	148.26	1.2418	28.22	
2	Amplitude	5.342	0.58341	24.907	0.086661	10.17	
3							
4							

This time all three files, each having multiple columns of data have been processed using the **My Gauss Results** analysis template we created earlier. This is a quick and easy way to process many files of data into a single custom report sheet.

3.8.2 Batch Processing with Summary Report in External Excel File

Summary

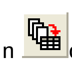
This tutorial is associated with the Sample Origin Project: **\Samples\Batch Processing\Batch Processing with Summary Report in External Excel File.OPJ**

Minimum Origin Version Required: Origin 8.1 SR2

What you will learn

- How to perform batch processing of multiple data files
- How to send results to an external Excel file and save that file

Steps

1. Make the 'Raw Data' Worksheet in the 'Book1' Workbook active.
2. Select **File : Batch Processing...** from the menu or click the **Batch Processing** button  on the Standard Toolbar.
3. Select the **Repeatedly Import into Active Analysis Template Window** radio.
4. Set **Data Source** as **Import From Files**.
5. Check the **Use Import Setting in Workbook** check box.
6. Click the **...** browse button next to **File List**.
7. Select **All Files (*.*)** in Files of Type and browse to Origin's \Samples\Batch Processing folder.
8. Select all 10 .csv files in the folder, click **Add File(s)** then click **OK**.
9. **Data Sheet** should be set to 'Raw Data'. Note: 'Raw Data' is the first sheet of 'Analysis Template', it will be renamed as the file name of last import file by default if this template has already been used and re-saved.
10. **Result Sheet** should be set to 'My Results'.

11. Click the button to the right of the "Output Sheet" edit box. *This will minimize the main dialog.* Then click on the title bar of the Excel book (Book2), and press the button to the right of the scrolled up dialog to expand it again.
12. Expand the Options branch and check the **Clear Output Sheet on Start** check box and enter **7** as the **Starting Row of Output Sheet**.
13. Check the **Append Label Rows** check box.
14. Click OK

4 Statistics

Topics covered in this section:
1. Descriptive Statistics (Tutorials)
2. ANOVA (Tutorials)
3. Nonparametric Tests (Tutorials)
4. Multivariate Analysis (Tutorials)
5. Hypothesis Tests
6. Power and Sample Size

4.1 Hypothesis Tests

4.1.1 Summary

Hypothesis tests are frequently used to measure the quality of sample parameters or to test whether estimations for two samples on a given parameter are equal .

With parametric methods,assumptions are made about the underlying distribution from where the sample populations are selected. Usually, it requires that the data are independently sampled from a normal distribution.

4.1.2 What you will learn

This tutorial will show you:

- How to carry out hypothesis tests for practical data with Origin
- How to interpret the generated results

4.1.3 Steps

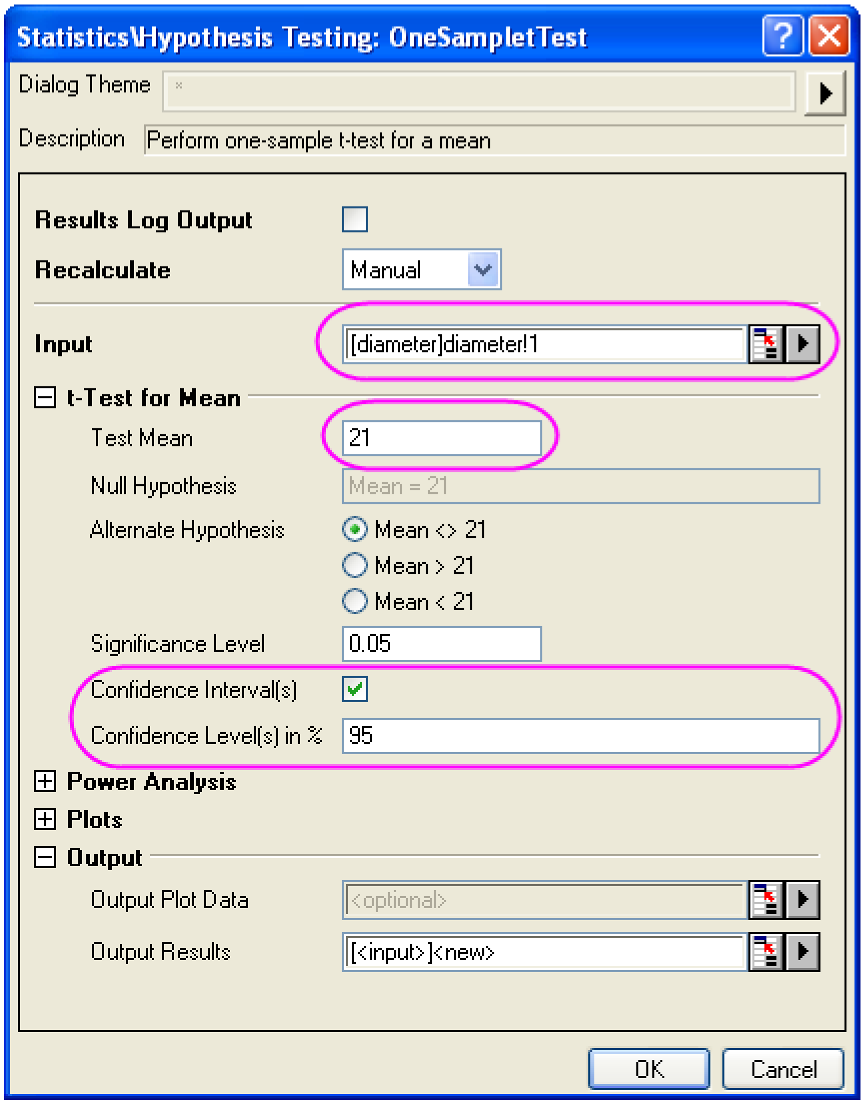
Hypothesis Tests in Origin

Data Type	Goal	Method
One Sample	Compare the mean with a given value	One-Sample t-Test
	Compare the varariance with a given value	One-Sample Test for Variance
Two Samples	Test whether the means are equal	Two-Sample t-Test
	Test whether the variance are equal	Two Sample Test for Variances
Paried Samples	Test whether the means are equal	Pair-Sample t-Test

One-Sample t-Test

Suppose a manufacturer produces high-quality screw nuts that must equal 21 millimeters in diameter. The quality control department randomly drew 120 nuts from the finished products, measureed the diameters for each and stored the results in Diameters.dat file.They want to examine whether the mean diameter of the nuts is equal to 21 or not. The distribution of the measured diameters is known as close to normal historically, but the standard deviation of the population is unknown. Hence they may use the One-Sample t-Test in Origin following the steps below:

1. Start with a new workbook and import the file **\Samples\Statistics\diameter.dat**.
2. Open the **One-Sample t-Test** dialog by using the menu item **Statistics:Hypothesis Testing:One-Sample t-Test**.
3. Select the first column as Input Data Range, specify a two-sided test and type 21 as the test mean; and select the confidence level as 95%.



4. Note that by default the test procedure provides descriptive statistics of the variable and the hypothesis test results. Additionally, it is possible to produce a histogram of the data and a confidence interval for the mean.
5. Click the **OK** button to finish the analysis and generate results.

The Descriptive Statistics table shows the sample size, mean, standard deviation, and standard error for the variable. The sample mean, 21.00459, is comparatively little bigger than the hypothesis mean 21, and the standard error of the mean(SEM) is 0.00156.

Descriptive Statistics

	N	Mean	SD	SEM
"diameter"	100	21.005	0.0156	0.00156

From the t-Test table, the t statistic (2.94337) and the associated p-value (0.00404) provide evidence that the average diameter of screw nuts is significantly different with 21 at the ?=0.05 level.

Test Statistics

	t Statistic	DF	Prob>|t|
"diameter"	2.9437	99	0.00404

Null Hypothesis: Mean = 21
Alternative Hypothesis: Mean <> 21
"diameter": At the 0.05 level, the population mean is significantly different from the test mean (21).

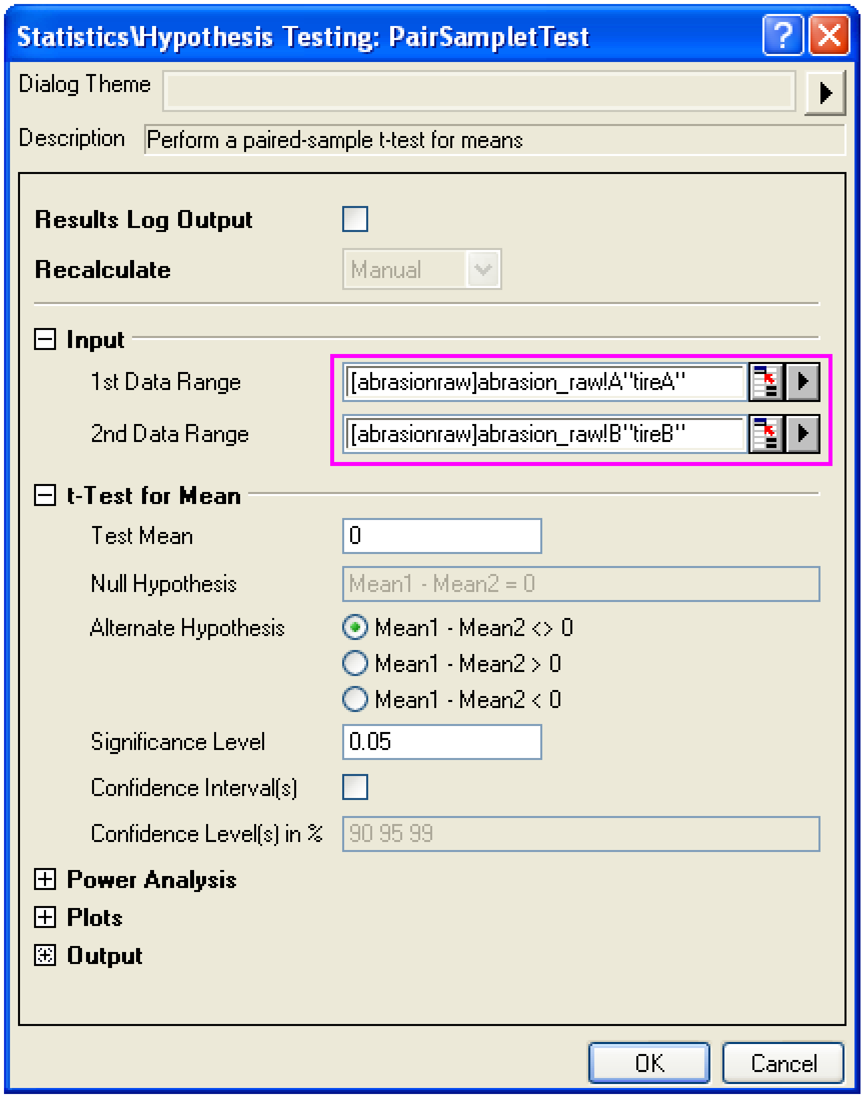
The confidence interval indicates that it is 95% confident that the true mean of the variable lies within the interval [21.0015, 21.00769].

Confidence Intervals for Mean

	Conf. Levels in %	Lower Limits	Upper Limits
"diameter"	95	21.0015	21.00769

Pair-sample t-Test

1. Start with a new workbook and import the file **\Samples\Statistics\abrasion_raw.dat**.
2. Open the **Pair-sample t-Test** dialog by using the menu item **Statistics:Hypothesis Testing:Pair-sample t-Test**.
3. Set column tireA as 1st Data Range and column tireB as 2nd Data Range, enter 0 to be Test Mean.



4. Accept other settings as default and click the **OK** button to generate results.

Descriptive Statistics

		N	Mean	SD	SEM
"tireA"		8	6145	1366.49709	483.12968
"tireB"		8	5825	1097.46461	388.01233
	Difference		320		

Test Statistics

	t Statistic	DF	Prob>|t|
	2.83119	7	0.02536

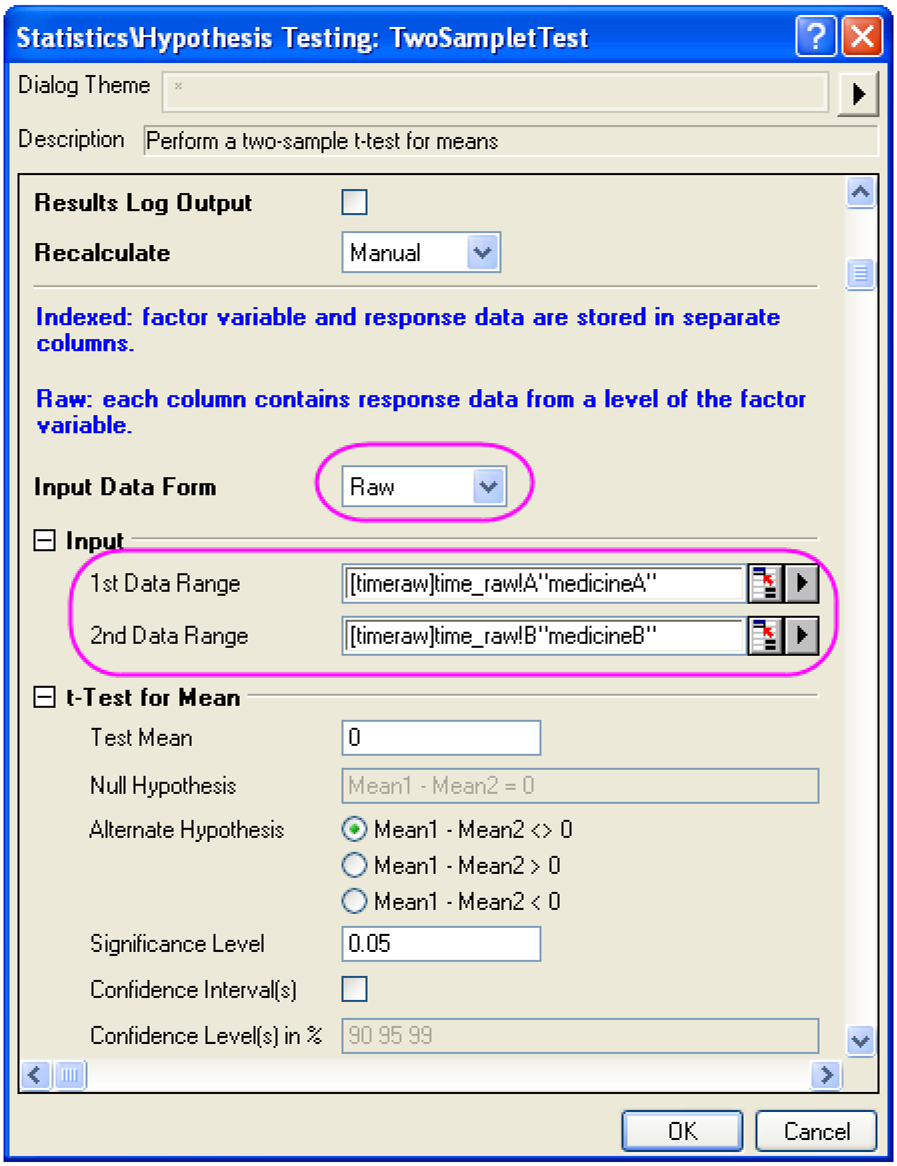
Null Hypothesis: mean1-mean2 = 0
Alternative Hypothesis: mean1-mean2 <> 0
At the 0.05 level, the difference of the population means is significantly different from the test difference(0).

From the t-Test table, the t statistic (2.83119) and the associated p-value (0.02536) indicate that the difference between the two means is significant, that is to say, the two types of tires have different abrasion resistance.

Two-Sample independent t-Test

A physician is evaluating the effect of two kinds of soporifics. To test the effectiveness of these two medicines, 20 insomniacs patients are randomly selected. Half took medicine A and the other half took medicine B, The extended sleeping time were recorded after each patient took the medicine. The result is saved as the time_raw.dat file.
To determine whether the two medicine have different effect on patients, we could carry out a two-sample independent t-test with the following steps:

1. Start with a new workbook and import the file **\Samples\Statistics\time_raw.dat**.
2. Open the **Two-Sample t-Test** dialog by using the menu item **Statistics:Hypothesis Testing:Two-Sample t-Test**.
3. Select "Raw" for the **Input Data Form**; set column A and column B as the first and second data range respectively.



4. Accept other settings as default and click the **OK** button to generate results.

T-Test procedure automatically provides two tests of the mean difference. One is based on the assumption that the variances of two samples are equal and the other is not. In this example, both tests indicate that there is no significant evidence for a difference in cure effects between medicine A and medicine B. (p-values are 0.0738 and 0.074, both greater than the significance level 0.05.)

t-Test Statistics

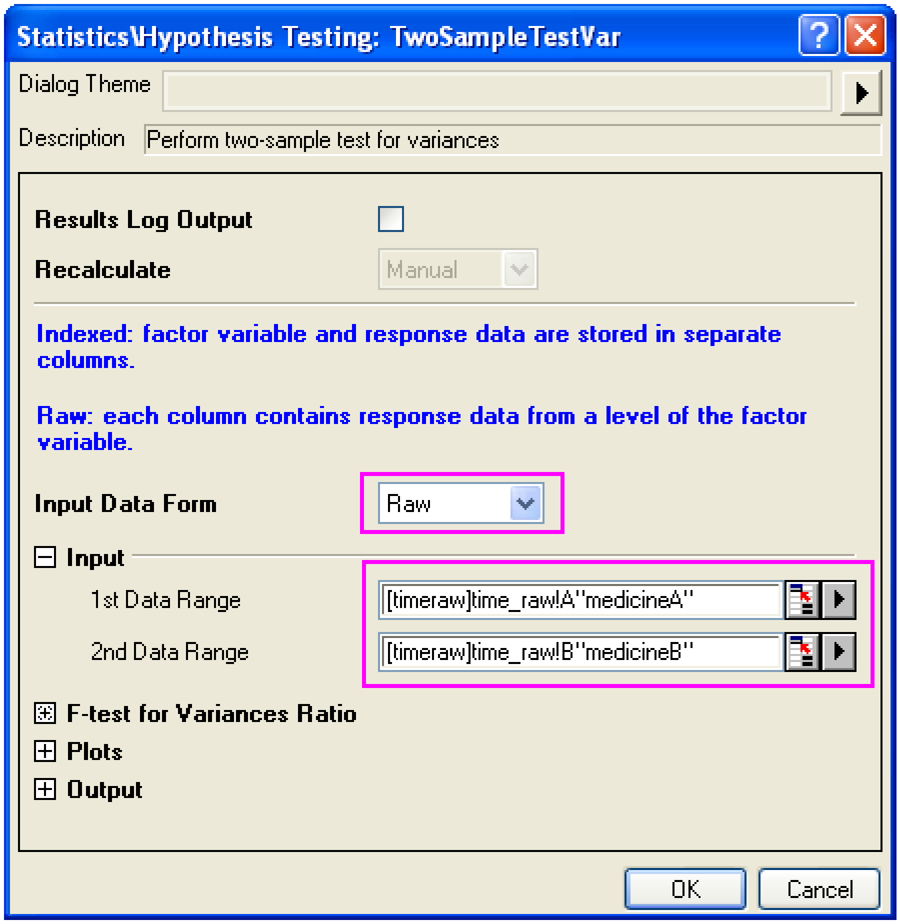
	t Statistic	DF	Prob>|t|
Equal Variance Assumed	1.89811	18	0.07384
Equal Variance NOT Assumed (Welch Correction)	1.89811	17.8248	0.074

Null Hypothesis: mean1-mean2 = 0
Alternative Hypothesis: mean1-mean2 <> 0
At the 0.05 level, the difference of the population means is NOT significantly different from the test difference(0).

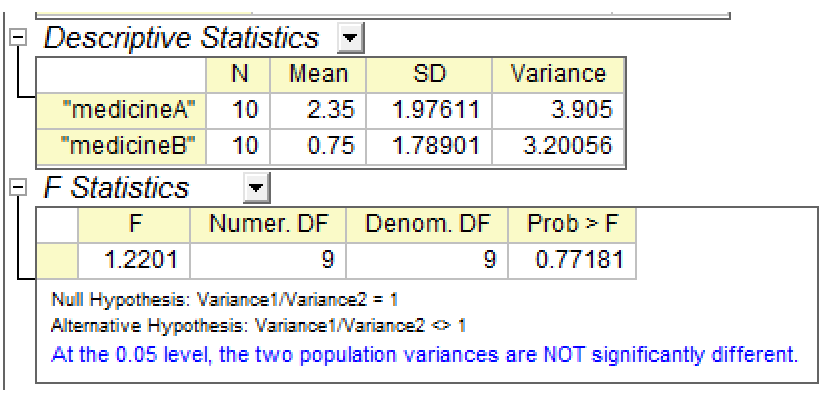
Note that both equal variance and unequal variance assumptions are supported.To determine whether the two samples have equal variances, we may select from top menu **Statistics:Hypothesis Testing:Two-Sample Test for Variance** to use the two-sample test for variance for testing

Two-Sample Test for Variance

1. Continue with a new workbook and import the file **\Samples\Statistics\time_raw.dat**.
2. Open the **Two-Sample Test for Variance** dialog by using the menu item **Statistics:Hypothesis Testing:Two-Sample Test for Variance**.
3. Select "Raw" for the **Input Data Form**; set column A and column B as the first and second data range respectively.



4. Accept other settings as default and click the **OK** button to generate results.



Descriptive Statistics

	N	Mean	SD	Variance
"medicineA"	10	2.35	1.97611	3.905
"medicineB"	10	0.75	1.78901	3.20056

F Statistics

	F	Numer. DF	Denom. DF	Prob > F
	1.2201	9	9	0.77181

Null Hypothesis: Variance1/Variance2 = 1
Alternative Hypothesis: Variance1/Variance2 <> 1
At the 0.05 level, the two population variances are NOT significantly different.

According to result , P_vaule=0.77181 >0.05,therefore ,it fail to reject null hypothesis, so two population variances are not signification difference.

4.2 Power and Sample Size

4.2.1 Summary

Power and sample size analysis is useful for researchers to design their experiments. Insufficient data and lack of the power to reject a false null hypothes is may lead to wrong conclusion and too much data is a waste of time and money . Therefore, it is essential to determine the sample size requirements of a experiments. The power of the experiment can be computed for a given sample size, as well as the required sample size for given power values.

4.2.2 What you will learn

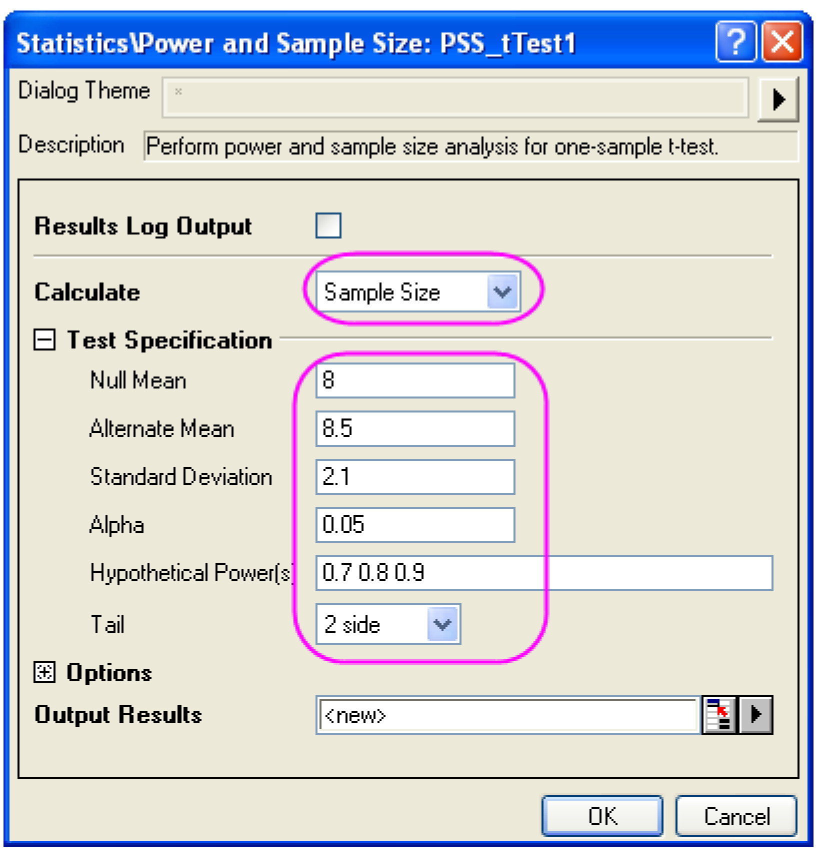
This tutorial will show you how to calculate sample size or estimate power value to design experiments under different practical situations.

4.2.3 (PSS)One-Sample t-Test

Background:
A sociologist wants to determine whether the average infant mortality rate in the United States is equal to 8. In experiment design, the difference of rate cannot vary more than 0.5. And it is already known that the standard deviation should be 2.1 from pilot studies.

Question:
What would the sample size be, in order to estimate the average infant mortality rate at a confidence level of 95% (α=0.05) for power values of 0.7, 0.8 and 0.9?
Steps in Origin:

1. Active an empty worksheet, select **Statistics: Power and Sample Size: (PSS) One-sample t-test**;
2. Use the settings in the following figure for the pop-up **PSS_tTest1** dialog and click **OK**.



Origin Output:
A result sheet will be generated, and listing the calculated sample size for hypothetical powers.

Sample Size(s) for Hypothetical Power(s)

Alpha	Power	Sample Size
0.05	0.7	111
0.05	0.8	141
0.05	0.9	188

Null Mean = 8; Alternate Mean = 8.5; SD = 2.1; 2-Sided Test

Result Interpretation:
According to the calculation, for experiment design, the sociologist should conduct a survey of 111 samples for power value 0.7, 141 samples for power value 0.8 and/or 188 samples for power value 0.9.

4.2.4 (PSS)Two-Sample t-Test

Background:
A doctor's office participates in two local insurance plans, Healthwise and Medcare. The purpose is to compare the mean time (in days) until reimbursement of claims for the two plans. Historical data shows that for Healthwise plan, the average time is 32 days and standard deviation is 7.5 days. For the Medcare plan, the average reimbursement time is 42 days and the standard deviation is 3.5 days.

Question:
If 5 claims from each plan were selected and the corresponding reimbursement times were recorded, what is the power to detect the difference in mean reimbursement times between the 2 plans by 5% or more?

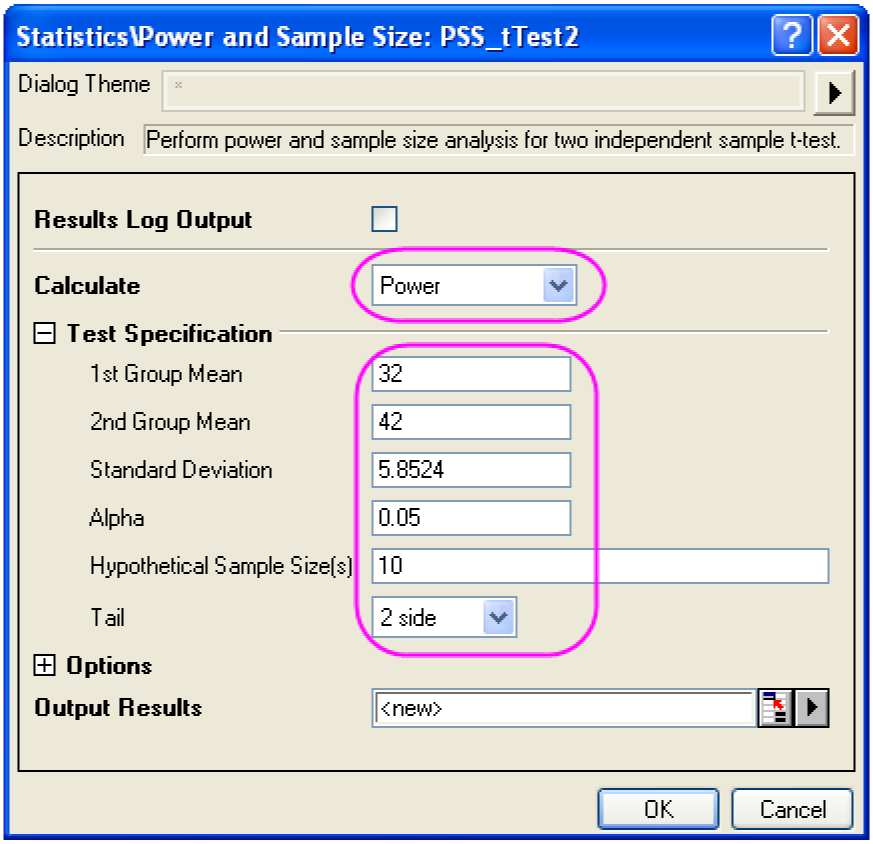
Steps in Origin:

1. Compute the pooled standard deviation as:

$$\sqrt{((5-1)*7.5\text{^}2+(5-1)*3.5\text{^}2)/(5+5-2)}=5.85235$$

*Note that this value will be used as the standard deviation later for the power calculation.

2. Sample size of 1st group and 2nd group should be 5+5=10,
3. Active an empty worksheet,and select **Statistics: Power and Sample Size: (PSS) Two-Sample t-Test**,
4. Use the settings as the following figure in the pop-up **PSS_tTest2** dialog and click **OK**.



Origin Output: A result sheet will be generated, showing the calculated power.

Power(s) for Hypothetical Sample Size(s)

Alpha	Sample Size	Power
0.05	10	0.95054

Group1 Mean = 32; Group2 Mean = 42; SD = 5.85235; 2-Sided Test

Results Interpretation:

According to the result, we can conclude that the doctor's office has a 0.95054:1(or 95%) chance of detecting a difference if it collects 5 claims for each plan,in other words, the chance that you will fail to reject the null hypothesis and incorrectly conclude that the two means are not different is 4.946%(1-0.95054)

4.2.5 (PSS)Paired-Sample t-Test

Background:

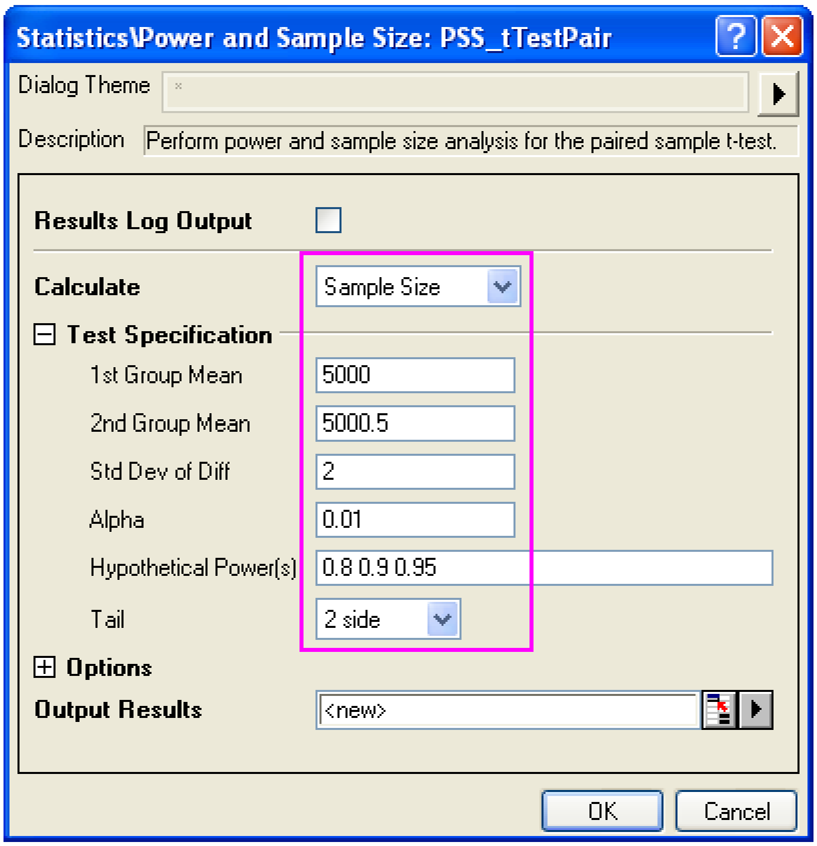
There are two same type measuring machines to measure the depth of a-Si thin film . In order to analysis whether there is any difference in the two machines measuring results , engineer would like to do an experiment to measure depth of a-Si thin film at the same position by the two machines in different products. According to a previous study on depth of a-Si thin film, the standard deviation of the difference was found to be 2µm . This will be used as an estimation of the standard deviation of the differences to plan this experiment. The difference in the measuring result of the two machines cannot be more than 0.5,and the average depth of measured by machine 1 is 5000µm.

Question:

How many samples must be taken at a confidence level of 99% for power values of 0.8, 0.9, 0.95?

Steps in Origin:

1. According to the information above, it is concluded that the mean of 1st group is 5000 µm and mean of 2nd group is 5000.5 µm
2. Active an empty worksheet and select **Statistics: Power and Sample Size: (PSS) Paired t-Test**
3. Set following in the pop-up **PSS_tTestPair** dialog and click **OK**



Origin Output:

A result sheet will be generated, listing the calculated sample size(i.e. number of samples) according to different power value.

Sample Size(s) for Hypothetical Power(s)

Alpha	Power	Sample Size
0.01	0.8	191
0.01	0.9	242
0.01	0.95	289

Group1 Mean = 5000; Group2 Mean = 5000.5; SD = 2; 2-Sided Test

Results Interpretation:

According to result report, we can concluding that engineer has 80% chance of detecting a difference if measures 191 thin film ,90% chance if measures 242 thin film and 95% chance if measures 289 thin film .(Note?Each number you enter is considered to be the sample size for each set of paired observations.)

4.2.6 (PSS)One-Way ANOVA

Background:

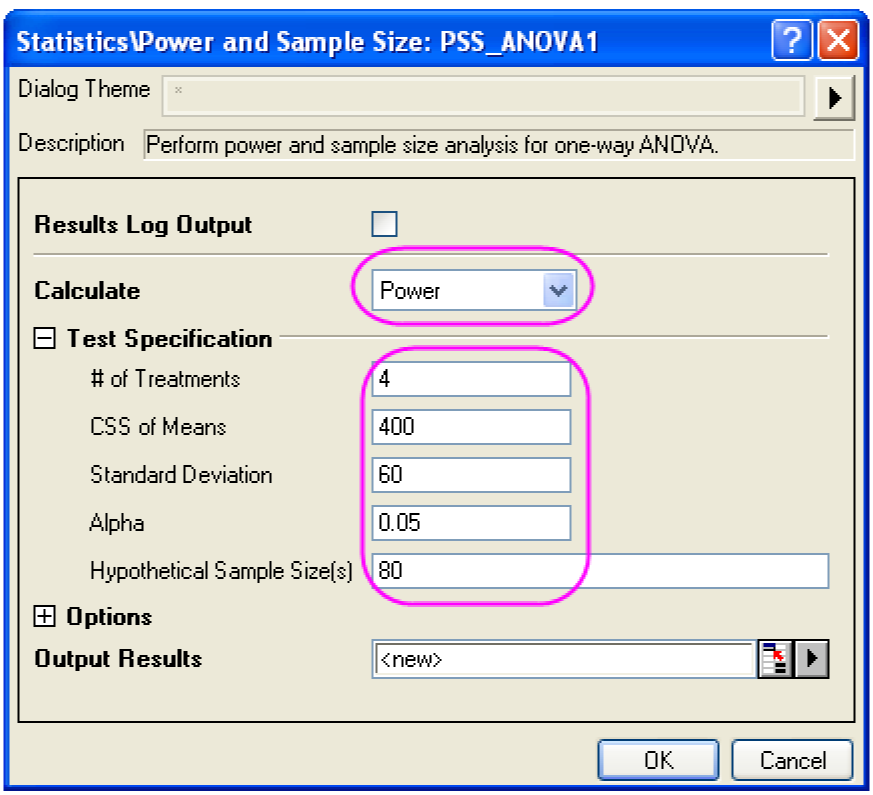
Researchers are interested in whether different plants have different nitrogen contents. They planned to record nitrogen contents in milligrams for 4 kinds of plants (20 observations per kind of plant). Previous research suggests that the square root of MSE(Mean Squared Error) is 60 and the CSS(corrected sum of squares) of mean is 400.

Question:

Is the plan feasible? (i.e. will the calculated power be acceptable?)

Steps in Origin:

1. Calculate the sum of sample size of 4 groups by 20*4=80
2. Active an empty worksheet and select **Statistics: Power and Sample Size: One Way ANOVA**
3. Use the settings in the following figure for the pop-up **PSS_ANOVA1** dialog and click **OK**.



Origin Output:

A result sheet is generated, and the power value is calculated from the known condition.

Power(s) for Hypothetical Sample Size(s)

Alpha	Sample Size	Power
0.05	80	0.6993

Treatments = 4; CSS of Means = 400; SD = 60

Results Interpretation:

The original research plan is not so good. There is only 69% chance to detect a difference from each group. To get more reliable results, researches should collect more samples per kind of plant.

4.3 Descriptive Statistics

4.3.1 Descriptive Statistics

- Descriptive Statistics
- 2D Binning

4.3.2 Descriptive Statistics

Summary

Origin provides comprehensive Descriptive Statistics support including basic statistics (mean, median, variance, etc.), frequency counts, and correlation coefficients of data you select. In addition to strong plotting features, Origin's statistical tools help you summarize and analyze your data.

This tutorial will show you how to:

- Use the **Statistics on Column** Dialog to calculate descriptive statistics for grouped data.
- Copy statistical results to a new worksheet for further processing.
- Unstack Columns to a graph.
- Analyze data sets with the **Correlation Coefficient** Tool.

Finding Frequency Information for Groups

Start with some data. We can use the **Discrete Frequency** Tool to quickly obtain frequency information for groups of data.

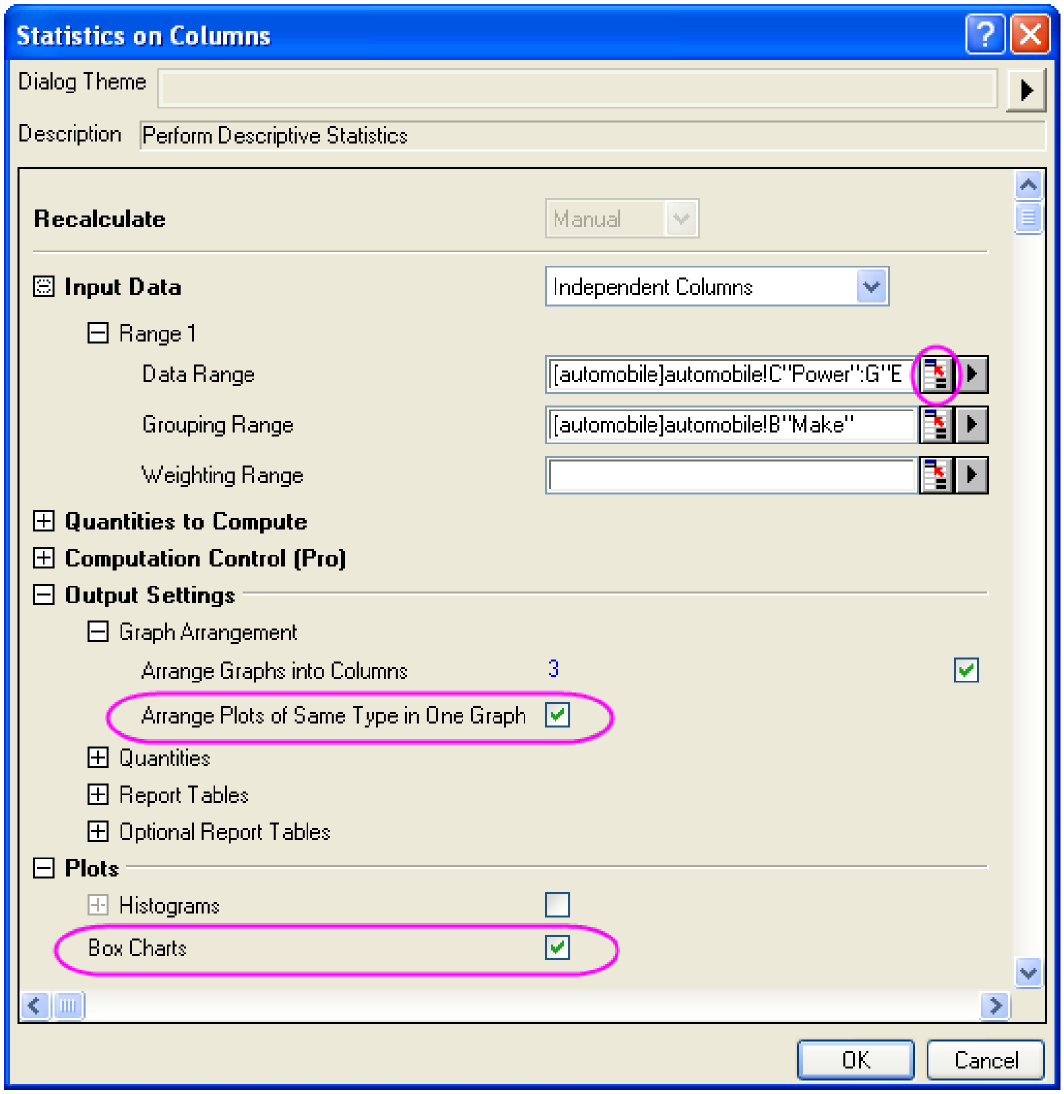
1. Start with a new project or a new workbook. Import the data file *\Samples\Statistics\automobile.dat* by using **Import Single ASCII** 
2. Highlight the first two columns. Select **Statistics: Descriptive Statistics: Discrete Frequency** to open a dialog. Column A and Column B are automatically picked as Input Data. Click **OK**

Results of discrete frequency are sorted in descending order of **Count**; the most frequently occurring data will appear first. You can rearrange the results by sorting worksheets even though there are locks on the columns.

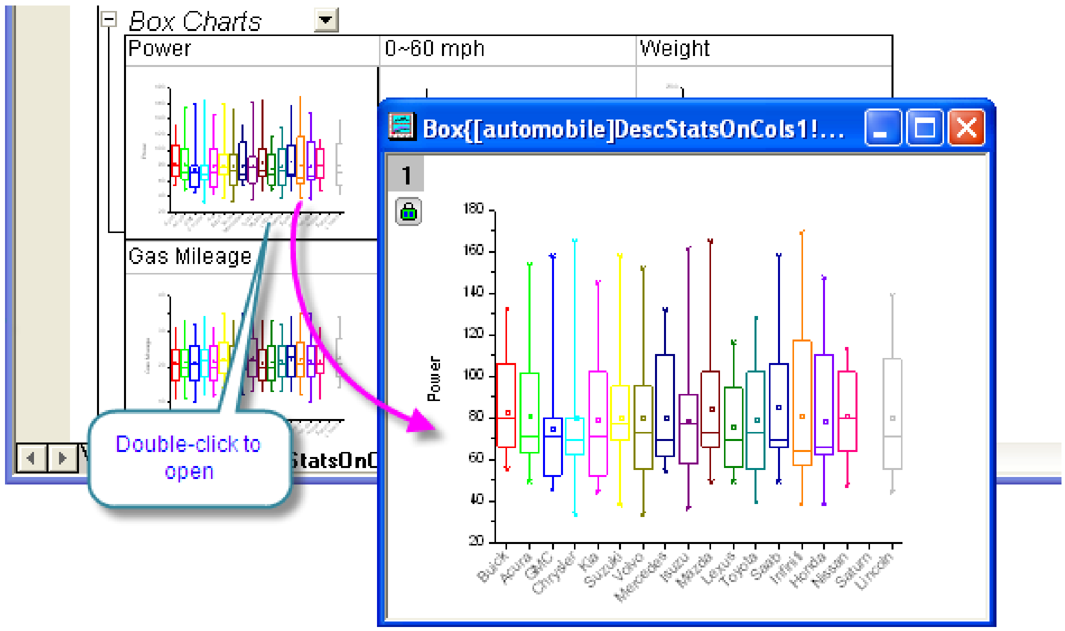
Calculating Descriptive Statistics on Grouped Data

Using the **Statistics on Columns** tool, we can find basic statistics for each group of data.

1. Switch back to the first sheet.
2. Select **Statistics: Descriptive Statistics: Statistics on Columns** to open the **Statistics on Columns** dialog.
3. Open the **Range 1** branch and click the **interactive button** . The dialog will "roll up" and you can set **Data Range** as Column C ~ Column G by selecting C(Y) and dragging to G(Y) in the Worksheet. Click the button in the rolled up dialog to restore the dialog. To set **Group Range** to **B(Y): Make**, click the **triangle button** next to **Grouping Range** and select **B(Y) : Make**.
4. Here, we will show how to make a box plot for the grouped data and put all groups in a graph for a quick comparison. Do the following: 1) Expand the **Output Settings** branch and the **Graph Arrangement** sub-branch. Select the **Arrange Plots of Same Type in One Graph** check box. 2) Expand the **Plots** branch, and select the **Box Charts** check box.


Statistics on Columns
Dialog Theme
Description
Perform Descriptive Statistics
Recalculate
Manual
Input Data
Independent Columns
Range 1
Data Range
[automobile]automobile!C"Power":G"E
Grouping Range
[automobile]automobile!B"Make"
Weighting Range
Quantities to Compute
Computation Control (Pro)
Output Settings
Graph Arrangement
Arrange Graphs into Columns
3
Arrange Plots of Same Type in One Graph
Quantities
Report Tables
Optional Report Tables
Plots
Histograms
Box Charts
OK
Cancel

5. Click the **OK** button to get the results in a report sheet.



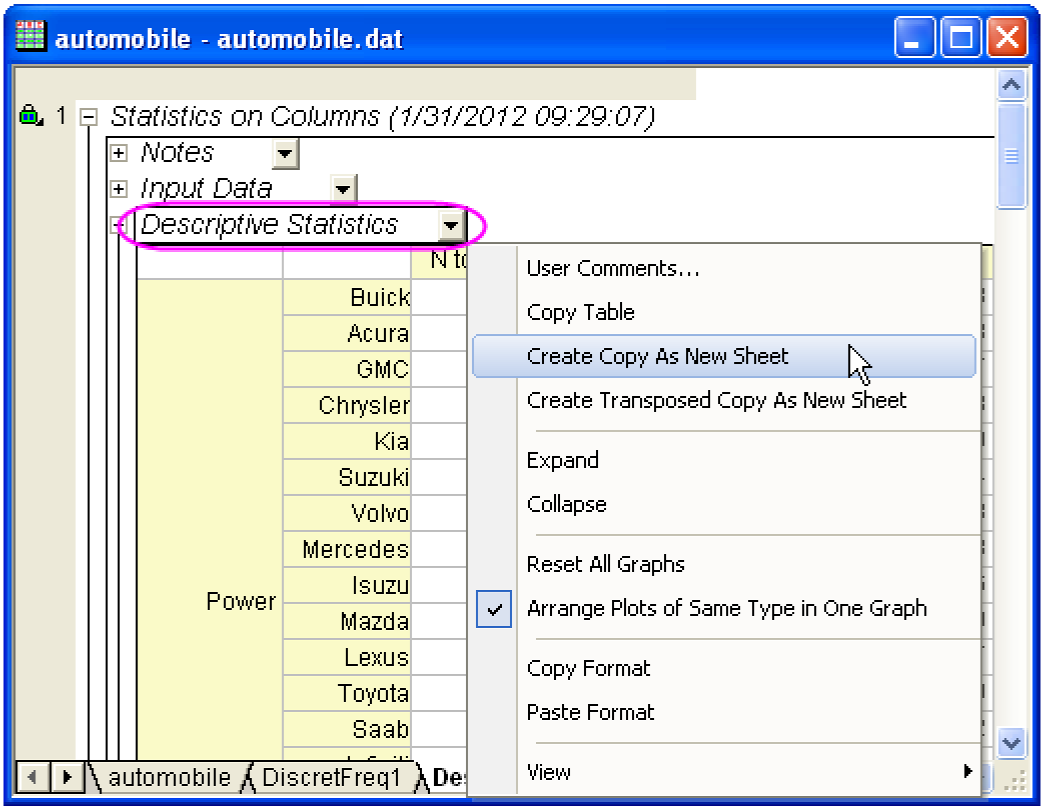
You can double-click to open the graph containing the box plot and customize the graph. Click the **Close** button on the graph to restore the modified graph to the Report Worksheet.

Using Statistical Results for Further Operations

After using the **Statistics on Columns** dialog to produce a report tree, you may wish to do further analysis and plotting on the statistical results.

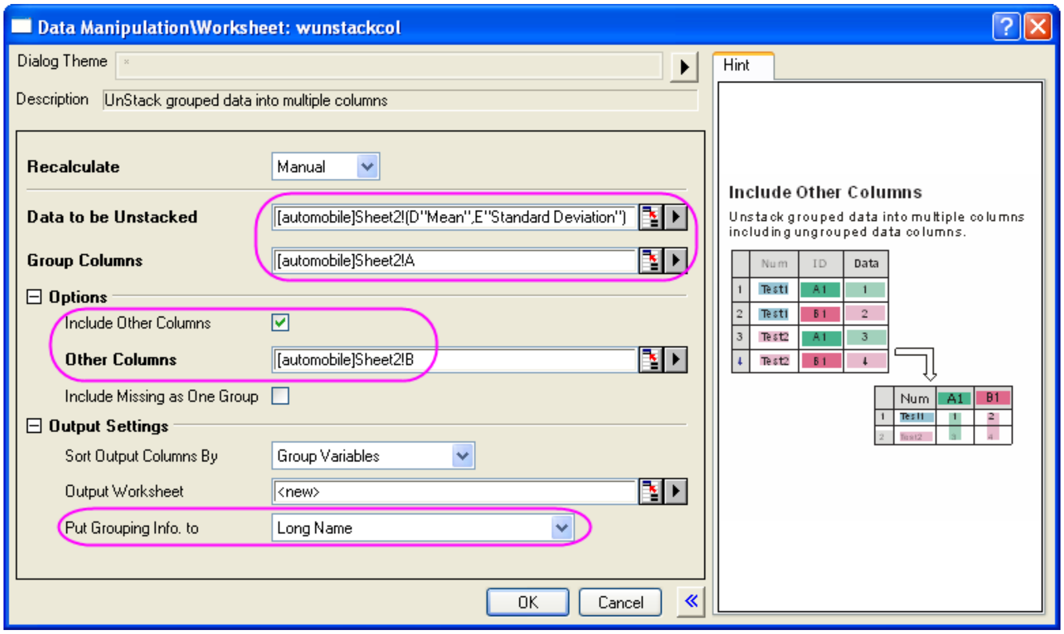
For example, to get average attribute values (i.e. horsepower, 0-60 mph time, weight, mileage) by vehicle Make from 1992 to 2004, perform the following:

1. In the report sheet, right-click on the title of the Descriptive Statistics table and select **Create Copy as New Sheet** from the short-cut menu.

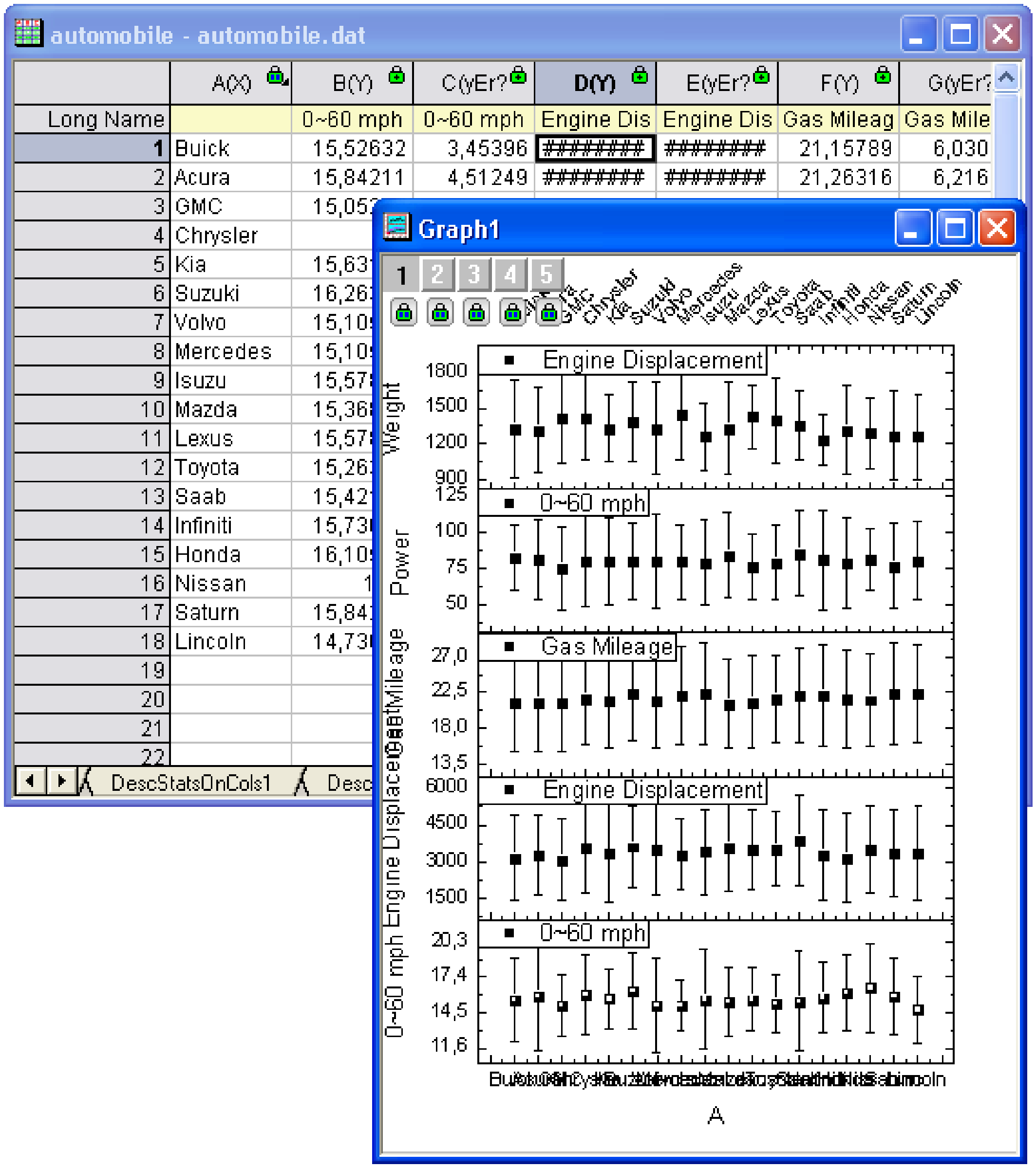


2. When the new sheet is active, select **Worksheet: Unstack Columns**.
3. In the dialog that comes up, set columns D and E as **Data to be Unstacked**. Since the **triangle button**  fly-out menu supports only one selection, you need to use the **interactive button** .
4. Set **column A** as **Group Variables**.
5. Select the **Include Other Columns** check box and set **Other Columns** to column B.

6. Set **Put Grouping Info. to** to **Long Name**. Click the **OK** button.



7. In the result of **Unstack Columns**, we get the mean and standard deviation of Power, 0~60 mph time, Weight, Gas Mileage and Engine Displacement for the 18 different car makes.
8. Highlight the whole result worksheet. Select **Plot: Multi-Curve: Stack** from the main menu.
9. In the pop-up dialog, all columns in the worksheet are automatically set as **Input**. Set **Plot Type** to **Scatter** and click the **OK** button.



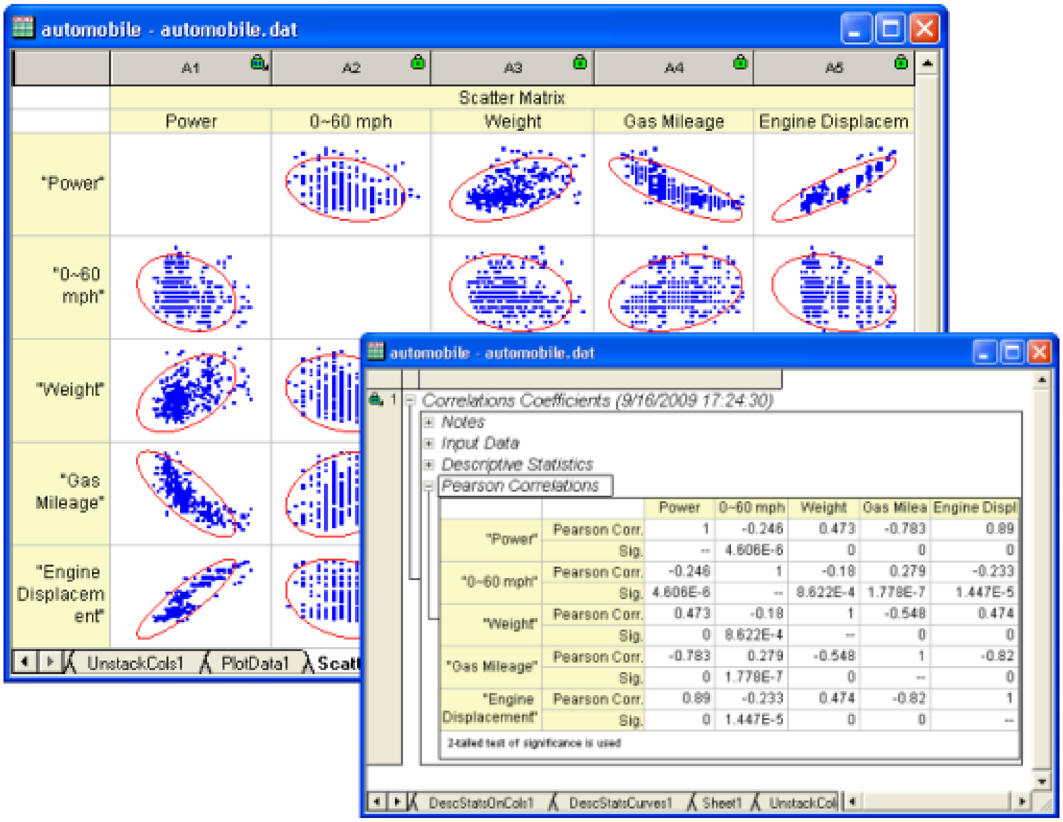
In the above screenshot, the top X-Axis Tick Labels have been rotated 45 degrees for clarity. To do this, double-click on the tick labels to open the **X-Axis** dialog. Set the **Rotation** on the **Custom Tick Labels** tab.

Analyzing the Relationship between different Indicators

We can use a correlation coefficient to explore the relationship between columns of our automobile data. In addition, we can plot a scatter matrix with a confidence ellipse to get a graphical representation of the correlation.

1. Go to the original worksheet with the source data. Highlight the last five columns.
2. Select **Statistics: Descriptive Statistics: Correlation Coefficient** from the Origin menu to open the **Correlation Coefficient** tool. Note that **Pearson** is the default selection. This method is suitable for quantitative data.

3. Under the **Plots** branch, select the **Add Confidence Ellipse** check box. The **Scatter Plot** check box should then be automatically selected. This means that the tool will create a scatter matrix with a confidence ellipse added to each scatter plot. Click **OK**.



Note the high positive correlation between **Engine Displacement** and **Power** and the high negative correlation between **Gas Mileage** and **Engine Displacement**.

4.3.3 2D_Binning

Summary

The 2D Frequency Count/Binning operation counts the frequencies for data with two variables. If needed, a 3D bar graph and/or an image plot of the result will be generated to provide an intuitive demonstration of the distribution of the data points.

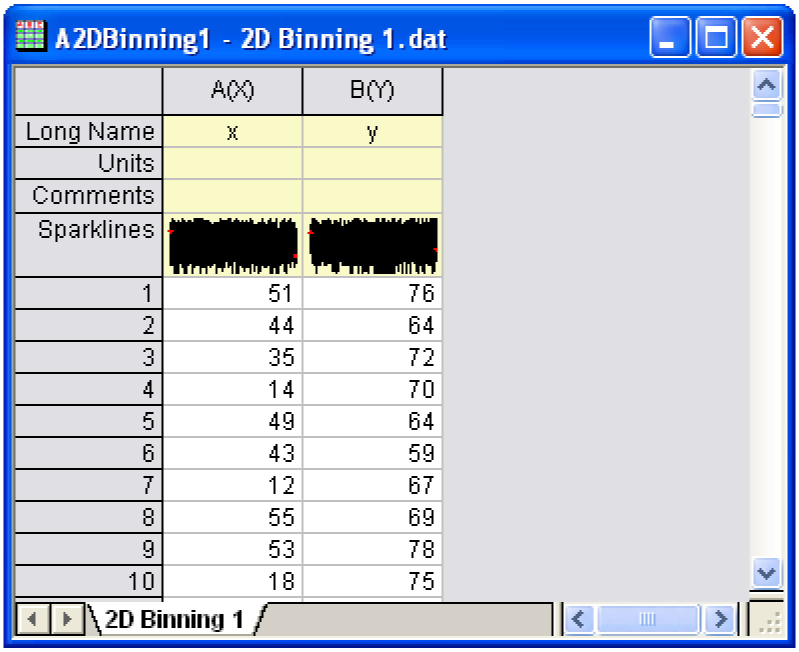
Minimum Origin Version Required: Origin 8.0 SR6

What you will learn

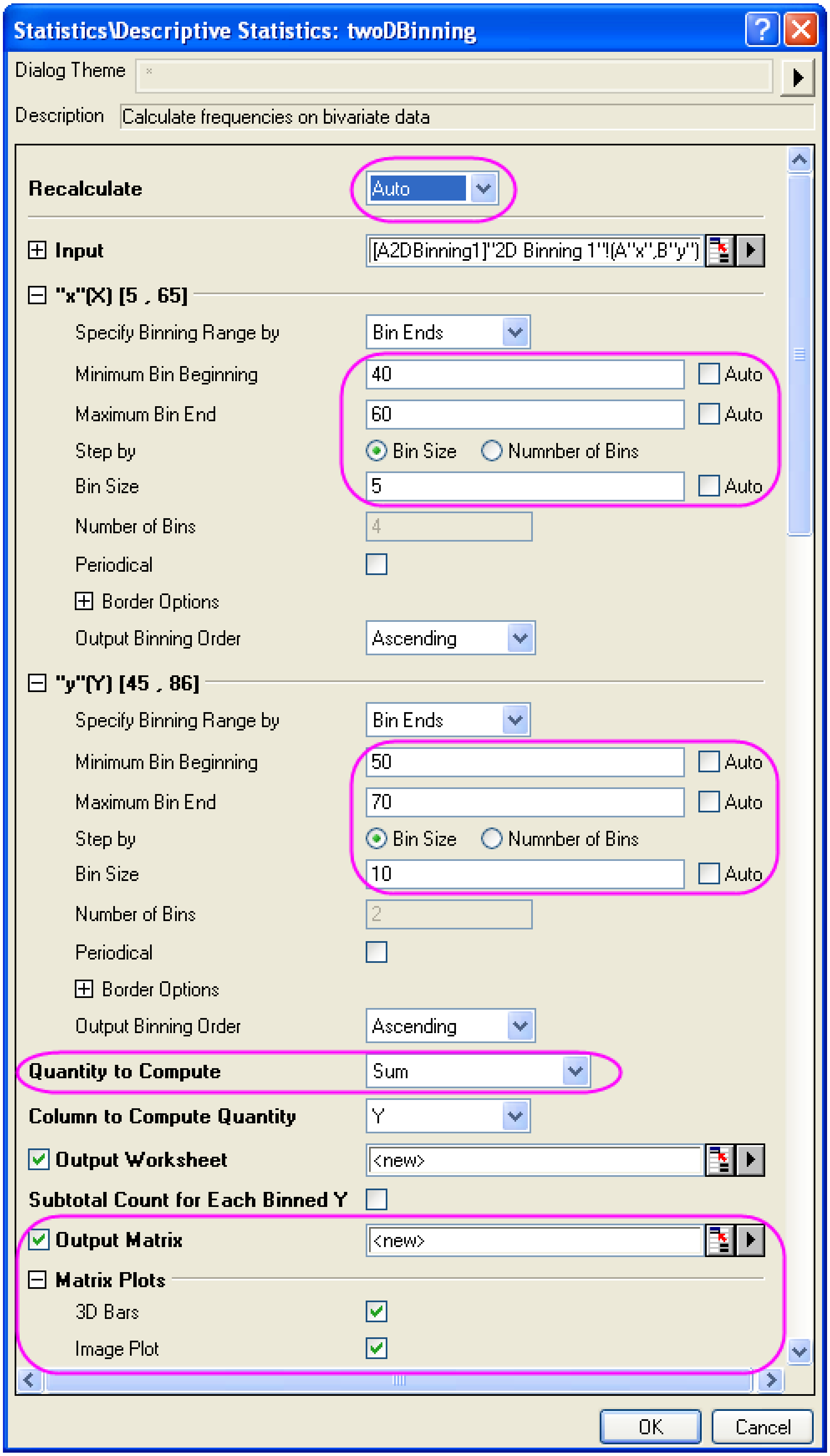
- How to count the frequencies for data with two variables.
- How to add the outliers into the bins.

Steps

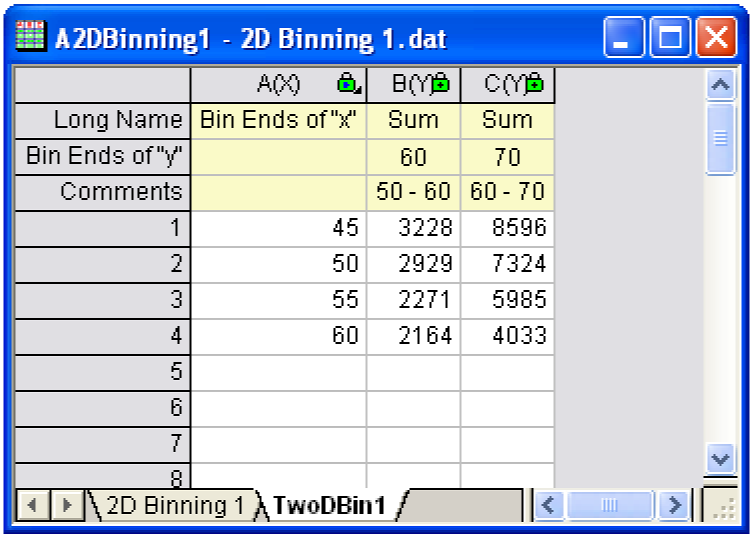
1. Create a new project and import the Origin sample data *2D Binning 1.dat* which is located in <Origin Program Folder> \Samples\Statistics.



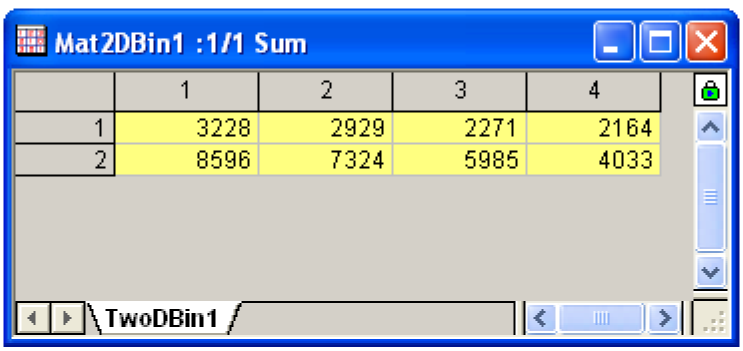
2. Highlight column A and column B, select menu **Statistics: Descriptive Statistics: 2D Frequency Count/Binning** to bring up the **TwoDBinning** dialog.
3. Specify the following settings to in the dialog:
   - Select **Auto** from the **Recalculate** drop-down list.
   - In the **X** branch, uncheck **Auto** for **Minimum Bin Beginning**, **Maximum Bin End**, and **Bin Size** and enter *40*, *60*, and *5* into the three text boxes, respectively. The same parameters in the **Y** branch are set to be *50*, *70*, and *10*, respectively.
   - Choose **Sum** from the **Quantity to Compute** drop-down list. Check **Output Matrix** check box. In the **Matrix Plots** branch, check both **3D Bars** and **Image Plot**.


Statistics\Descriptive Statistics: twoDBinning
Dialog Theme
Description
Calculate frequencies on bivariate data
Recalculate
Auto
Input
[A2DBinning1]"2D Binning 1"!(A"x",B"y")
"x"(X) [5 , 65]
Specify Binning Range by
Bin Ends
Minimum Bin Beginning
40
Auto
Maximum Bin End
60
Auto
Step by
Bin Size
Numnber of Bins
Bin Size
5
Auto
Number of Bins
4
Periodical
Border Options
Output Binning Order
Ascending
"y"(Y) [45 , 86]
Specify Binning Range by
Bin Ends
Minimum Bin Beginning
50
Auto
Maximum Bin End
70
Auto
Step by
Bin Size
Numnber of Bins
Bin Size
10
Auto
Number of Bins
2
Periodical
Border Options
Output Binning Order
Ascending
Quantity to Compute
Sum
Column to Compute Quantity
Y
Output Worksheet
<new>
Subtotal Count for Each Binned Y
Output Matrix
<new>
Matrix Plots
3D Bars
Image Plot
OK
Cancel

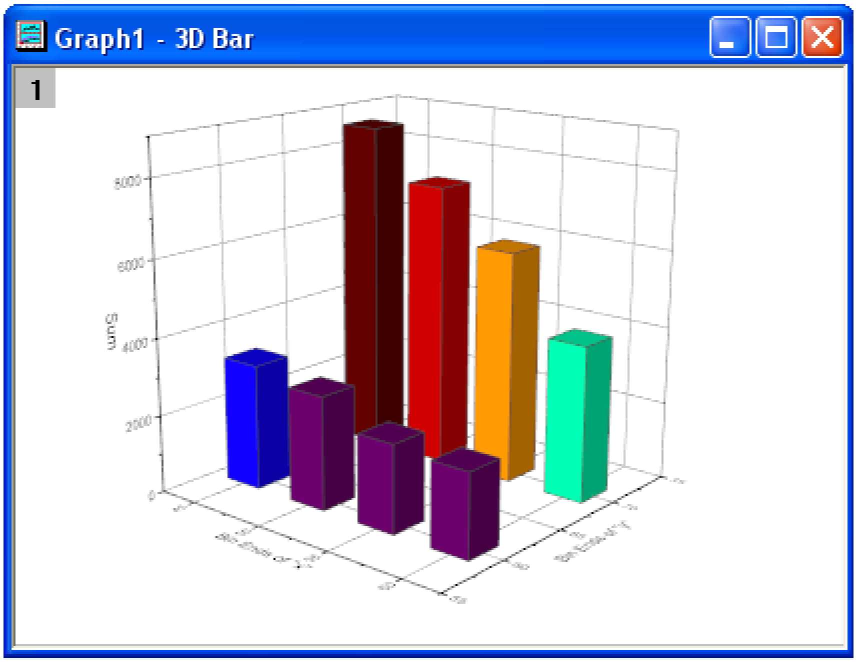
4. Click the **OK** button, then you will get the following outputs:
   - **Worksheet**



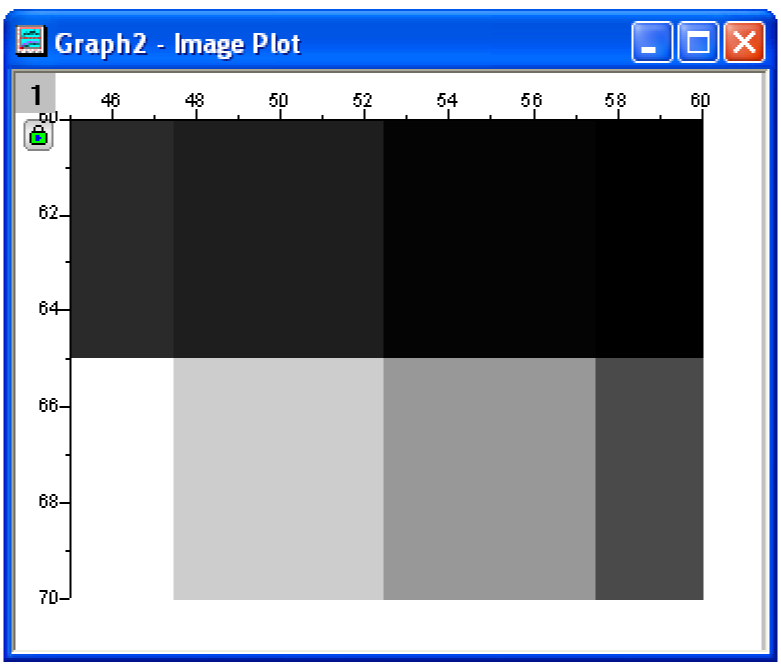
   - **Matrix**



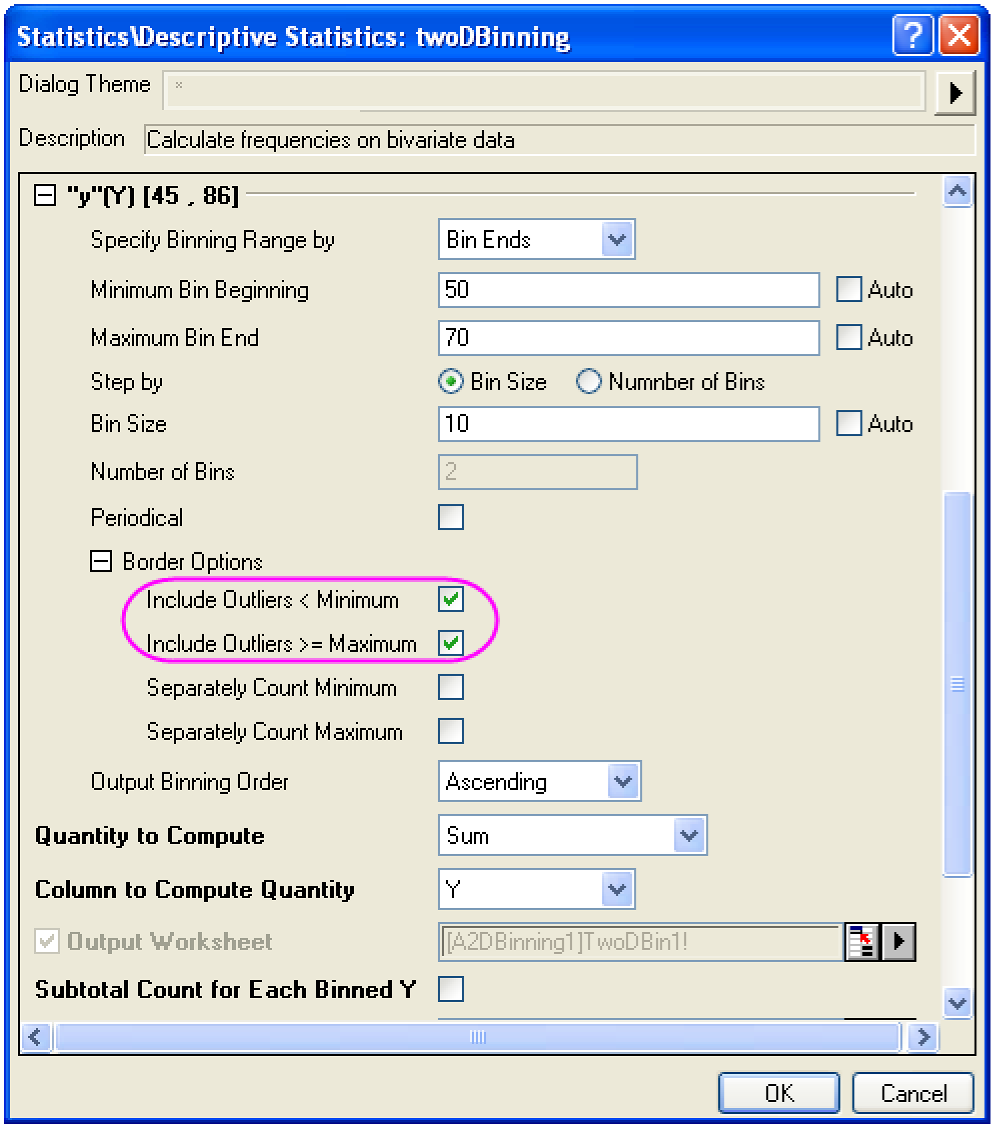
   - **3D Bars**



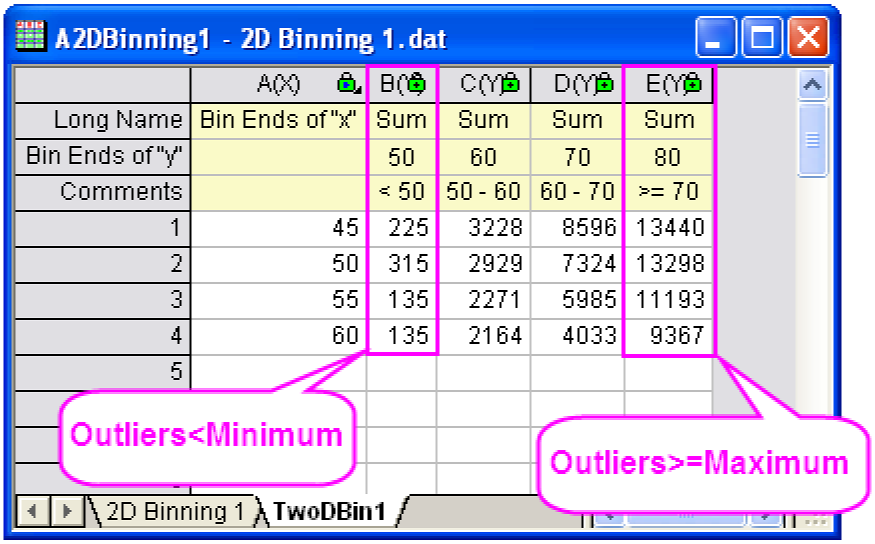
o **Image Plot**



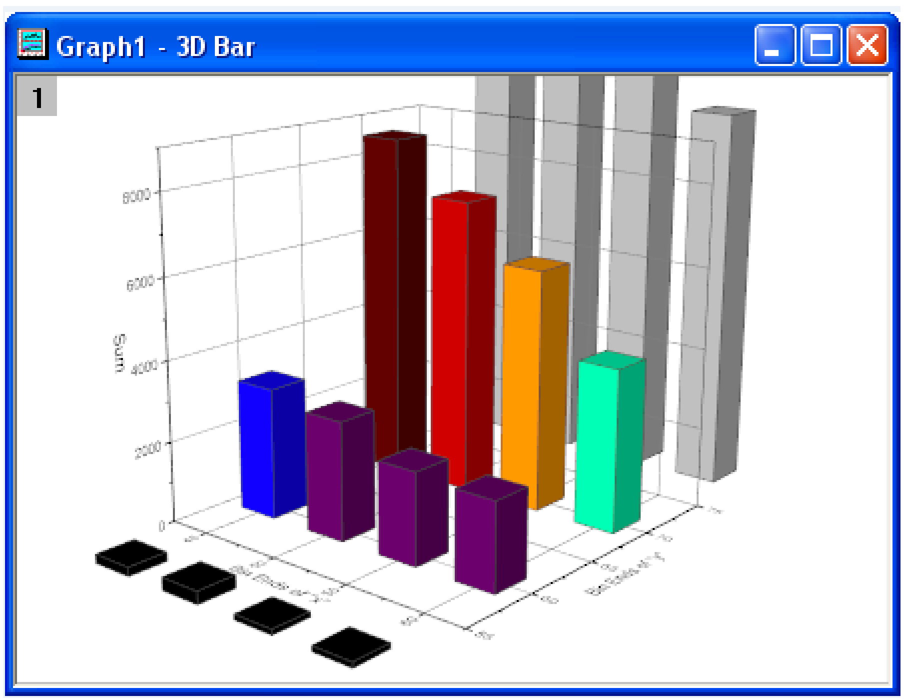
5. To add the outliers for Y variable into the bins, click on the lock icon in the **TwoDBin1** worksheet, and select **Change Parameters** to open the dialog again.
6. In the **Y** branch, expand **Border Options** node, then check both **Include Outliers<Minimum** and **Include Outliers>=Maximum**.

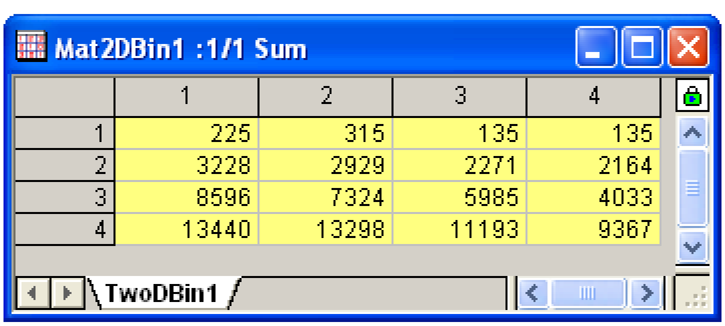


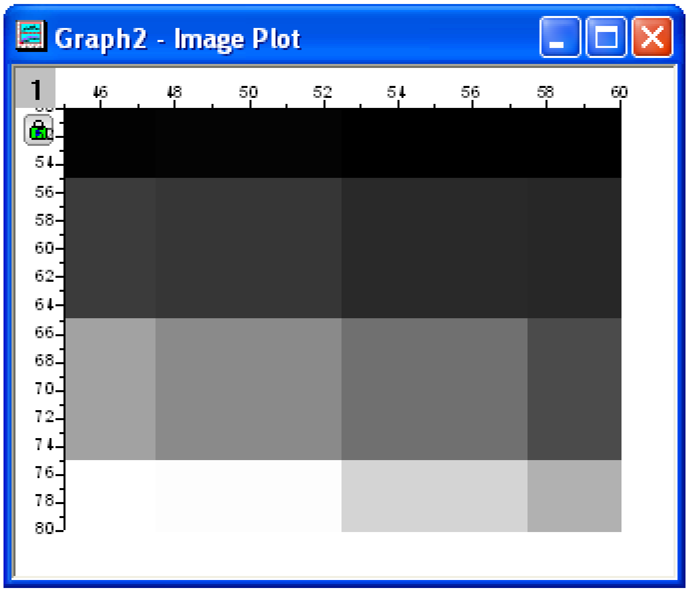
7. Click the **OK** button. Two columns for the outliers are added to the **TwoDBin1** worksheet.



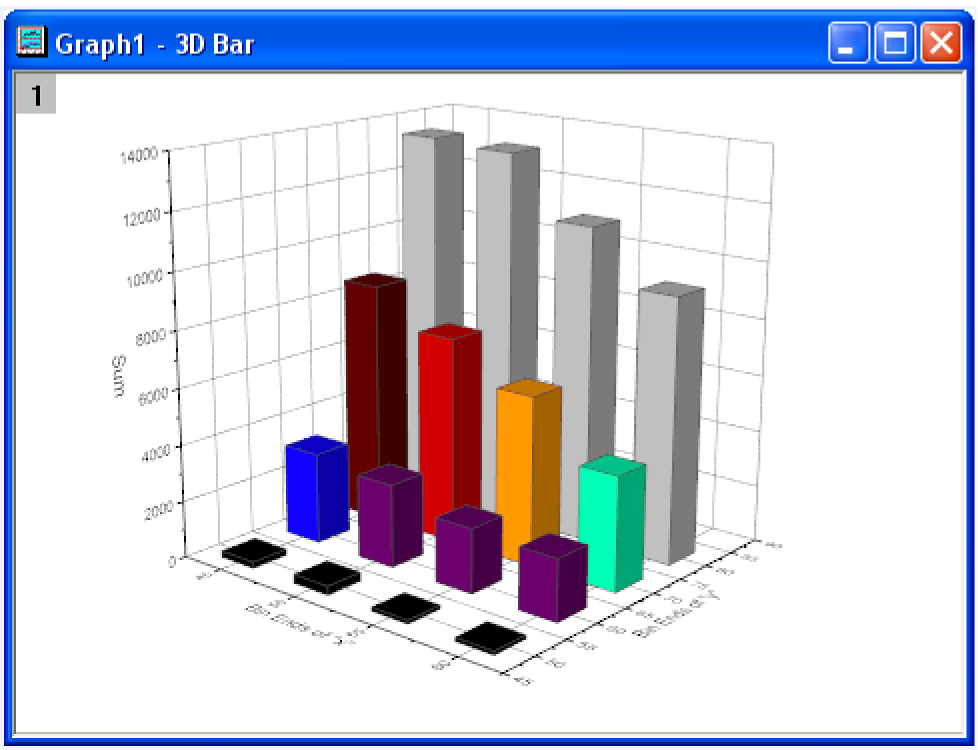
Matrix, 3D Bars Graph, and the Image Plot will be updated as well.







8. Double click on Y axis and open **Axes Dialog** ,chage **scale from**and **to** to 45 and 90. Click Z axis scale, and chage **to** to be 14000.



4.4 ANOVA

- One Way ANOVA

4.4.1 One Way ANOVA

Summary

There are two main modes of datasets in Statistics - indexed and raw. When you perform an analysis, you do not need to use the whole dataset, so Origin provides several ways to select data. For example, you can use the interactive Regional Data Selector button to graphically select the data or you can use the **Column Browser** dialog to make your selection.

In this tutorial, you'll use the Analysis of Variance (ANOVA) statistical test, to learn how to use these two different modes of data to perform analysis and how to select data by using the **Column Browser** dialog.

ANOVA is a kind of parametric method for means comparison and is an extension of t-test. When there are more than two groups to be compared, pairwise t-test is not appropriate and ANOVA should be used. ANOVA requires normality and equal variance. Otherwise, non-parametric analysis should be used.

Minimum Origin Version Required: Origin 8.0 SR6

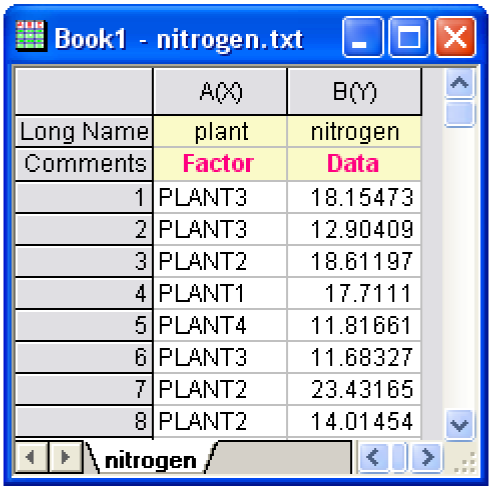
What you will learn

This tutorial will show you how to:

- Use different input data mode on statistical analysis dialog
- Test normality for special part of dataset
- Perform one-way ANOVA
- Select data by Column Browser

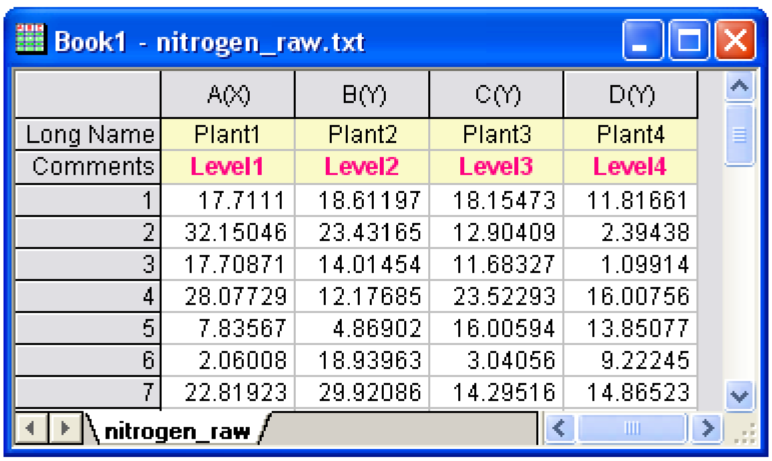
Steps

Origin can calculate ANOVA in indexed as well as raw data mode. For One-Way ANOVA, when using indexed mode, data should be organized in two columns : one for Factor and the other for data.



	A(X)	B(Y)
Long Name	plant	nitrogen
Comments	Factor	Data
1	PLANT3	18.15473
2	PLANT3	12.90409
3	PLANT2	18.61197
4	PLANT1	17.7111
5	PLANT4	11.81661
6	PLANT3	11.68327
7	PLANT2	23.43165
8	PLANT2	14.01454

When using Raw data mode, the different levels are in different columns.



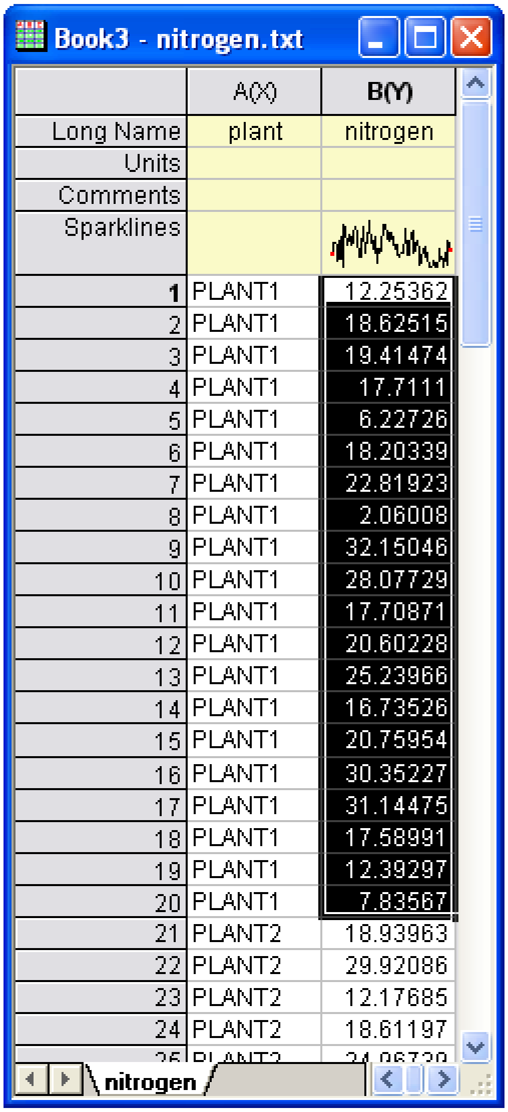
	A(X)	B(Y)	C(Y)	D(Y)
Long Name	Plant1	Plant2	Plant3	Plant4
Comments	Level1	Level2	Level3	Level4
1	17.7111	18.61197	18.15473	11.81661
2	32.15046	23.43165	12.90409	2.39438
3	17.70871	14.01454	11.68327	1.09914
4	28.07729	12.17685	23.52293	16.00756
5	7.83567	4.86902	16.00594	13.85077
6	2.06008	18.93963	3.04056	9.22245
7	22.81923	29.92086	14.29516	14.86523

Indexed data mode

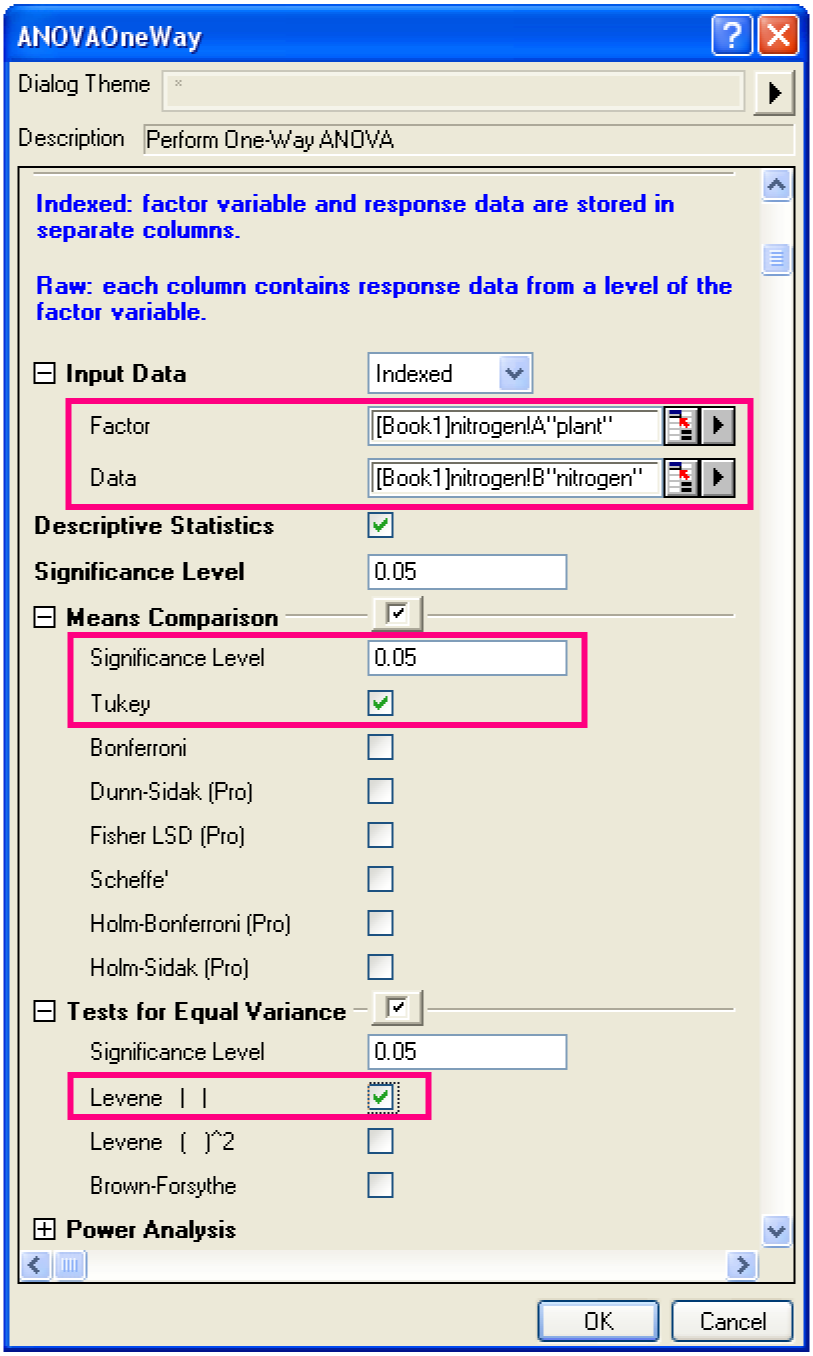
Nitrogen content has been recorded in milligrams for 4 kinds of plant, and we are interested in whether different plants have different nitrogen content. We will perform One-Way ANOVA using index data mode for this example.

1. Start with a new workbook and import the file *\Samples\Statistics\nitrogen.txt*. Make sure you select *.txt* from the drop-down menu Files of type. First, we should perform a normality test on each group of data to determine if they are from a normal distribution.
2. Highlight the first column, right-click and select **Sort Worksheet** from the Worksheet menu and choose **Ascending**.
3. Highlight the second column from row 1 to row 20 - which belongs to "PLANT1" - and open the **Normality Test** dialog by choosing the menu item **Statistics: Descriptive Statistics:**

Normality Test.



4. Use the default setting of the dialog and click **OK**. From the p-value of result,p-value=0.58545, we can see "PLANT1" follows a normal distribution.
5. In a similar way, you can highlight the range of data "PLANT2", "PLANT3" and "PLANT4" and test for Normality. Our sample data has normal distribution for all plants.
6. With our nitrogen data worksheet active, open the **ANOVAOneWay** dialog by using the menu item **Statistics: ANOVA: One-Way ANOVA**. Set the **Input Data** mode as **Indexed**, assign the "plant" and "nitrogen" column as **Factor** and **Data** respectively using the right-arrow buttons. Click the + to expand the **Means Comparison** node, set **Significance Level** as 0.05 and check the **Tukey** Means Comparison method. Check **Levene | |** from **Tests for Equal Variance** branch. Click the **OK** button to perform One-Way ANOVA.



Explaining the result:

- From the "Homogeneity of Variance Test" table of one-way ANOVA result, we can see that the four groups have equal variance, since the p-value is bigger than 0.05.

Homogeneity of Variance Test

Levene's Test(Absolute Deviations)

	DF	Sum of Squares	Mean Square	F Value	Prob>F
Model	3	18.06843	6.02281	0.34578	0.79229
Error	76	1323.76846	17.41801		

At the 0.05 level, the population variances are not significantly different.

- From the result of Overall ANOVA we can conclude that at least two groups of the four have significant different means, since the p-value is smaller than 0.05.

Overall ANOVA

	DF	Sum of Squares	Mean Square	F Value	Prob>F
Model	3	1996.36652	665.45551	12.86214	6.99338E-7
Error	76	3932.05317	51.73754		
Total	79	5928.41969			

Null Hypothesis: The means of all levels are equal.
Alternative Hypothesis: The means of one or more levels are different.
At the 0.05 level, the population means are significantly different.

- To research further, we expand the results of "Means Comparisons".

Means Comparisons

Tukey Test

	MeanDiff	SEM	q Value	Prob	Alpha	Sig	LCL	UCL
PLANT2 PLANT1	2.26308	2.27459	1.40706	0.75274	0.05	0	-3.71181	8.23796
PLANT3 PLANT1	-2.46538	2.27459	1.53284	0.70039	0.05	0	-8.44027	3.5095
PLANT3 PLANT2	-4.72846	2.27459	2.93989	0.16935	0.05	0	-10.70334	1.24643
PLANT4 PLANT1	-10.93833	2.27459	6.80085	4.38499E-5	0.05	1	-16.91322	-4.96345
PLANT4 PLANT2	-13.20141	2.27459	8.20791	8.24355E-7	0.05	1	-19.1763	-7.22653
PLANT4 PLANT3	-8.47295	2.27459	5.26801	0.00207	0.05	1	-14.44784	-2.49807

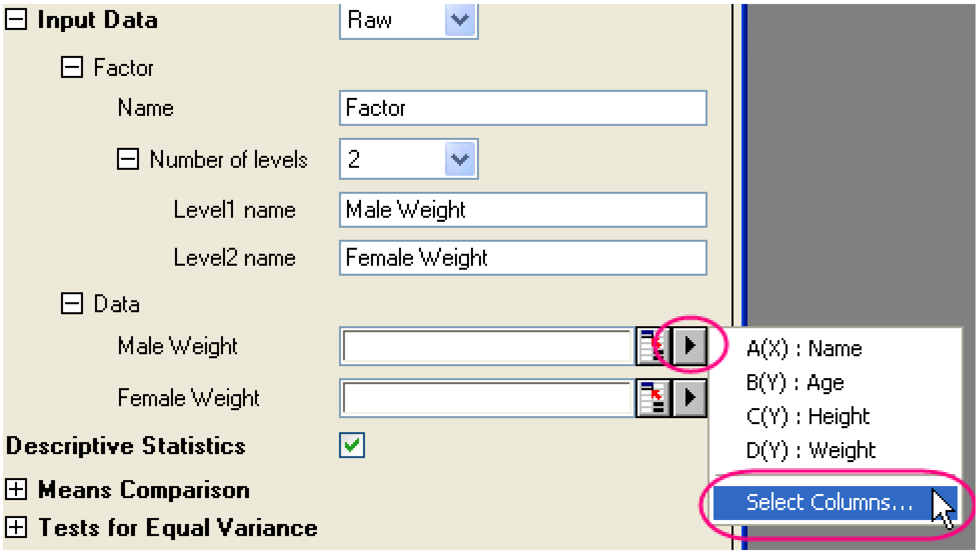
Sig equals 1 indicates that the means difference is significant at the 0.05 level.
Sig equals 0 indicates that the means difference is not significant at the 0.05 level.

Here we see that PLANT4 has significantly different means when compared to each of the other three groups.

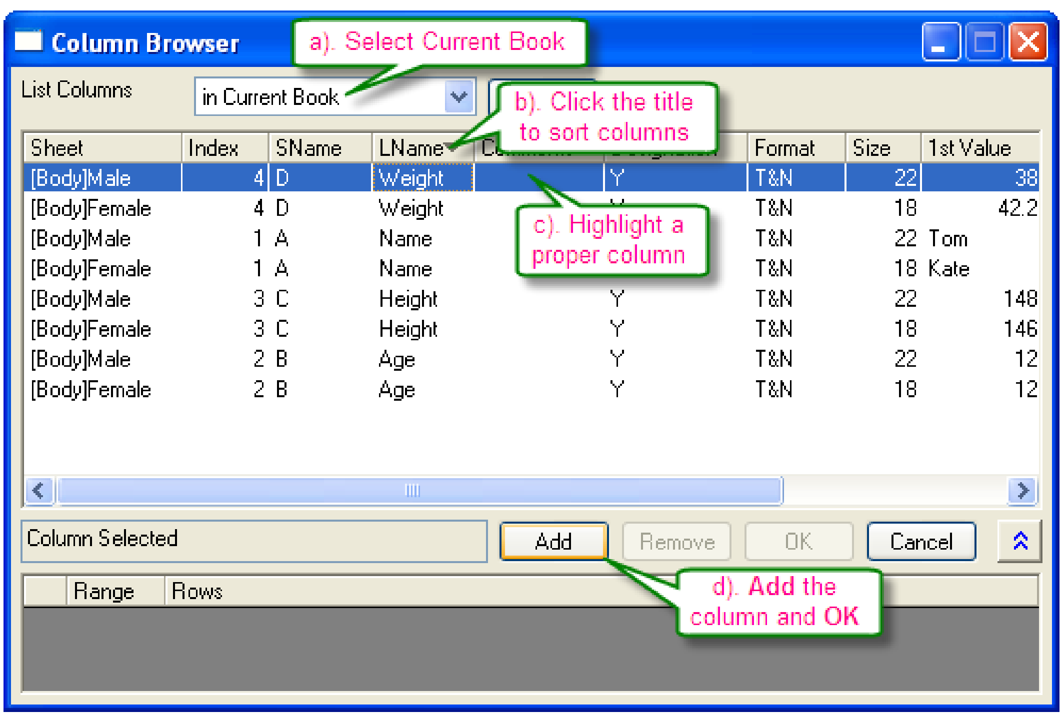
raw data mode

1. Select **File : Open** and choose **WorkBooks** from **Files of type** drop-down list, and browse to *\Samples\Statistics* folder and open the file *Body.ogw*
2. Select menu item **Statistics : ANOVA : One-Way ANOVA** to bring up the **ANOVAOneWay** dialog. Choose **Raw** as **Input Data** mode. Enter the **Level1 Name** and **Level2 Name** as "Male Weight" and "Female Weight" respectively.
3. Now we will use the **Data Browser** to select data in the **Data** branch. Click the triangle icon beside **Male Weight** edit box, in the fly-out menu, select **Select Columns...** to open the

Column Browser dialog.



In the Column Browser dialog, you can select **in Current Book** from **List Columns** drop-down list to see all available worksheet columns in the current book. Select *Weight* in the sheet *[Body]Male* and click **Add** and **OK** to add it to **Male Weight** edit box. Similarly, assign *Weight* from *[Body]Female* to **Female Weight** edit box.



4. Accept other default settings in the ANOVAOneWay dialog and click **OK**. From the output report footnote, we can conclude that at the 0.05 level, the population weight means between male and female are not significantly different.

4.5 Nonparametric Tests

4.5.1 Non-parametric Statistics Overview

Summary

Nonparametric tests is widely used when you don't know whether your data follows normal distribution, or you have confirmed that your data do not follow normal distribution. Meanwhile, hypothesis tests are parametric tests based on the assumption that the population follows a normal distribution with a set of parameters.

What you will learn

This tutorial will show you:

- An introduction on non-parametric tests in Origin
- How to run the non-parametric tests for different practical situations
- How to calculate correlation coefficient in non-parametric statistics

Introduction: Nonparametric Tests in Origin

Nonparametric tests does not require the normality assumption. It is commonly used in the following situations:

- Small sample size
- Categorical/Binary/Ordinal data
- Normal distribution can not be assumed

		Nonparametric	**Parametric**
		Data from any distribution	**Data from normal distribution**
		Small Samples	**Large Samples**
One Sample		Wilcoxon Signed Rank Test	One Sample T-Test
Two Samples	Independent Samples	• Mann-Whitney test • Kolmogorov-Smirnov test	Two Sample T-Test
	Paired Samples	• Wilcoxon signed rank Test • Sign Test	Paired Sample T-Test
Multiple Samples	Independent Samples	• Kruskal-Wallis ANOVA • Mood's Median Test	One Way ANOVA
	Related Samples	Friedman ANOVA	One Way Repeated Measure ANOVA

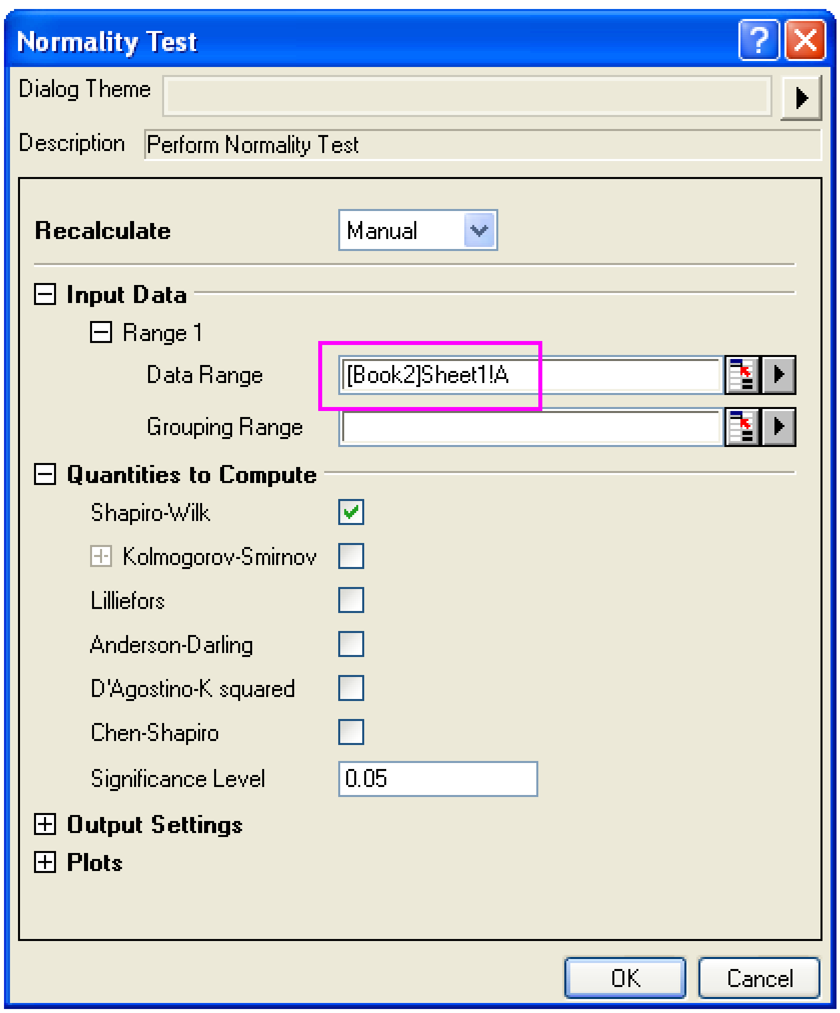
	Correlation in Samples	• Spearman • Kendall	Pearson

Examples

One Sample Independent Tests

The One-Sample Wilcoxon Signed Rank test is designed to examine the population median relative to a specified value. You may choose a one- or two-tailed test. The Wilcoxon signed rank test hypotheses are H0: median = hypothesized median versus H1: median ? hypothesized median.
In this example, a quality engineer in a production shop is interested in whether median (or average) of the weight of product is equal to 166. So select 10 product at random and measured their weight . The data measured as following:
151.5 152.4 153.2 156.3 179.1 180.2 160.5 180.8 149.2 188.0
The engineer perform **Normality Test** to determine if the distribution of the data is normal distribution

1. Open a new worksheet and input the above data in col(A). Select **Statistics: Descriptive Statistics: Normality Test...** to open the **Normality Test** dialog.
2. Select A(X) as **Data range**.



3. Click the **OK** button to generate results.

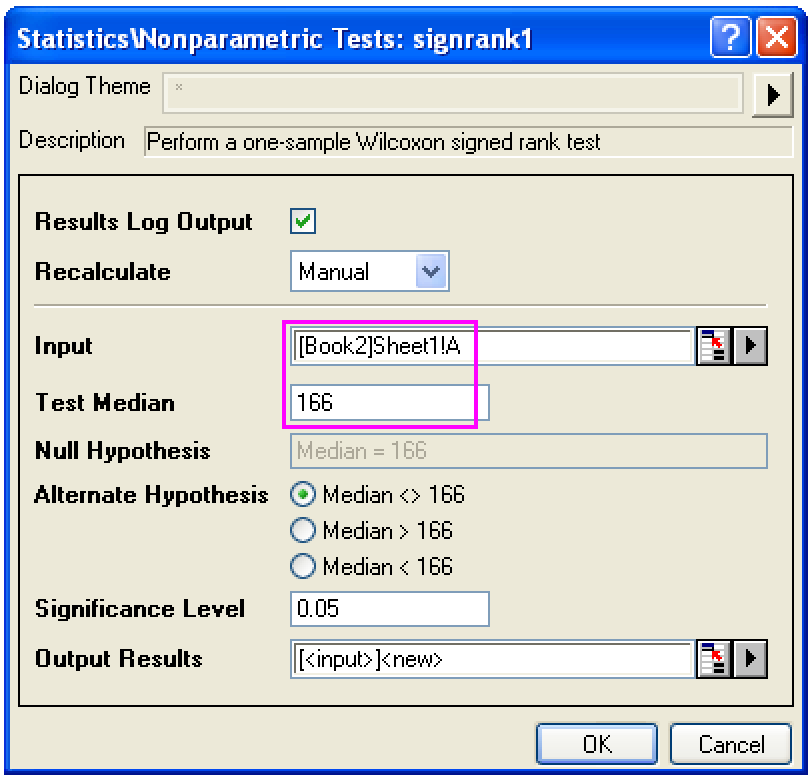
Shapiro-Wilk

	DF	Statistic	p-value	Decision at level(5%)
B	10	0.83472	0.03814	Reject normality

B: At the 0.05 level, the data was not significantly drawn from a normally distributed population.

According to result,P-value=0.03814,the distribution of the data is not normal distribution at the 0.05 level. So, perform One-Sample Wilcoxon Signed Rank test:

1. Select **Statistics: Nonparametric Tests: One-sample Wilcoxon Signed Rank Test...** to open the dialog.
2. Set column A as **Data Range**.
3. Input **166** in **Test Median** text box.



4. Click the **OK** button to generate results

Descriptive Statistics

	N	Min	Q1	Median	Q3	Max
A	10	149.2	152.175	158.4	180.35	188

Test Statistics

	W	Z	Extact Prob>=|W|	Asymp. Prob >=|W|
A	28	0	1	1

Null Hypothesis: Median = 166
Alternative Hypothesis: Median <> 166
A: At the 0.05 level, the population median is NOT significantly different from the test median (166).

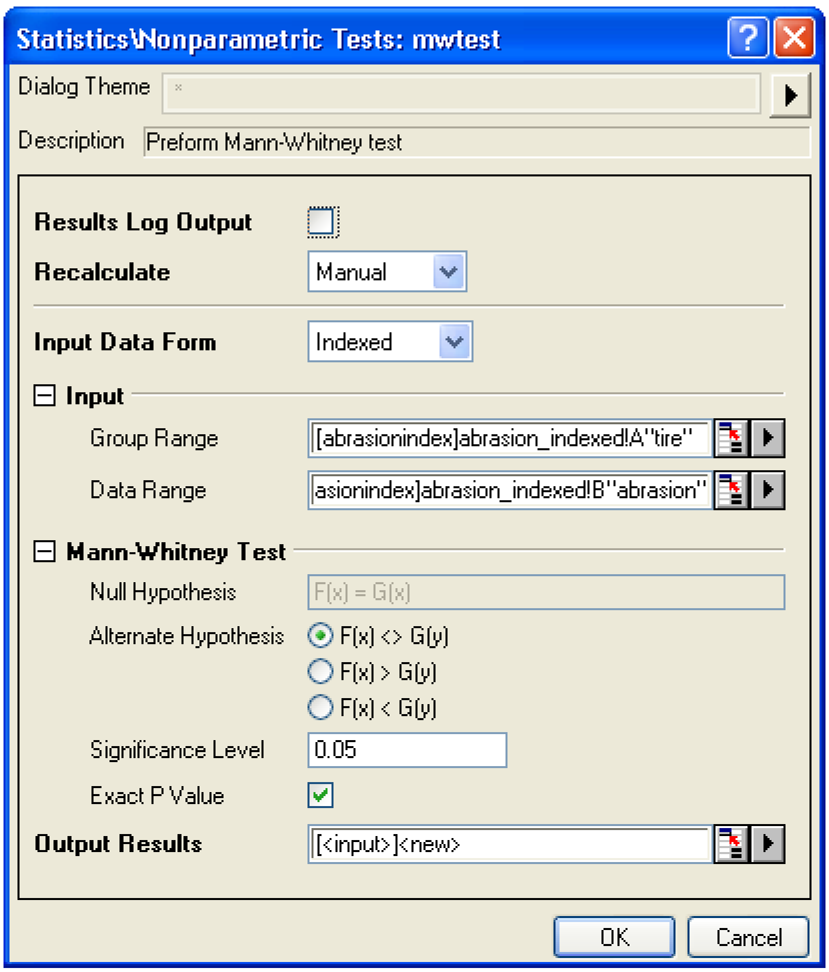
According to the result, it fails to reject null hypothesis at the 0.05 level and concludes that the median is equal to 166.

Two Sample Independent Tests

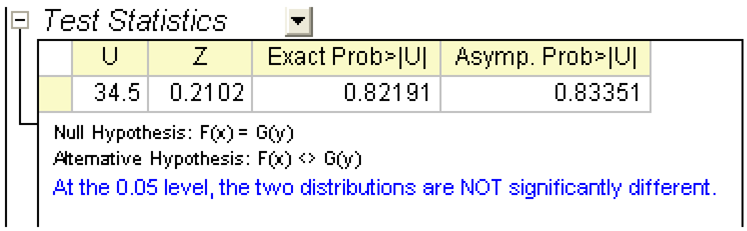
Origin provides two tests for non-parametric statistics of two sample independent system: the **Mann-Whitney Test** and **Two Sample Kolmogorov-Smirnov Test**.

This following example shows the practical use of **Mann-Whitney Test**. The abrasions(in mg) are measured for two types of tires(A and B), 8 experiments were carried out for each tire type. The data is indexed and stored in abrasion_indexed.dat file.

1. Import the abrasion_indexed.dat file from **\Samples\Statistics**
2. Select **Statistics: Nonparametric Tests: Mann-Whitney Test** to open the dialog.
3. Keep **Input Data Form** as **Index**
4. Set column A as **Group Range**, set column B as **Data Range**.
5. Select the **Exact P Value** check box.



6. Click the **OK** button to generate results, which should be in the **MannWhitney1** sheet



Test Statistics

	U	Z	Exact Prob>|U|	Asymp. Prob>|U|
	34.5	0.2102	0.82191	0.83351

Null Hypothesis: F(x) = G(y)
Alternative Hypothesis: F(x) <> G(y)
At the 0.05 level, the two distributions are NOT significantly different.

- **U**: The **U** statistic can be simply calculated from the rank of two groups. It is the number of times a score in the 2nd group is larger than a score in the 1st group.
- **Z**: The approximate Normal test statistic. It provides an excellent approximation as the sample size grows.
- **Exact Prob**: The exact p-value, only available when **Exact P Value** is selected in the dialog. However, it could be very CPU-time consuming for large sample size.
- **Asymp.Prob**: The asymptotic p-value calculated from the approximate Normal test statistic, **Z**

Non-parametric Measures of Correlation

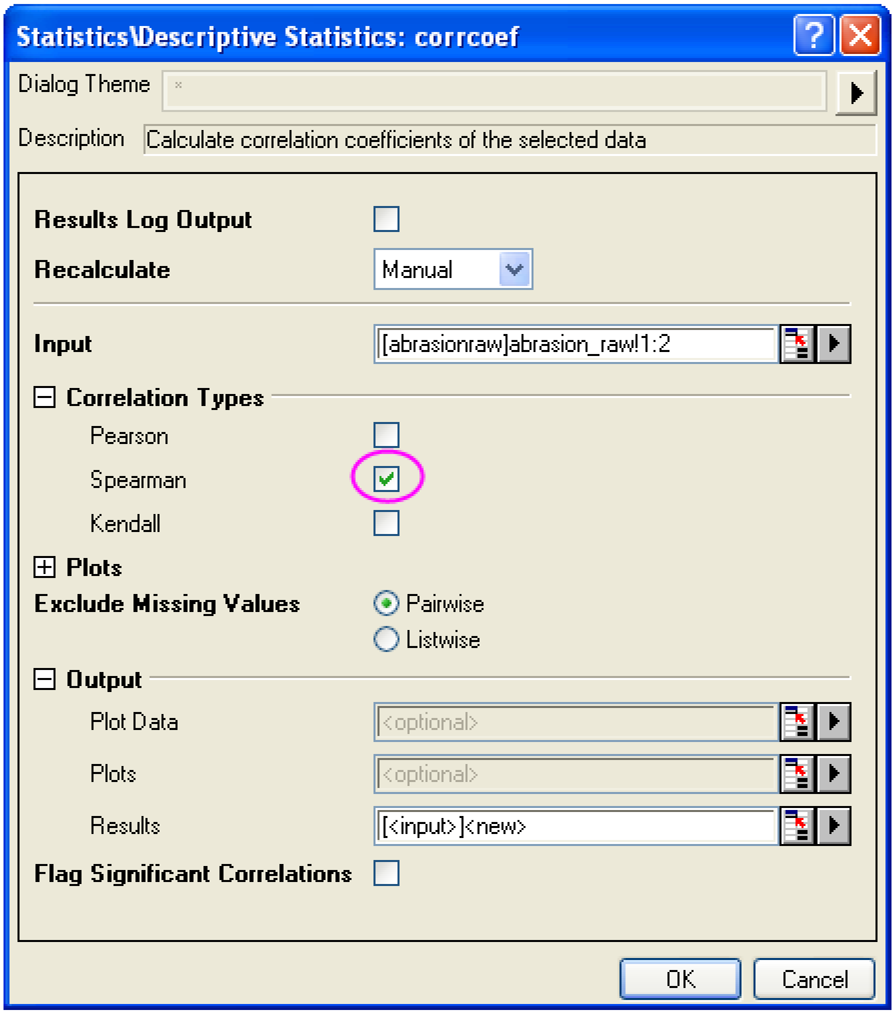
Correlation coefficient is used as a measure of relationship between two variables.It is possible to calculate the correlation coefficient for non-parametric statistics.

Origin provides two non-parametric methods to measure the correlations between variables:

- **Spearman**: common substitution of Pearson correlation coefficient,Spearman's coefficient can be used when both dependent (variable and independent variable are ordinal numeric, or when one variable is a ordinal numeric and the other is a continuous variable. However, it can also be appropriate to use Spearman's correlation when both variables are continuous.
- **Kendall**: Used with ordinal variables for assessing agreement among raters

The following example shows how to calculate correlation coefficient for non-parametric situations.

1. Import the abrasion_raw.dat file from **Samples\Statistics**;
2. Highlight Column A and column B. Select **Statistics:Descriptive Statistics:Correlation Coefficient** to open the corrcoef dialog;
3. Check **Spearman** and uncheck **Pearson**;



4. Click the **OK** button to generate the results, in the **CorrCoef1** sheet.

From the value of **Spearman Corr.**, it can be concluded that the abrasion between tire A and tire B are strongly related.

Spearman Correlations

		tireA	tireB
"tireA"	Spearman Corr.	1	0.90476
	Sig.	--	0.00201
"tireB"	Spearman Corr.	0.90476	1
	Sig.	0.00201	--

2-tailed test of significance is used

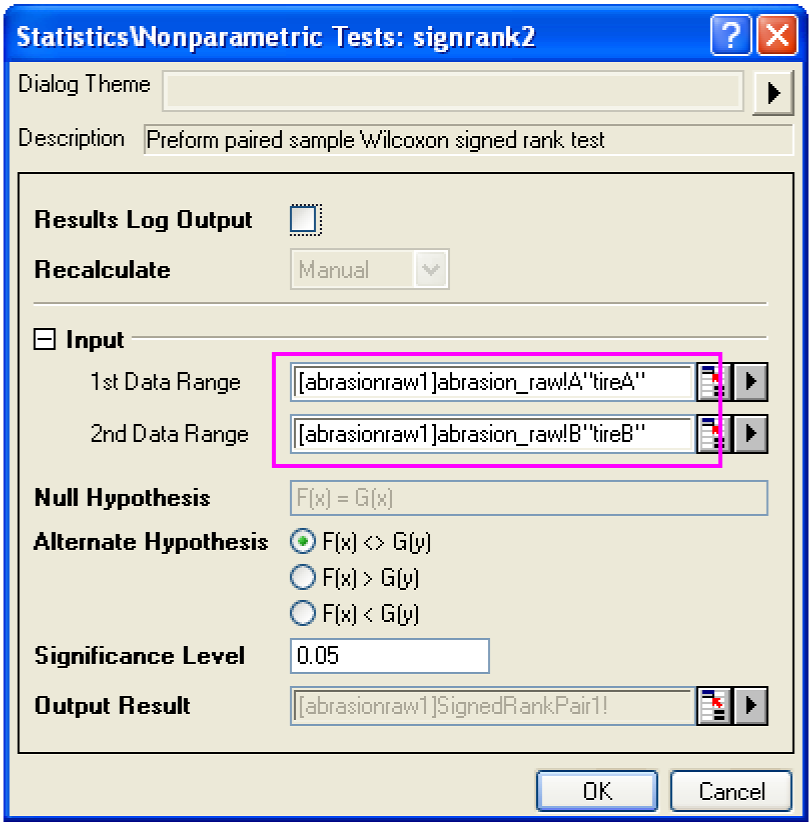
Paired Sample wilcoxon Signed Rank Tests

We will compare the two medians of *tire A* and *tire B* in above example.

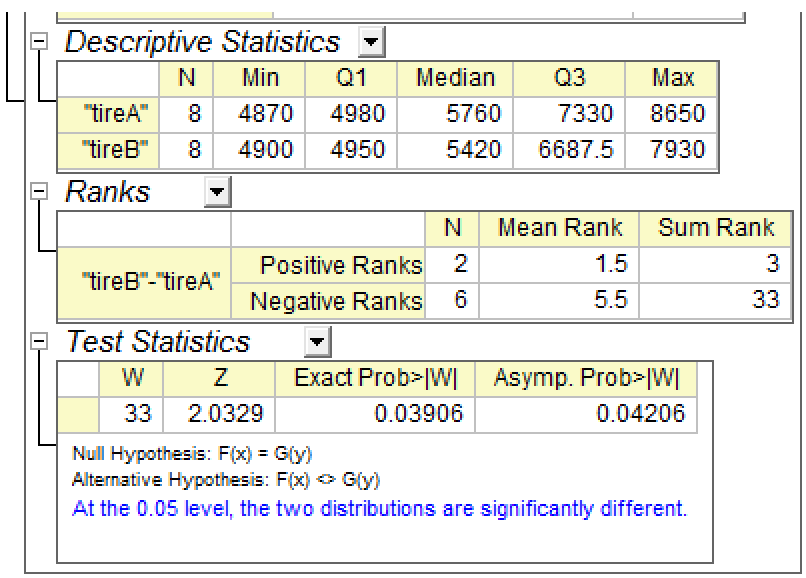
1. Continue with the abrasion_raw.dat file from **Samples\Statistics**;

2. Select **Statistics: Nonparametric Tests: Paired Sample wilcoxon Signed Rank Tests** to open the dialog;

3. Selcet Column A as **1st Range Data** and column B as **2nd range Data**;



4. Click the **OK** button to generate the results.



Descriptive Statistics

	N	Min	Q1	Median	Q3	Max
"tireA"	8	4870	4980	5760	7330	8650
"tireB"	8	4900	4950	5420	6687.5	7930

Ranks

		N	Mean Rank	Sum Rank
"tireB"-"tireA"	Positive Ranks	2	1.5	3
	Negative Ranks	6	5.5	33

Test Statistics

	W	Z	Exact Prob>|W|	Asymp. Prob>|W|
	33	2.0329	0.03906	0.04206

Null Hypothesis: F(x) = G(y)
Alternative Hypothesis: F(x) <> G(y)
At the 0.05 level, the two distributions are significantly different.

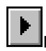
We can conclude that two medians are significantly different. Obviously, median of group A is larger than that of group B.

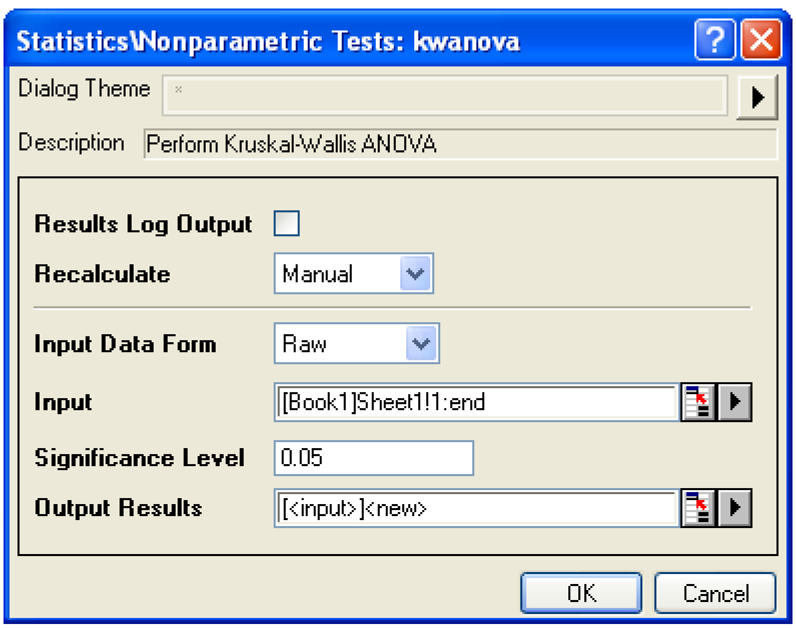
Multiple Independent Samples Test

In this example, the gas mileage of four car makers are measured. Several experiments are carried out for each car makers. The results are listed in the sample data table.

GMC / mpg	Infinity / mpg	Saab / mpg	Kia / mpg
26.1	32.2	24.5	28.4
28.4	34.3	23.5	34.2
24.3	29.5	26.4	29.5
26.2	35.6	27.1	32.2
27.8	32.5	29.9	
30.6	30.2		
28.1			

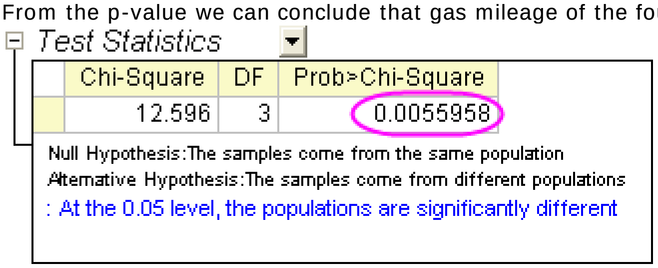
To evaluate whether the gas mileage of the four car makers are equal, and which one is the most efficient, Kruskal-Wallis ANOVA is chosen as the nonparametric test method.

1. Create a new workbook in Origin, copy the sample data and paste into it;
2. Select **Statistics:Nonparametric Tests:Kruskal-Wallis ANOVA** to open the kwanova dialog;
3. Specify Raw as **Input Data Form**;
4. Click the **triangle button**  next to **Input**, and select **All Columns** in the context menu;



5. Click the **OK** button to generate results, the results are stored in a new worksheet **KWANOVA1**.

From the p-value we can conclude that gas mileage of the four car makers are significant different.



From the rank table we can conclude that **Infinity** is the most efficient one.

Ranks

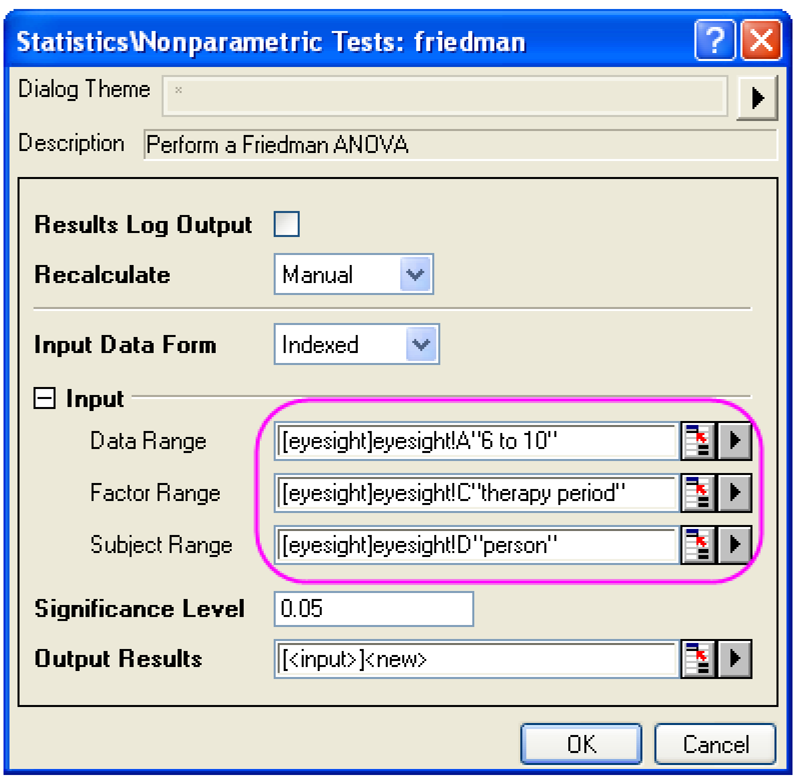
	N	Mean Rank	Sum Rank
"GMC/mpg"	7	7.7857	54.5
"Infinity/mpg"	6	17.833	107
"Saab/mpg"	5	6.2	31
"Kia/mpg"	4	15.125	60.5

Multiple Related Samples Test

Ophthalmologists are investigating whether laser He-Ne therapy works for children. They have data from 2 groups, 6-10 Years Old and 11-16 Years Old. Each data set contains study of 5 persons' naked-eye eyesight difference after 3 period of therapy. The results are stored in the eyesight.dat.

Due to the small sample size, non-parametric statistics would be needed in analysis, following the steps below:

1. Import the eyesight.dat file from **\Samples\Statistics**;
2. Select **Statistics:Nonparametric Tests:Friedman ANOVA** to open the friedman dialog;
3. Select Column A as **Data Range**, Column C as **Factor Range**, and Column D as **Subject Range**;



4. Click the **OK** button to generate results.

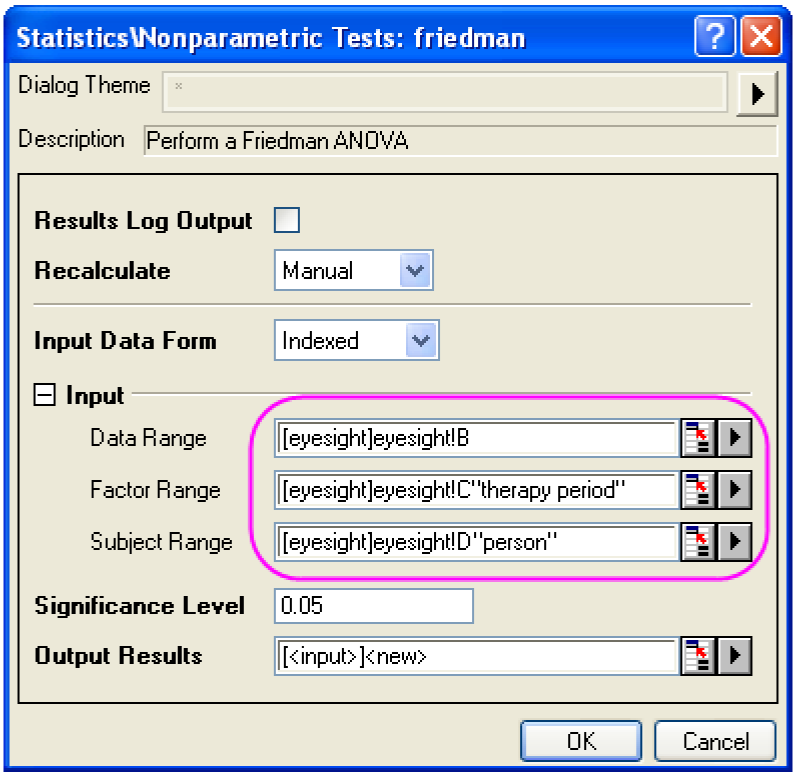
The p-value of χ_{6to10} is 0.0067379, which is less than 0.05. The populations are significantly different, indicating that the therapy are effective for the age group 6-10.

Test Statistics

	Chi-Square	DF	Prob>Chi-Square
	10	2	0.0067379

Null Hypothesis:The samples come from the same population
Alternative Hypothesis:The samples come from different populations
At the 0.05 level, the populations are significantly different

In a similar way, choose column B as **Data Range** and the rest setting of **Input** are the same with Step 3 previously.



Check the result, we can see that p-value of χ_{11to16} is 0.02599, less than 0.05 or 0.10. So we can also conclude that eyesight of 11-16 years old kids is better after 3 period of therapy.

Test Statistics

	Chi-Square	DF	Prob>Chi-Square
	7.3	2	0.025991

Null Hypothesis:The samples come from the same population
Alternative Hypothesis:The samples come from different populations
At the 0.05 level, the populations are significantly different

And we can see that $\chi_{6to10} > \chi_{11to16}$,that means, laser He-Ne therapy works better on 6-10 years old kids. The earlier children are to be involved in therapy, the more their eyesight can be improved.

4.6 Multivariate Analysis

4.6.1 Principal Component Analysis

Summary

Principal Component Analysis is useful for reducing and interpreting large multivariate data sets with underlying linear structures, and for discovering previously unsuspected relationships.

We will start with data measuring protein consumption in twenty-five European countries for nine food groups. Using Principal Component Analysis, we will examine the relationship between protein sources and these European countries.

Selecting Principal Methods

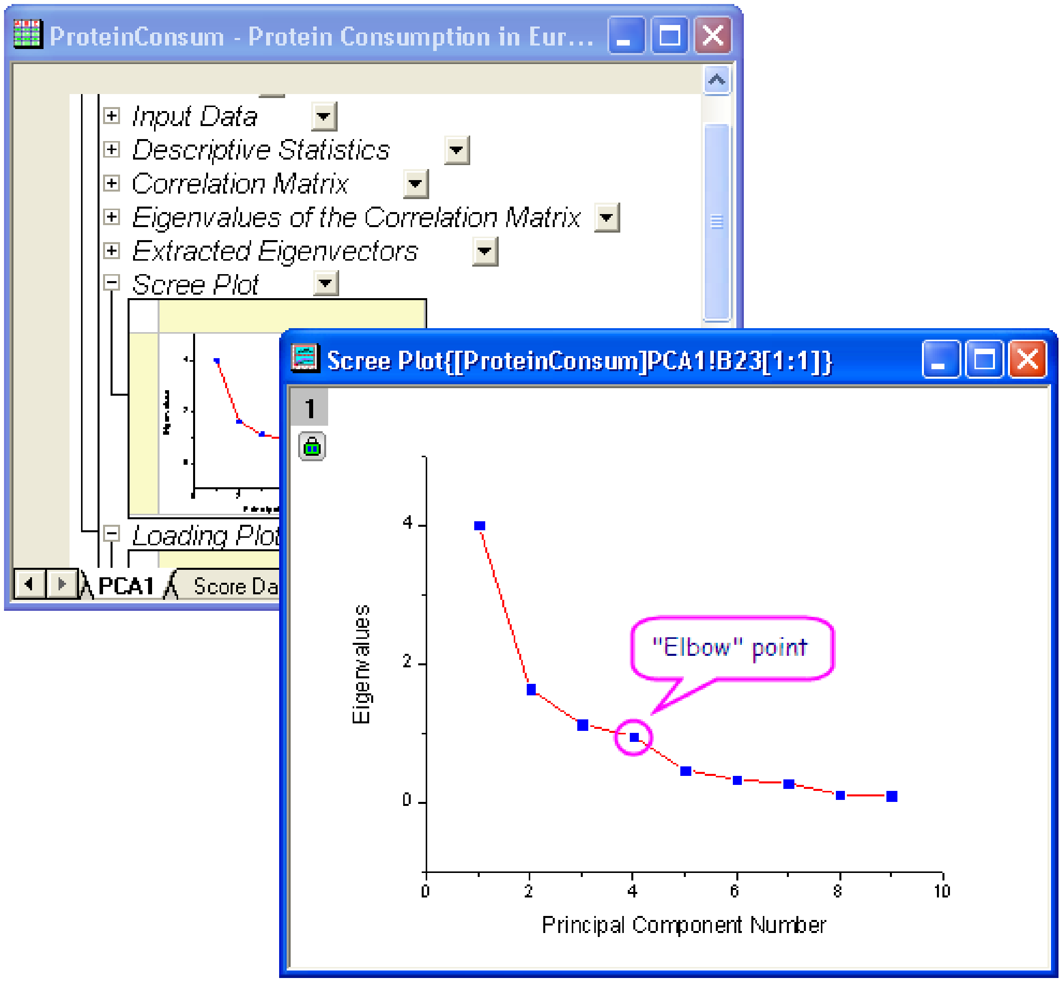
To determine the number of principal components to be retained, we should first run Principal Component Analysis and then proceed based on its result:

1. Open a new project or a new workbook. Import the data file **\samples\Statistics\Protein Consumption in Europe.dat**
2. Select the entire worksheet and then select **Statistics: Multivariate Analysis: Principal Component Analysis**.
3. Accept the default settings in the open dialog box and click **OK**.
4. Select sheet **PCA1**.
5. In the **Eigenvalues of the Correlation Matrix** table, we can see that the first four principal components explain 86% of the variance and the remaining components each contribute 5% or less. We will keep four main components.

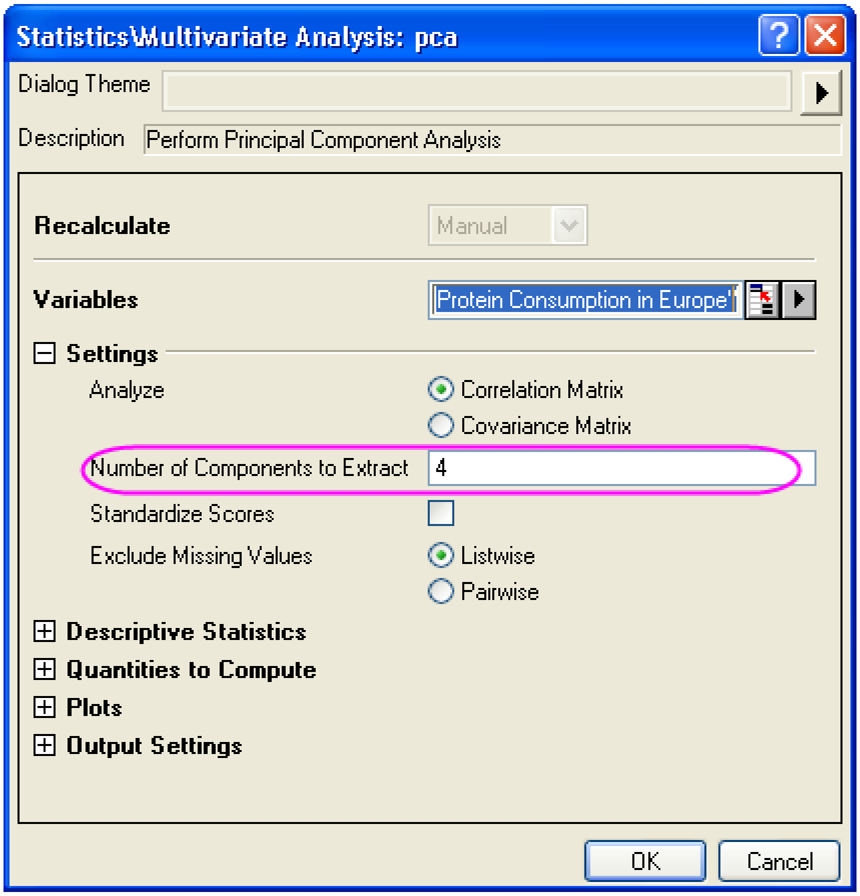
Eigenvalues of the Correlation Matrix

	Eigenvalue	Percentage of Variance	Cumulative
1	4.00644	44.52%	44.52%
2	1.635	18.17%	62.68%
3	1.12792	12.53%	75.22%
4	0.95466	10.61%	85.82%
5	0.46384	5.15%	90.98%
6	0.32513	3.61%	94.59%
7	0.27161	3.02%	97.61%
8	0.11629	1.29%	98.90%
9	0.09911	1.10%	100.00%

6. A scree plot can be a useful visual aid for determining the appropriate number of principal components. The number of components depends on the "elbow" point at which the remaining eigenvalues are relatively small and all about the same size. This point is not very evident in the scree plot, but we can still say the fourth point is our "elbow" point.



7. Click the lock icon in the results tree and select **Change Parameters** in the context menu. Set **Number of Components to Extract** to **4**. Do not close the dialog; in the next steps, we will retrieve component diagrams.



Request Principal Component Plots

In the **Plots** branch of the dialog, users can choose whether they want to create a scree plot or a component diagram.

- **Scree Plot**

  The **scree plot** is a useful visual aid for determining an appropriate number of principal components.

- **Component Plot**

  Component plots show the component score of each observation or component loading of each variable for a pair of principal components. In the **Select Principal Components to Plot** group, users can specify which pair of components to plot. The component plots include:

  - **Loading Plot**

  The **loading plot** is a plot of the relationship between the original variables and the subspace dimension. It is used to interpret relationships between variables.

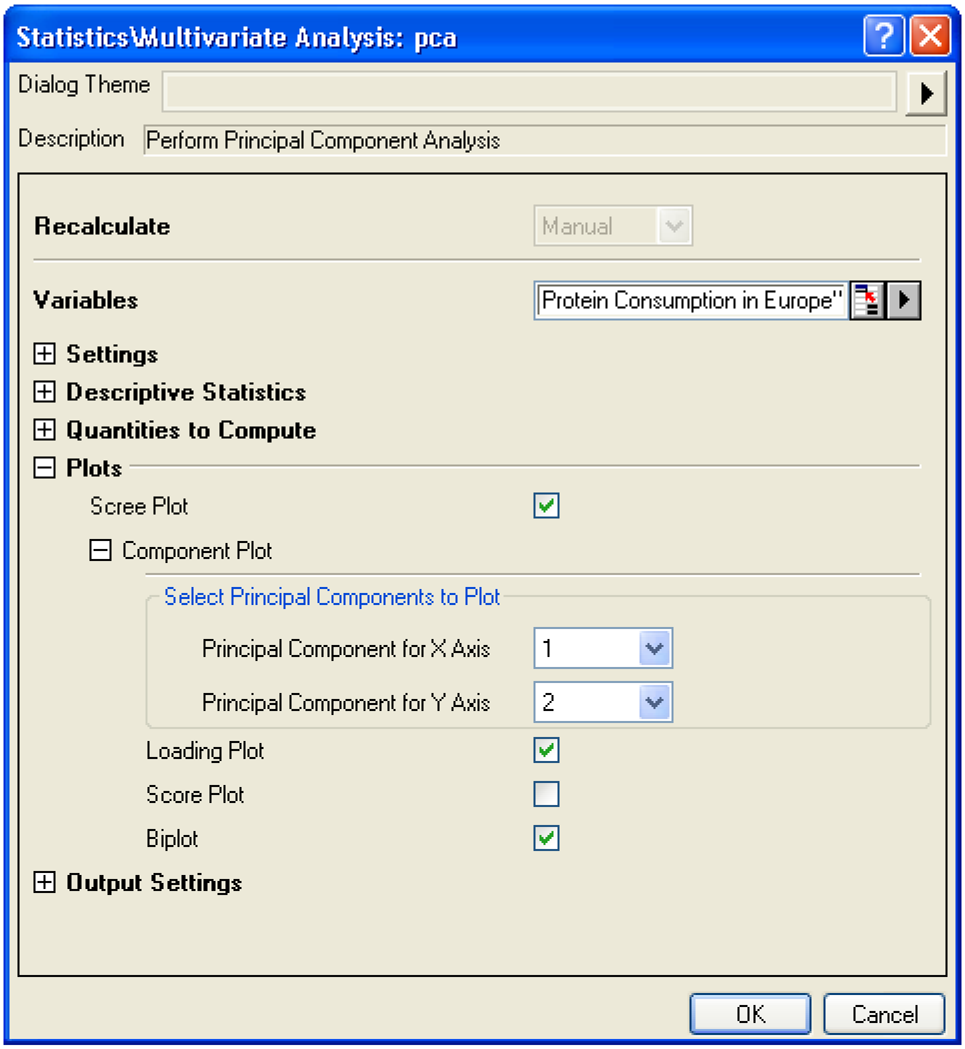
  - **Score Plot**

  The **score plot** is a projection of data onto subspace. It is used to interpret relationships between observations.

  - **BiPlot**

The **biplot** shows both the loadings and the scores for two selected components in parallel.

1. In the dialog that was opened in the preceeding steps, open the **Plots** branch. Make sure **Scree Plot**, **Loading Plot**, and **Biplot** are selected.
2. The first two components are usually responsible for the bulk of the variance. This is why we are going to plot the component plot in the space of the first two principal components. In the **Select Principal Components to Plot** group, set **Principal Component for X Axis** to **1**, and set **Principal Component for Y Axis** to **2**. Click **OK**.



Interpreting The Results

1. In the **Correlation Matrix**, we can see that the variables are highly correlated. Many values are greater than **0.3**. Principal Component Analysis is an appropriate tool for removing the collinearity.

Correlation Matrix

	Red Meat	White Meat	Eggs	Milk	Fish	Cereals	Starch	Nuts	Fruits & Vegetables
Red Meat	1	0.153	0.58561	0.50293	0.06096	-0.49988	0.13543	-0.34945	-0.07422
White Meat	0.153	1	0.62041	0.28148	-0.23401	-0.4138	0.31377	-0.63496	-0.06132
Eggs	0.58561	0.62041	1	0.57553	0.06557	-0.71244	0.45223	-0.55978	-0.04552
Milk	0.50293	0.28148	0.57553	1	0.13788	-0.59274	0.22241	-0.62109	-0.40836
Fish	0.06096	-0.23401	0.06557	0.13788	1	-0.52423	0.40385	-0.14715	0.26614
Cereals	-0.49988	-0.4138	-0.71244	-0.59274	-0.52423	1	-0.53326	0.651	0.04655
Starch	0.13543	0.31377	0.45223	0.22241	0.40385	-0.53326	1	-0.47431	0.08441
Nuts	-0.34945	-0.63496	-0.55978	-0.62109	-0.14715	0.651	-0.47431	1	0.37497
Fruits & Vegetables	-0.07422	-0.06132	-0.04552	-0.40836	0.26614	0.04655	0.08441	0.37497	1

2. The main component variables are defined as linear combinations of the original variables. The **Extracted Eigenvectors** table provides coefficients for equations.

Extracted Eigenvectors

	Coefficients of PC1	Coefficients of PC2	Coefficients of PC3	Coefficients of PC4
Red Meat	0.30261	-0.05625	-0.29758	0.64648
White Meat	0.31056	-0.23685	0.6239	-0.03699
Eggs	0.42668	-0.03534	0.18153	0.31316
Milk	0.37773	-0.18459	-0.38566	-0.00332
Fish	0.13565	0.64682	-0.32127	-0.21596
Cereals	-0.43774	-0.23349	0.09592	-0.0062
Starch	0.29725	0.35283	0.24298	-0.33668
Nuts	-0.42033	0.14331	-0.05439	0.33029
Fruits & Vegetables	-0.11042	0.53619	0.40756	0.46206

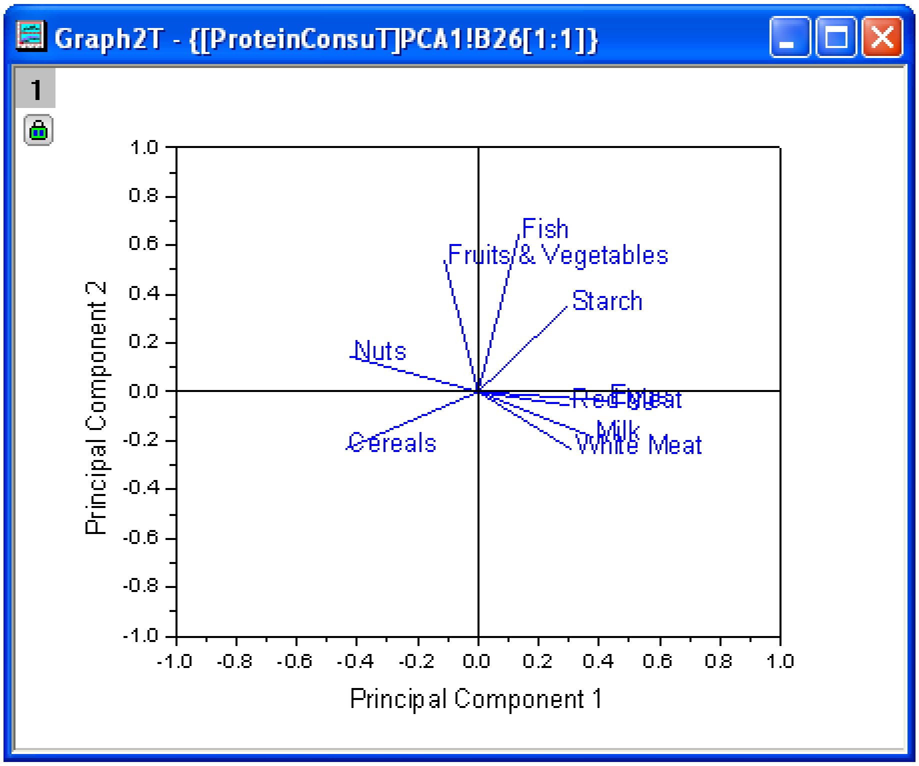
*PC*1 = 0.30261 * *RedMeat* + 0.31056 * *WhiteMeat* + 0.42668 * *Eggs* + 0.37773 * *Milk* + 0.13565 * *Fish* - 0.43774 * *Cereals* + 0.29725 * *Starch* - 0.42033 * *Nuts* - 0.11042 * *FruitsVegetables*

*PC*2 = - 0.05625 * *RedMeat* - 0.23685 * *WhiteMeat* - 0.03534 * *Eggs* - 0.18459*Milk* + 0.64682 * *Fish* - 0.23349 * *Cereals* + 0.35283 * *Starch* + 0.14331 * *Nuts* + 0.53619 * *FruitsVegetables*

*PC*3 = - 0.29758 * *RedMeat* + 0.6239 * *WhiteMeat* + 0.18153 * *Eggs* - 0.38566 * *Milk* - 0.32127 * *Fish* + 0.09592 * *Cereals* + 0.24298 * *Starch* - 0.05439 * *Nuts* + 0.40756 * *FruitsVegetables*

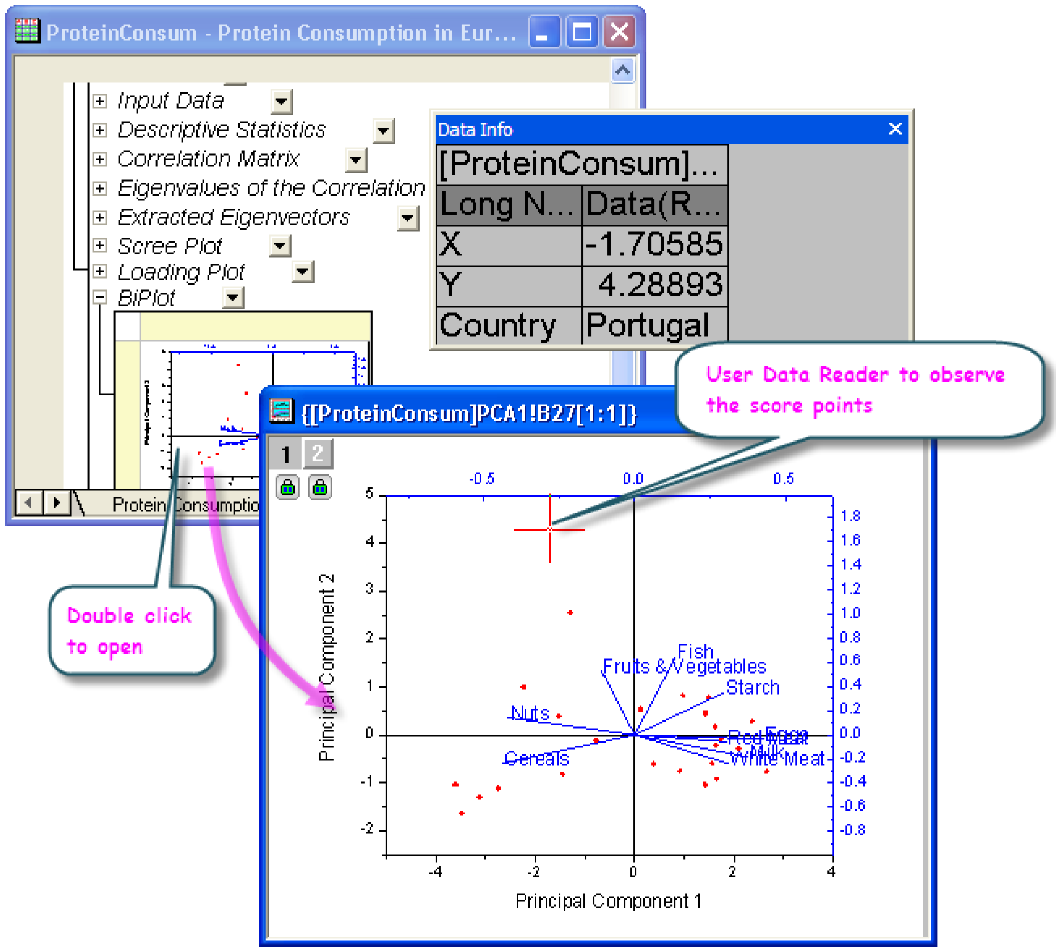
*PC*4 = 0.64648 * *RedMeat* - 0.03699 * *WhiteMeat* + 0.31316*Eggs* - 0.00332 * *Milk* - 0.21596 * *Fish* - 0.0062 * *Cereals* - 0.33668 * *Starch* + 0.33029 * *Nuts* + 0.46206 * *FruitsVegetables*

3. The **Loading Plot** reveals the relationships between variables in the space of the first two components. In the loading plot, we can see that Red Meat, Eggs, Milk, and White Meat have similar heavy loadings for principal component 1. Fish, fruit, and vegetables, however, have similar heavy loadings for principal component 2.



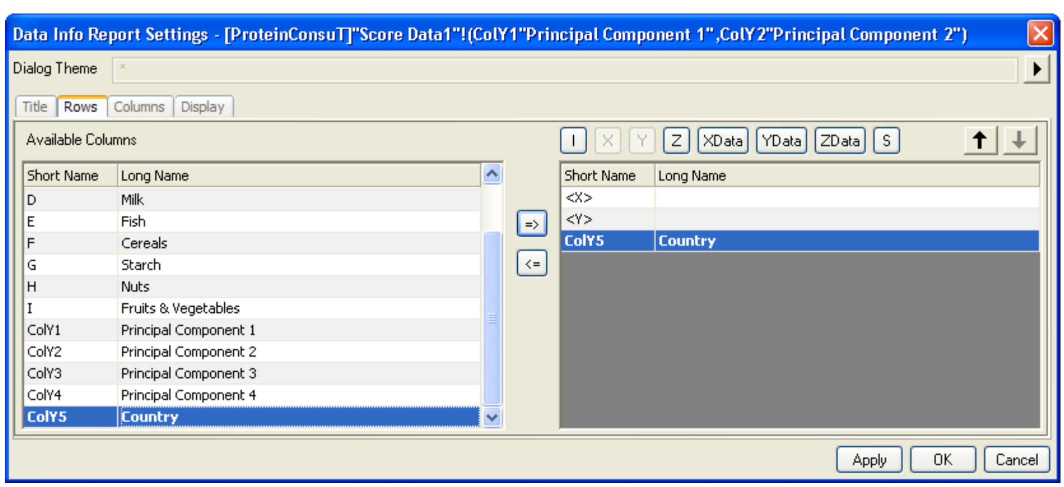
4. The **biplot** shows both the loadings and the score for two selected components in parallel. It can reveal the projection of an observation on the subspace with the score points. It can also find the ratio of observations and variables in the subspace of the first two components. (Note: Double-click the graph to open and customize.)

5. Use the Data Reader tool  to open the Data Info window and examine the plot in greater detail. We can see that Spain and Portugal's protein sources differ from those of other European countries. Spain and Portugal rely on fruits and vegetables, while eastern European countries such as Albania, Bulgaria, Yugoslavia, and Romania prefer cereals and nuts.



To display country information in the **Data Info** window, as in the image above:

1. Right-click the **Data Info** window and select **Preferences...**.
2. In the **Rows** tab, move **Country** from the left panel to the right. Click **OK**.



4.6.2 Cluster Analysis

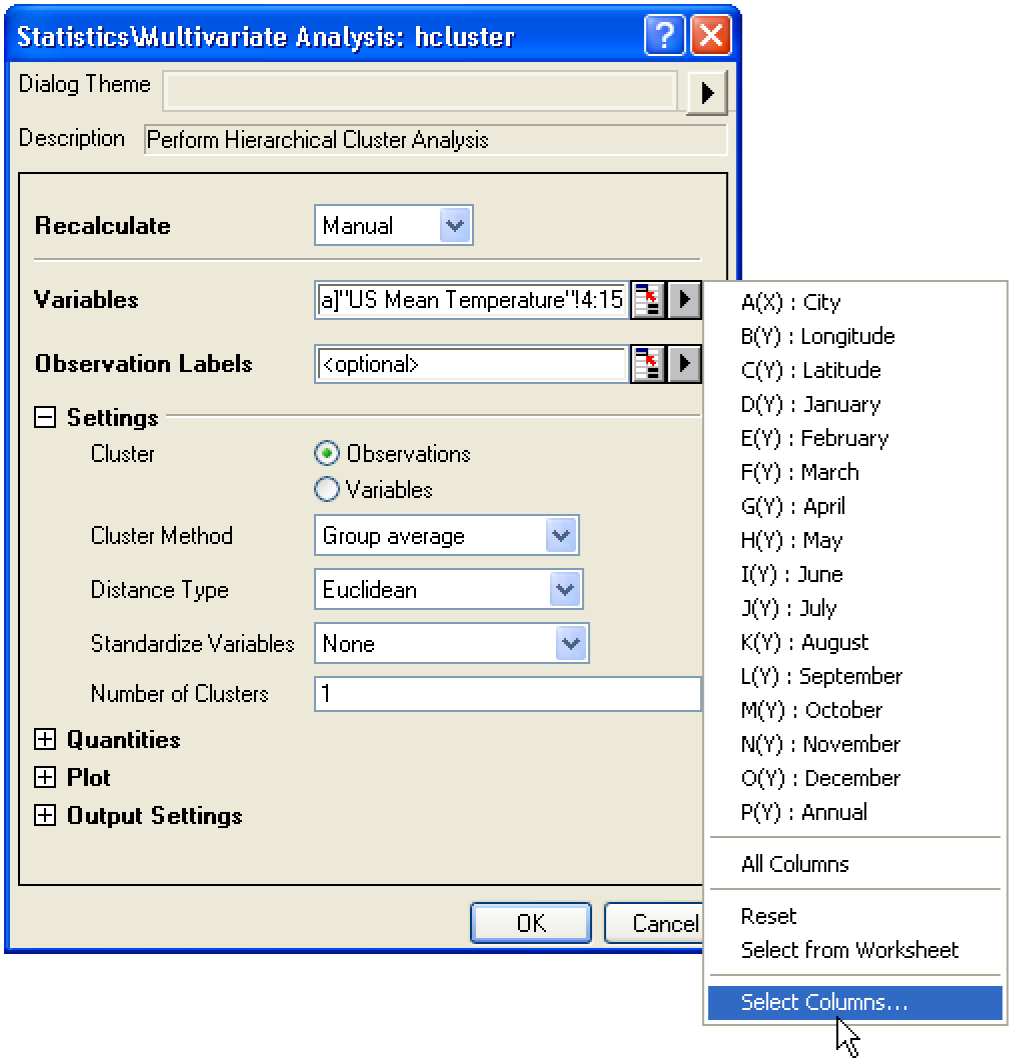
<u>Summary</u>

We will perform cluster analysis for the mean temperatures of US cities over a 3-year-period.

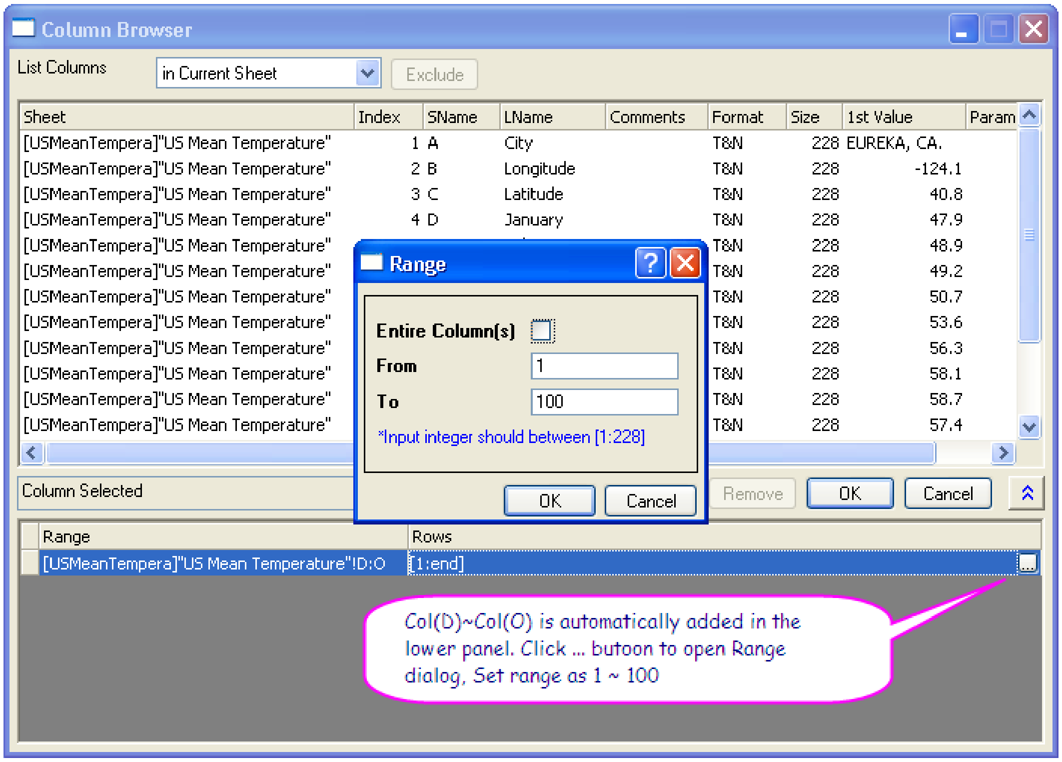
The starting point is a hierarchical cluster analysis with randomly selected data in order to find the best method for clustering. K-means analysis, a quick cluster method, is then performed on the entire original dataset.

<u>Hierarchical Cluster Analysis</u>

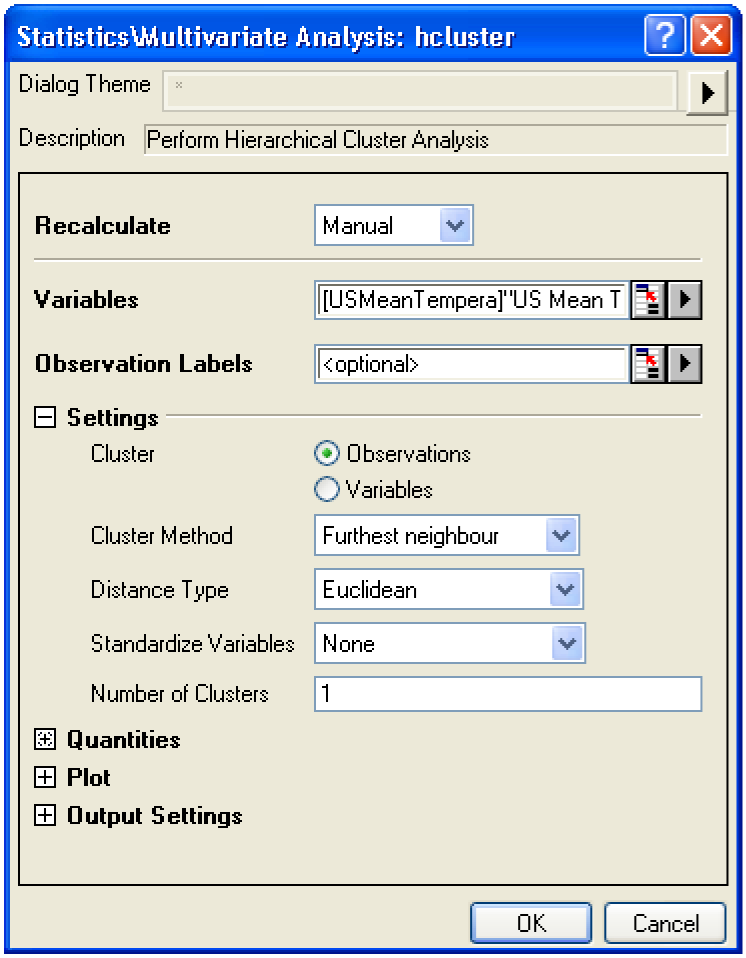
1. Start with a new project or a new workbook. Import the data file **\Samples\Graphing\US Mean Temperature.dat**.
2. Highlight Column D through Column O.
3. Select **Statistics: Multivariate Analysis: Hierarchical Cluster Analysis**.
4. Click the **triangle button** next to **Variables**, and then click **Select Columns...** in the context menu.



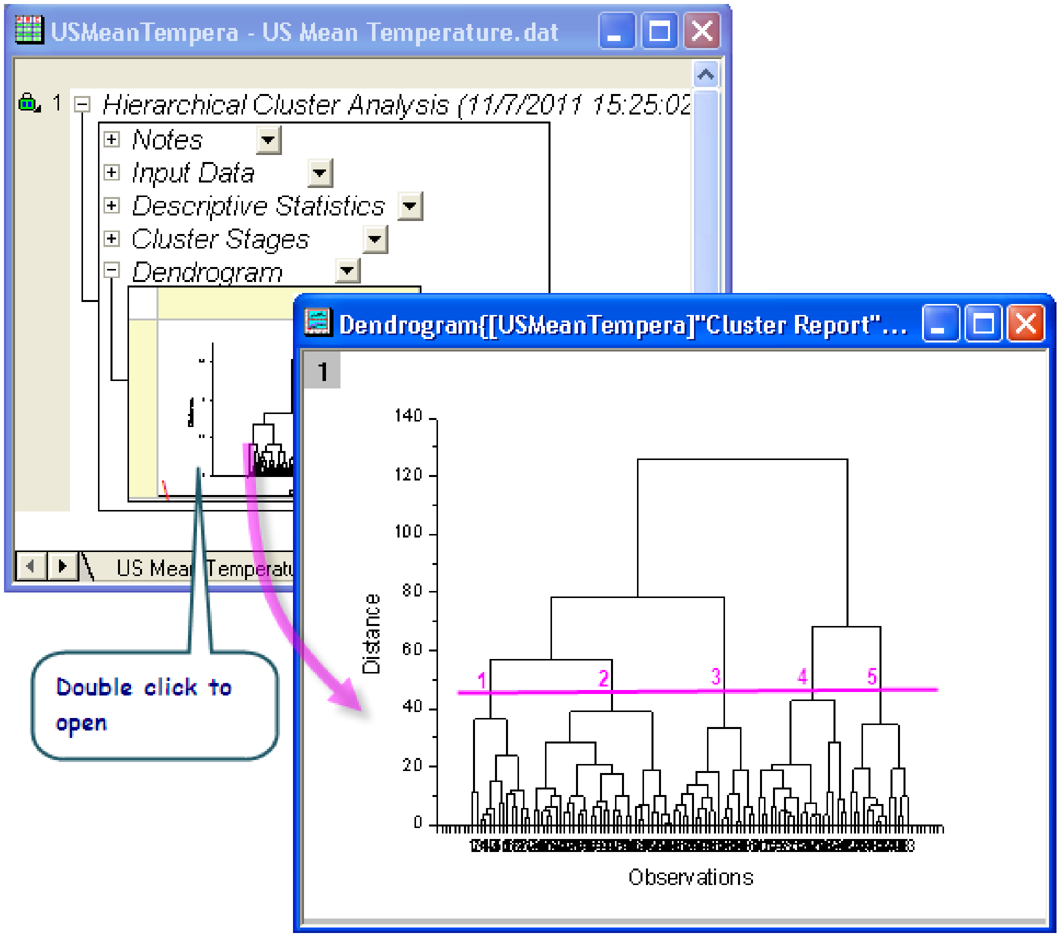
5. In the lower panel of the Column Browser dialog, click the **...** button. Set the data range from **1** to **100**. Click **OK**.



6. In the dialog, make sure **Cluster** is set to **Observations**, and **Number of Clusters** is **1**. Select **Furthest Neighbour** for **Cluster Method** and then click **OK**.

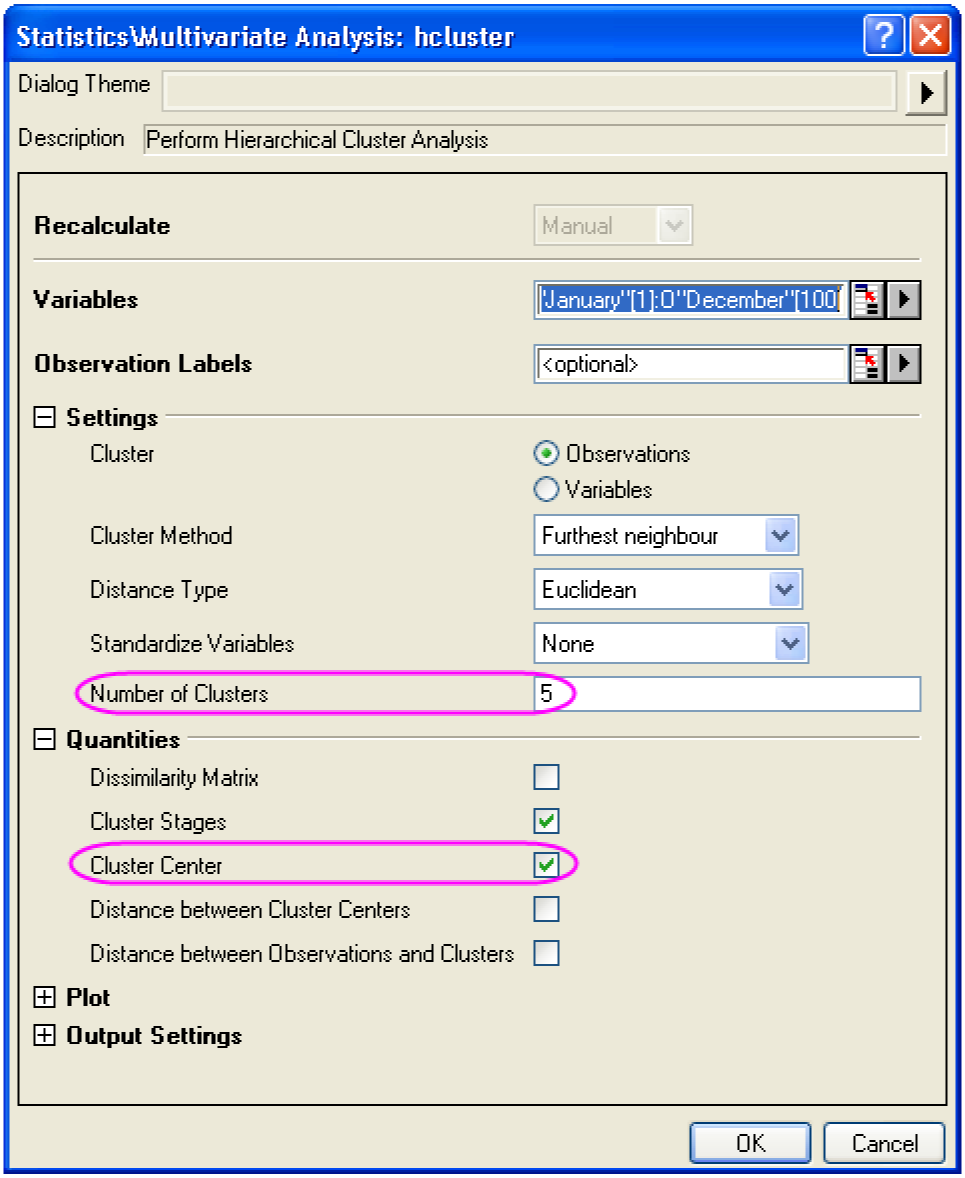


7. Go to the **Cluster 1** sheet. Based on the resulting dendrogram, we choose to cluster data into 5 groups.

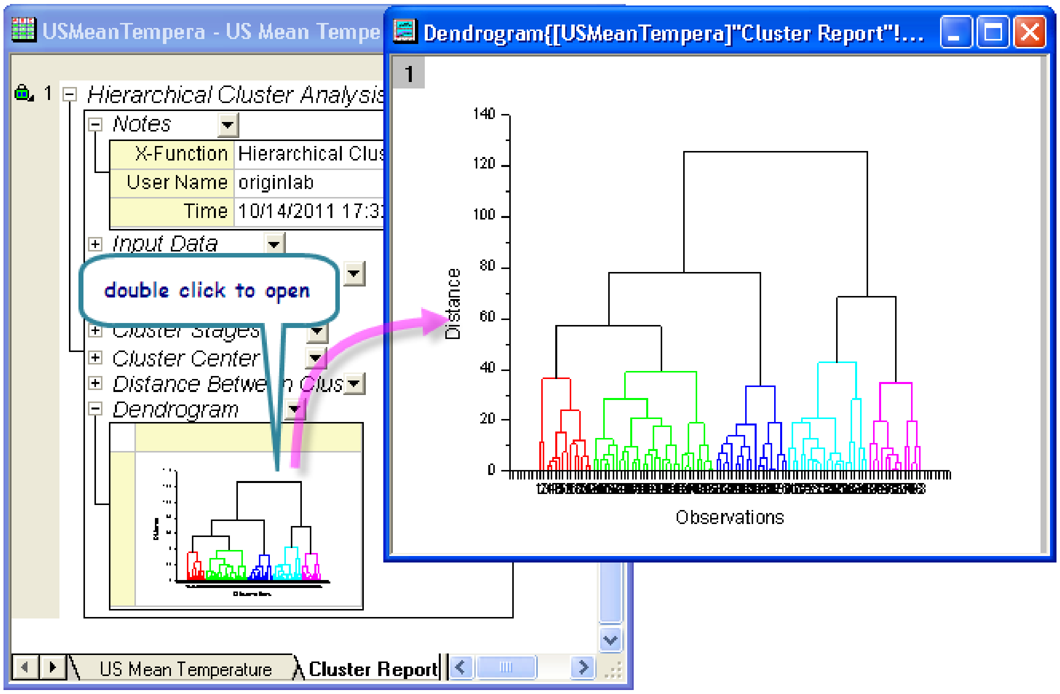


8. Click the lock icon in the dendrogram or the result tree, and then click **Change Parameters** in the context menu.

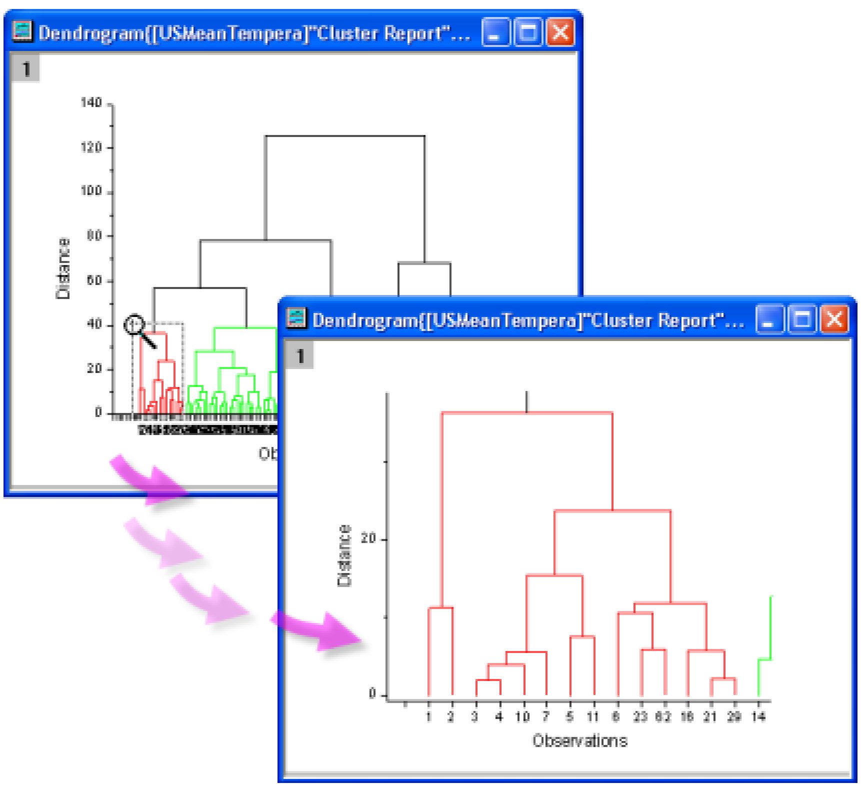
9. Set **Number of Clusters** to **5** and then select the **Cluster Center** check box in the **Quantities** branch. Click **OK**.



10. In the resulting dendrogram, we can clearly see how observations are clustered. (Note, you can double-click to open and customize the dendrogram.)

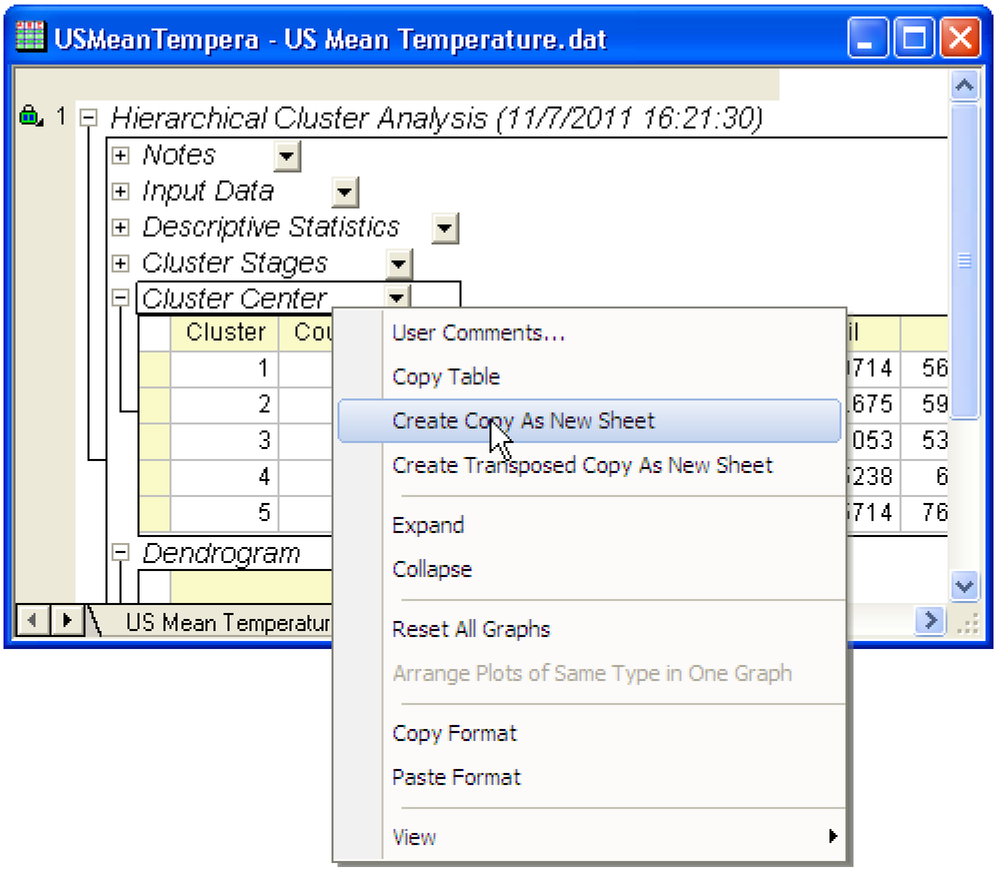


11. Due to the large number of observations, tick labels overlap in this dendrogram. Use the **Scale In**  tool to select an area to magnify.



Analyzing Original Data with K-Means Cluster

1. Right-click on **Cluster Center** and select **Create Copy as New Sheet** in the context menu. We are going to use the newly created **Sheet2** as the **Initial Cluster Centers** in our k-means cluster analysis.



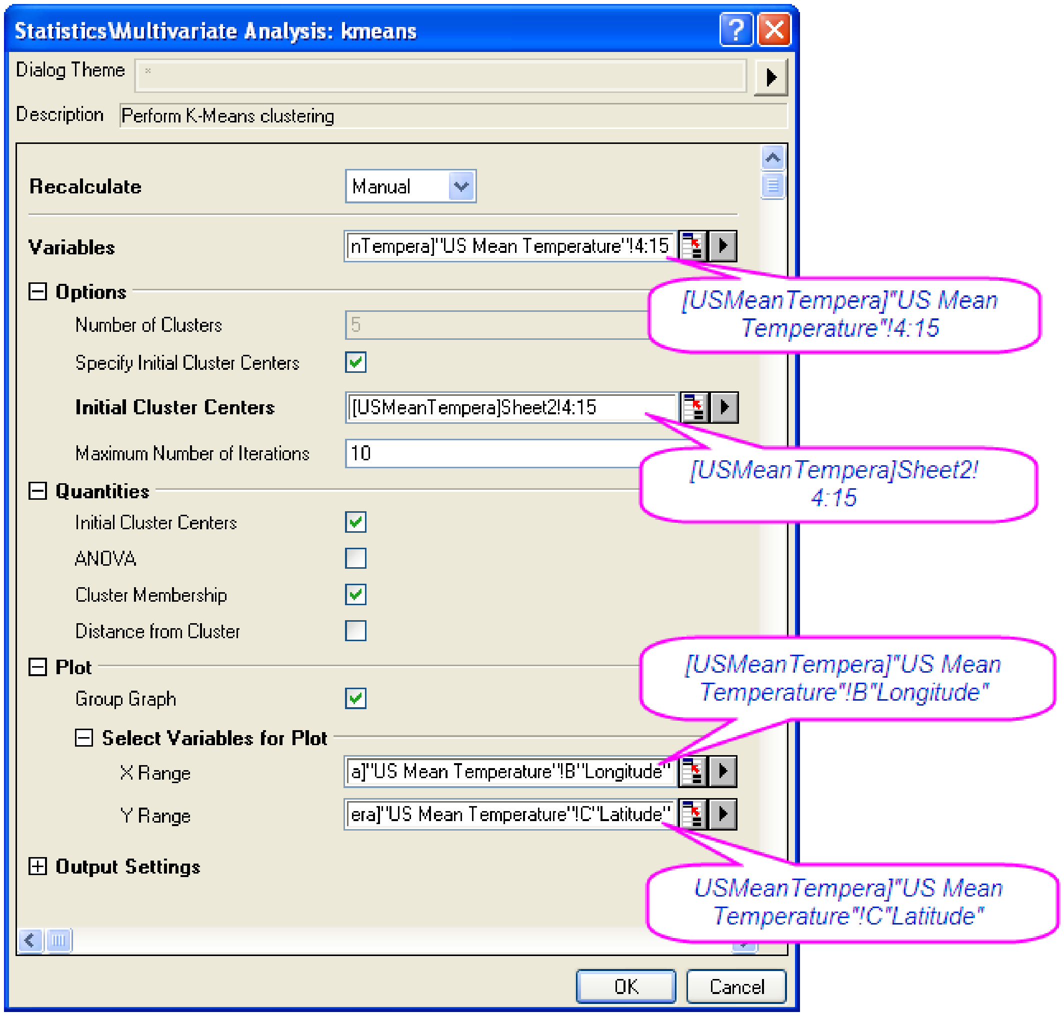
2. Go back to the worksheet with the source data (US Mean Temperature), and highlight col(D) through col(O). Select **Statistics: Multivariate Analysis: K-Means Cluster Analysis**.

3. Select the **Specify Initial Cluster Centers** check box. Click the **interactive button** next to **Initial Cluster Centers**. The dialog will "roll up".

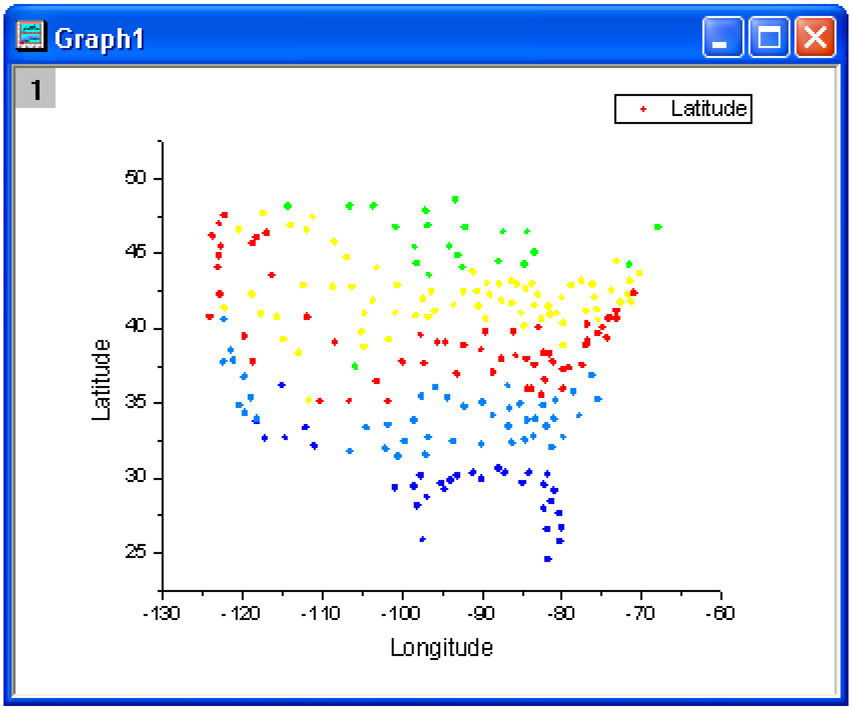
4. Go to **Sheet 2** and hightlight Col(D) through Col(O). Click the button on the rolled-up dialog to restore the dialog.

5. In the **Plot** branch, select **Group Graph**. Click the **interactive button** next to **X Range**. The dialog will "roll up". Go back to the source worksheet **US Mean Temperature**, and highlight **Col(B):Longtitude**. Click the button in the rolled up dialog to restore.

6. Click the **triangle button** next to **Y Range**, and then select **C(Y), Latitude**. Click **OK**.



7. Activate the worksheet **K-Means1**. Observe that data has been clustered into 5 groups corresponding to the latitudes of the cities.



4.6.3 Discriminant Analysis

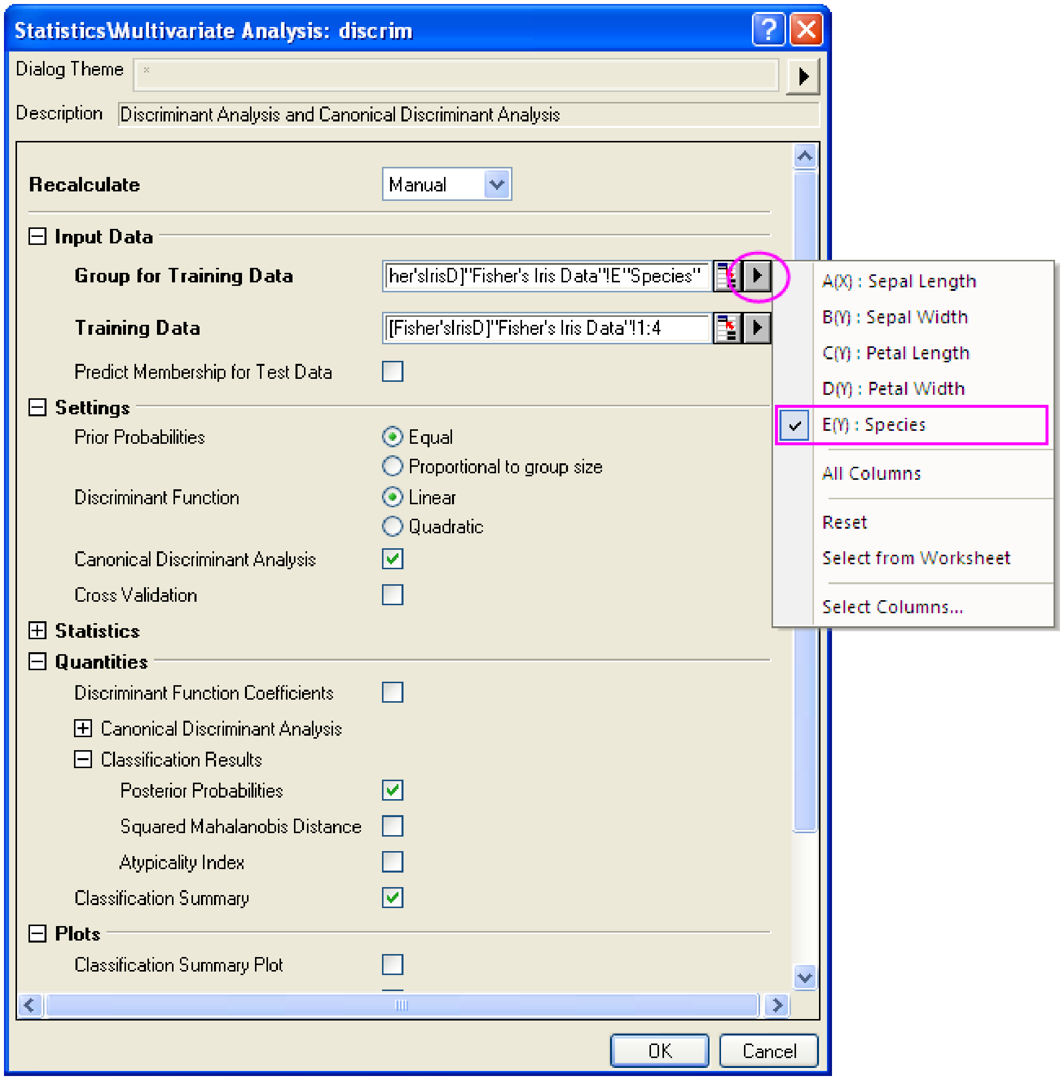
Summary

The Iris flower data set, or Fisher's Iris dataset, is a multivariate dataset introduced by Sir Ronald Aylmer Fisher in 1936. This dataset is often used for illustrative purposes in many classification systems. The dataset consists of fifty samples from each of three species of Irises (iris setosa, iris virginica, and iris versicolor). Four characteristics, the length and width of sepal and petal, are measured in centimeters for each sample. We can use discriminant analysis to identify the species based on these four characteristics.

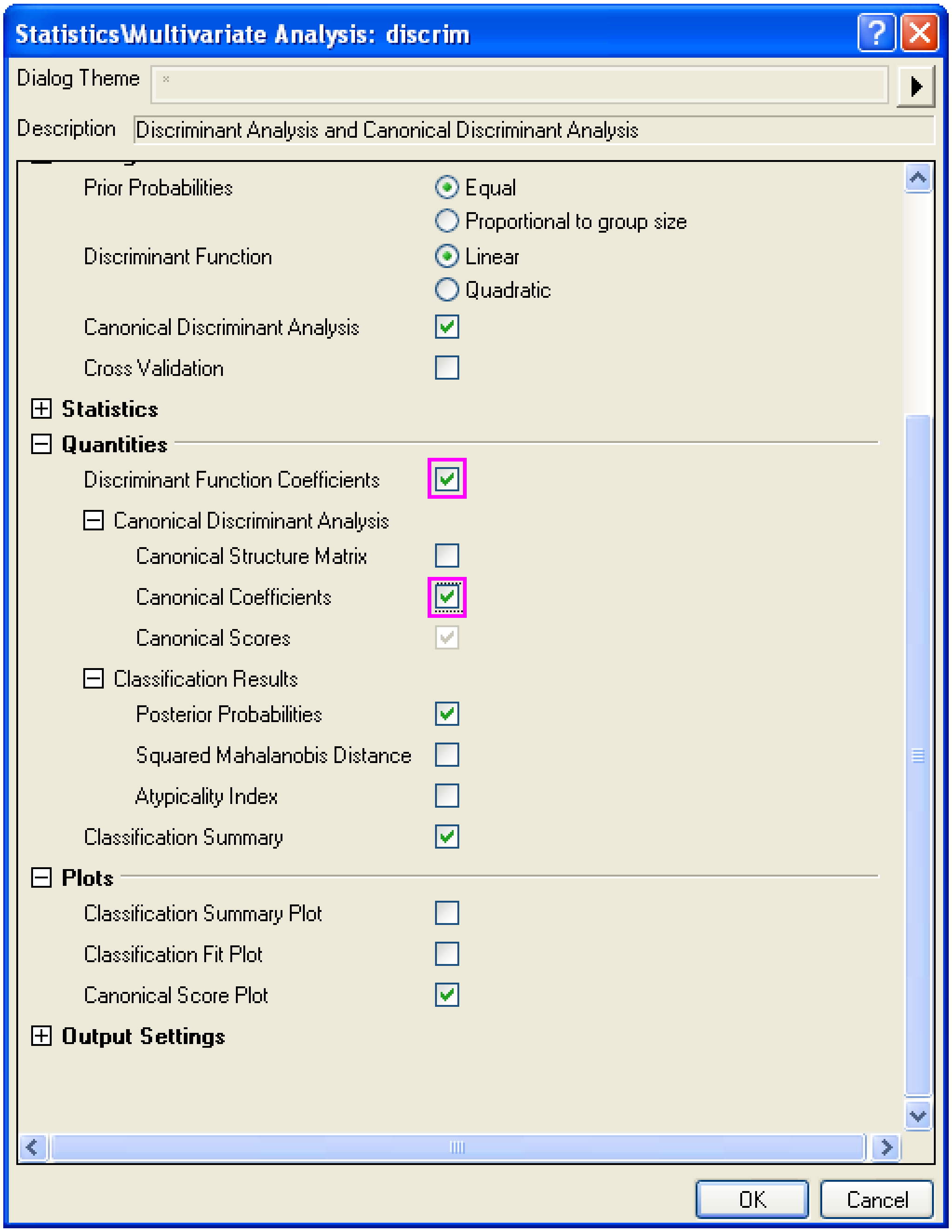
We will use a random sample of 120 rows of data to create a discriminant analysis model, and then use the remaining 30 rows to verify the accuracy of the model.

Discriminant Analysis

1. Open a new project or a new workbook. Import the data file **\Samples\Statistics\Fisher's Iris Data.dat**
2. Highlight columns A through D. and then select **Statistics: Multivariate Analysis: Discriminant Analysis** to open the **Discriminant Analysis** dialog. **Column A ~ D** are automatically added as **Training Data**. Click the **triangle button** next to **Group for Training Data** and select **E(Y):Species** in the context menu



3. Open **Quantities** branch , and then select **Discriminant Function Coefficients** check box. Select **Canonical Coefficients** check box under **Canonical Discriminant Analysis** branch. Accept all other default settings and click **OK**



Interpreting Results

Go to sheet **Discrim1**

Canonical Discriminant Analysis

The **Canonical Discriminant Analysis** branch is used to create the discriminant functions for the model.

1. Using the **Unstandardized Canonical Coefficient** table we can construct the canonical discriminant functions.

Unstandardized Canonical Coefficients

	Canonical Variable 1	Canonical Variable 2
Constant	-2.10511	-6.66147
Sepal Length	-0.82938	0.0241
Sepal Width	-1.53447	2.16452
Petal Length	2.20121	-0.93192
Petal Width	2.81046	2.83919

*D*1 = - 2.10511 - 0.82938 * *SL* - 1.53447 * *SW* + 2.20121 * *PL* + 2.81046 * *PW*

*D*2 = - 6.66147 + 0.0241 * *SL* + 2.16452 * *SW* - 0.93192 * *PL* + 2.83919 * *PW*

where SL = Sepal Length, SW = Sepal Width, PL = Petal Length, PW = Petal Width

2. The Eigenvalues table reveals the importance of the above canonical discriminant functions. The first function can explain 99.12% of the variance, and the second can explain the remaining 0.88%.

Eigenvalues

	Eigenvalue	Percentage of Variance	Cumulative	Canonical Correlation
1	32.19193	99.12%	99.12%	0.98482
2	0.28539	0.88%	100.00%	0.4712

3. The Wilk's Lambda Test table shows that the discriminant functions significantly explain the membership of the group. We can see that both values in the **Sig** column are smaller than 0.05. Both values should therefore be included in the discriminant analysis.

Wilks' Lambda Test

	Wilks' Lambda	Chi-square	df	Sig.
1 to 2	0.02344	546.1153	8	8.87078E-113
2 to 2	0.77797	36.52966	3	5.78605E-8

At the 0.05 level, the dimensionality is significantly 2.

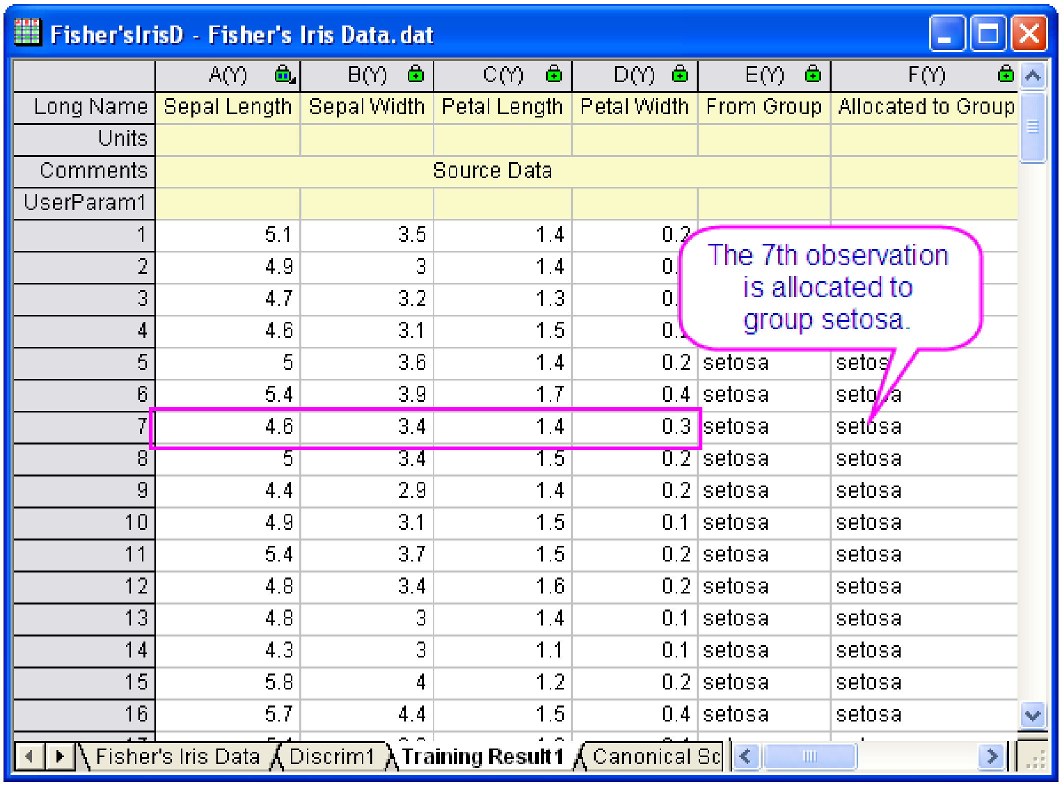
Classification

1. In order to classify observations, the score of the observations from the coefficients of the linear discriminant function is calculated and then evaluated.

Coefficients of Linear Discriminant Function

	setosa	versicolor	virginica
Constant	-86.30847	-72.85261	-104.36832
Sepal Length	23.54417	15.69821	12.44585
Sepal Width	23.58787	7.07251	3.68528
Petal Length	-16.43064	5.21145	12.76654
Petal Width	-17.39841	6.43423	21.07911

2. Switch to the worksheet **Training Result**. For the seventh observation, we can compute the score of each group from the **Coefficient of Linear Discriminant Function** table (above).



	A(Y)	B(Y)	C(Y)	D(Y)	E(Y)	F(Y)
Long Name	Sepal Length	Sepal Width	Petal Length	Petal Width	From Group	Allocated to Group
Units						
Comments			Source Data			
UserParam1						
1	5.1	3.5	1.4	0.2		
2	4.9	3	1.4	0.		
3	4.7	3.2	1.3	0.		
4	4.6	3.1	1.5	0.		
5	5	3.6	1.4	0.2	setosa	setos
6	5.4	3.9	1.7	0.4	setosa	seto a
7	4.6	3.4	1.4	0.3	setosa	setosa
8	5	3.4	1.5	0.2	setosa	setosa
9	4.4	2.9	1.4	0.2	setosa	setosa
10	4.9	3.1	1.5	0.1	setosa	setosa
11	5.4	3.7	1.5	0.2	setosa	setosa
12	4.8	3.4	1.6	0.2	setosa	setosa
13	4.8	3	1.4	0.1	setosa	setosa
14	4.3	3	1.1	0.1	setosa	setosa
15	5.8	4	1.2	0.2	setosa	setosa
16	5.7	4.4	1.5	0.4	setosa	setosa

$Score(setosa)$ = - 86.30847 + 23.54417 * 4.6 + 23.58787 * 3.4 - 16.43064 * 1.4 - 17.39841 * 0.3 = 73.971051

$Score(versicolor)$ = - 72.85261 + 15.69821 * 4.6 + 7.07251 * 3.4 + 5.21145 * 1.4 + 6.43423 * 0.3 = 32.631989

$Score(virginica)$ = - 104.36832 + 12.44585 * 4.6 + 3.68528 * 3.4 + 12.76654 * 1.4 + 21.07911 * 0.3 = - 10.390569

3. We can see that the score (setosa) 73.971051 is the maximum value, i.e., the seventh observation should be assigned to the group **setosa**.

4. The **Classification Summary for Training Data** shows that the classification in the groups **setosa** is 100% correct. For **versicolor**, only two observations are mistakenly classified as virginica, and for **virginica**, only one is mistakenly classified. The error rate is only 2.00%. This model is good.

Classification Summary for Training Data

Classification Count

	Predicted Group			
	setosa	versicolor	virginica	Total
setosa	50	0	0	50
	100.00%	0.00%	0.00%	100.00%
versicolor	0	48	2	50
	0.00%	96.00%	4.00%	100.00%
virginica	0	1	49	50
	0.00%	2.00%	98.00%	100.00%
Total	50	49	51	150
	33.33%	32.67%	34.00%	100.00%

Error Rate

	setosa	versicolor	virginica	Total
Prior	0.33333	0.33333	0.33333	
Rate	0.00%	4.00%	2.00%	2.00%

Error rate for classification of training data is 2.00%.

Model Validation

The **Classification Summary of Training Data** evaluate the observation via discriminant functions derived from the same data. But usually "error rate" is larger when user evaluate the test data, which are not used for discriminant function estimation. There are two methods to correct this.

- **Cross-validation**:

  In cross-validation, each training data is treated as the test data, exclude it from training data to judge which group it should be classified as, and then verify whether the classification is correct or not.

- **Subset Validation**:

  Usually we will randomly divide the set of observations into subsets, the first of which is used for the estimation of discriminant model (training set) and the second is for testing the reliability of the results (test set).

Preparing Data for Analysis

We are going to sort the data in random order, and then use the first 120 rows of data as training data and the last 30 as test data.

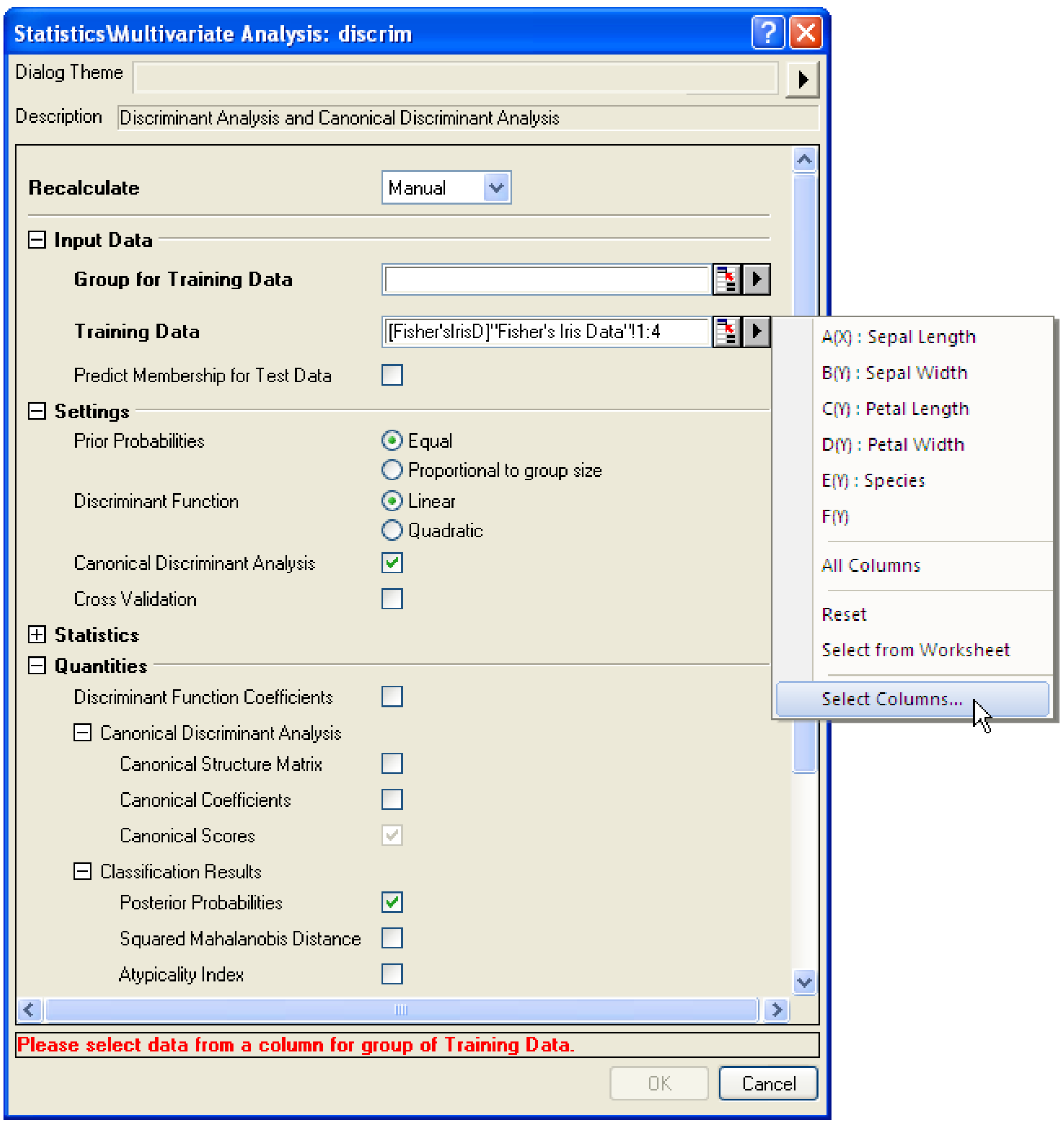
1. Go back to sheet **Fisher's Iris Data**
2. Add a new column and fill the column with **Normal Random Numbers**.
3. Select the newly added column. Right-click and select **Sort Worksheet: Ascending** from the shortcut menu.

Notes: Origin will generate different random data each time, and different data will result in different results.

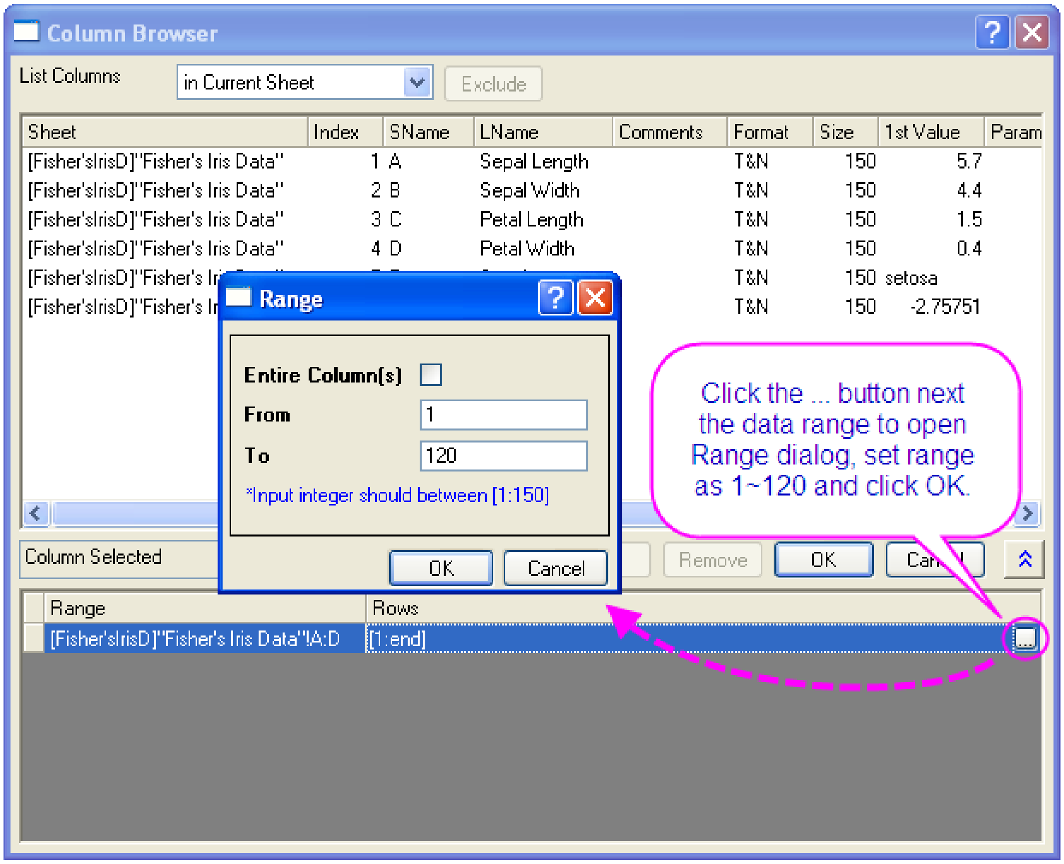
In order to get the same results as shown in this tutorial, you could open the *Analysis.opj* under the **Samples** folder, browse in the **Project Explorer** and navigate to the **Discriminant Analysis** subfolder under the **Analysis-Origin Pro** folder, then use the data from column (F) in the **Fisher's Iris Data** worksheet, which is a previously generated dataset of random numbers.

Run Discriminant Analysis

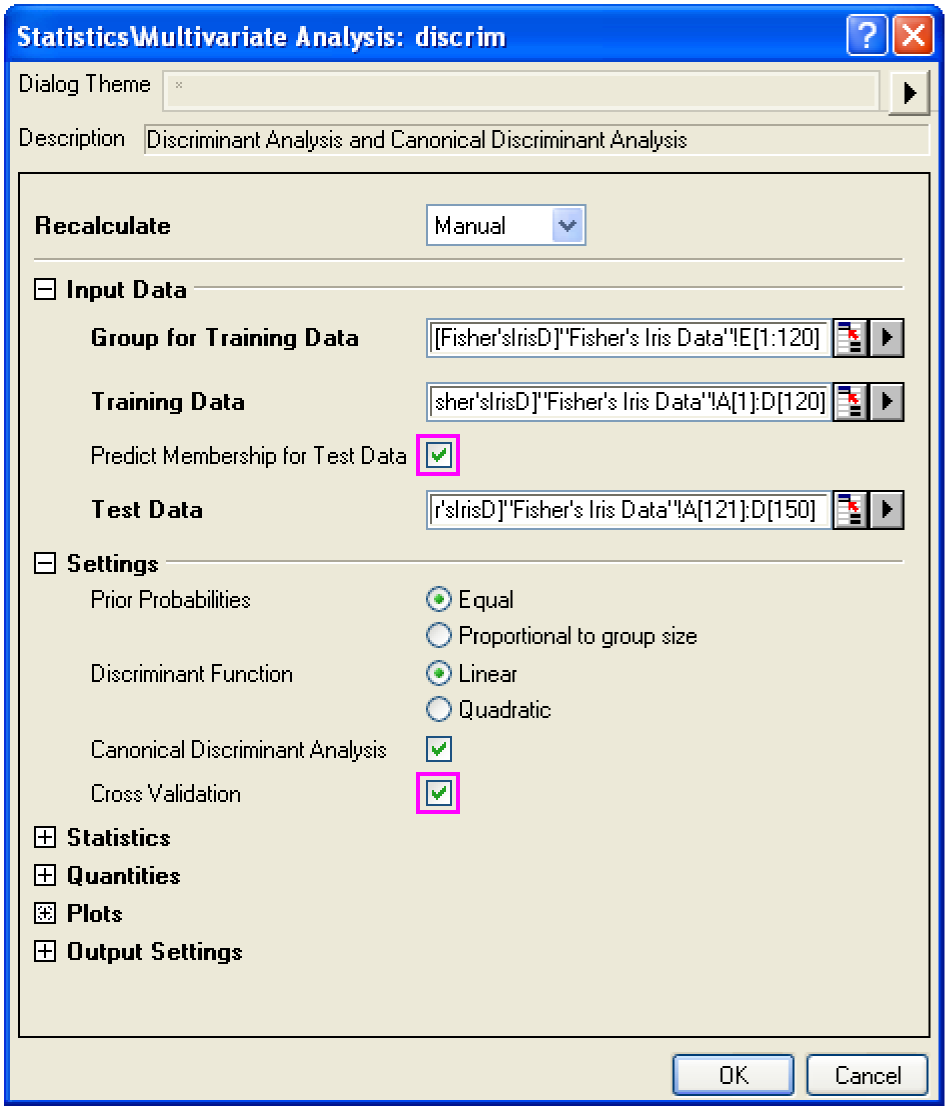
1. Select columns A through D.
2. Select **Statistics: Multivariate Analysis: Discriminant Analysis** to open the **Discriminant Analysis** dialog.
3. To set the first 120 rows of columns A through D as **Training Data**, click the **triangle button** next to **Training Data**, and then select **Select Columns** in the context menu.



4. In the **Column Browser** dialog, click the **...** button in the lower panel. Set data range from **1** to **120**. Click **OK**.



5. To set first 120 rows of Col(E) as **Group for Training Data**, click the **triangle button** next to **Group for Training Data** and select **E(Y): Species** in the context menu. Then click the Group for Training Data **triangle button** again, select **Select Columns** in the context menu, and set range from **1** to **120** with column browser. Click **OK**.
6. Select **Predict Membership of Test Data** check box. Click the Test Data **interactive button** . The dialog will "roll up". Select columns A through D in the worksheet. Click the button in the rolled up dialog to restore the dialog. Then click the **triangle button** to open **Column Browser** by selecting **Select Columns** in the context menu. Click **...** button in lower panel, and set range from **121** through **150**.
7. Open **Settings** branch , and then select **Cross Validation** check box. Click **OK**.



Cross-validation

Go to sheet **Discrim2**. **Cross-validation Summary for Training Data** table provides prediction error rate by classifying each case while leave it out from the model calculations. However, this method is still "optimistic" than subset validation.

Cross-validation Summary for Training Data

Classification Count

	Predicted Group			
	setosa	virginica	versicolor	Total
setosa	44	0	0	44
	100.00%	0.00%	0.00%	100.00%
virginica	0	36	2	38
	0.00%	94.74%	5.26%	100.00%
versicolor	0	2	36	38
	0.00%	5.26%	94.74%	100.00%
Total	44	38	38	120
	36.67%	31.67%	31.67%	100.00%

Error Rate

	setosa	virginica	versicolor	Total
Prior	0.33333	0.33333	0.33333	
Rate	0.00%	5.26%	5.26%	3.51%

Error rate for Cross-validation of training data is 3.51%.

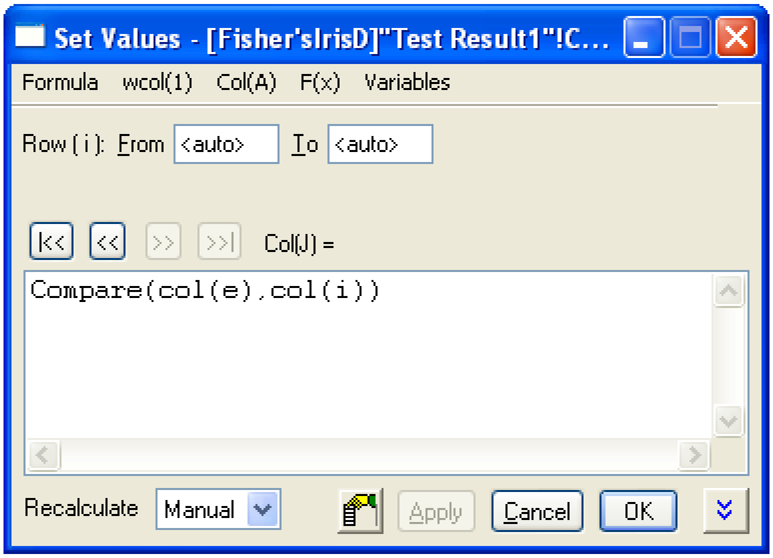
Subset Validation

1. The **Classification Summary for Test Data** provide information that how the test data are classified.

Classification Summary for Test Data

	setosa	virginica	versicolor	Total
Count	6	12	12	30
Percent	20.00%	40.00%	40.00%	100.00%

2. On the worksheet **Fisher's Iris Data**, copy the last 30 rows (121 through 150) of **Col(E): Species**.
3. On the worksheet **Test Result**, add one column, **Col(I)**, to the worksheet. Paste the copied values in the new column.
4. Add a new column, **Col(J)** to the worksheet, right click on it and select Set Column Values in the context menu. In the opened dialog, type **Compare(col(e),col(i))** in the pop-up dialog and click OK.



5. None of 30 values is 0, it means the error rate the testing data is 0. Our discriminant model is pretty good.

Adjusting Prior Probabilities

Discriminant analysis assumes that prior probabilities of group membership are identifiable. If group population size is unequal, prior probabilities may differ. We can use **Proportional to group size** for the **Prior Probabilities** option in this case.

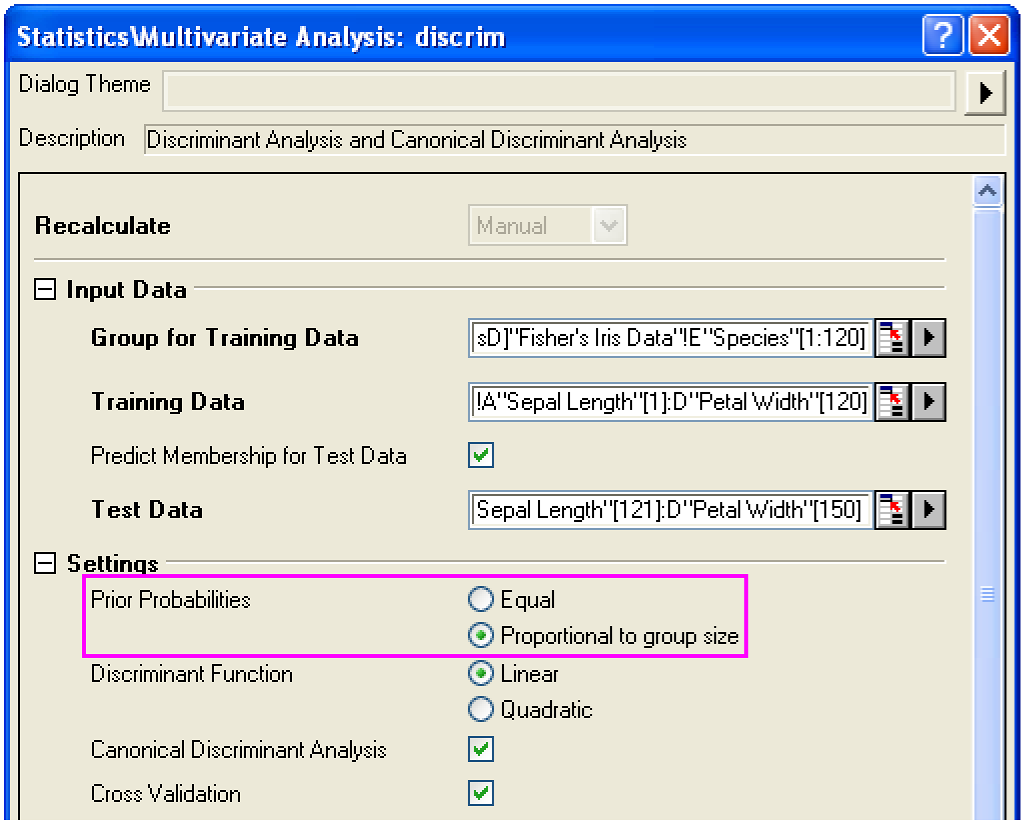
1. Go to sheet **Discrim2**, **Prior** row of the **Error Rate** table under **Classification Summary for Training Data** branch indicate the prior probabilities for membership in groups. It is assumed that a case is equally likely to be one of the three groups. Adjusting the prior probabilities according to the group size can improve the overall classification rate.

Error Rate

	setosa	virginica	versicolor	Total
Prior	0.33333	0.33333	0.33333	
Rate	0.00%	2.63%	5.26%	2.63%

Error rate for classification of training data is 2.63%.

2. Click on the button and select **Change Parameter** from the context menu. Select **Proportional to group size** for **Prior Probabilities** radio box. Click **OK** button.



3. We can see the classification error rate is 2.50%, it is better than 2.63%, error rate with equal prior probabilities.

Prior Probabilities = Poportional to group

Error Rate

	setosa	virginica	versicolor	Total
Prior	0.36667	0.31667	0.31667	
Rate	0.00%	2.63%	5.26%	2.50%

Prior Probabilities = Equal

Error Rate

	setosa	virginica	versicolor	Total
Prior	0.33333	0.33333	0.33333	
Rate	0.00%	2.63%	5.26%	2.63%

5 Graphing

Topics covered in this section:

5.1 Graphing Data From Multiple Sheets

5.1.1 Summary

Origin provides close to 150 built-in graph templates that can be used to create a wide variety of plots. You can modify these templates or create your own to add to the collection. Creating a graph in Origin is as simple as selecting the desired data and then selecting a template from a menu or from the Graphing toolbars. The Plot Setup dialog offers more flexibility in creating plots, such as plotting data from multiple books or sheets.

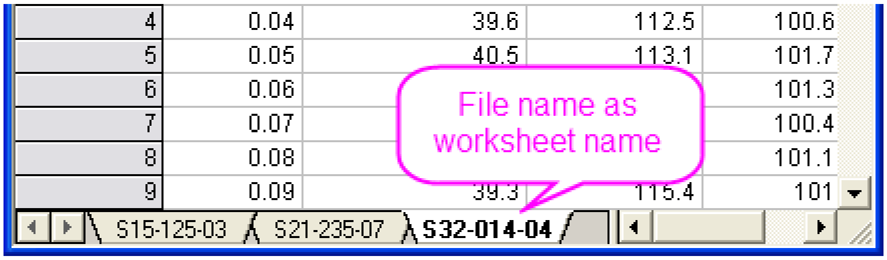
This tutorial will show you how to:

- Select data in a worksheet and quickly create a plot
- Add data to an existing graph with drag-and-drop plotting
- Use Plot Setup to plot data from multiple sheets
- Create and save a custom graph template
- Plot groups of plots by label

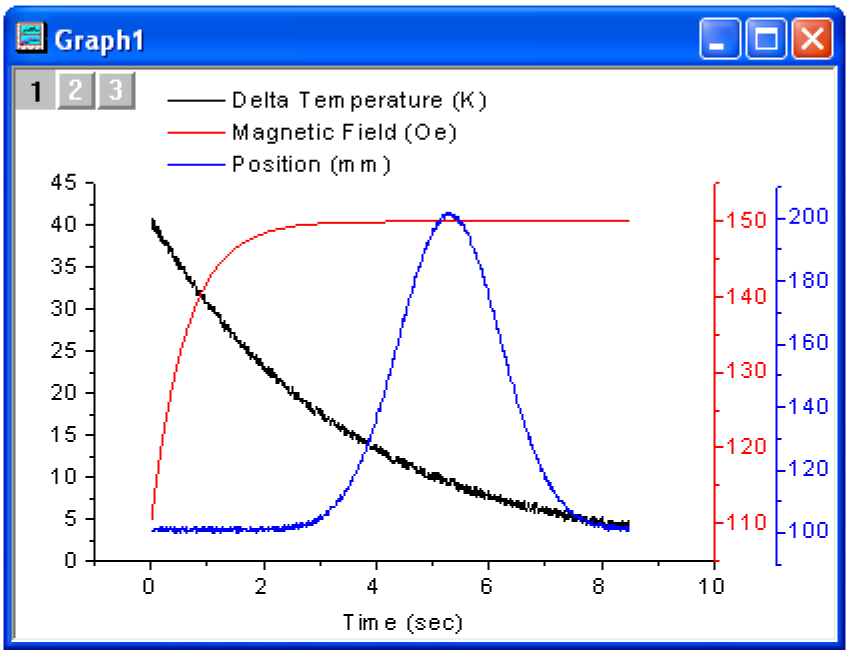
5.1.2 Create a Plot Quickly by Selecting Data

1. Select **File: Recent Imports: impASC: My Multifile Import** from the main menu. (**My Multifile Import** is a theme saved in another tutorial. If you have never saved it, please perform the steps in the tutorial to create it.)

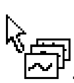
2. Import the files *S15-125-03.dat*, *S21-235-07.dat* and *S32-014-04.dat* from **\Samples\Import and Export**. Note that the settings in the theme are used to import the files, so Origin will import each file to a different worksheet of the same workbook. The file names are used as the worksheet names.

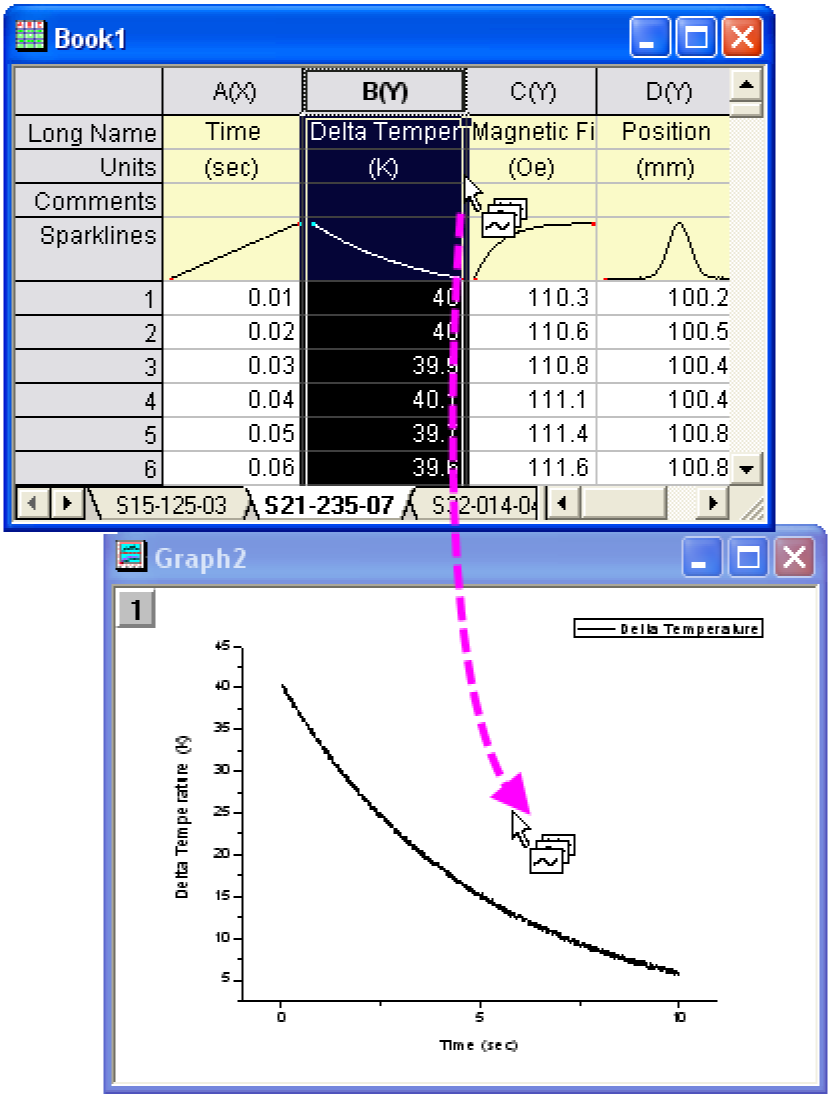


3. Make the third worksheet, S32-014-4, active. To create a three layer graph, highlight the three Y columns, Delta Temperature, Magnetic Field, and Position, and then select **Plot: Multi-Curve: 3Ys Y-YY**. Note: There is no need to highlight the Time column, as Origin will automatically plot the Y columns against the associated X column in the worksheet.

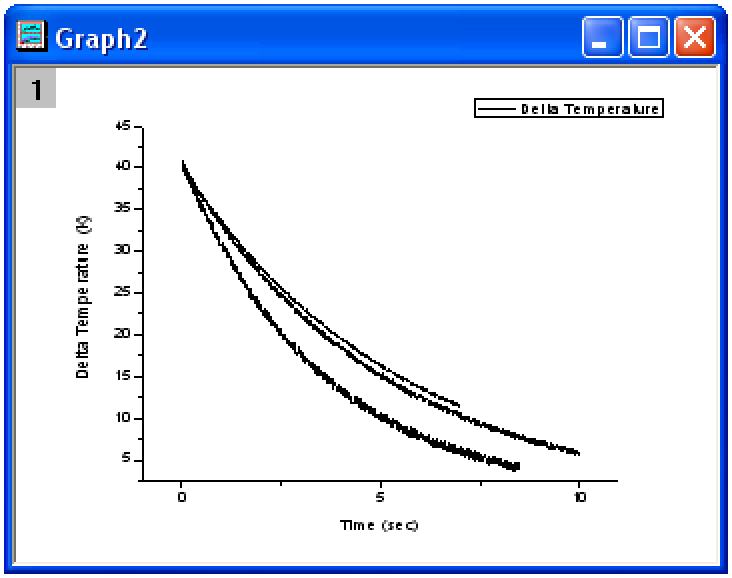


5.1.3 Add Data to an Existing Graph with Drag and Drop Plotting

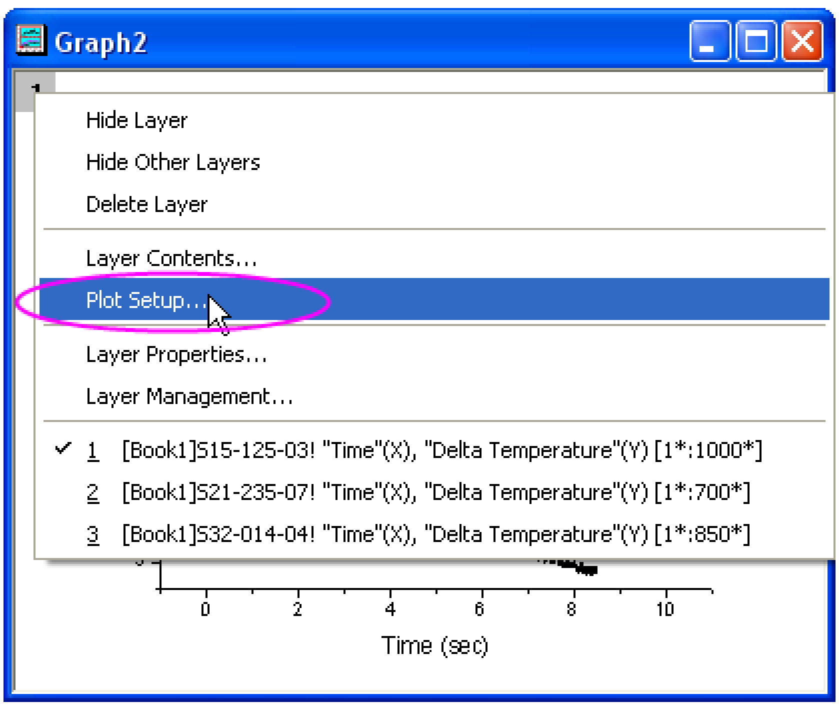
1. Go back to the book with the three sheets of data from the above example.
2. Highlight the **Delta Temperature** column (Column B) from the first sheet and then select **Plot: Line: Line** to create a line plot.
3. Go back to the workbook, and for the other two sheets, one at a time, select the **Delta Temperature** column, and position the cursor at the edge of the column, until you see the cursor turn into .



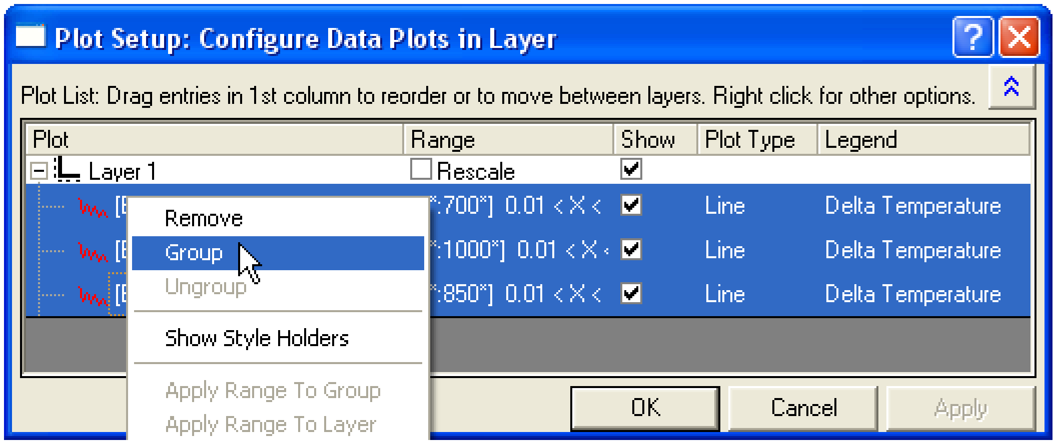
Then you can drag-and-drop the column into the graph page. Another curve will be added to the current layer.



4. After adding the other curves, right click on the layer icon and select **Plot Setup** from the context menu.

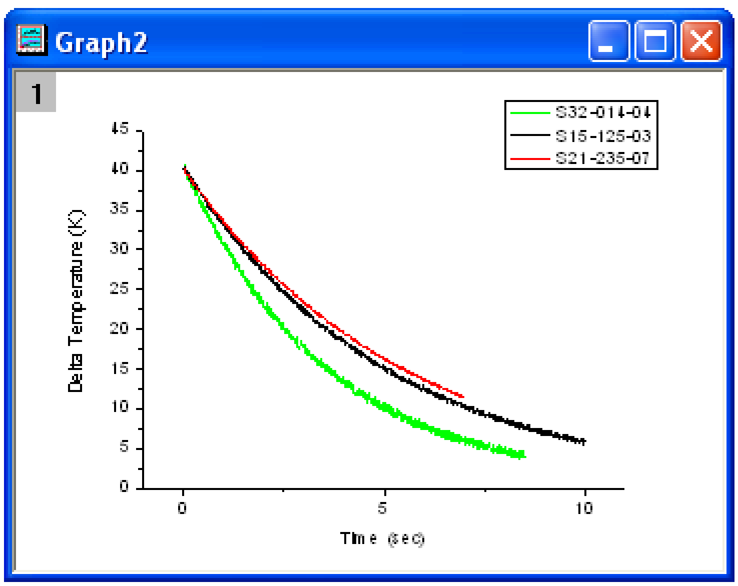


5. The **Plot Setup** dialog opens with the bottom panel visible. (Note: The **Plot List** panel is the only one you need, but if you wish, you can click the  button to expand and show the other two panels of the dialog.) Hold the **Shift** key while selecting all three data plot entries, and then right-click and select **Group**. Click **OK** to close the dialog.



Grouping the plots allows for quick creation of presentation-ready graphs, because each selection in the group is assigned a differentiating set of plot attributes (line color = black, red, green; symbol shape = square, circle, triangle; etc...).

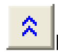
6. Select **Graph: Update Legend** to open the *legendupdate* X-Function dialog. Set the **Auto Legend Translation Mode** to *Custom*. Enter **@WS** for **Legend Custom Format**. Click **OK** and the legend will now contain the worksheet name for each data plot.

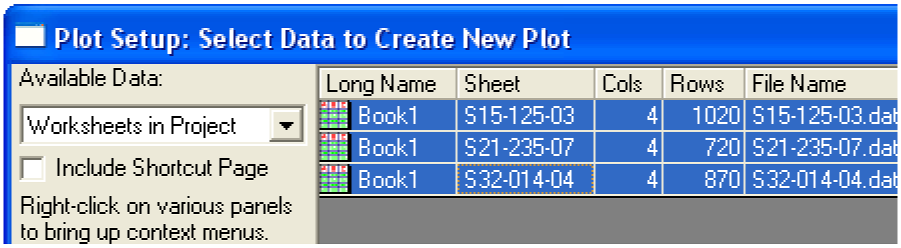


5.1.4 Create a Plot using Plot Setup

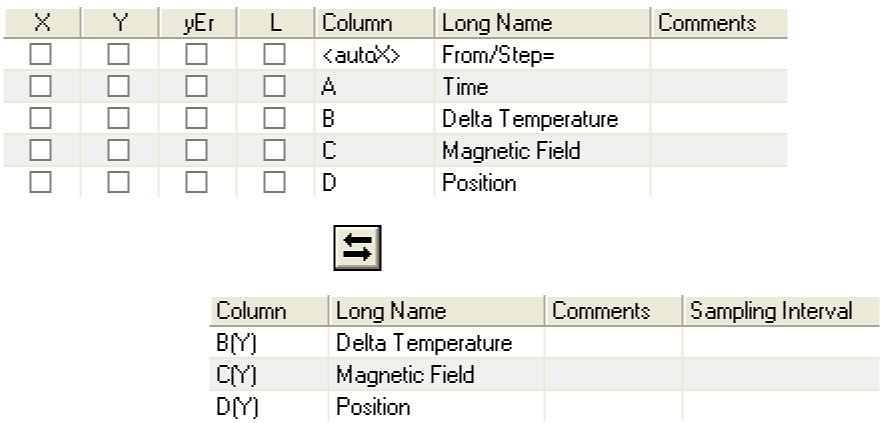
The Plot Setup dialog box is useful for a variety of plotting tasks, including creating graphs, modifying the plot type, adding plots to or removing plots from the graph, grouping or ungrouping plots, and editing the plot range.

1. Using the same workbook as in the above examples, select no data. It doesn't matter what worksheet is active; what is important is that no columns are highlighted.
2. Select **Plot: Multi-Curve: 4 Panel** from the menu. Without any data selected, Origin will open the **Plot Setup** dialog, allowing you to choose the data you wish to plot.

3. Expand the top panel by clicking the  button. Hold down the **Shift** key to highlight the three worksheets. Alternatively, you can just click and drag your selection so that all three worksheets are highlighted.

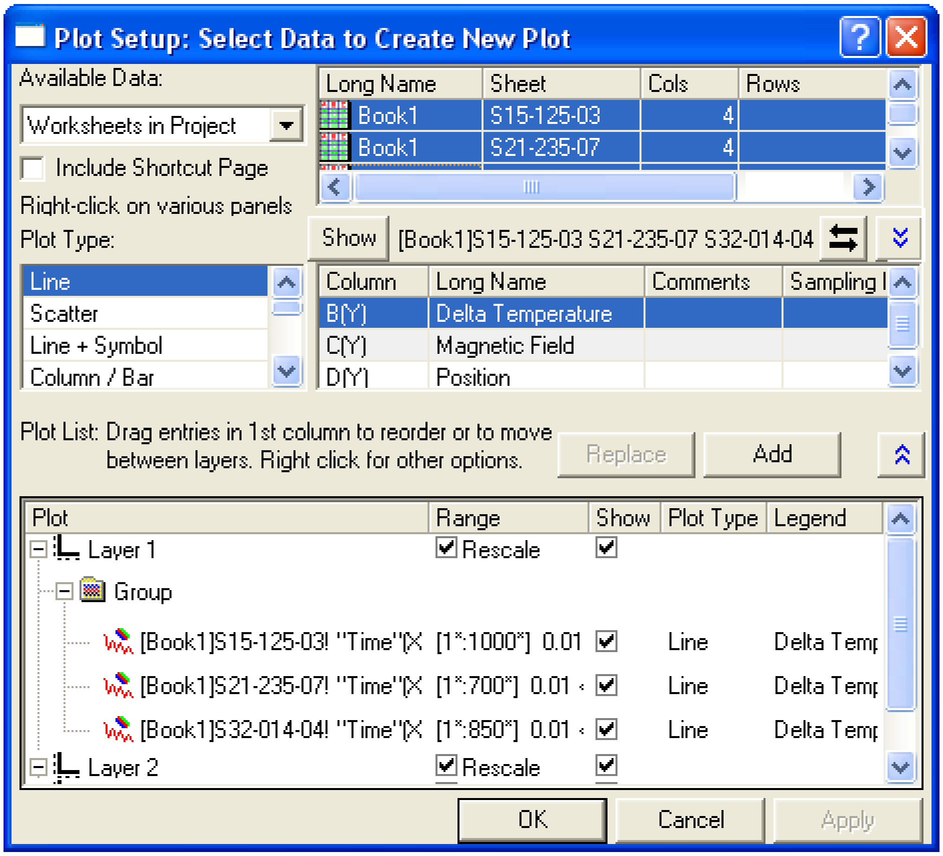


4. In the middle panel, common columns in all three sheets are displayed. In this case, all three sheets have similar data with matching column names. Click the double arrow button  to show the plot column list. This mode is easier because you don't have to check the X and Y designation check boxes.

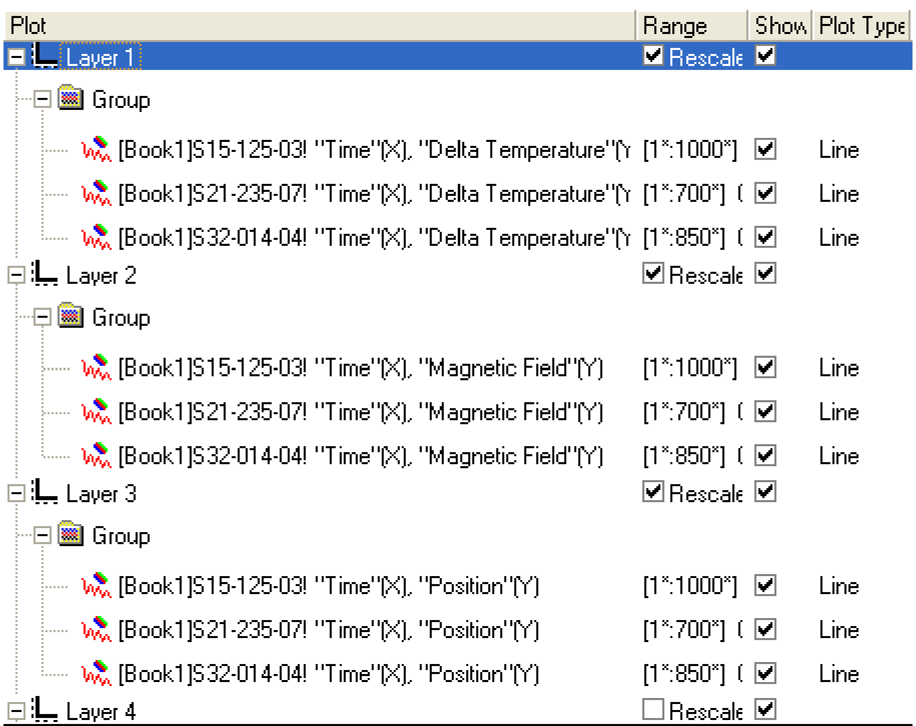


5. Open the bottom panel of the dialog, the Plot List, if it is not already visible. **Layer 1** is highlighted. Select **Delta Temperature** in the middle panel, and then click the **Add** button. Since you already selected the three worksheets in Step 3, this will add the **Delta Temperature**

column from each of the three worksheets to layer 1.

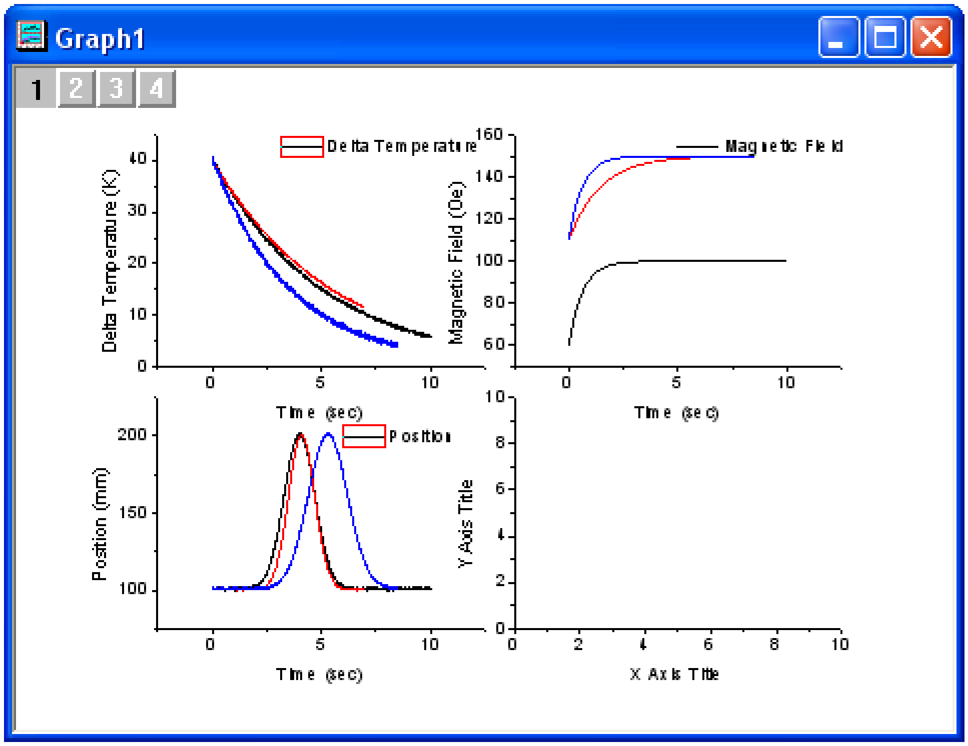


6. Repeat these steps to add **Magnetic Field** and **Position** into layer 2 and layer 3, respectively.



Note that in each layer, the three data plots are automatically grouped.

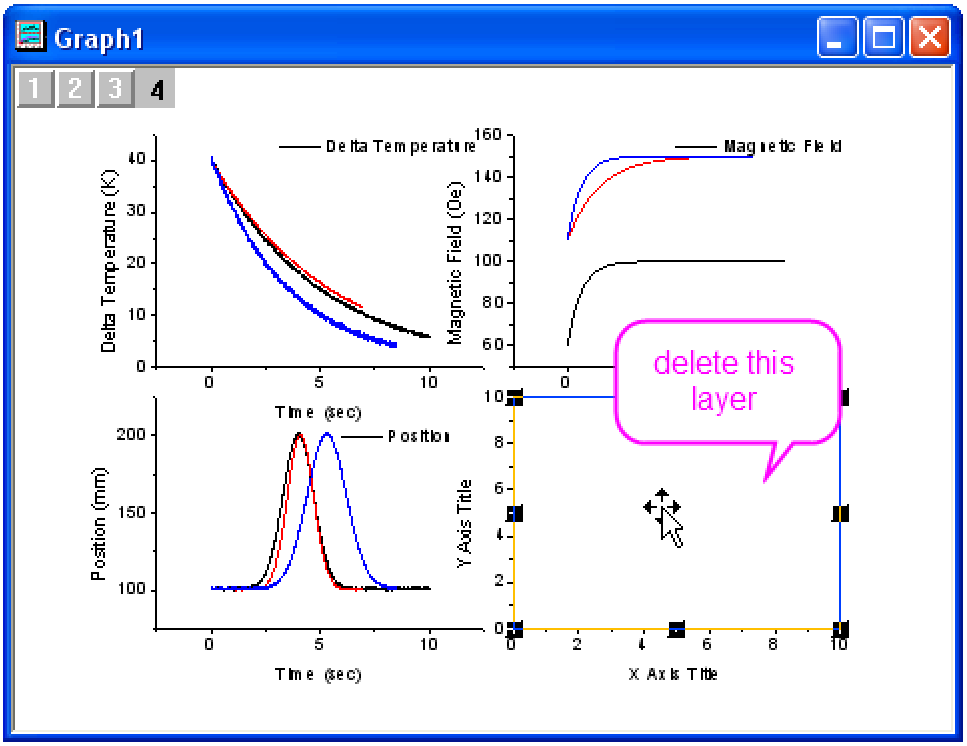
7. Click the **OK** button to create the graph.



5.1.5 Customize and Save a Graph Template

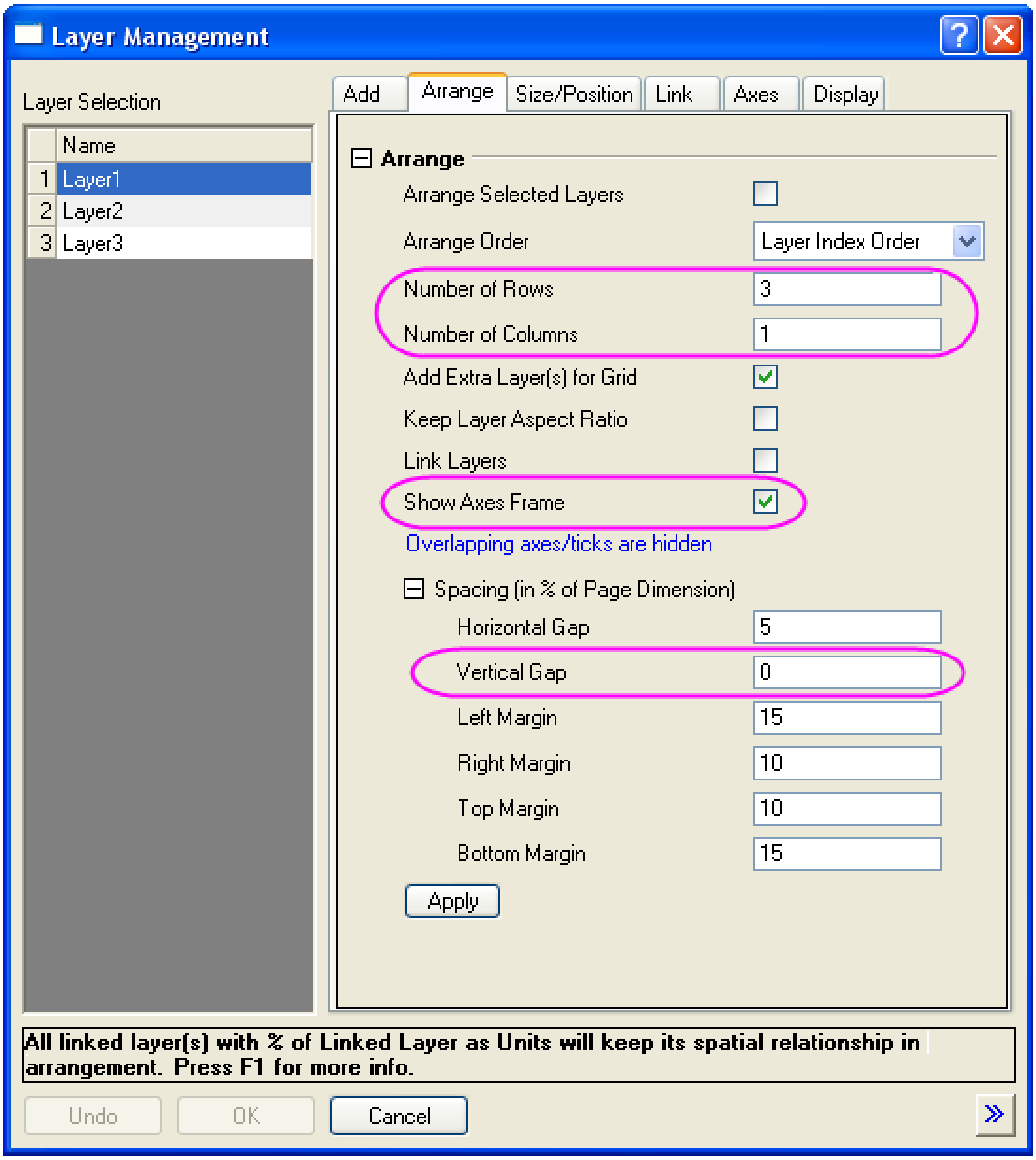
In the above examples, the **3Ys Y-YY**, **Line** and **4 Panel** plots are each created from a different, specific built-in plot template. If you don't see the exact graph you need, you can often create it and save it as your own template to reuse later with similar data.

1. Continue with the 4 panel plot from above. Select the empty 4th layer and press the **Delete** key on the keyboard to get rid of it.



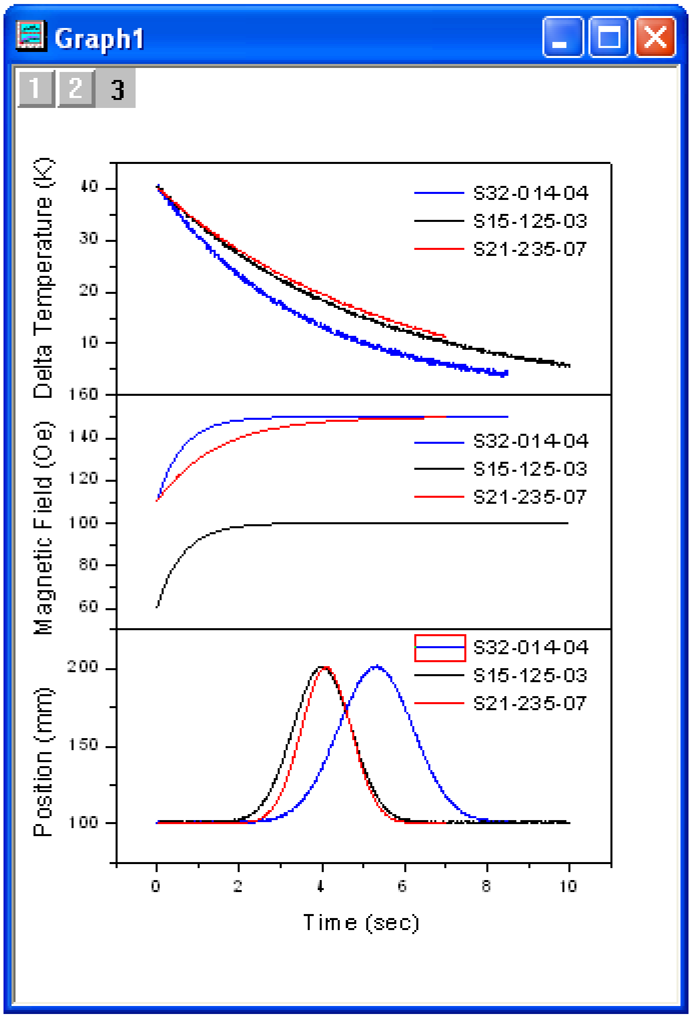
2. Select **Graph: Layer Management** from the menu to bring up the Layer Management dialog. On the **Arrange** tab, set column=1, row=3; check the **Show Axes Frame** check box; expand the

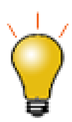
Spacing(% of Page) branch and set the **Vertical Gap** to 0. Click **Apply** and then click **OK**.



3. Select **File: Page Setup** from the menu and change the **Orientation** to **Portrait**.
4. Select **Graph: Update Legend: Open Dialog**. Leave the destination as Whole Page to update the legend in all layers on the graph page. Set the **Auto Legend Translation Mode** to *Custom*.

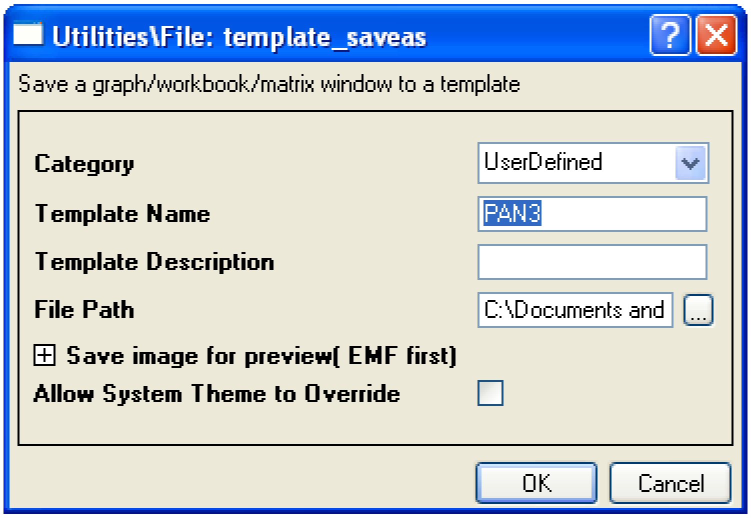
Enter **@WS** for **Legend Custom Format**, and click OK. Your graph will look as follows:





To align the left Y-axis titles, you can hold down Shift and select them, and then use the Left Align button on the Object Edit toolbar.

5. Now that you have customized the graph, select **File: Save Template As**. In the dialog that opens, save as a new template with a new name such as **PAN3** under the **UserDefined** category.

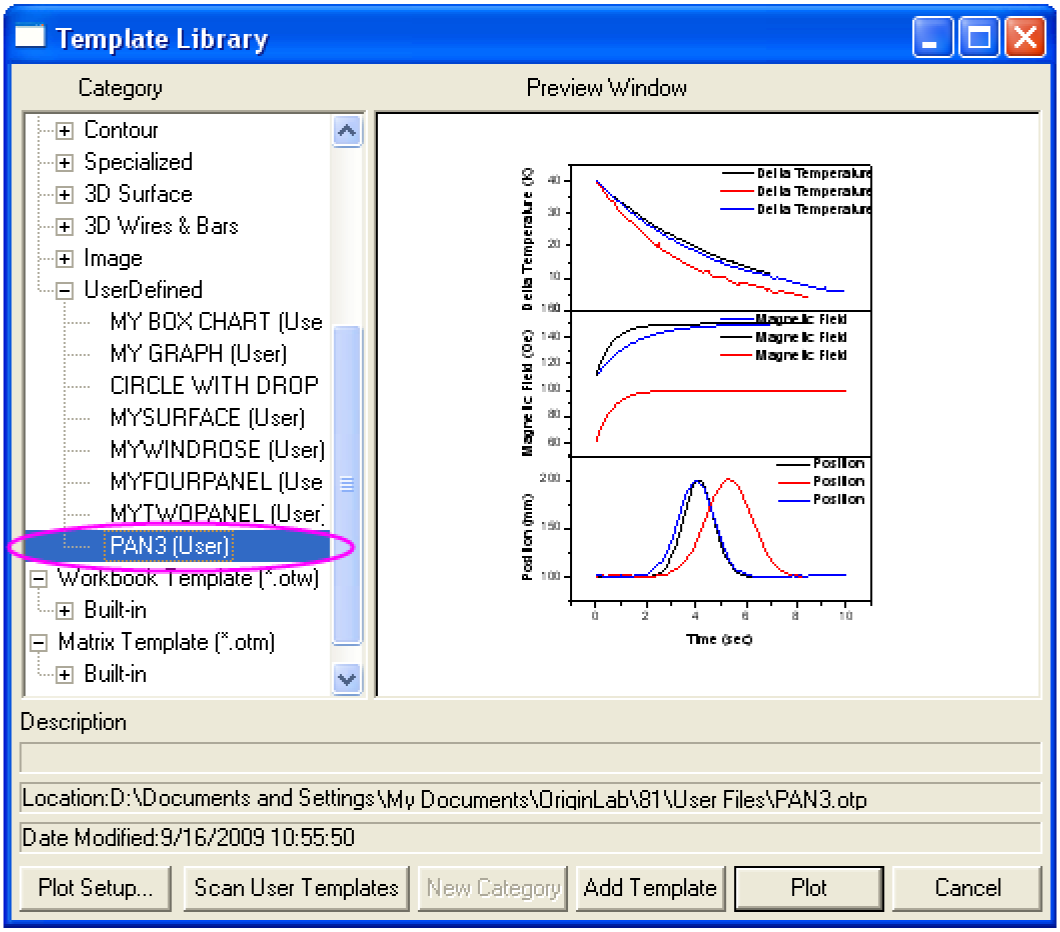


Now you can reuse this template in the next example.

5.1.6 Plot into a Saved Custom Template

1. Go back to the workbook, and in the menu, select **Plot: Template Library** to bring up the Template Library dialog. All of the Origin templates, including Graph, Workbook, and Matrix templates, are listed here. Expand the **UserDefined** category under **Graph Template**, and you'll

see the one that you just saved in the last example.

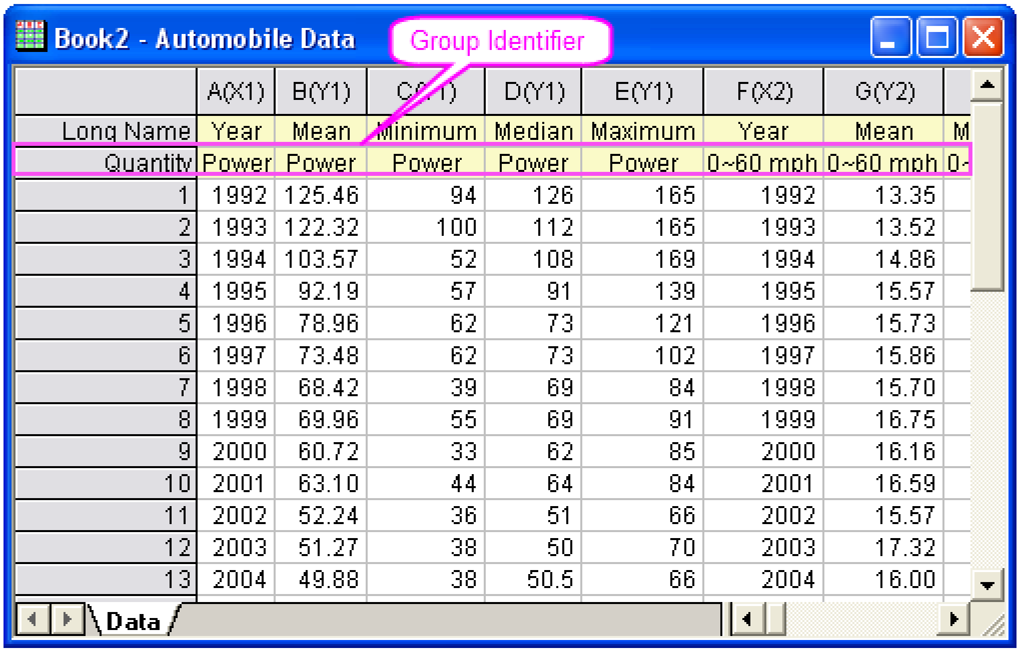


2. Click the **Plot Setup** button. Then you can use the same steps from the **Create a Plot using Plot Setup** section to create a three-panel graph directly.

5.1.7 Plot Groups

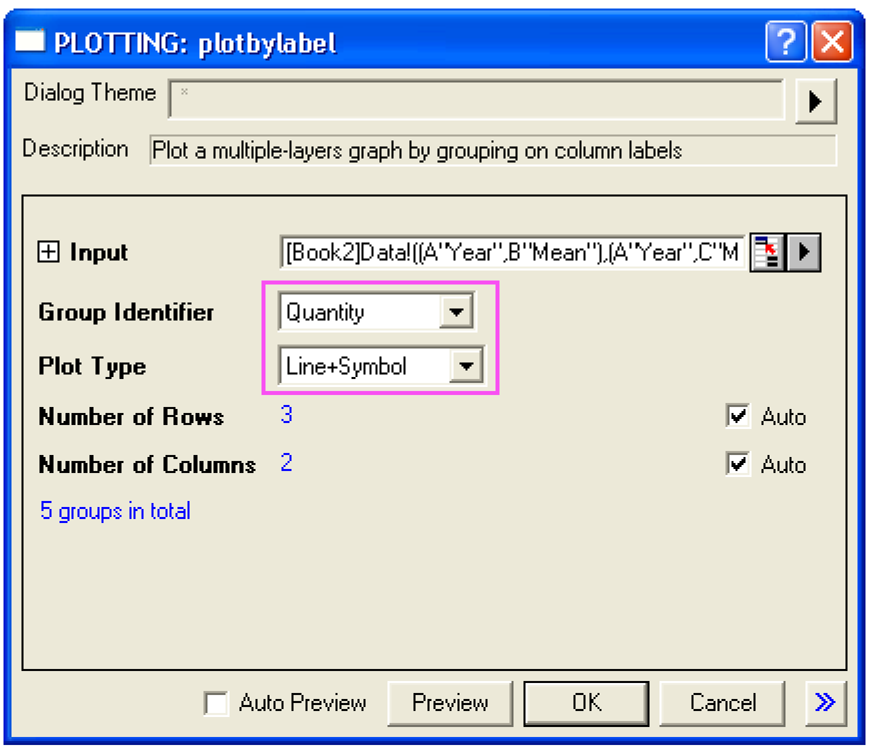
Origin offers a plotting option, **Multiple Panels by Label**, that allows you to create a multilayer graph, each layer of which contains multiple plots, identified by the same label.

1. Open the file **Samples\Graphing\Automobile Data.ogw**.

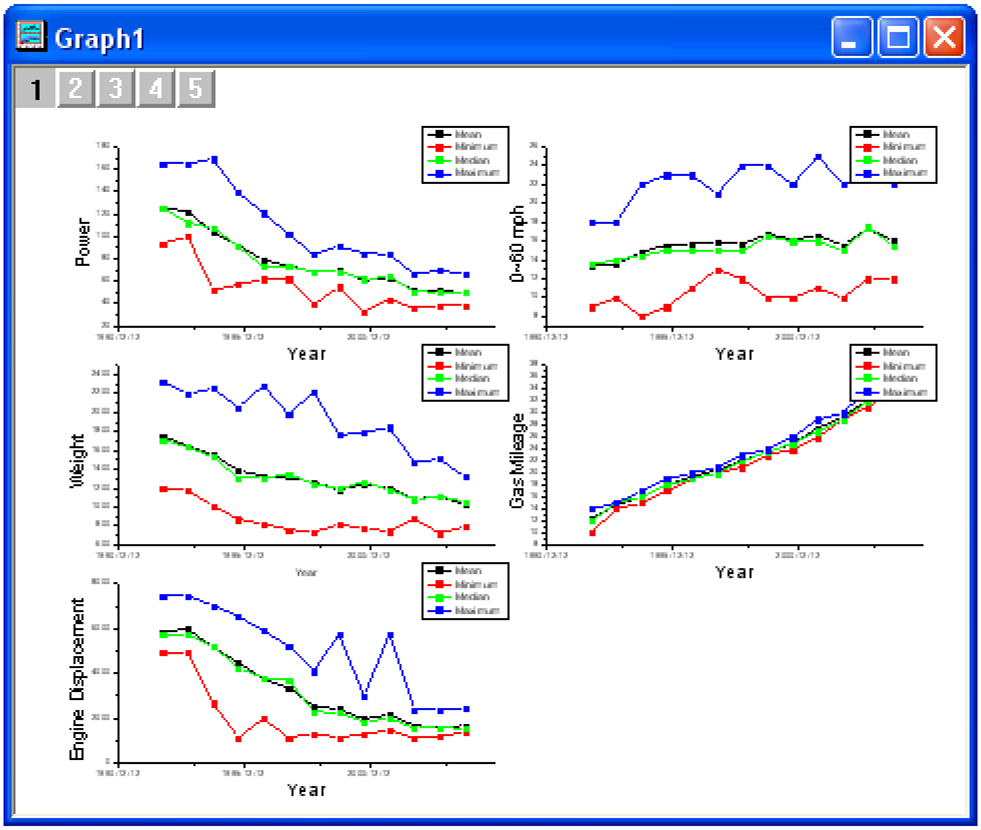


	A(X1)	B(Y1)	C(Y1)	D(Y1)	E(Y1)	F(X2)	G(Y2)	M
Long Name	Year	Mean	Minimum	Median	Maximum	Year	Mean	M
Quantity	Power	Power	Power	Power	Power	0~60 mph	0~60 mph	0~
1	1992	125.46	94	126	165	1992	13.35	
2	1993	122.32	100	112	165	1993	13.52	
3	1994	103.57	52	108	169	1994	14.86	
4	1995	92.19	57	91	139	1995	15.57	
5	1996	78.96	62	73	121	1996	15.73	
6	1997	73.48	62	73	102	1997	15.86	
7	1998	68.42	39	69	84	1998	15.70	
8	1999	69.96	55	69	91	1999	16.75	
9	2000	60.72	33	62	85	2000	16.16	
10	2001	63.10	44	64	84	2001	16.59	
11	2002	52.24	36	51	66	2002	15.57	
12	2003	51.27	38	50	70	2003	17.32	
13	2004	49.88	38	50.5	66	2004	16.00	

2. Click on top left corner of the worksheet to select the entire sheet. Select **Plot: Multi-Curve: Multiple Panels by Label** to open the **plotbylabel** dialog.
3. Set the dialog options as follows, and click the OK button to create the graph.



The graph should look like this:



5.2 Line Symbol

Topics covered in this section:

1. 2D Plotting
2. Line Graph with Recession Bars
3. Scatter Plot of Decay and Recovery Curves
4. Micro-Raman spectroscopy of complex nanostructured mineral systems
5. Scatter Central Plot
6. Line Graph with Masked Data
7. Mark out a segment of plot with different plot style
8. Adding Plot
9. Error Bars with Fill Area

5.2.1 Basic 2D Plotting

Summary

Origin provides flexible ways to create 2D plots. You can easily customize plot attributes, arrange layers, and select different datasets for each layer. This tutorial will teach you the basic plotting skills.

Minimum Origin Version Required: Origin 8.0 SR6

What you will learn

This tutorial will show you how to:

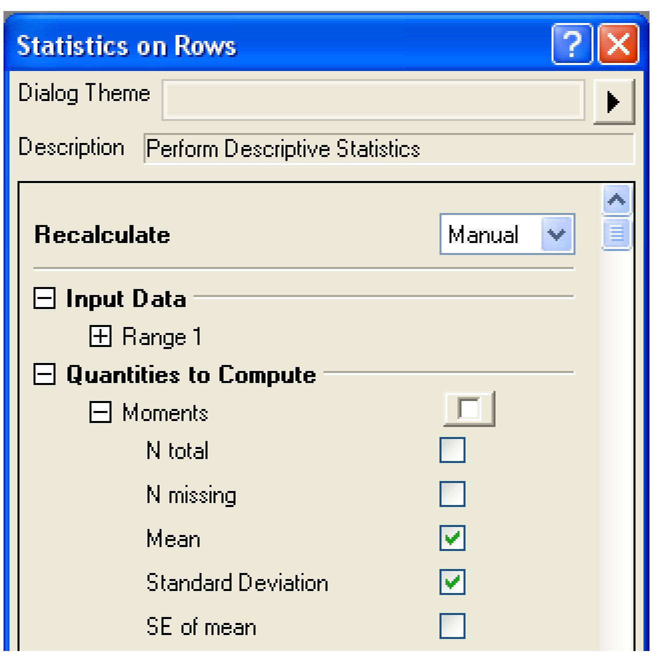
- Perform simple row statistics
- Create a graph and save as a template
- Plot into your template
- Use the Plot Setup dialog

Steps

Simple row statistics

1. Start with an empty worksheet, select **File: Import: Single ASCII...** to open the *Import Single ASCII* dialog, browse to the *\Samples\Curve Fitting* subfolder of the Origin program folder, and import the file *Dose Response - No Inhibitor.dat*.

2. Highlight columns 2 through 4 and select **Statistics: Descriptive Statistics: Statistics on Rows**. Make sure to check the **Mean** and **Standard Deviation** check boxes on *Quantities to Compute* > *Moments* branch to output these results.

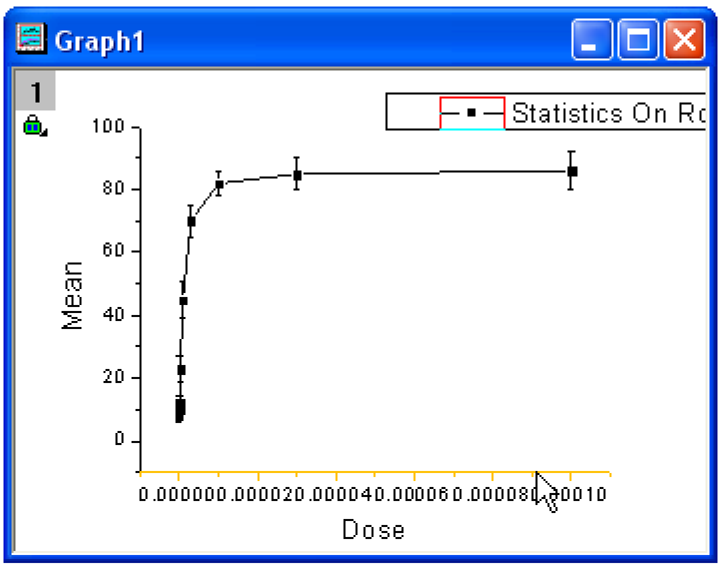


3. After you click the **OK** button, two new columns, **Mean(Y)** and **SD(yErr)** are added to the source worksheet. Here, **yErr** means that this is an error column and the data in this column can be used to plot error bars.

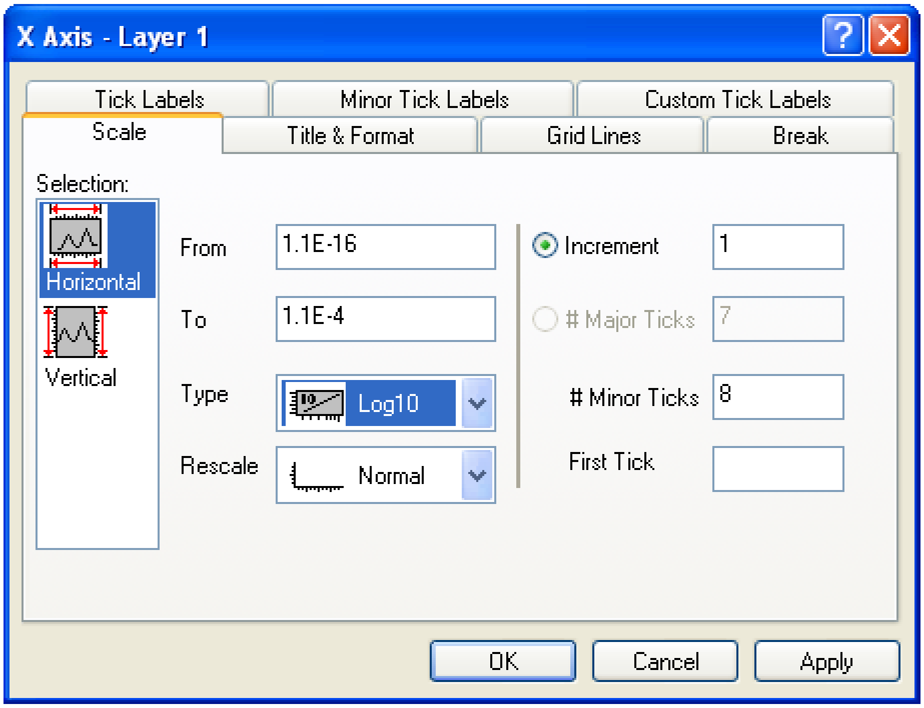
   Note: To simplify plotting, each column in an Origin worksheet has a plotting designation. To change a column's plotting designation, select the column and click on the **Column** menu. Alternatively, right-click on the column and choose **Set As** from the context menu.

Create a graph and save as a template

1. Highlight the Mean(Y) and SD(yEr-) columns and select **Plot: Line+Symbol: Line+Symbol** to create the plot:

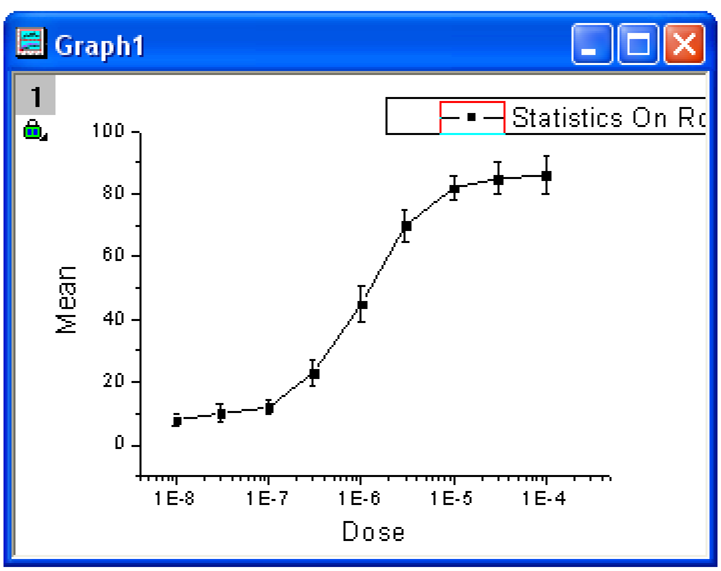


2. To change the X scale to Log, double-click on the **X** axis to bring up the **X Axis** dialog. On the **Scale** tab, change the axis **Type** to **Log10**:

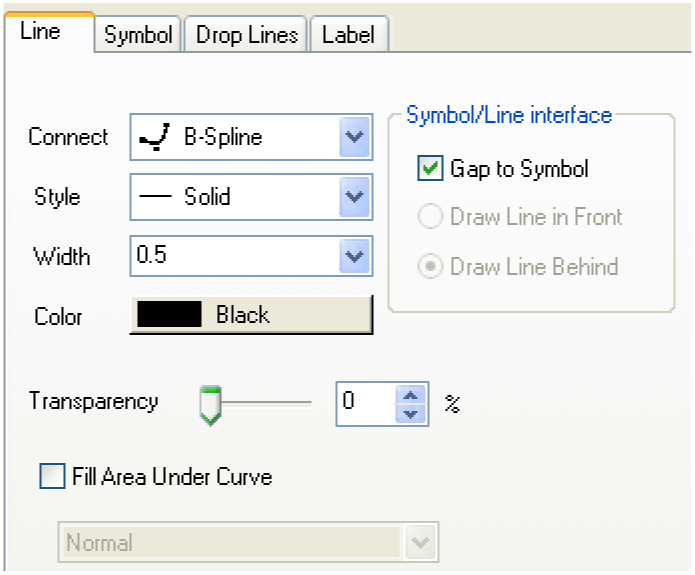


Click the **OK** button to close the dialog.

3. Select **Graph: Rescale to Show All** from the menu, which will rescale the X and Y axes of the graph. :

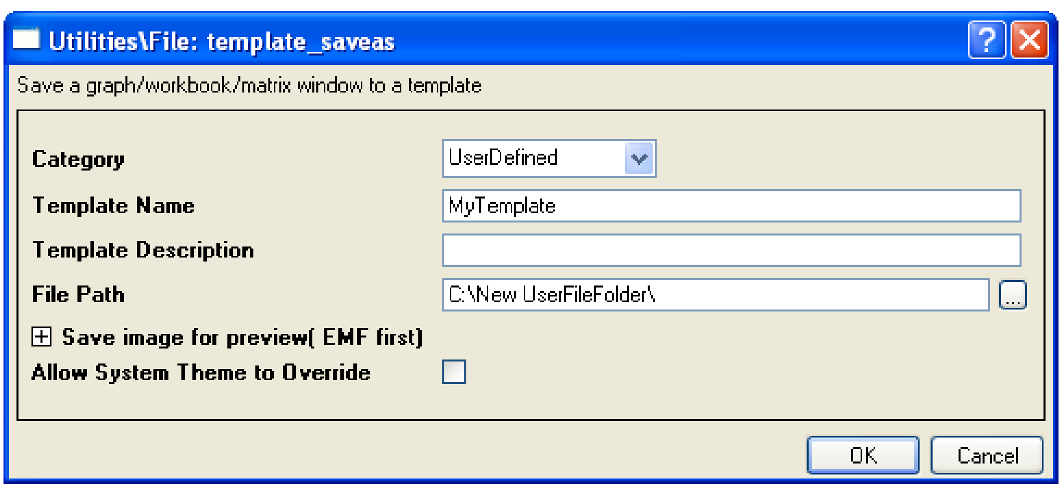


4. To edit the curve, double-click on any plot symbol to bring up the **Plot Details** dialog. Alternatively, right-click inside the graph and choose **Plot Details** from the context menu. On the **Line** tab of right panel, select **B-Spline** as connect line to get a smoother curve.



Click the **OK** button to close the dialog.

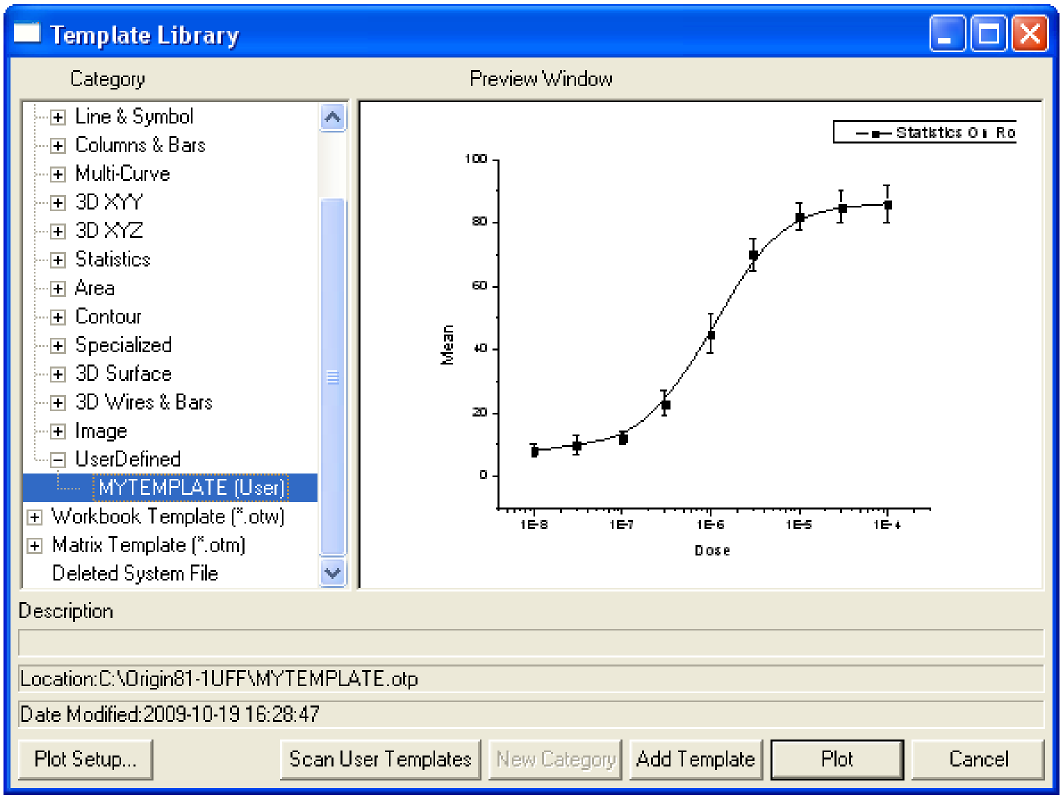
5. When all modifications have been made and the graph looks the way you want it, you can use this graph to create a template, to be used in the future with similar data. Select **File: Save Template as** to open the *Save Template* dialog. In the **Category** drop-down list, select *UserDefined*; and then type a proper **Template Name**. In this example, we use *MyTemplate*. Click **OK** to save the template.



Plot into graph template with the Plot Setup dialog

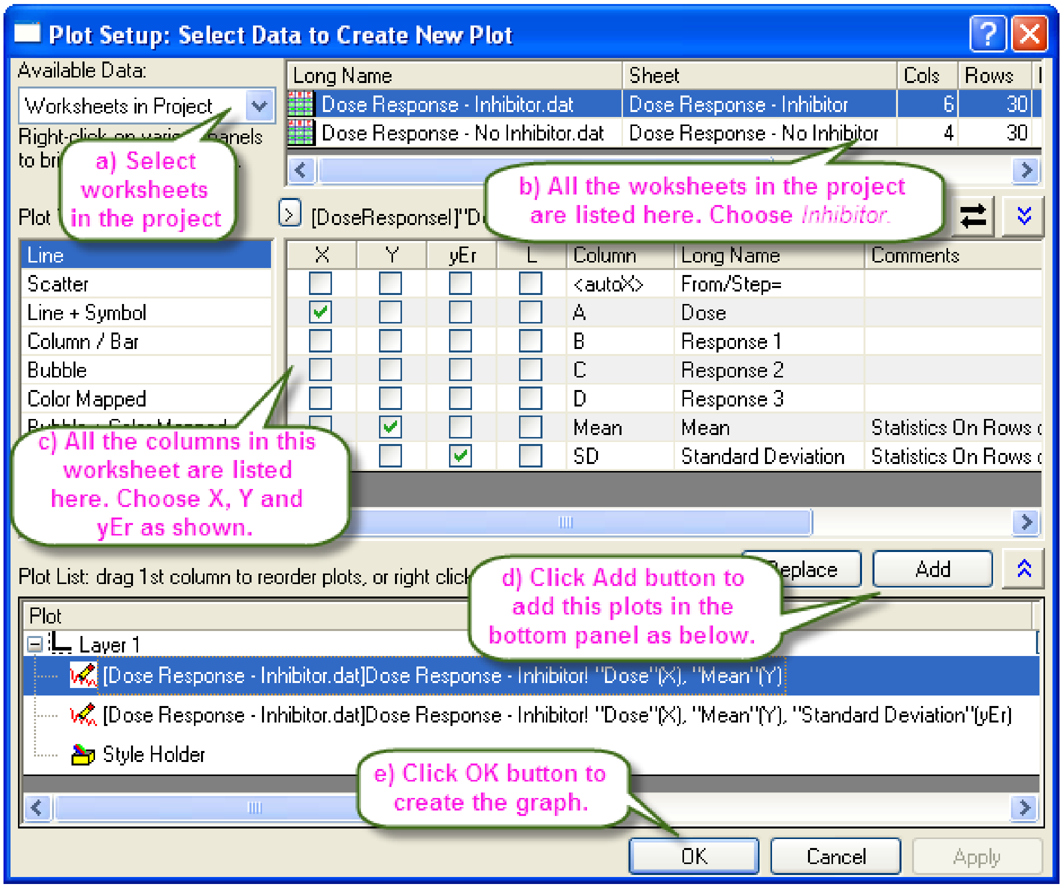
1. Click the  button to open a new workbook, and import the file *\Samples\Curve Fitting\Dose Response - Inhibitor.dat* as above. Perform **Statistics on Rows**, calculating the Mean and SD of this worksheet as you did above and by following the same steps.

2. Select **Plot: Template Library** to open the *Template Library*. Select *MyTemplate* from the *UserDefined* category.

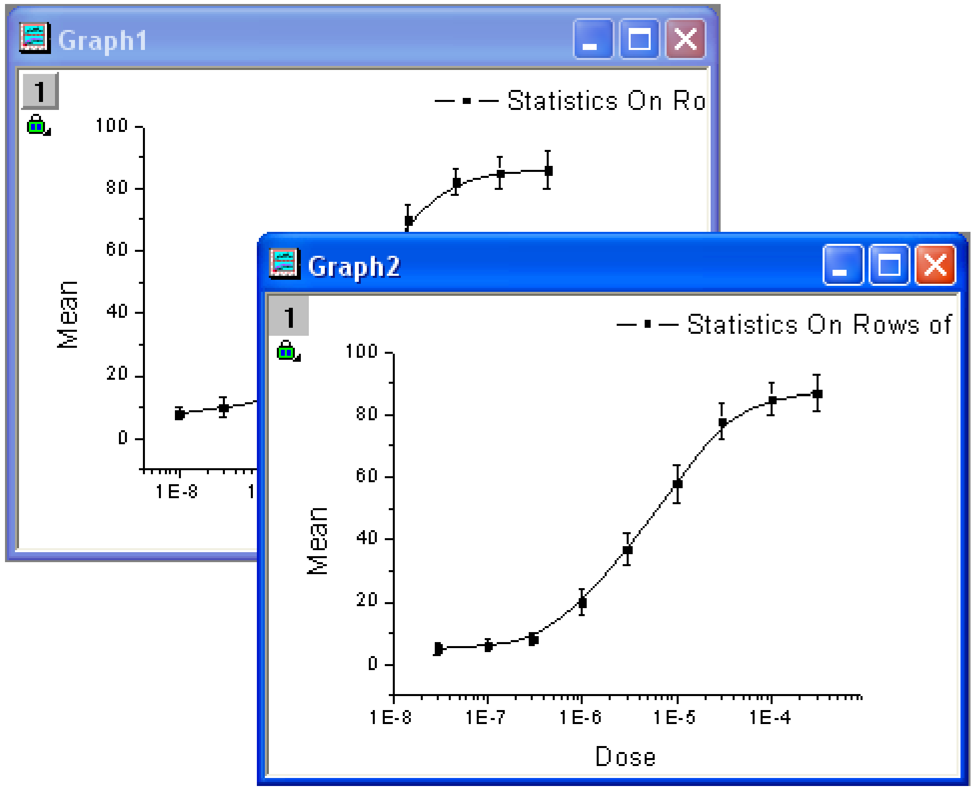


Click the **Plot Setup** button to select the data from which to create the plot. If you click the Plot button, Origin will plot whatever data is highlighted in the worksheet.

3. In the *Plot Setup* dialog, you can choose which columns are to be plotted. (There are three panels in *Plot Setup* dialog, click the  or  button to expand them) To finish creating the plot from your template, please follow the steps a - e outlined on the picture below.



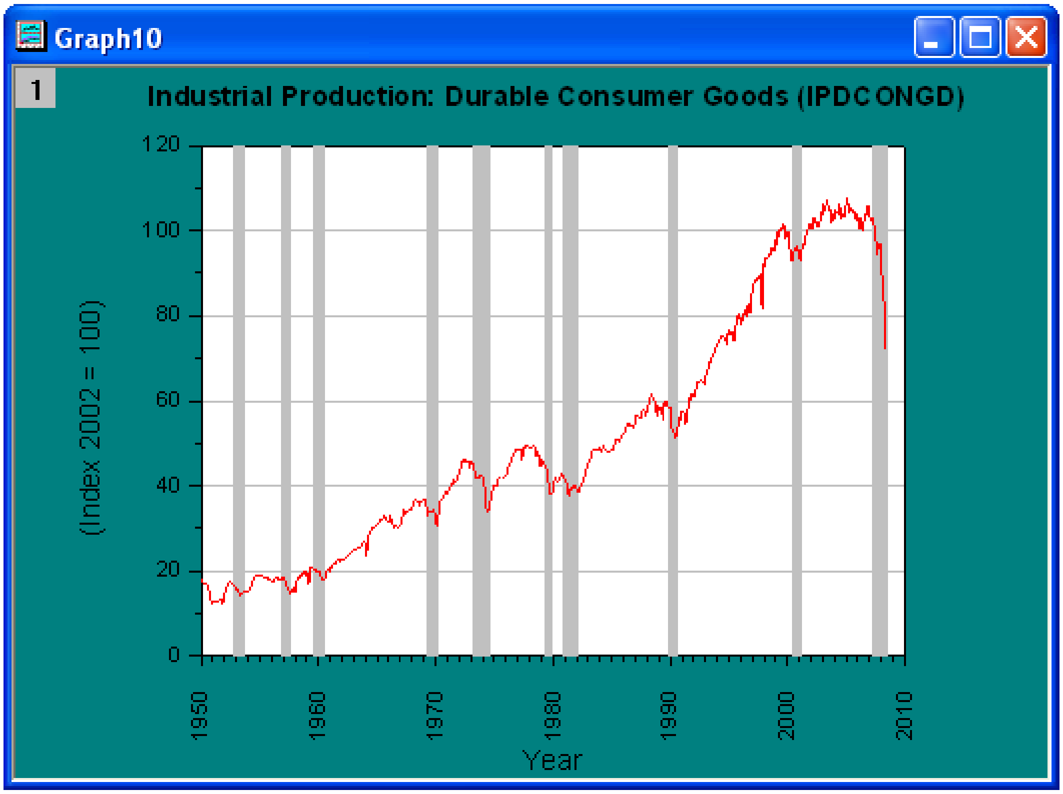
And then you will have:



5.2.2 Line Graph with Recession Bars

<u>Summary</u>

This tutorial will show how to create a line graph with vertical bars spanning across the layer from top to bottom. This type of graph is common when plotting economics data with periods of recession marked as vertical bars.



Minimum Origin Version Required: Origin 8.0 SR6

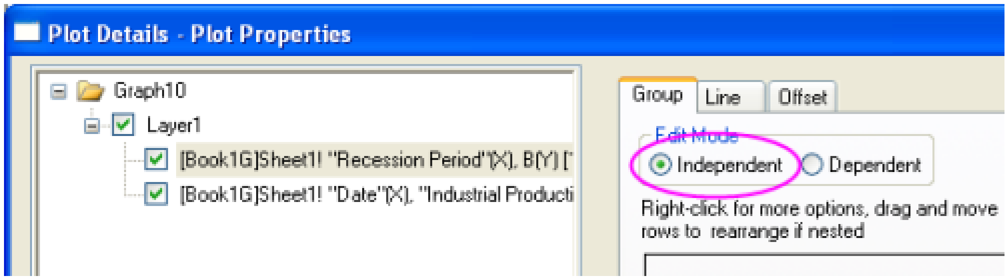
What you will learn

- Plot a line graph
- Use **Span Vertical Bars between Missing Values** to show recession bars
- Set date display format in the **Axes** dialog

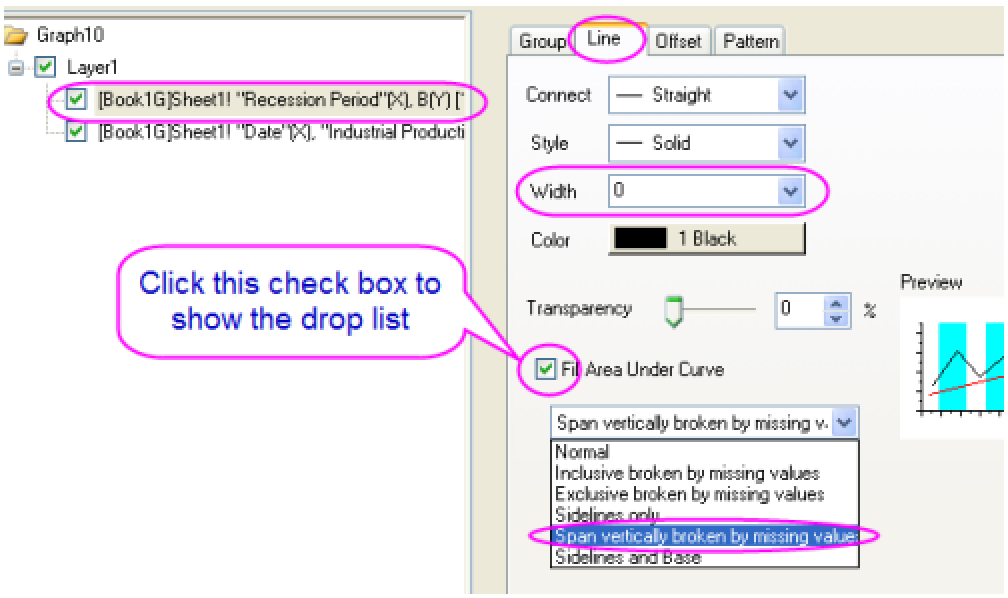
Steps

This tutorial is associated with the **2D and Contour Graphs: Line and Symbol: Recession Bars** in the 2D and Contour Graphs project(\Samples\2D and Contour Graphs.opj).

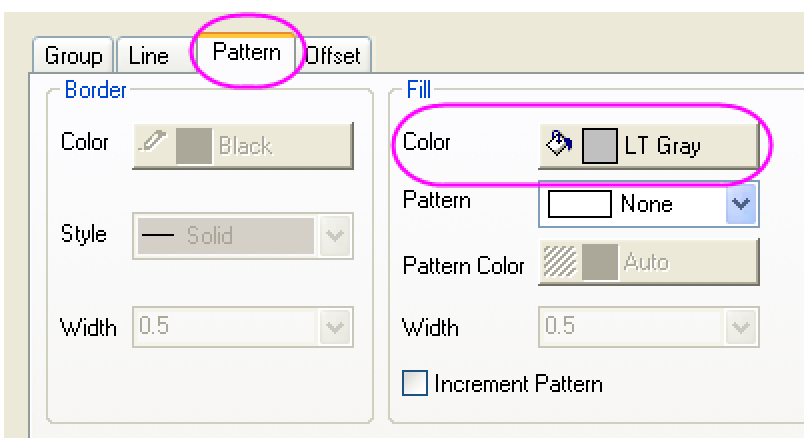
1. Open the **Recession Bars** folder in 2D and Contour Graphs.opj and active the workbook Book1G.
2. Highlight all four colums in the worksheet and then right-click to select **Set as: XY XY** from the context menu. After that, choose "**Plot: Line: Line**" menu to plot a line graph.
3. Double-click the plot to show the **Plot Details** dialog box. In the **Group** tab of right panel select **Independent**.



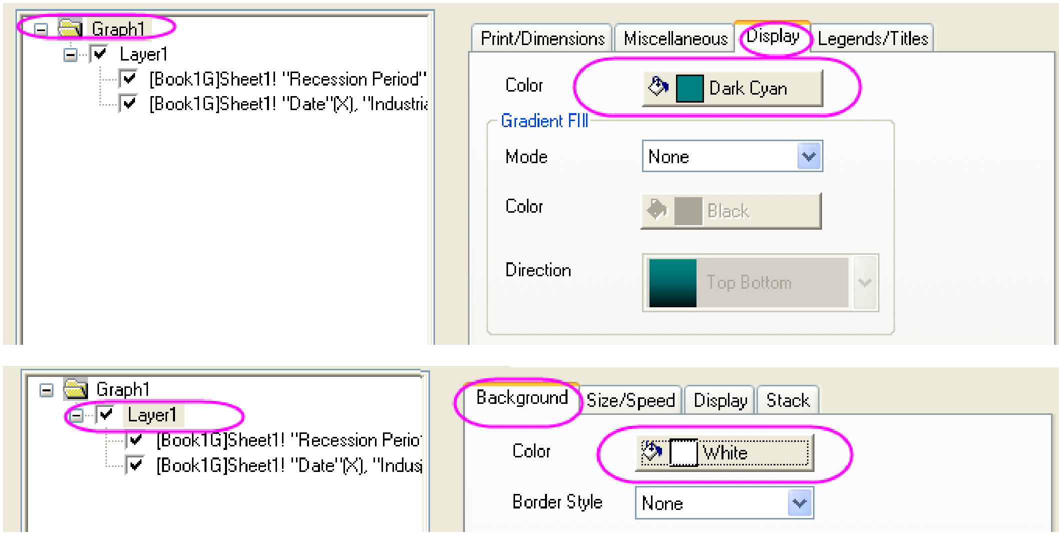
4. In the left panel of the dialog box, choose the first plot node of recession data (A(X):B(Y)). Select the **Line** tab and set **Width** to *0* and click the **Fill Area Under Curve** check box to show the drop-down list. Then choose **Span Vertical Bars between Missing Values** from the drop down list.



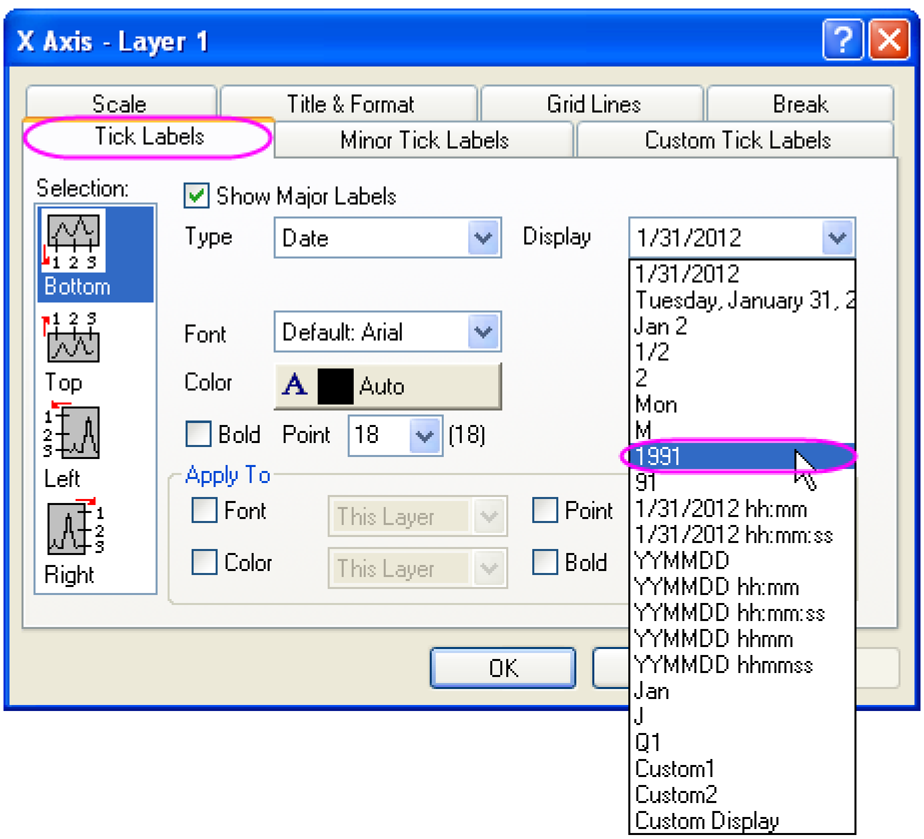
5. Go to the **Pattern** tab, set **Color** in the Fill group as **LT Gray**. Click **Apply** button to apply these settings.



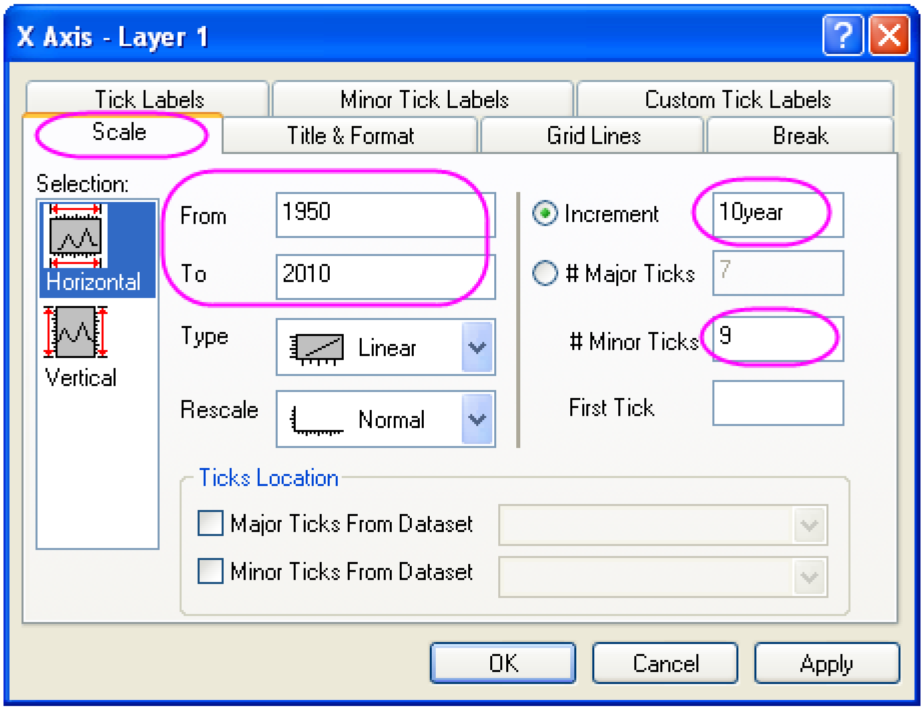
6. After that, go to the Graph level, and set **Color** in the **Display** tab as **Dark Cyan**. Then go to the Layer level, set the **Color** in **Backgroud** tab as **White**. Click **OK** button to close this dialog.



7. Double-click the X axis to open the **Axis** dialog. Go to **Tick Labels** tab to change the **Display** item as 1991.



8. Go to **Scale** tab to set the **From**, **To**, **Increment** and **#Minor Ticks** item as *1950*, *2010*, *10year* and *9*. Then choose **Vertical** in the **selection**, set the **From** item to *0*. Click **OK** button.



9. Delete the Legend for this graph. Double-click the X title and edit the title as *Year*. Do the same to Y title and edit Y title as *(Index 2002 = 100)*.
10. Do the following steps to customize the aixs label and grid lines:
    1. Double-click the x axis to open the **Axis** dialog. Go to the **Custom Tick Labels** Tab and set the **Rotation(degree)** as *90*.
    2. Click the **Grid lines** tab, choose **Horizontal** in the **Selection**, check **Major Grids** box and set **Line Color** as **LT-Gray**.
    3. Go to the **Title & Format** tab, select **Top** in the **Selection**, check **Show Axis & Ticks** box and set **Major ticks** and **Minor Ticks** as *None*. Do the same to **Right** in the **Selection**. Click **OK**.
11. Right-click on the layer, and select **Add / Modify Layer Title** to add title. Type *Industrial Production: Durable Consumer Goods (IPDCONGD)* in the text box. Highlight the text and click the  button in the **Format** toolbar. Adjust the text to an appropriate size by selecting number from 

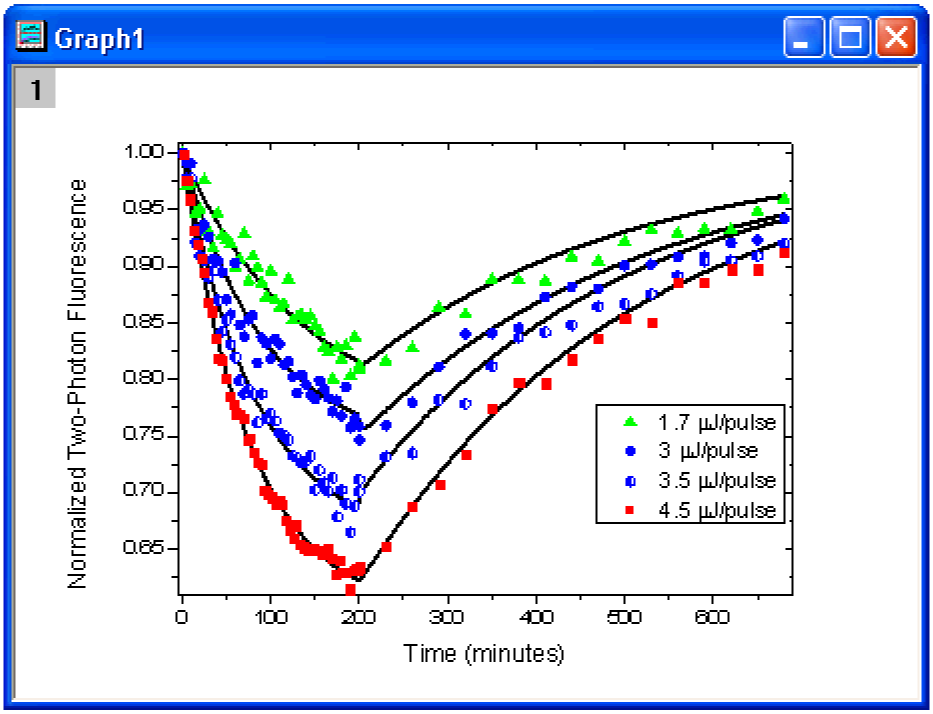
in the **Format** toolbar.

After all, you will get the final graph.

5.2.3 Scatter Plot of Decay and Recovery Curves

Summary

The scatter plot below depicts 3 decay and recovery curves obtained after taking two-photon fluorescence measurements of reversible photodegradation in a dye-doped polymer. To learn more about the graph, please read the case study.



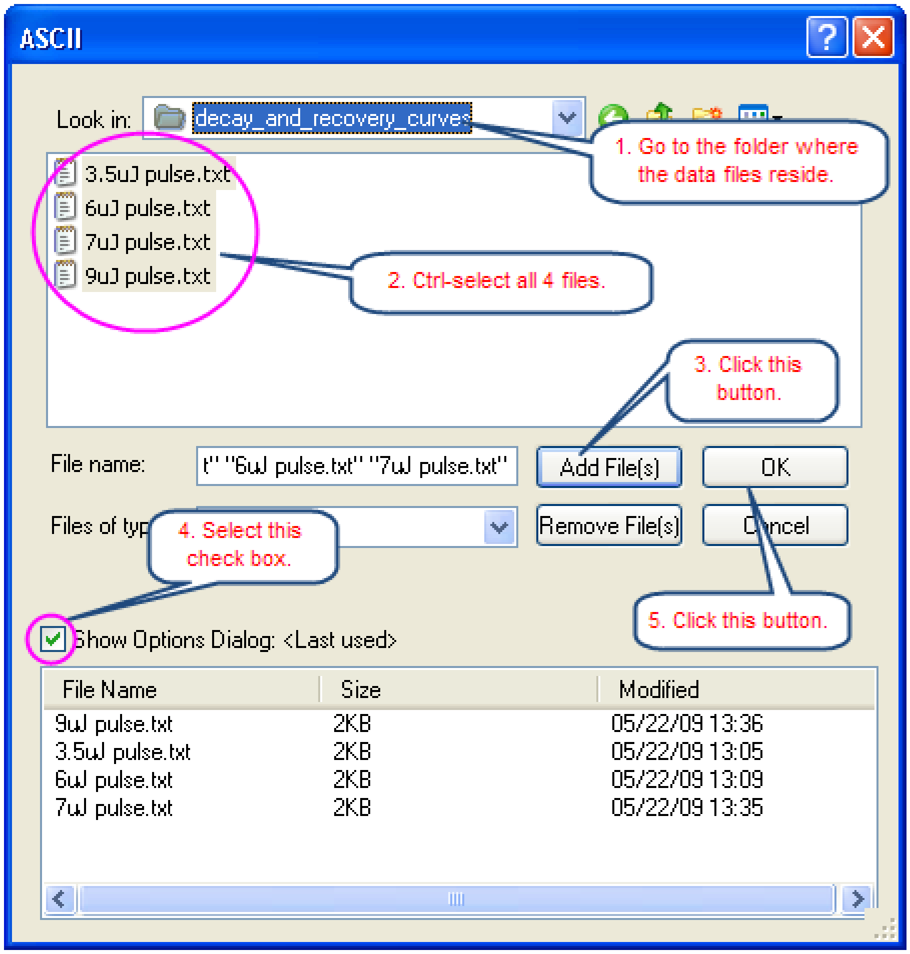
Minimum Origin Version Required: Origin 8.5.1 SR0

What you will learn

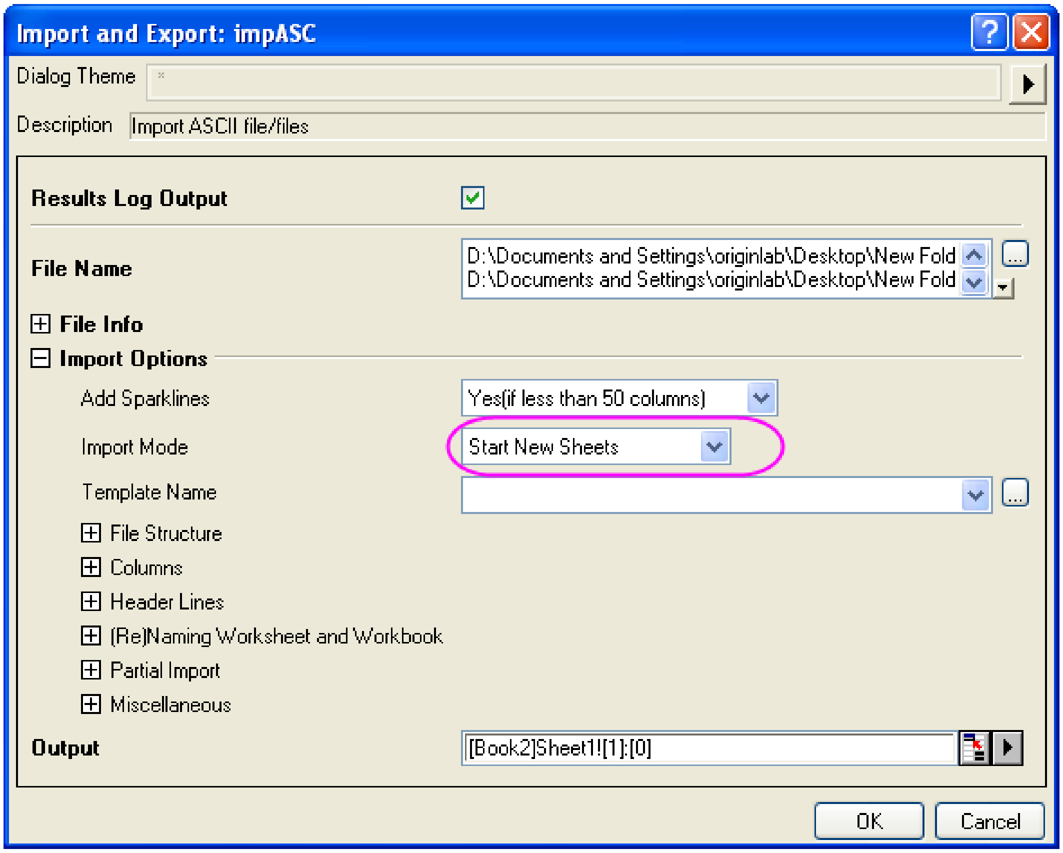
- How to use the **Plot Setup** dialog to arrange plots in a layer
- How to customize the symbols in your graph

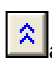
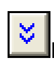
Steps

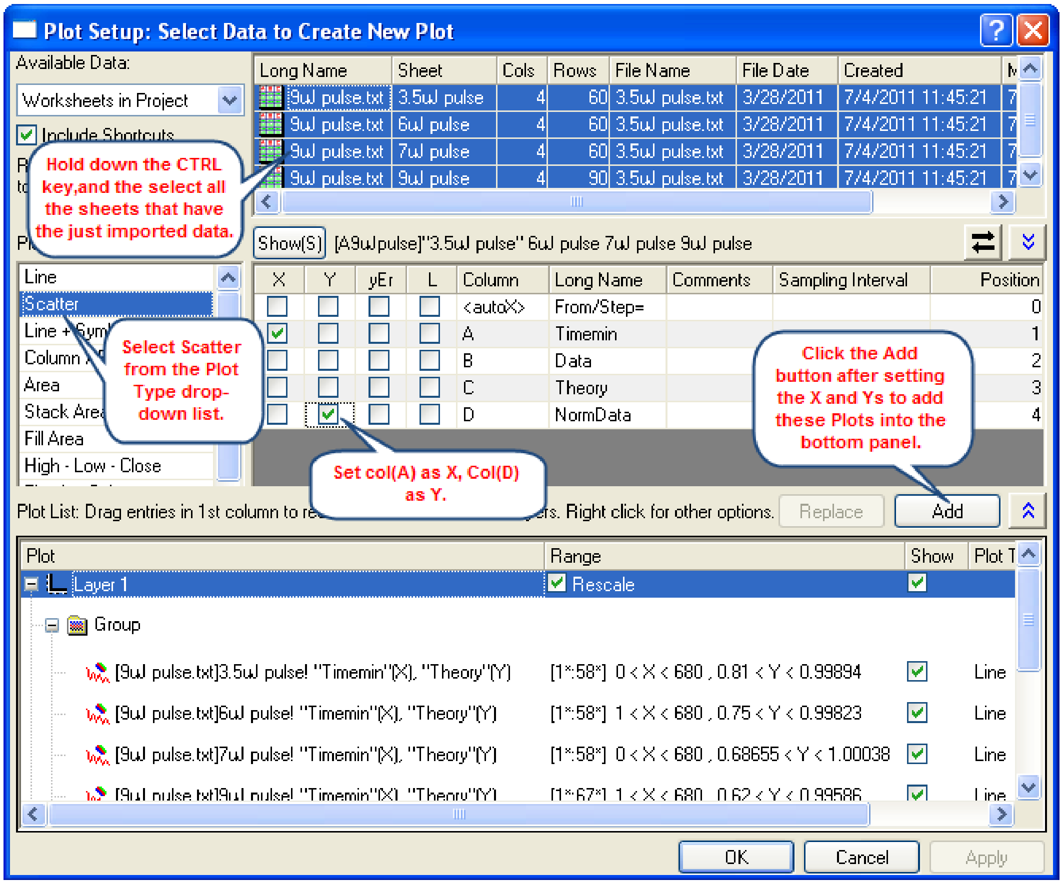
1. Download the zip file from here and extract the text files .
2. Open Origin and click the **Import Multiple ASCII** button  on the **Standard** toolbar to open the **ASCII** dialog and then import the text files.



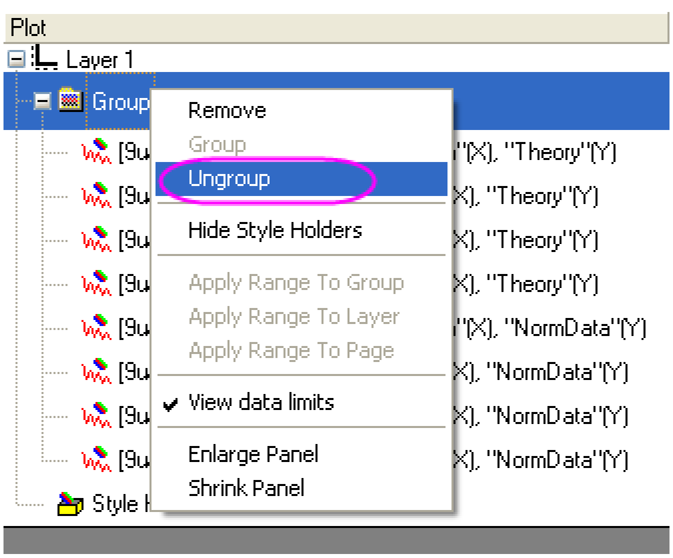
3. In the **impASC** dialog, set **Import Mode** as **Start New Sheets**. Click OK to finish importing.



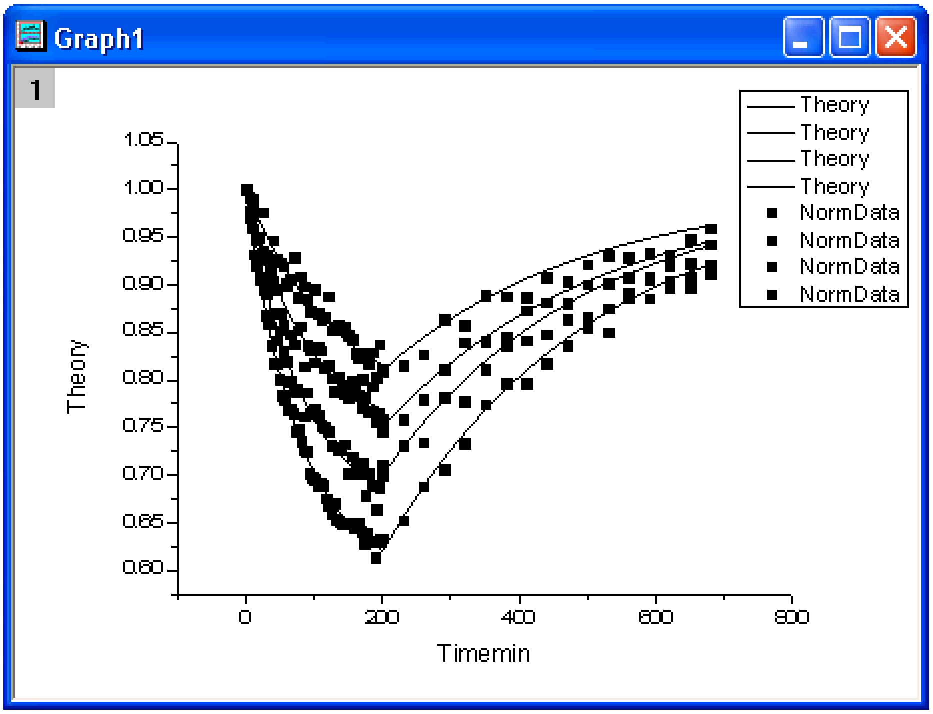
4. We will use the Plot Setup dialog to create a graph with 8 plots. Active the workbook, and make sure that no datasets are now selected. Click the  button on the **2D Graphs** toolbar to open the **Pot Setup** dialog.
   - Show all of the three panels of **Plot Setup** dialog (if not all of them are shown) by clicking the  and  buttons.
   - First, we will add 4 line plots into a graph by using the Plot Details dialog. Highlight all dataset in the top panel, and then select column **Timemin** as X, column **Theory** as Ys in the middle panel. Then add them into the bottom panel.
   - Then we will add 4 scatter plots in to the same graph. Select **Scatter** from the **Plot Type** drop-down list, make sure all dataset in the top panel has been selected, and then select column **Timemin** as X, column **NormData** as Ys in the middle panel. Then add them into the bottom panel.



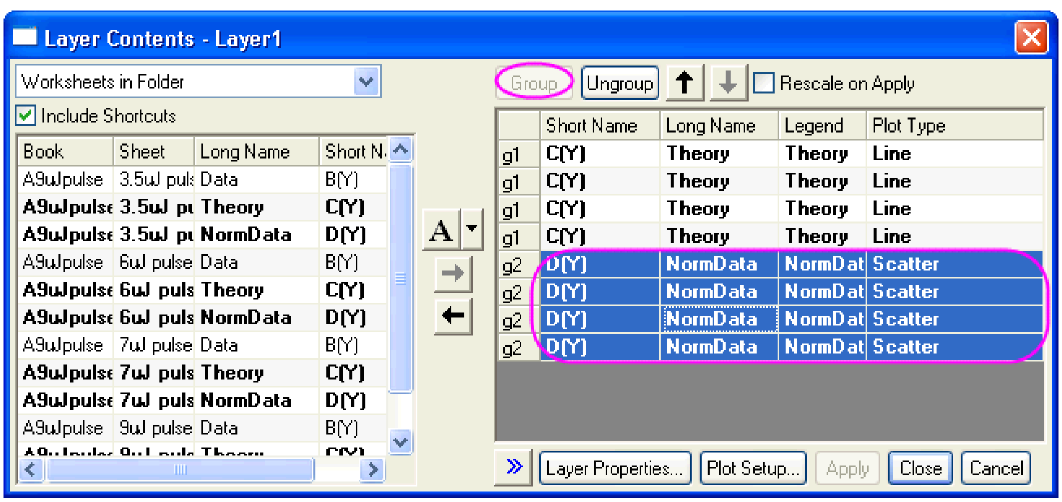
o In bottom panel, if there is a **Group** branch under **Layer1**, right-click on it to select **Ungroup** from the short-cut menu to ungroup these plots.



o Click **OK** to generate the graph which look like the following image shows.

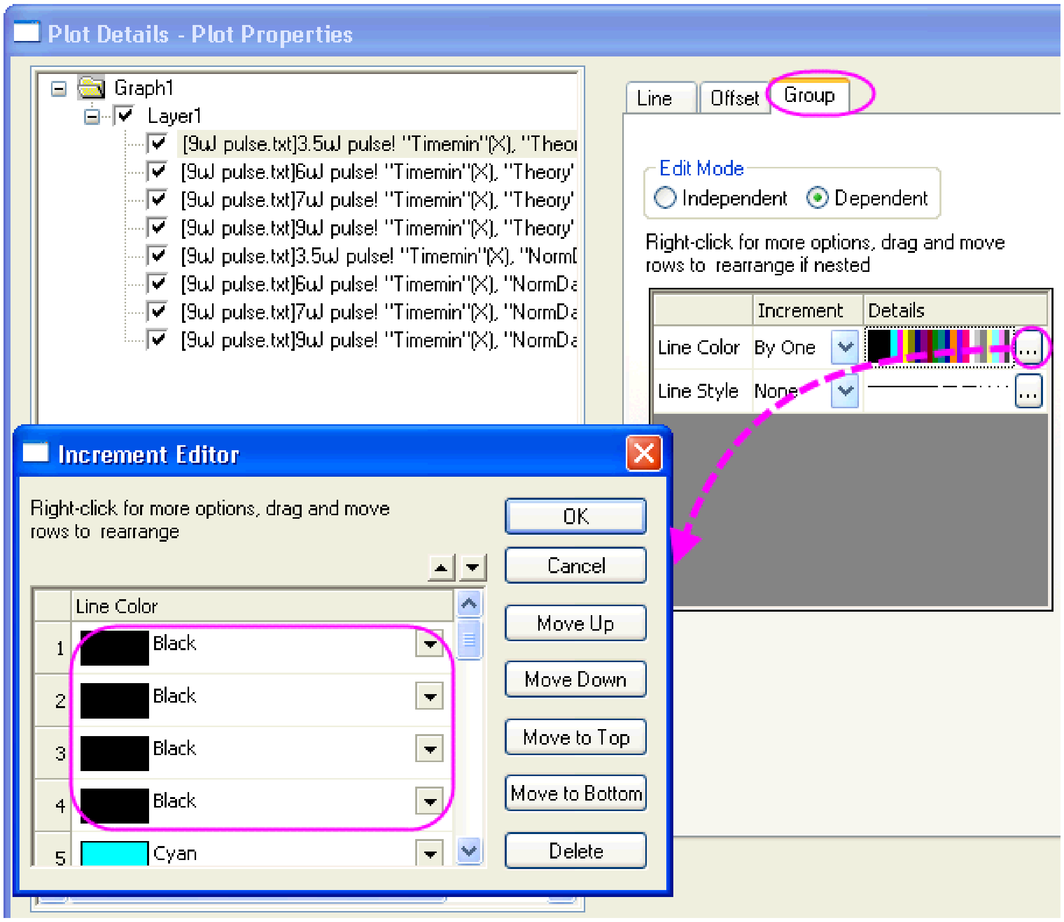


5. Double-click the layer icon at the top-left corner of graph window to open the **Layer Contents** dialog. Then group the **Theory** plots and **NormData** Plots as Group 1 and Group 2 using the **Group** button after highlight the 4 Theory/Norma Data plots separately by using mouse and the **Shift** key,



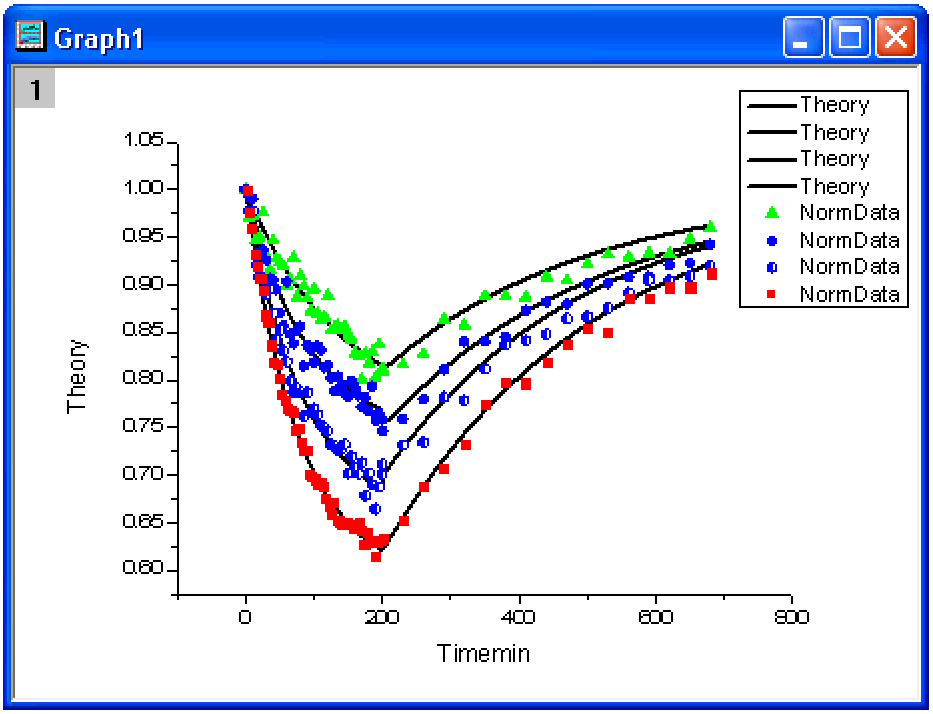
6. Then we will customize the 8 plots in the Plot Details dialog. Select **Format: Plot Properties** to open the Plot Details dialog. In the left panel of the dialog, you could see there are 8 plots: the first 4 plots contain a word *Theory* are line plots, the other 4 contain *NormData* are scatter plots.

7. We will first customize the 4 line plots. Select the first line plot, in the **Line** tab, select **B-Spline** from the **Connect** drop down list and set the **Width** to **3**. Then click the **Apply** button. In the **Group** tab, click the button that next to the **Line Color** as the screenshot shows to change the color of the 4 lines to **Black**. Click OK to apply these settings.



8. Then customize the 4 scatter plots. Select the first Scatter plot, in the **Symbol** tab, change the **Size** to 8.
   - In the **Group** tab, we will mainly customize the symbols in the list box that in the middle of the tab. In the **Symbol Type** row, select **By one** in the **Increment** column. Click the ... button to open the **Increment Editor** dialog, in the dialog select **UpTriangle**, **Circle**,**Hexagon** and **Square** for the first 4 rows.
   - In the **Symbol Edge Color** row, select **By one** in the **Increment** column. Click the ... button to open the **Increment Editor** dialog, in the dialog select **Green**, **Blue**,**Blue** and **Red** for the first 4 rows.
   - In the **Symbol Interior** row, select **By one** in the **Increment** column. Click the ... button to open the **Increment Editor** dialog, in the dialog select **Solid**, **Solid**,**Half Left** and **Solid** for the first 4 rows.

Click OK button to close the Plot Details dialog, then the graph will look like.



9. Then customize the Axes of the graph. Double click on the X axis, in the **Scale** tab, change **From** to **-5**, **To** to **690**. Then click **Vertical** icon in the **Selection** box, change **From** to 0.61 and **To** to **1.01**. In the **Title&Format** tab, select **Top** in the Selection box, enable the **Show Axis&Tick** check box, and select **In** from both **Major Ticks** and **Minor Ticks** drop-down lists. Then select **Right** from the **Selection** box, and do the same thing as we did for the Top.

10. Then we will customize the titles and the legend. Change the titles as the following images shows. Right click on the legend and select **Properties** from the context menu, and input the following strings into it.

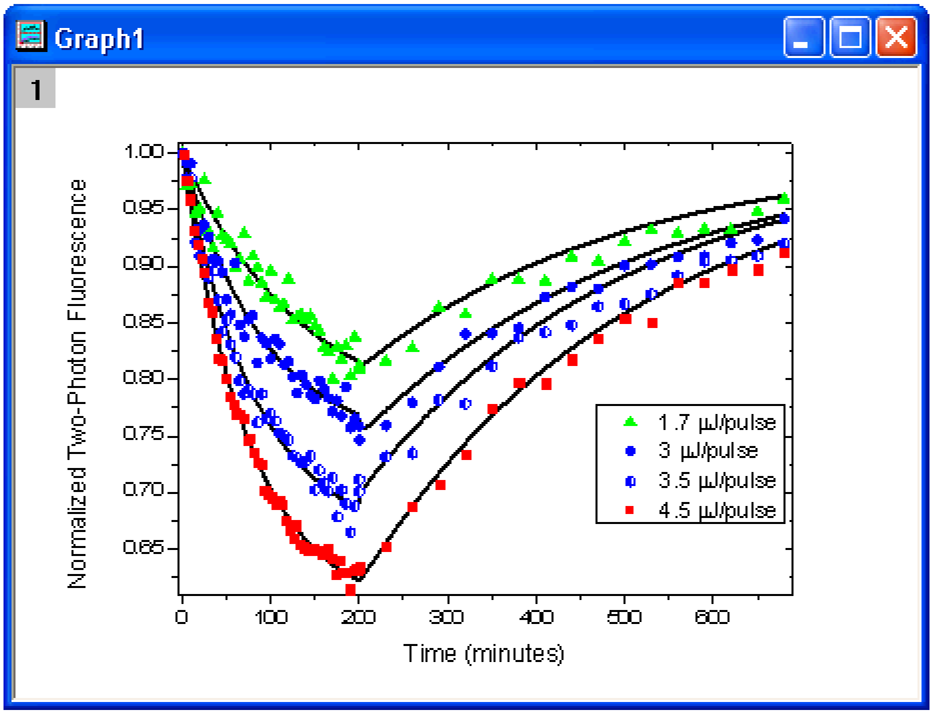
\l(5) 1.7 \g(m)J/pulse

\l(6) 3 \g(m)J/pulse

\l(7) 3.5 \g(m)J/pulse

\l(8) 4.5 \g(m)J/pulse

The final graph will look like

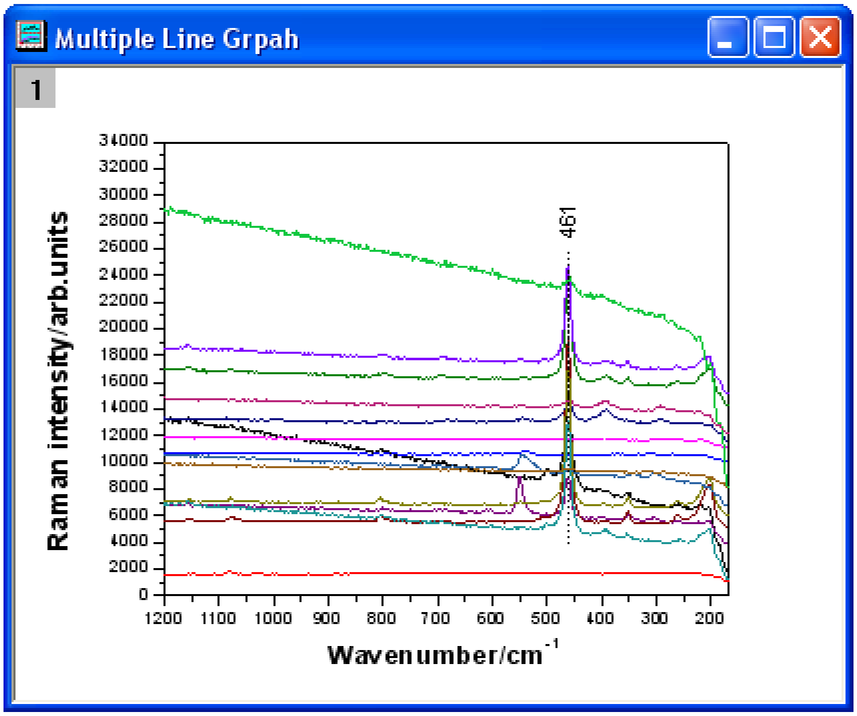


5.2.4 Micro-Raman spectroscopy of complex nanostructured mineral systems

Contents

Summary

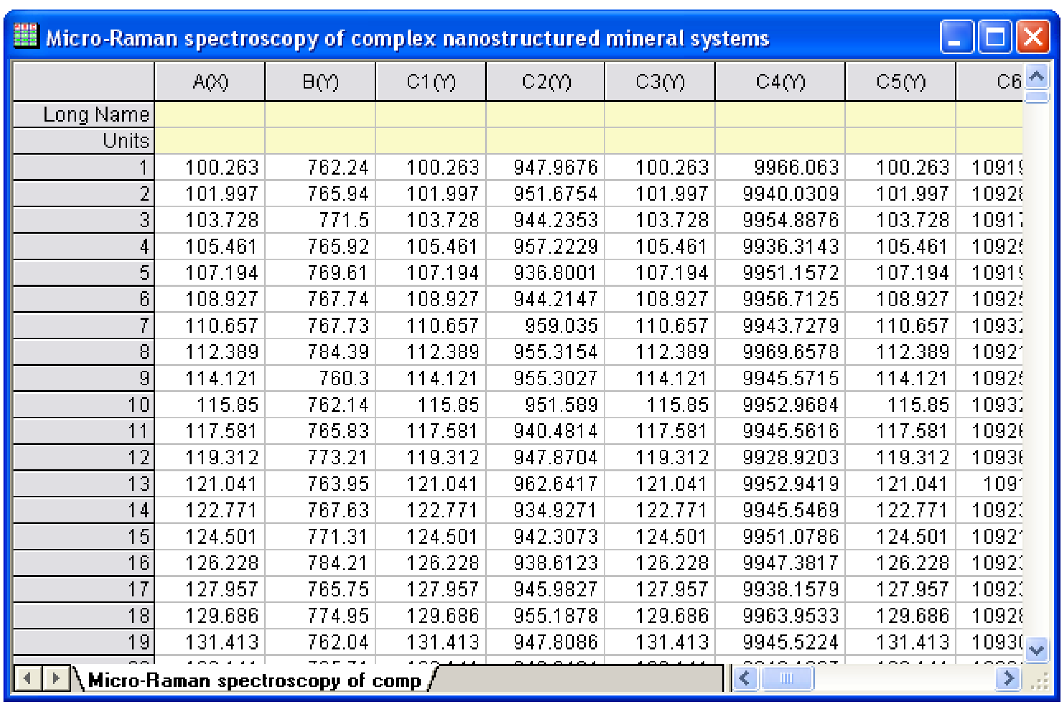
This tutorial will show you how to create a multiple line plot and how to customize it.



Minimum Origin Version Required: Origin 8.5.1

<u>Steps</u>

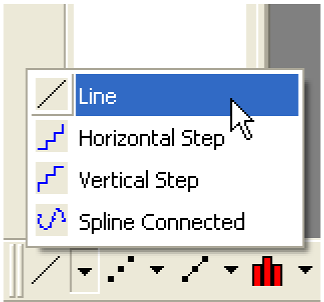
1. Create a new worksheet. Import the binned data.


Micro-Raman spectroscopy of complex nanostructured mineral systems

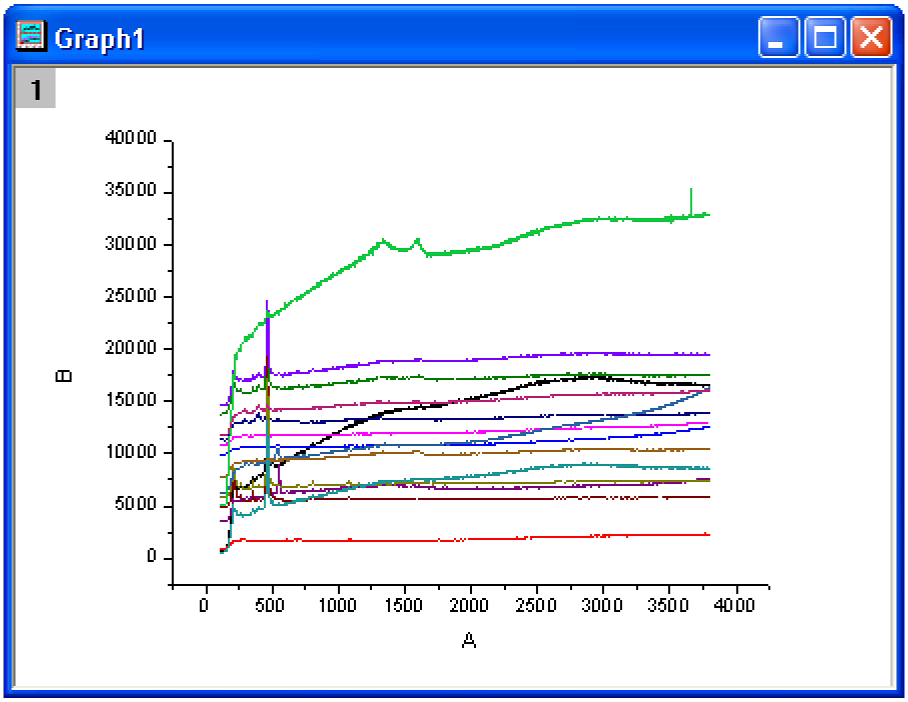
	A(X)	B(Y)	C1(Y)	C2(Y)	C3(Y)	C4(Y)	C5(Y)	C6
Long Name								
Units								
1	100.263	762.24	100.263	947.9676	100.263	9966.063	100.263	1091
2	101.997	765.94	101.997	951.6754	101.997	9940.0309	101.997	1092
3	103.728	771.5	103.728	944.2353	103.728	9954.8876	103.728	1091
4	105.461	765.92	105.461	957.2229	105.461	9936.3143	105.461	1092
5	107.194	769.61	107.194	936.8001	107.194	9951.1572	107.194	1091
6	108.927	767.74	108.927	944.2147	108.927	9956.7125	108.927	1092
7	110.657	767.73	110.657	959.035	110.657	9943.7279	110.657	1093
8	112.389	784.39	112.389	955.3154	112.389	9969.6578	112.389	1092
9	114.121	760.3	114.121	955.3027	114.121	9945.5715	114.121	1092
10	115.85	762.14	115.85	951.589	115.85	9952.9684	115.85	1093
11	117.581	765.83	117.581	940.4814	117.581	9945.5616	117.581	1092
12	119.312	773.21	119.312	947.8704	119.312	9928.9203	119.312	1093
13	121.041	763.95	121.041	962.6417	121.041	9952.9419	121.041	109
14	122.771	767.63	122.771	934.9271	122.771	9945.5469	122.771	1092
15	124.501	771.31	124.501	942.3073	124.501	9951.0786	124.501	1092
16	126.228	784.21	126.228	938.6123	126.228	9947.3817	126.228	1092
17	127.957	765.75	127.957	945.9827	127.957	9938.1579	127.957	1092
18	129.686	774.95	129.686	955.1878	129.686	9963.9533	129.686	1092
19	131.413	762.04	131.413	947.8086	131.413	9945.5224	131.413	1093

Micro-Raman spectroscopy of comp

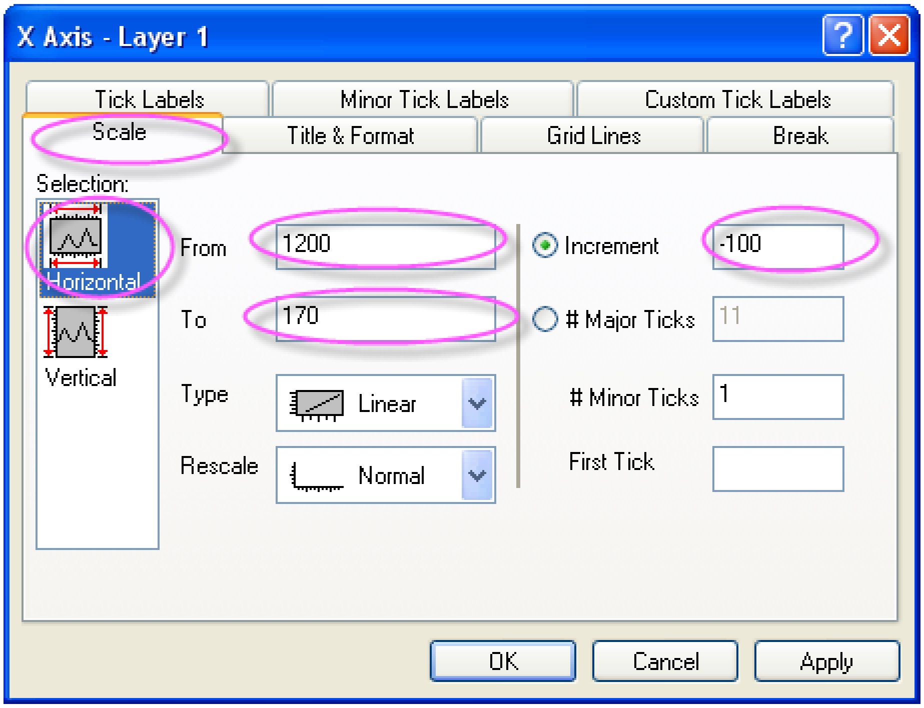
2. Highlight all the columns in the worksheet. Right-click and select **Set As: XY XY** from the context menu. Then click the **Line** button on the **2D Graphs** toolbar.



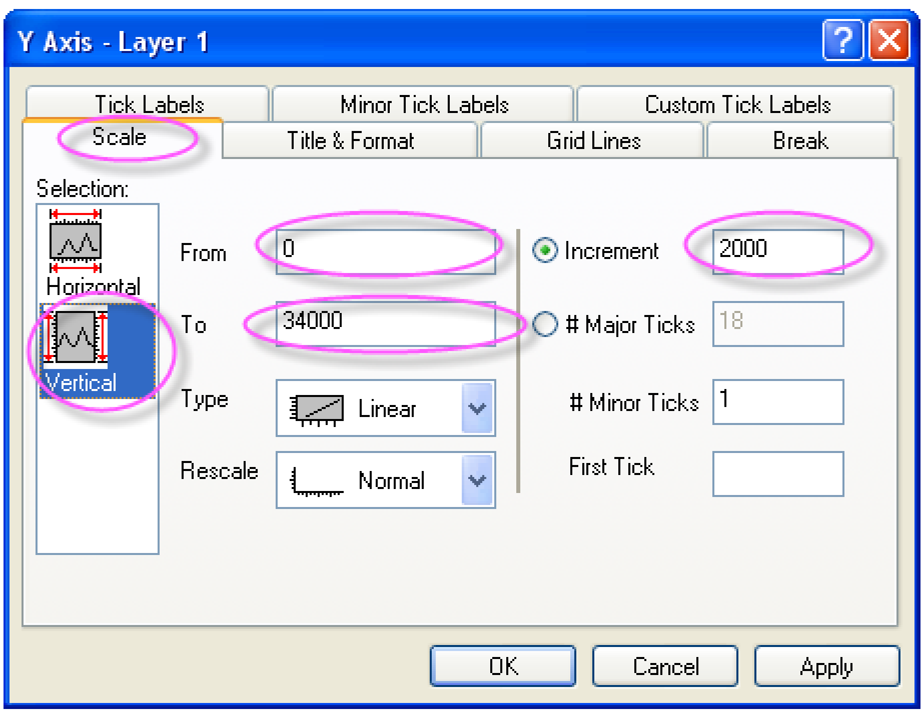
Delete the legend. The graph should look like:



3. Double-click on the **X** axis to bring up the **X Axis** dialog box. In the **Scale** tab, select **Horizontal** in the **Selection** list box. Then set the options as the screenshots below.

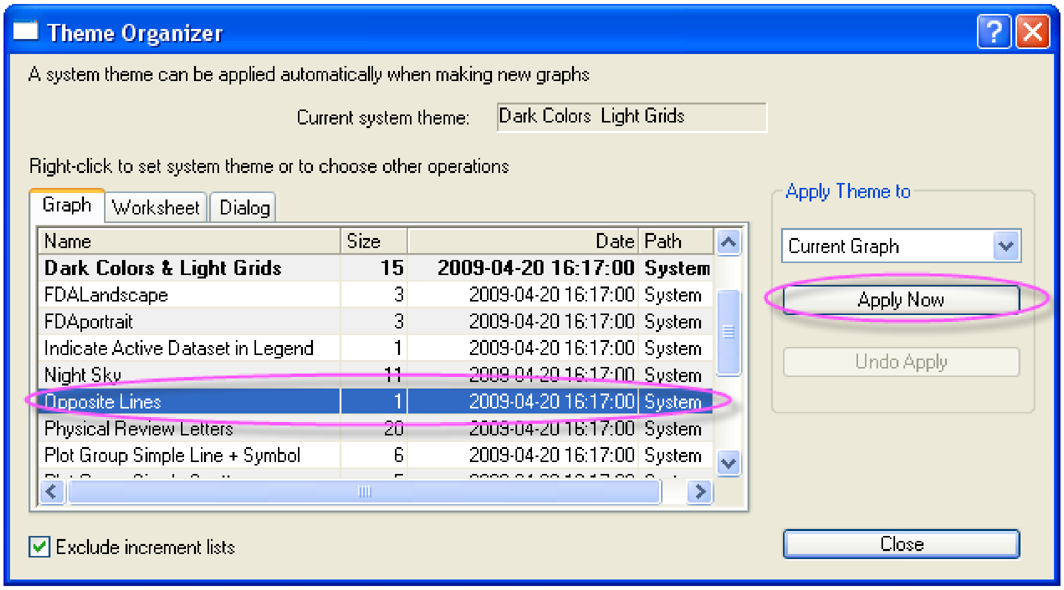


Select **Vertical** in the **Selection** list box. Then set the options as the screenshots below.

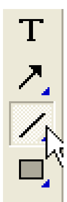


Click **OK**.

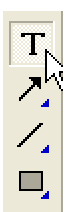
4. Then we apply a graph theme to add a top **X** axis and a right **Y** axis. Select **Tool: Theme Organizer** to open the **Theme Organizer** dialog. Activate the **Graph** tab and select **Opposite Lines** from the table. Then click the **Apply Now** button. Click the **Close** button to close the dialog.



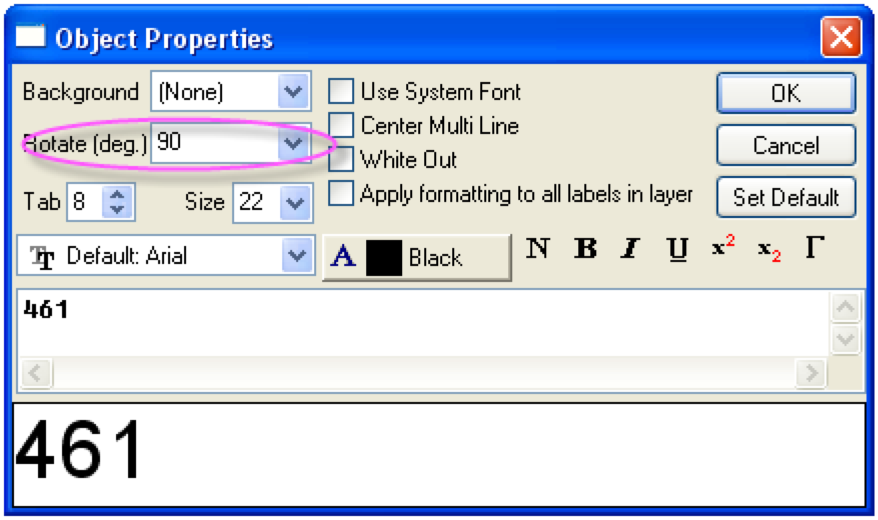
5. Click the **Line** button on the **Tools** toolbar and draw a line across the peaks' centers as the sample image shows. Please hold down the **SHIFT** key while drawing the line to force it to be a vertical line. Double-click on the line. In the **Line** tab, select **Dash** with the **Type** drop-down list. Click **OK**.



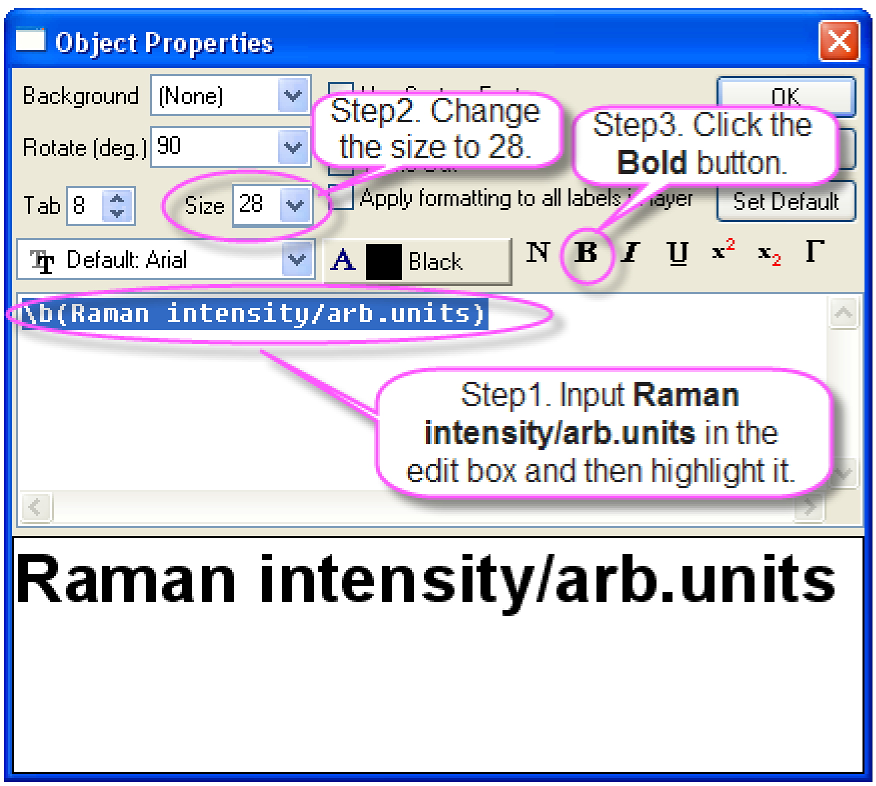
6. Click the **Text** button in the **Tools** toolbar. Add a text object near the line object and enter **461** in it.



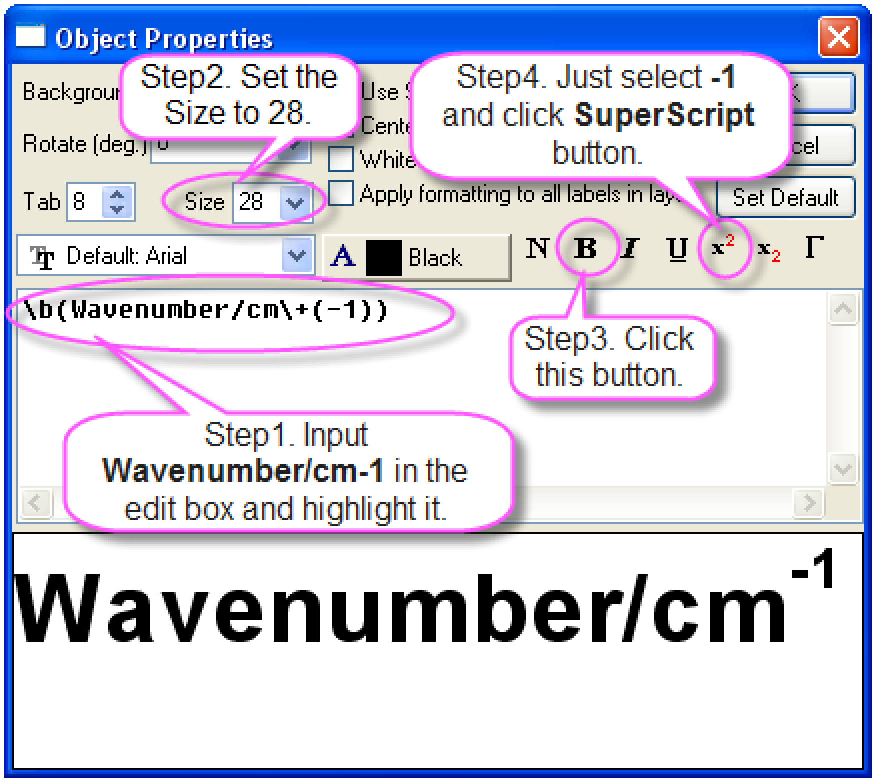
Right-click on the label and select **Properties** from the short-cut menu. Then set the dialog options as the screenshot below. Click **OK**.



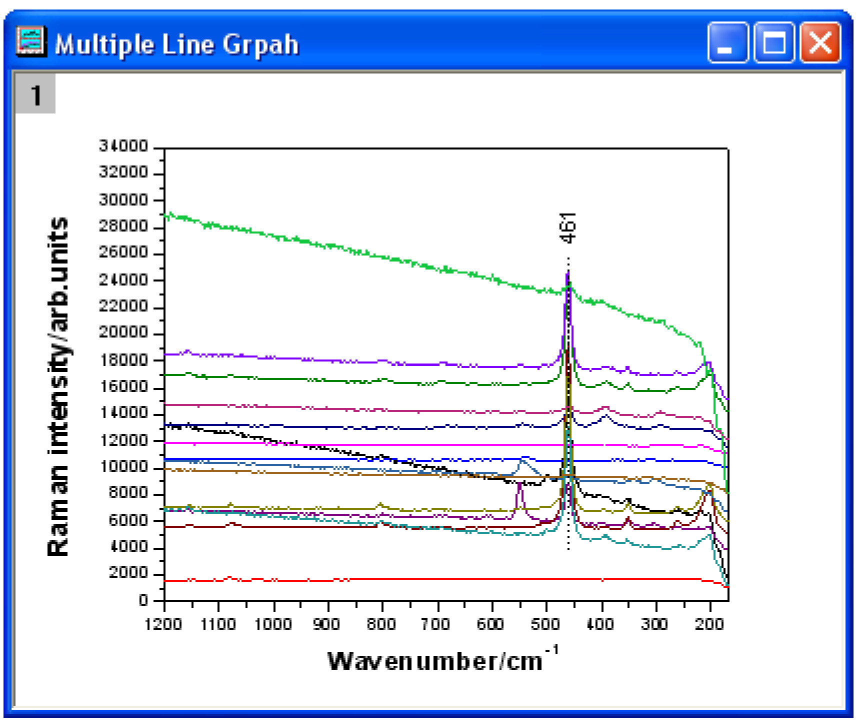
7. Right-click on the title of the Y axis and select **Properties** from the short-cut menu. Then set the dialog options as the screenshot below. Click **OK**.



8. Right-click on the title of the X axis and select **Properties** from the short-cut menu. Then set the dialog options as the screenshot below. Click **OK**.



The graph should look like



Sample Data

Download the **Micro_Raman_Spectroscopy.txt** file from http://www.originlab.com/ftp/graph_gallery/data/Micro_Raman_Spectroscopy.txt. Click the **Import Single ASCII button**  and select the file to import it into Origin.

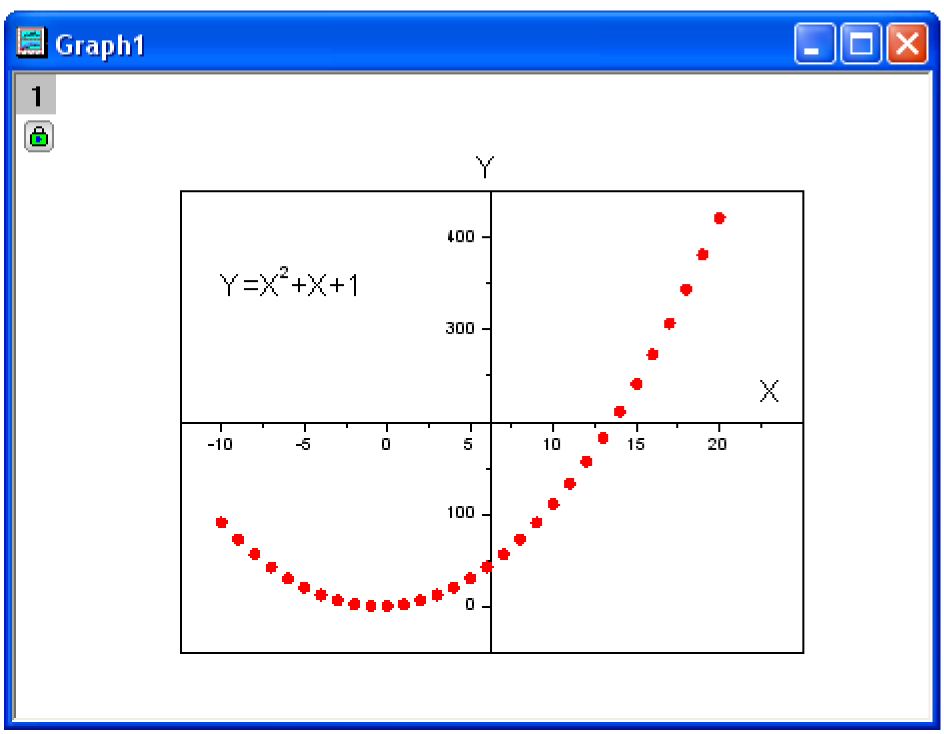
The following table contains part of the sample data.

A(X1)	B(Y1)	A(X2)	B(Y2)	A(X3)	B(Y3)
100.263	762.24	100.263	947.9676	100.263	9966.063
101.997	765.94	101.997	951.6754	101.997	9940.0309
103.728	771.5	103.728	944.2353	103.728	9954.8876
105.461	765.92	105.461	957.2229	105.461	9936.3143
107.194	769.61	107.194	936.8001	107.194	9951.1572
108.927	767.74	108.927	944.2147	108.927	9956.7125
110.657	767.73	110.657	959.035	110.657	9943.7279
112.389	784.39	112.389	955.3154	112.389	9969.6578
114.121	760.3	114.121	955.3027	114.121	9945.5715
115.85	762.14	115.85	951.589	115.85	9952.9684

5.2.5 Scatter Central Plot

<u>Summary</u>

Scatter central plot is a graph with the X and Y axes located in the middle of the layer. In this tutorial, a scatter central plot will be created, then the symbol and the axes will be customized.



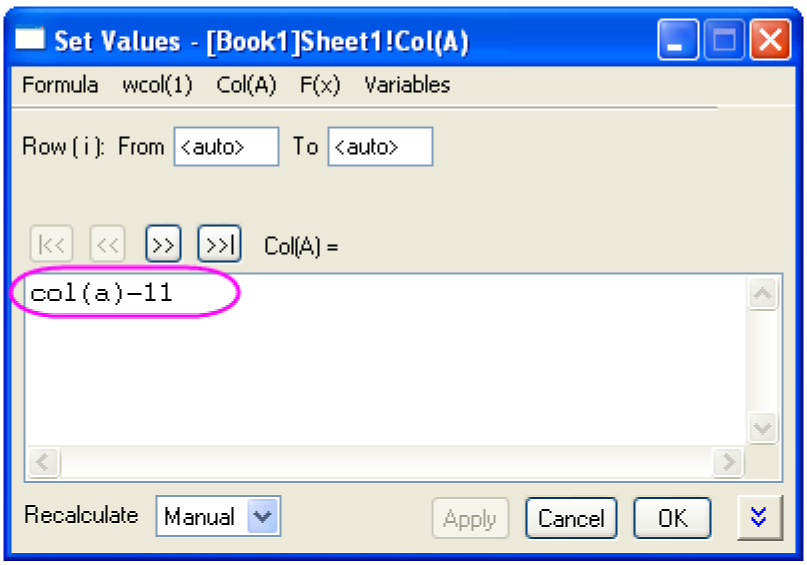
Minimum Origin Version Required: Origin 8.1 SR0

What you will learn

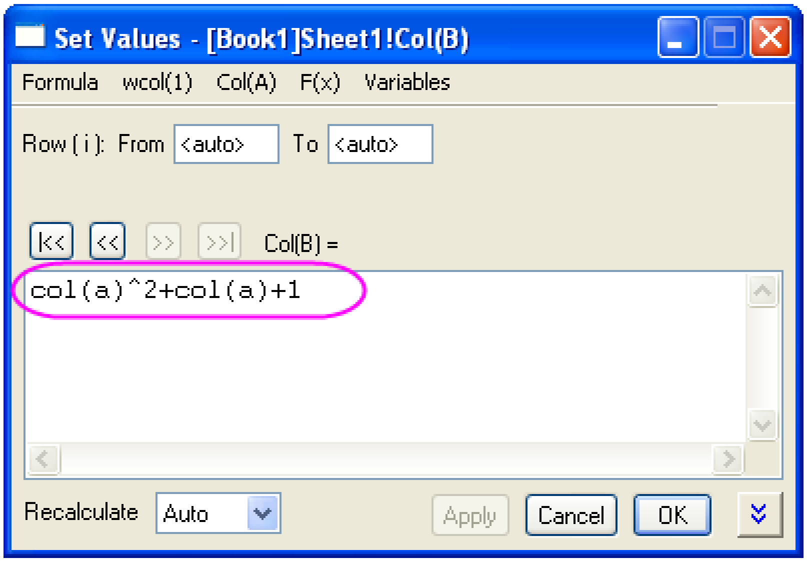
- Set column values by using **Set Values** dialog
- Create a scatter central plot
- Change the color and shape of the scatter points
- Customize the tick label of axis

Steps

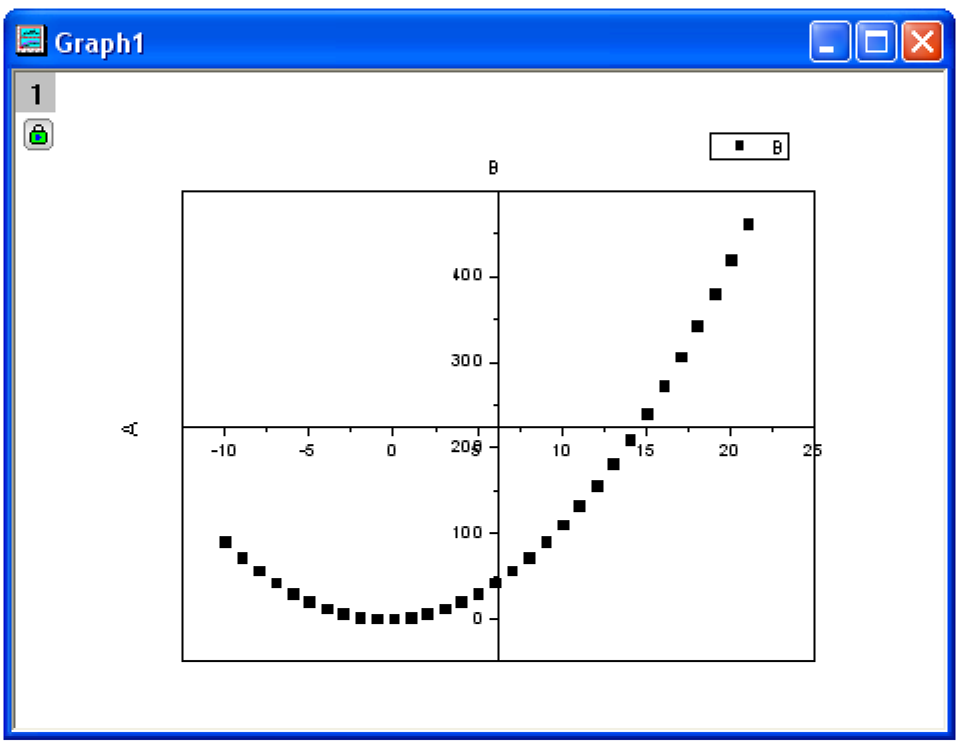
1. Set column values by using **Set Values** dialog. Highlight column A, right-click and select **Fill Column with: Row Numbers** from the context menu. Right-click column A again and select **Set Column Values** to open the **Set Values** dialog. Type **Col(a) - 11** in the middle text box and then click the **OK** button to finish setting values for column A.



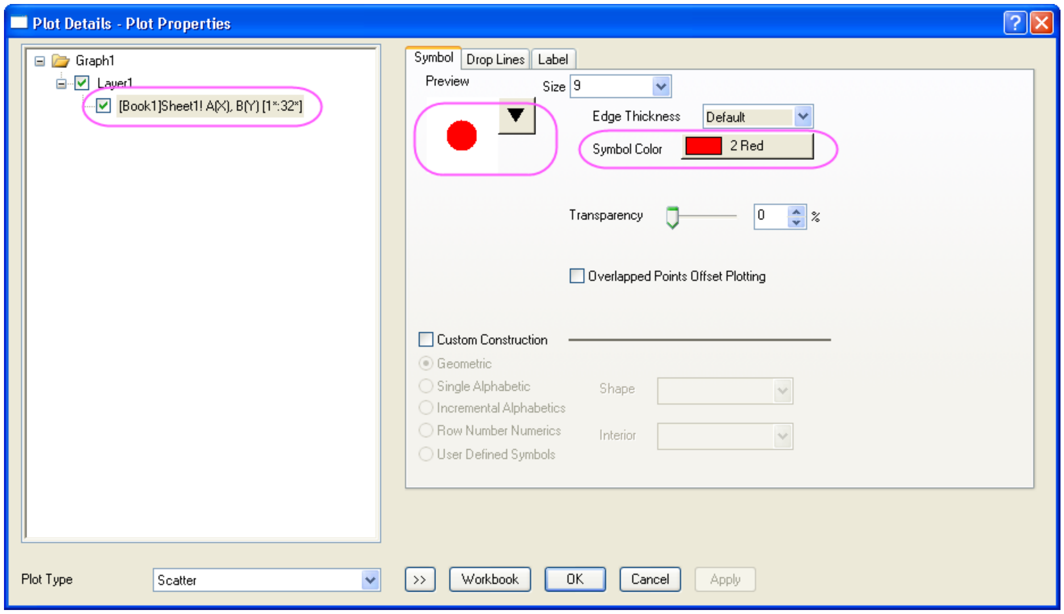
2. Do the same thing for column B, highlight it and select **Set Column Values** dialog from the right-clicking menu to open the **Set Values** dialog again. Type **Col(a)^2 + Col(a) + 1** in the middle text box this time. Change the **Recalculate** to **Auto**, then click the **OK** button to finish setting values for column B.



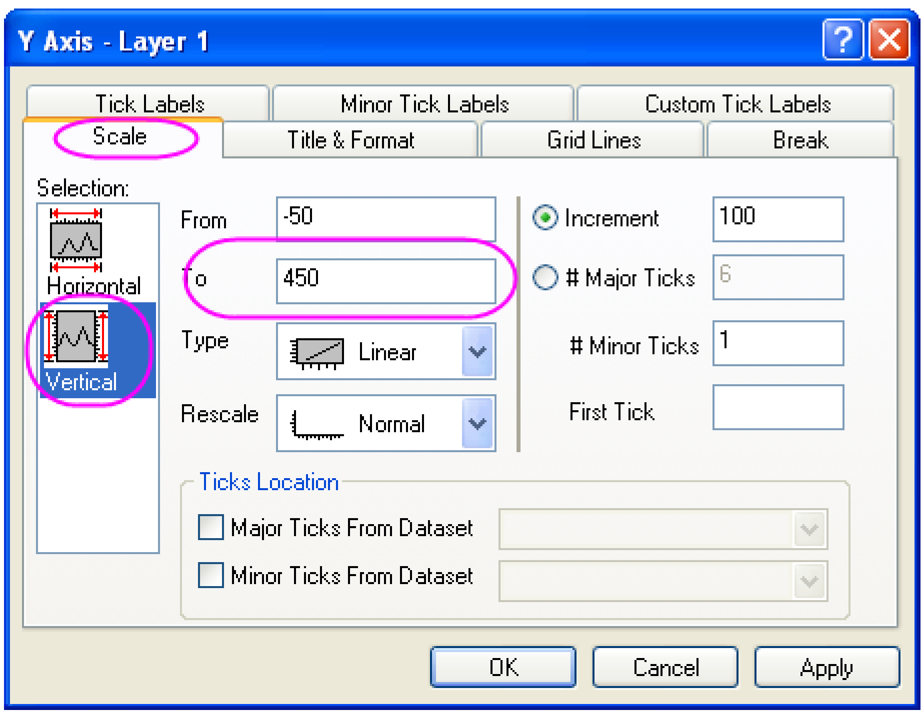
3. Create a scatter central plot. highlight column B and select **Plot: Symbol: Scatter Central** from the Origin main menu to create a scatter central plot.



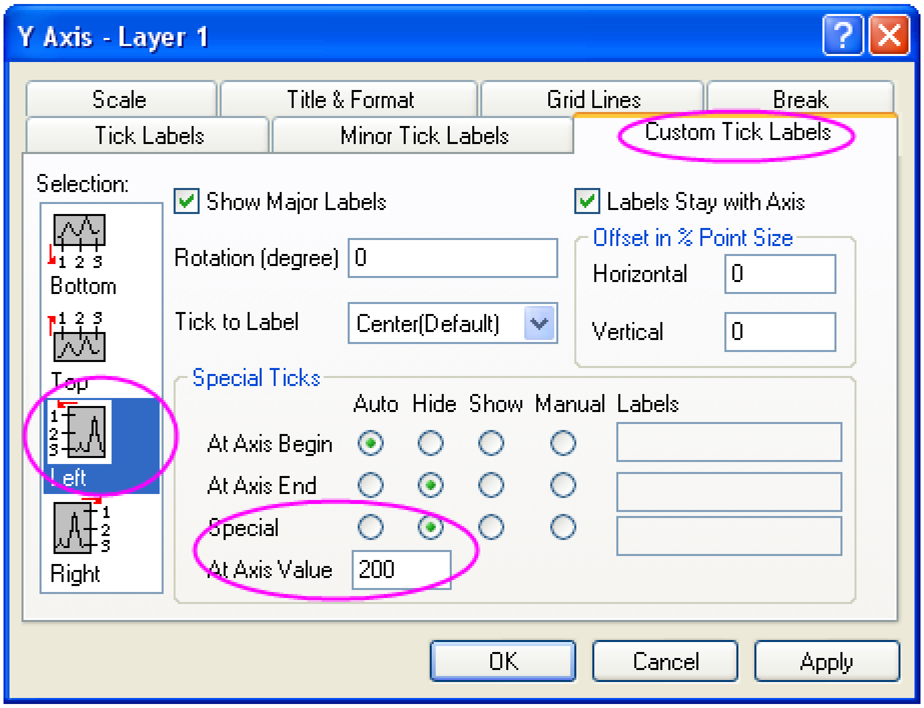
4. Change the color and shape of the scatter points. Double click on the scatter plot to open the **Plot Details** dialog. In the **Symbol** tab of right panel, click the button beside **Preview** and choose the solid circle symbol. And then click the button beside **Symbol Color** and choose **Individual Color: Red**. Click the **OK** button to close the **Plot Details** dialog.



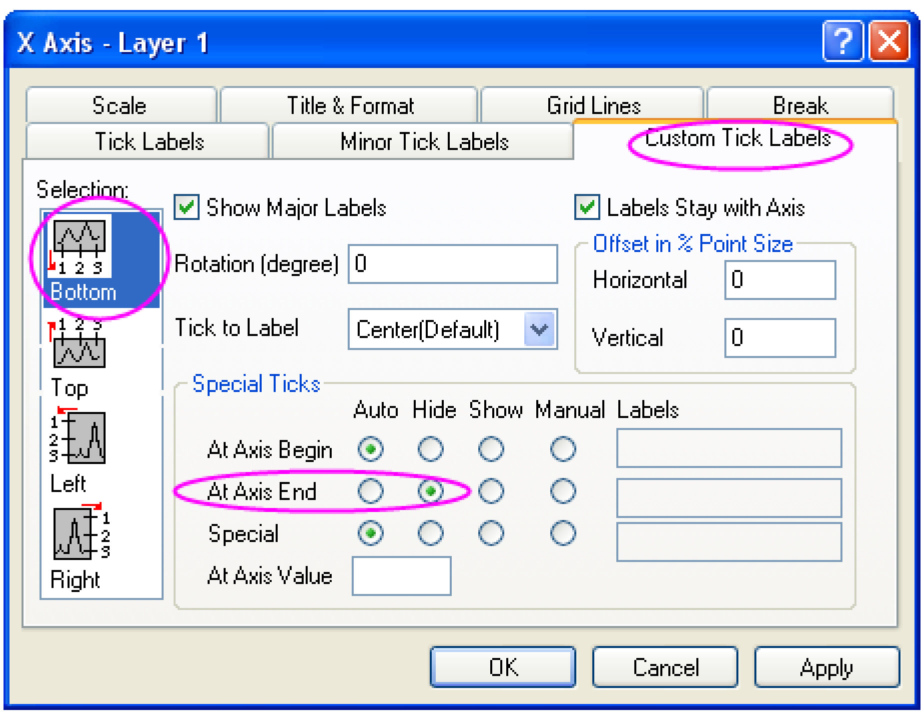
5. Customize the tick label of axis. Double click on the vertical axis to open the dialog for axes settings. Change the **To** value to 450.



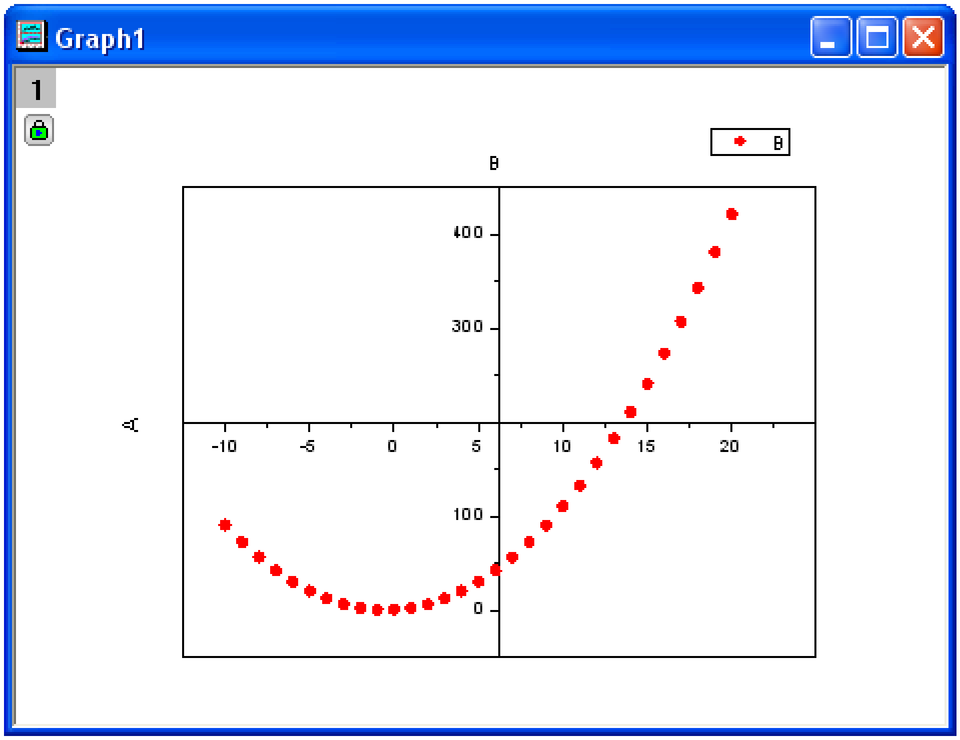
6. Switch to the **Custom Tick Labels** tab, select the **Hide** radio on the **Special** row and then type **200** in the **At Axis Value** box.



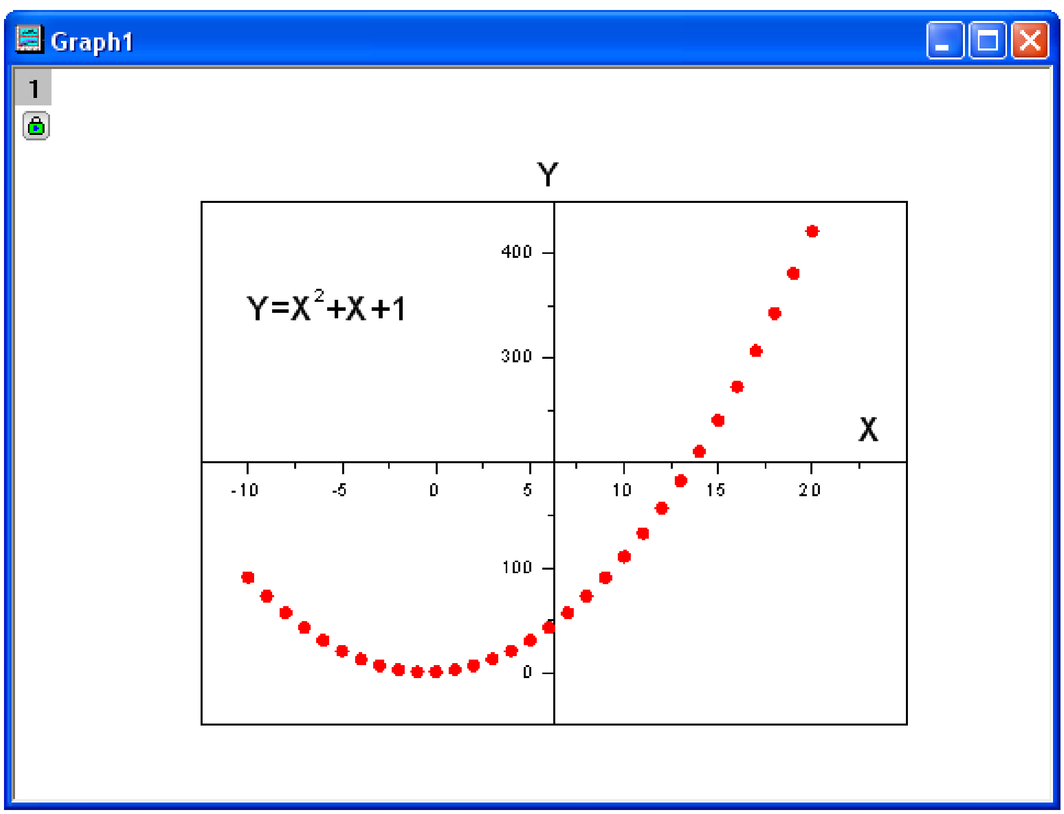
7. Click the **Bottom** option in the **Selection** panel, and then select the **Hide** radio on the **At Axis End** row.



8. Click the **OK** button to finish axes settings and the scatter central graph looks like below.



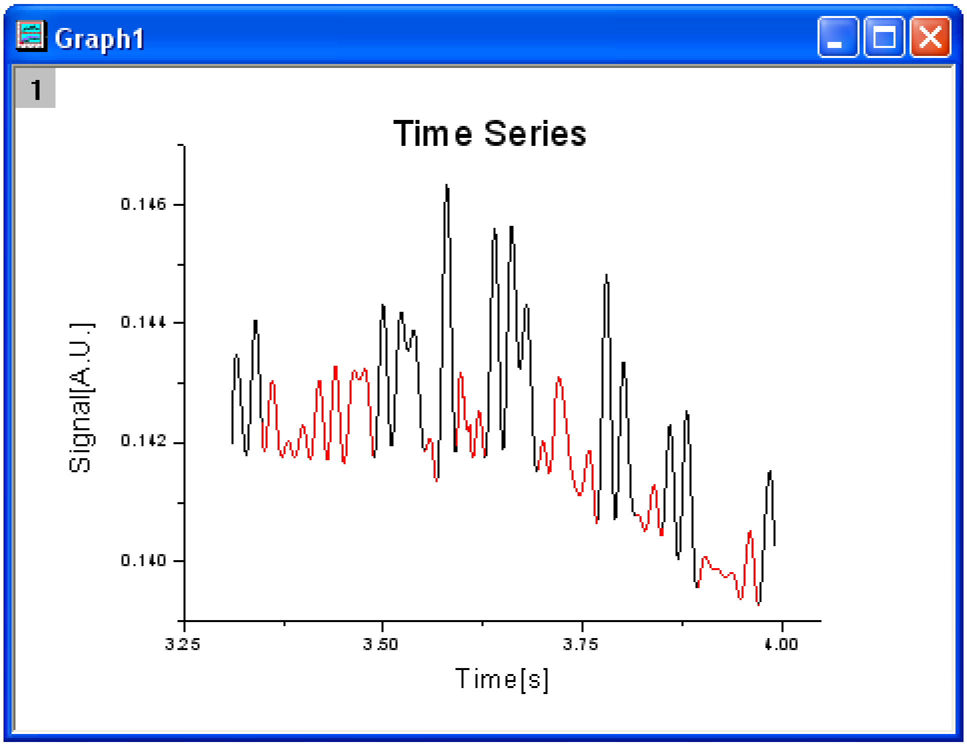
9. Now, delete the legend and change the label of axes, **A** to **X** and **B** to **Y**. Move the **X** to the end of X axis. Add a text object to the graph, the text says $\boldsymbol{Y = X^2 + X + 1}$. The final graph looks like below.



5.2.6 Line Graph with Masked Data

Summary

In a line graph, masking data is supported. In this tutorial, a line graph will be created first, then the masking tool is used to mask the low peaks on the graph.



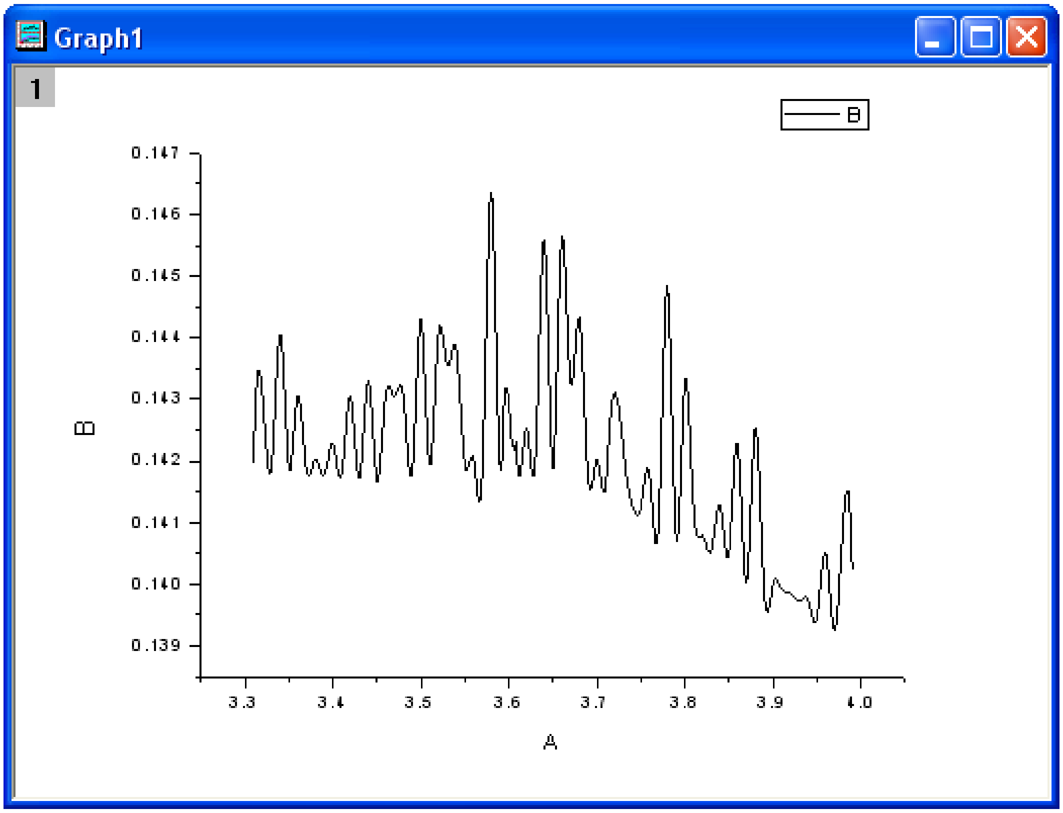
Minimum Origin Version Required: Origin 8.1 SR0

What you will learn

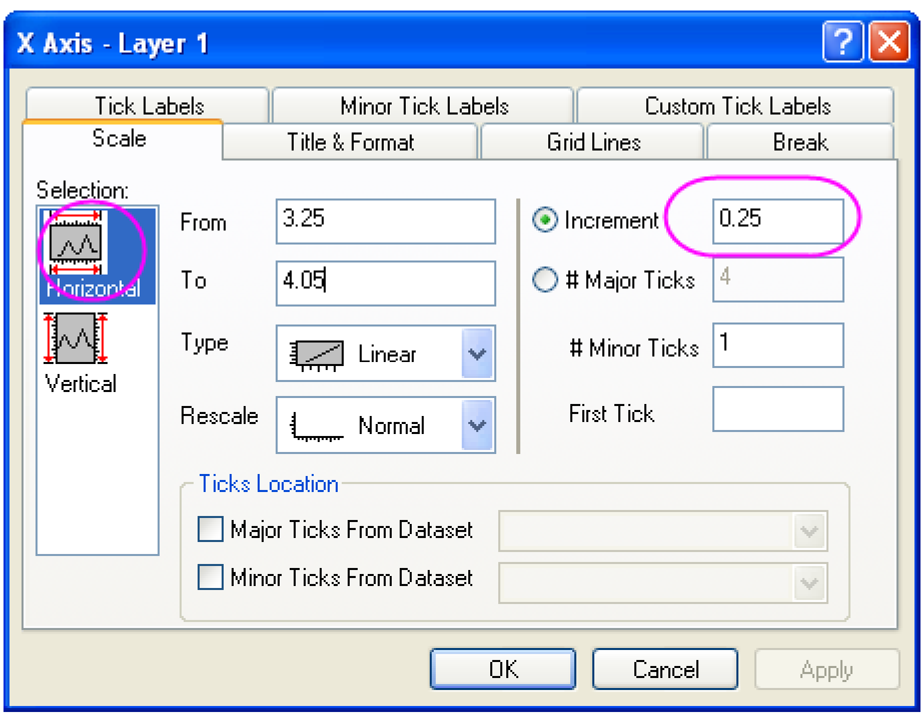
- Plot a line graph
- Set the scales of axes
- Mask data on the line graph

Steps

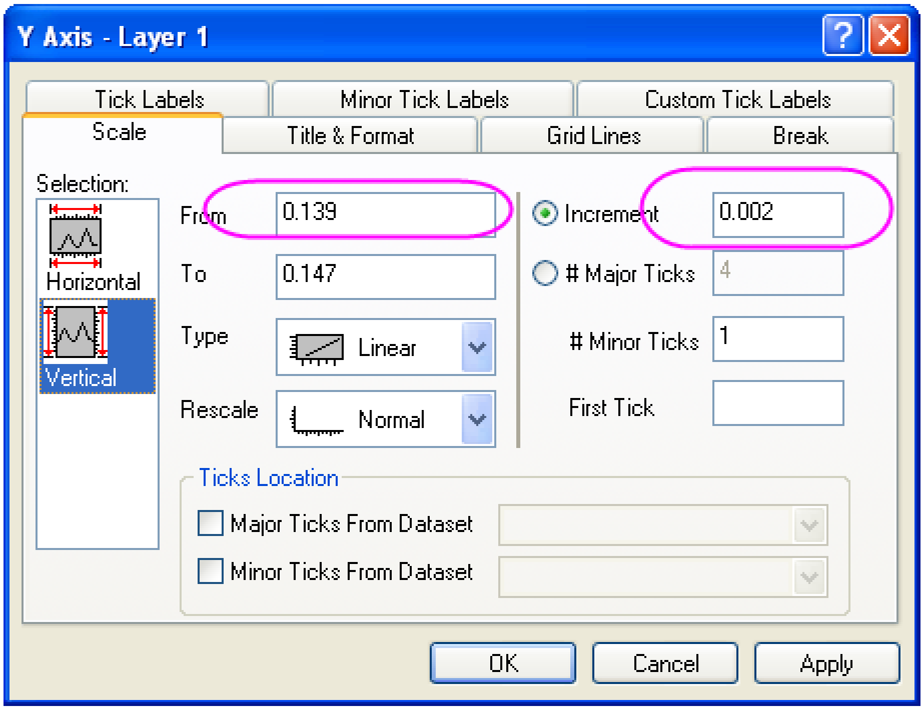
1. To start this tutorial, please download the data file from ftp.
2. Click the **Import Single ASCII**  button to open the file browser, then select the file just downloaded and keep the default settings to import this file.
3. Highlight both columns in the worksheet and select **Plot: Line: Line** from the Origin main menu to plot a line graph.



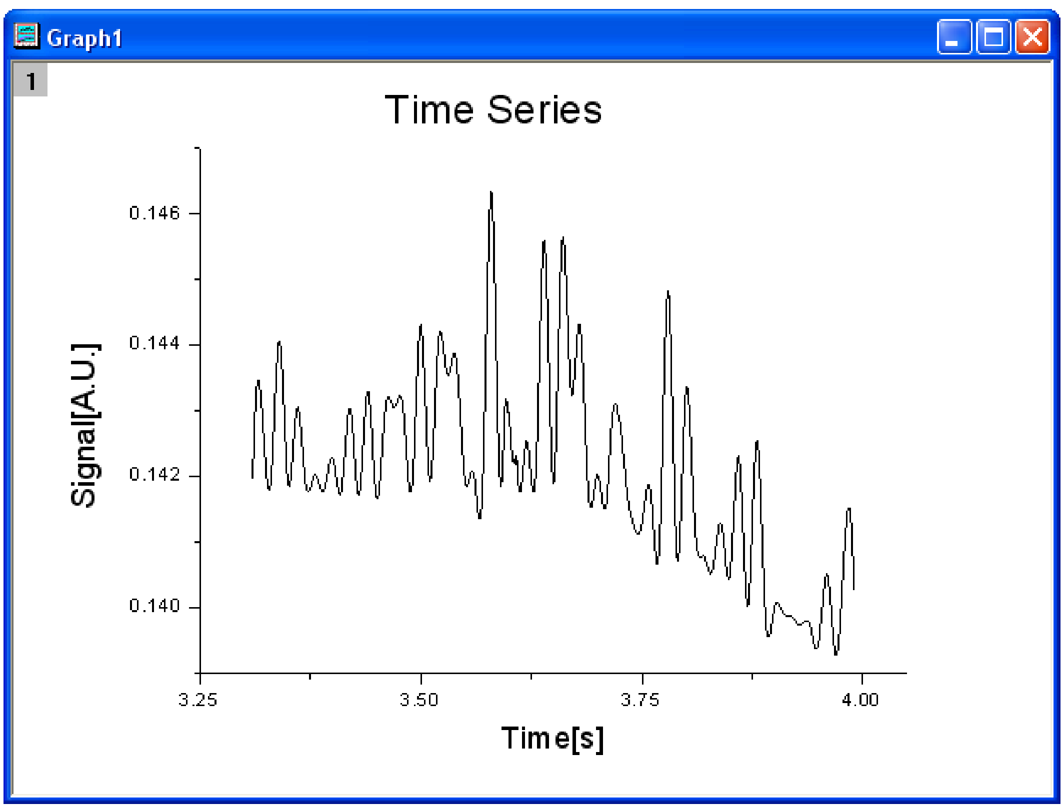
4. Double click on the X axis to open the axes setting dialog, then change the **Increment** to 0.25.



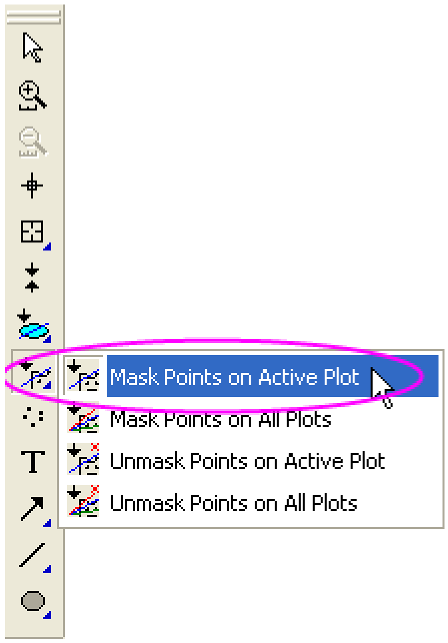
5. Select the **Vertical** in the **Selection** panel, then change the **From** to 0.139 and the **Increment** to 0.002. Click the **OK** button to finish axes settings.



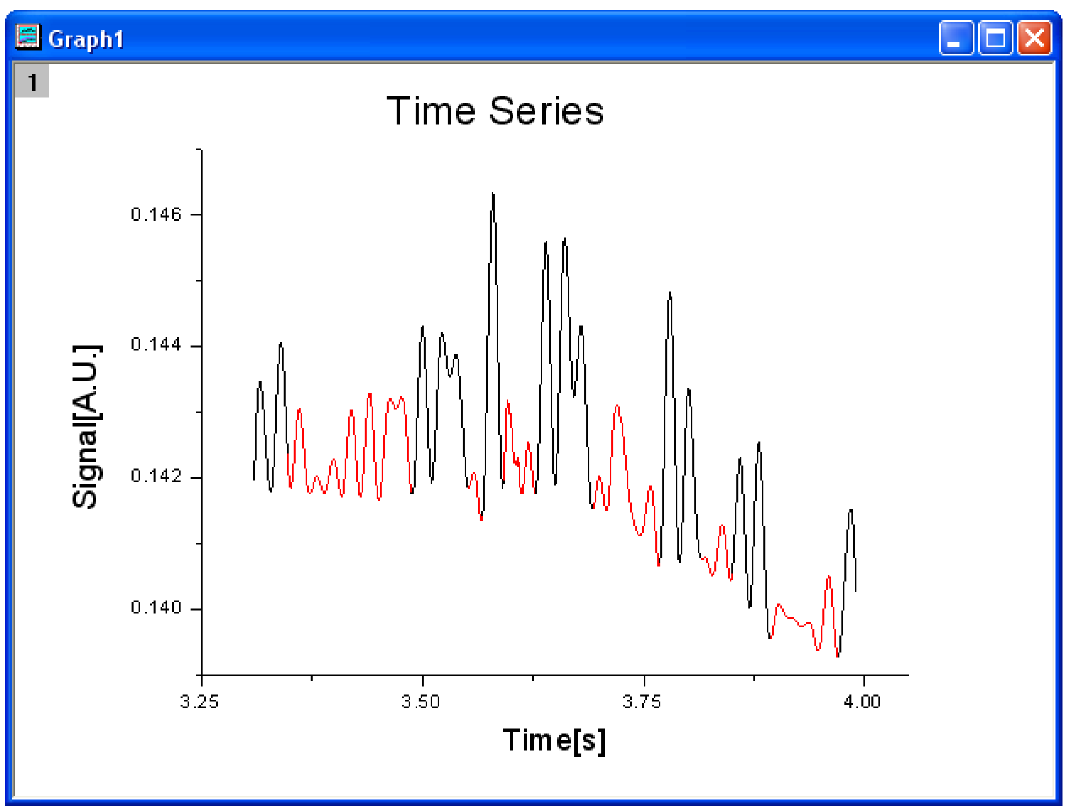
6. Delete the legend on the graph. Then change the X label to **Time[s]** and the Y label to **Signal[A.U.]**. And then add a text object as the title, say **Time Series**. Set font size of both axes labels to 28 and the title to 36.



7. Activate this graph, then click down on the **Regional Mask Tool** button on the Tool toolbar until a context menu appears. Then select **Mask Points on Active Plot** from this menu to get in the masking mode.



8. Click the proper place on the graph and drag a regional to select the low peaks, at the same time, mask them. Repeat it until all low peaks are masked. Then click the **Pointer** button on the Tool toolbar to end the masking mode.



5.2.7 Mark out a segment of plot with different plot style

Summary

In Origin, you can mark out a segment of plot with different plot style, such as a segment of dashed in a solid line plot.

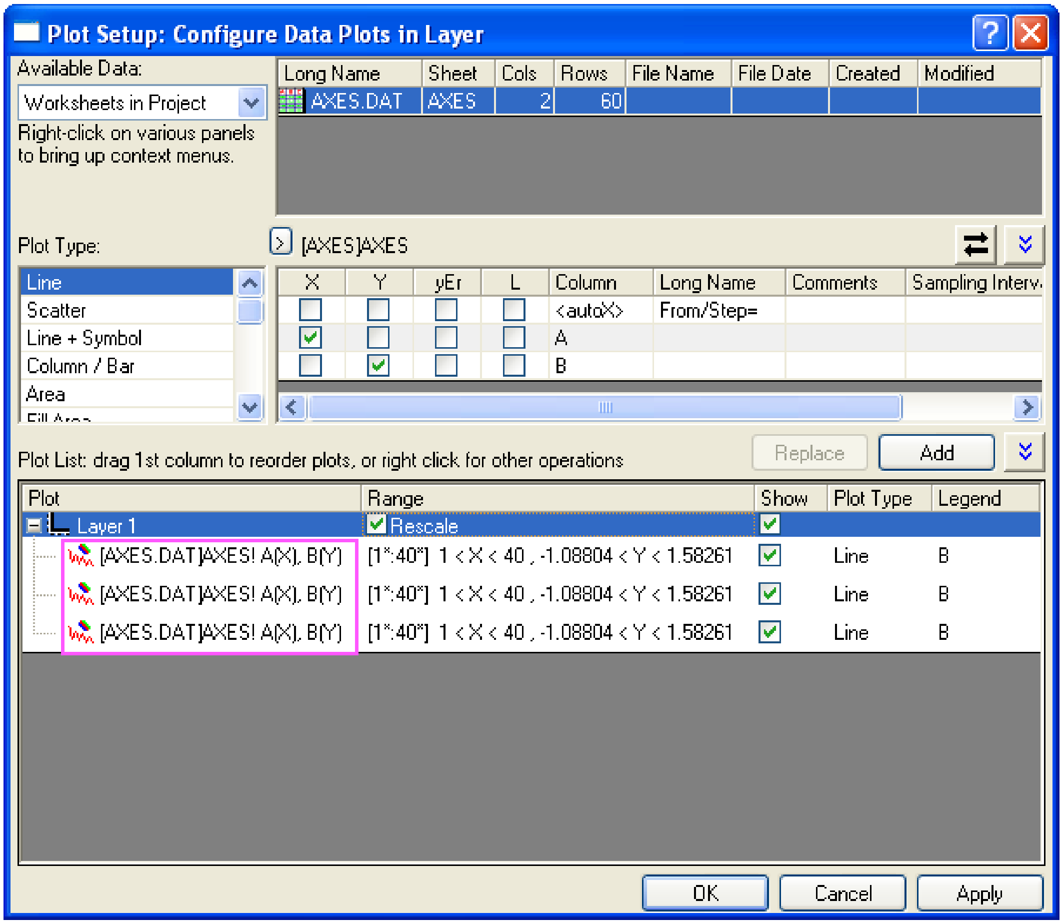
Minimum Origin Version Required: Origin 8.0 SR6

What you will learn

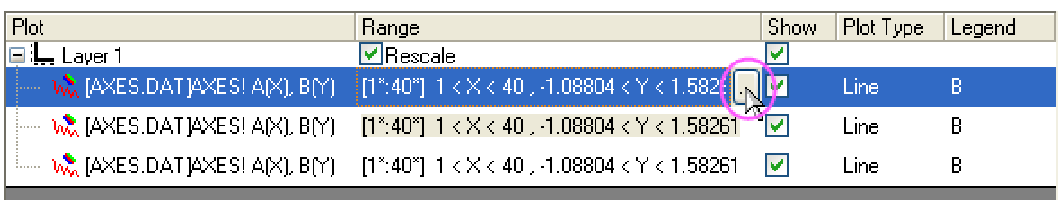
- Use the **Plot Setup** dialog to create a graph
- Mark out the special segment of a plot

Steps

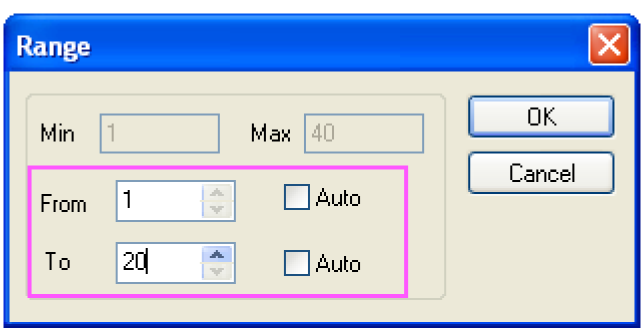
1. Start with an empty worksheet. Select **File: Import: Single ASCII...** from the Origin menu to open the **Import Single ASCII** dialog. Browse to the *\Samples\Graphing* subfolder of the Origin program folder, and import the file *AXES.DAT* .
2. Click the  button on the **Standard** toolbar to create a new graph window and then select **Graph: Plot Setup** from the main menu to bring up the **Plot Setup** dialog.
3. Show all of the three panels of the **Plot Setup** dialog. Select the **AXES** worksheet in the top panel. Then go to the middle panel to select **A** as X and **B** as Y. After that, click **Add** to add this data plot to the bottom panel. Repeat this step three times. Three data plots should be listed in the bottom panel.



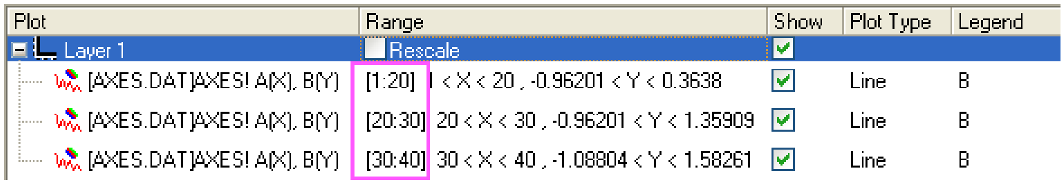
4. In the lower panel, click in the **Range** column that corresponds to the first data plot. The [...] button should be activated. Then click this button to open the **Range** dialog box.



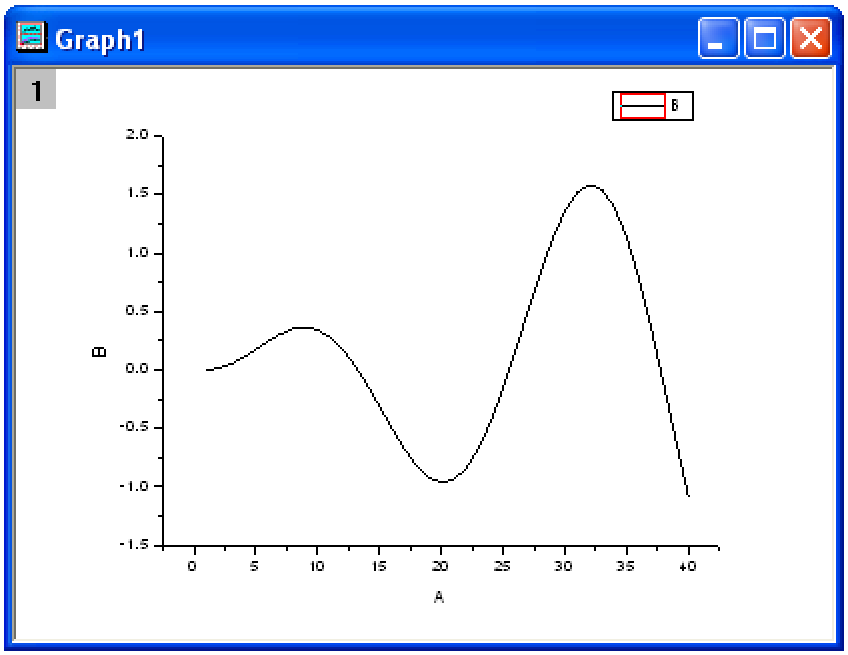
5. Clear the **Auto** check boxes (if they are selected) and then set **From** to **1** and **To** to **20**. Click **OK** to close the dialog.



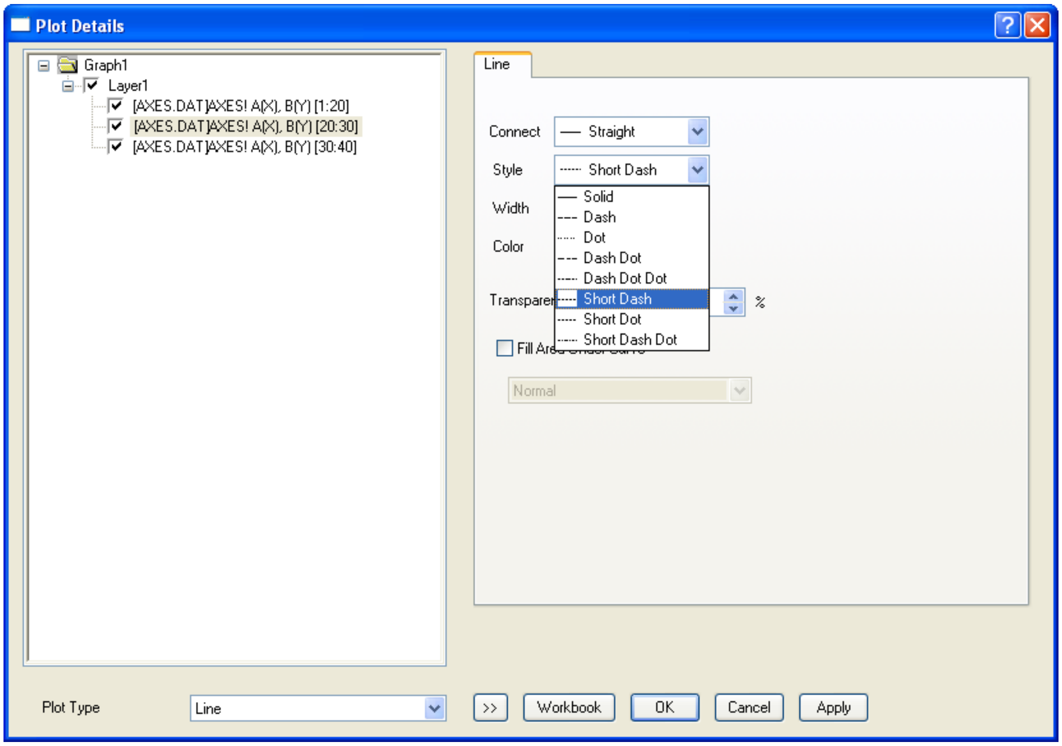
6. Similarly, set the ranges for other two data plots to "20 to 30" and "30 to 40" respectively.



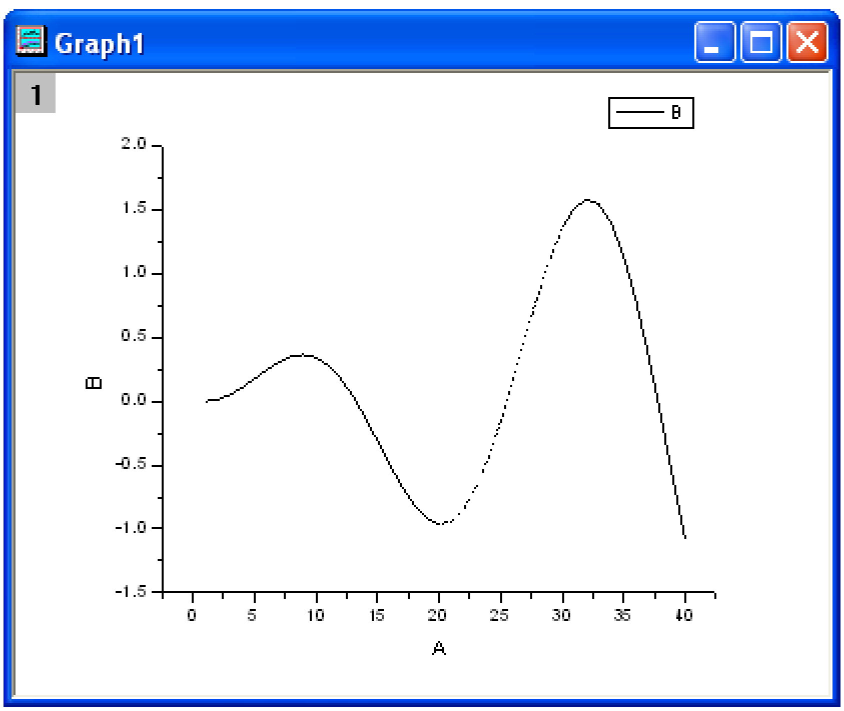
7. Click the **OK** button to close the **Plot Setup** dialog. You should get a graph like this:



8. Double-click on the curve in the graph window to open the **Plot Details** dialog. Select the second data plot from the left panel. In the right panel, change the **Style** to **Short Dash**, and then click the **Ok** button.



9. Finally, we get the plot with a range marked out.



5.2.8 Adding a Data Plot to Existing Graph

Summary

The Plot Setup dialog can be used to add/reorder/arrange data plots in an exiting graph. This dialog provides flexibility in selecting the desired data sheet and then selecting data to be plotted using column meta data such as long name.

Minimum Origin Version Required: Origin 8.0 SR6

What you will learn

This tutorial will show you how to:

- Use the Plot Setup Dialog Top Panel to find your dataset
- Add the chosen dataset to existing graph

Steps

Choosing the Data Source

1. Click the **New Project** button on the Standard toolbar, to begin with a new project.
2. Click the **Import Wizard** button on the Standard toolbar. The Import Wizard opens. (Note that if this is the first time that you have started the Import Wizard, you will experience a slight delay as Origin compiles the necessary files.)
3. Verify that the **ASCII** radio button is selected in the **Data Type** group.
4. Click the browse button to the right of the File text box. Navigate to the Origin folder; browse to the **Samples** folder and then the **Import and Export** folder.
5. Double-click to select S15-125-03.dat from the list of files. Repeat for files, S21-235-07.dat and S32-014-04.dat.
6. Click **OK**.
7. Leave the **Import Filters for current Data Type** as **Data Folder: VarsFromFileNameAndHeader**. (This filter has the settings to use when importing the file.)
8. Set the **Import Mode** as **Start New Sheets**.
9. Click the **Finish** button. The three data files import into the workbook, each as a new sheet. You will have a book with three sheets: **Trial Run 1**, **Trial Run 2**, and **Trial Run 3**.

Plotting the Data

1. Select **Trial Run 1** sheet.
2. Highlight the D(Y) column.
3. Click the **Line** button on the **2D Graphs** toolbar. A new graph is created.

Adding data to the graph

1. Double-click on the layer 1 icon in the upper-left hand corner of the graph. Click **Plot Setup** button in the opened **Layer Contents** dialog.
2. Select **Layer 1** in the Plot List.
3. Click the blue arrows in the upper right corner of the dialog to **Show Plot Designations**.
4. Again click the blue arrows in the upper right corner of the dialog to **Show Available Data**.
5. Select **Trial Run 2** from the Available Data list.
6. Check **Time** as X and **Position** as Y.

7. Click **Add**.
8. Check the **Rescale** checkbox.
9. Click **OK**.

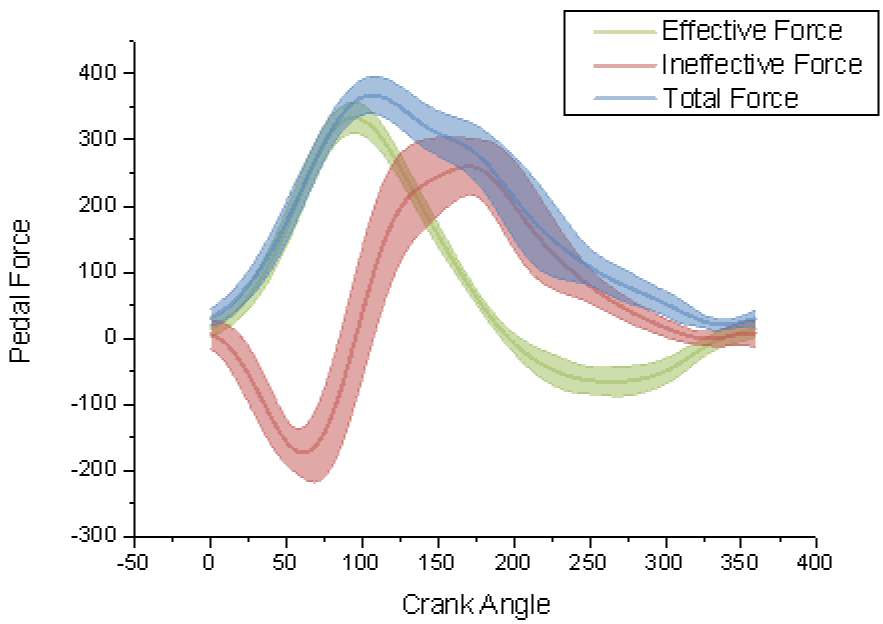
Updating the Legend and Formatting the Plot

1. Select **Graph:New Legend**.
2. Double-click on the line symbol for the second data plot in the legend. The Plot Details dialog opens.
3. Change the **Color** from Black to Red.
4. Click **OK**.

5.2.9 Error Bars with Fill Area

Summary

This graph displays three datasets with error bars. The error bars have been set to draw as lines with fill areas. Transparency has been set for all three curves so that data in overlapping regions can be clearly seen.



Minimum Origin Version Required: Origin 8.5.1 SR0

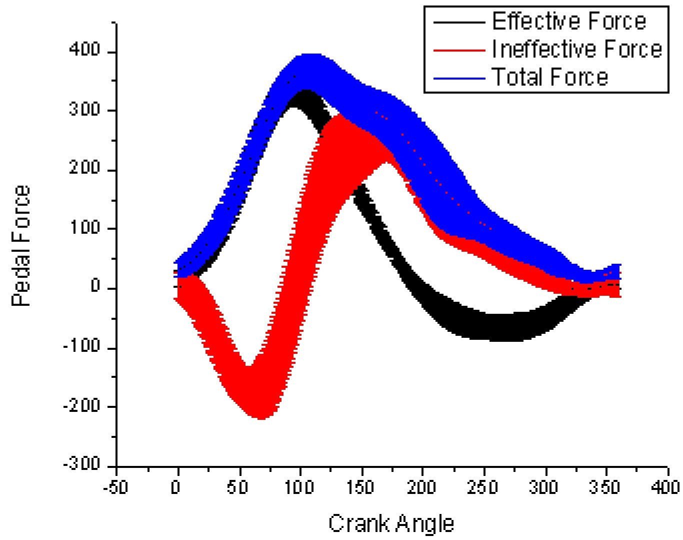
What you will learn

- Plot the error bar with fill area
- Set transparency for error bar
- Set and save custom color

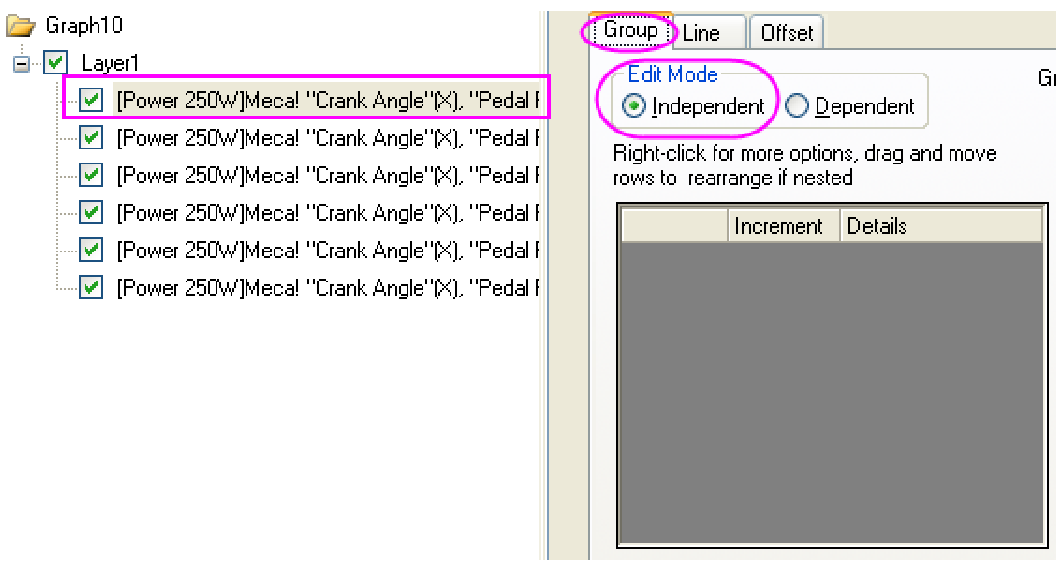
Steps

This tutorial is associated with the *2D and Contour Graphs: Line and Symbol: Error Bars with Fill Area* folder in the 2D and Contour Graphs project(\Samples\2D and Contour Graphs.opj).

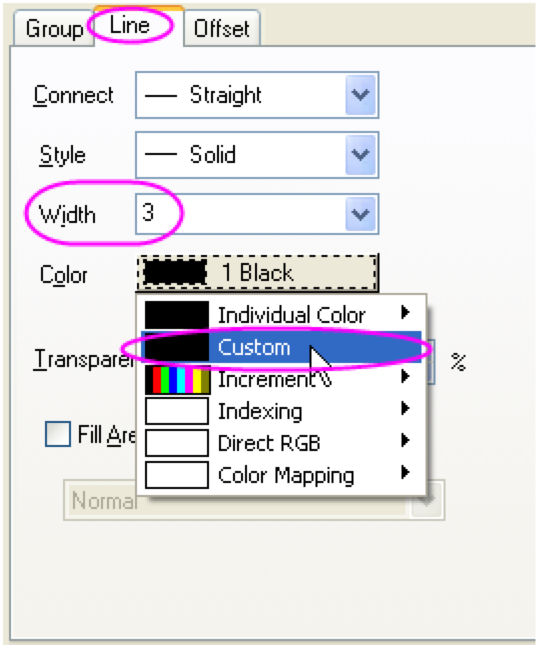
1. Highlight all columns in the worksheet. In the main menu, click **Plot: Line**, and then click **Line**.



2. Double-click on the plot to open the **Plot Details** dialog. Select the first plot node under Layer1 in the left panel. In the right panel activate the **Group** tab. Choose **Independent** in **Edit Mode**.

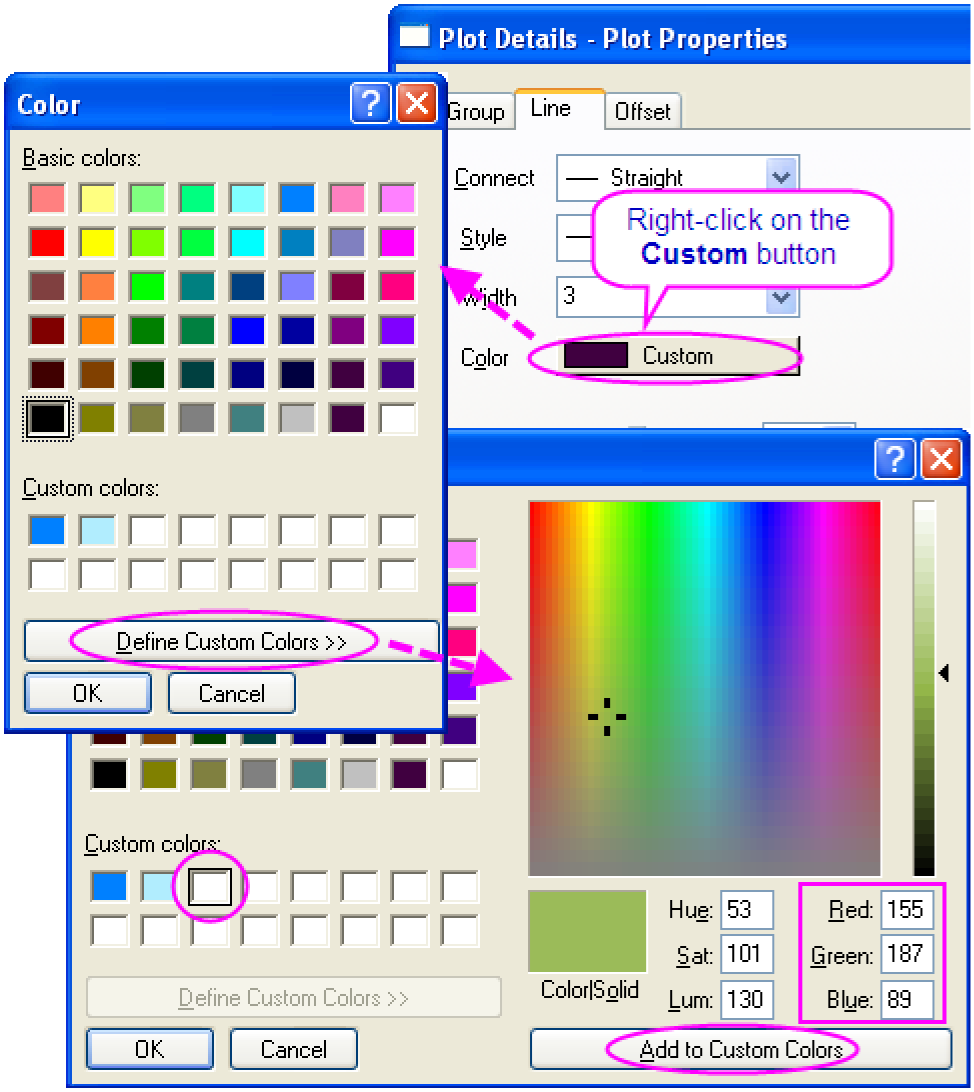


3. On the **Line** tab set **Width** as **3**. To set a custom color click on the button right to **Color** and select **Custom** from the drop list.



Right-click on the **Custom** button to bring up **Color** dialog. Click on **Define Custom Colors** to expand the right panel. For further use save the custom color in the **Custom colors:** palette:

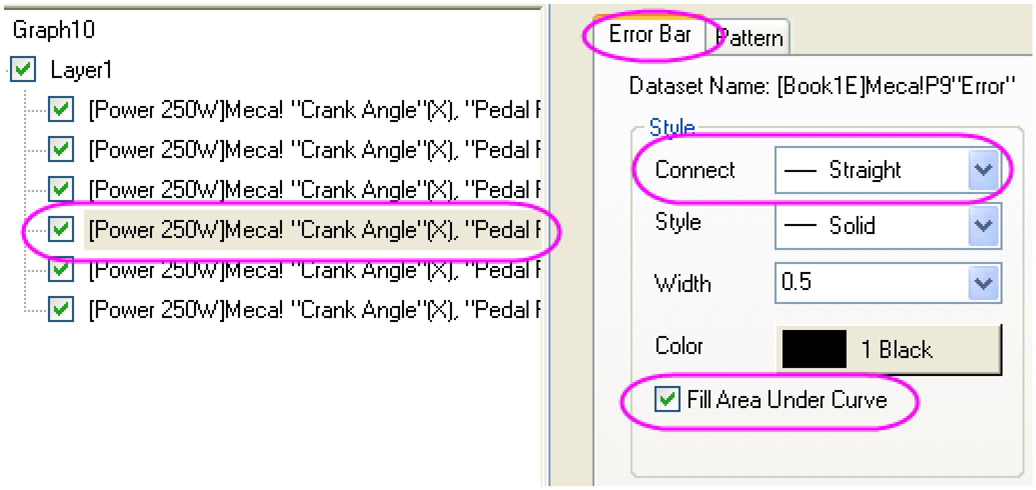
1. Select an blank box under the **Custom colors:** palette.
2. Set **Red**, **Green**, **Blue** as **155**, **187**, **89** respectively in the right panel.
3. Click **Add to Custom Colors** to save this color to the **Custom colors** palette.



Click **OK** to close the dialog. Set **Transparency** as **50**

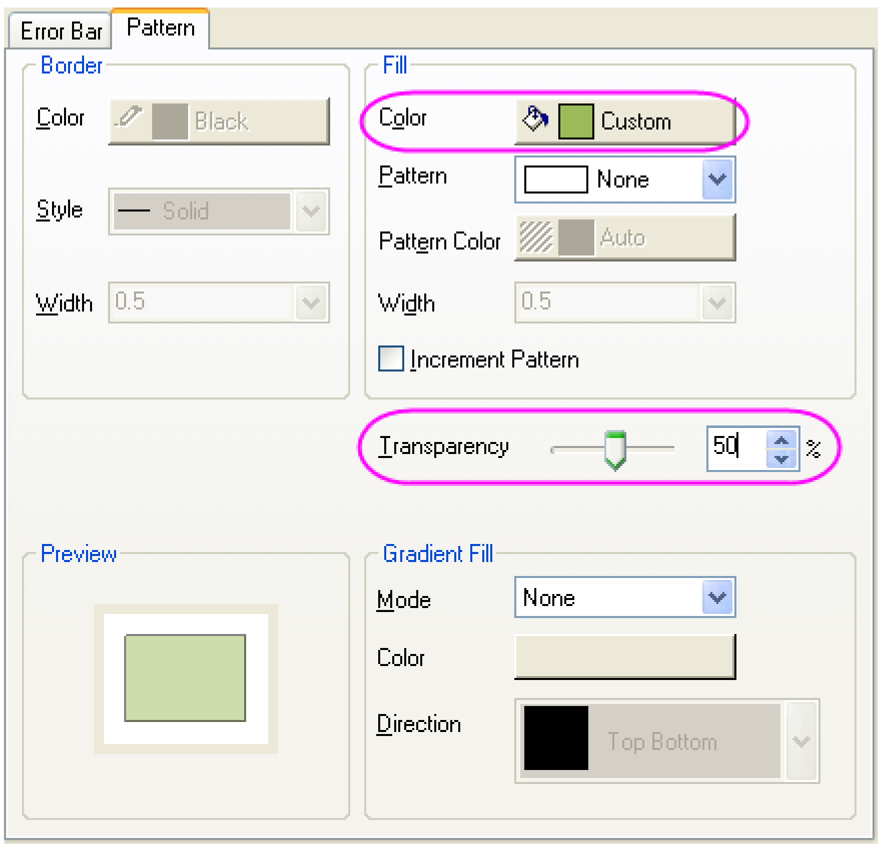
4. Repeat the step 3 for the other two plots. Except, set the colors of the second plot and the third plot as **RGB(192, 80, 77)**, **RGB(79, 129, 189)** respectively.

5. In the left panel, select the first of the three error plots. On the **Error Bar** tab, set **Connect** as **Straight**, then the **Fill Area Under Curve** option will show up in **Style** section. Check the box before it and the **Pattern** tab should now be available.



Set **Color** as **Automatic**.

6. On the **Pattern** tab, change the fill color to the custom color saved before to match the line color, and set Transparency to 50.



7. Repeat the last two steps for the other two error plots. Click **OK** to complete the graph.

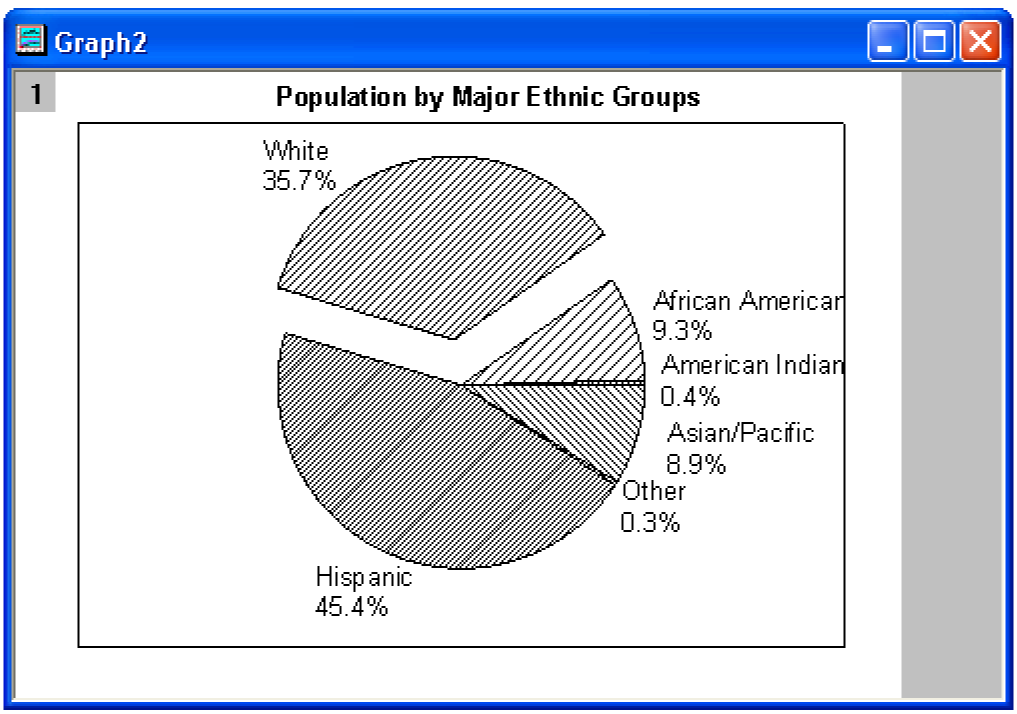
5.3 Column Bar Pie

Topics covered in this section:

5.3.1 2D Pie Chart of a Population Study

Summary

Pie charts created by the default template has a 3D view. However, you can easily convert it to a 2D chart. In addition, you can set the size and rotation of the chart to further customize it. This tutorial will show you how to create and customize a **2D Pie** chart.



Minimum Origin Version Required: Origin 8.5.1 SR0

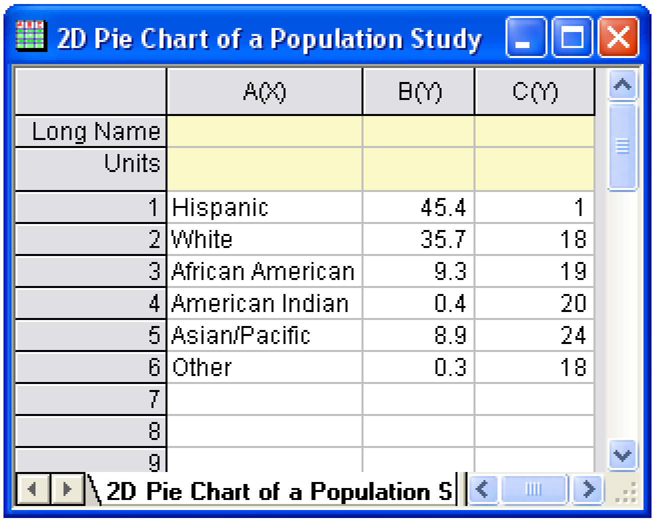
What will you learn

This tutorial will show you how to

- Create a 2D Pie chart
- Customize the Pie chart

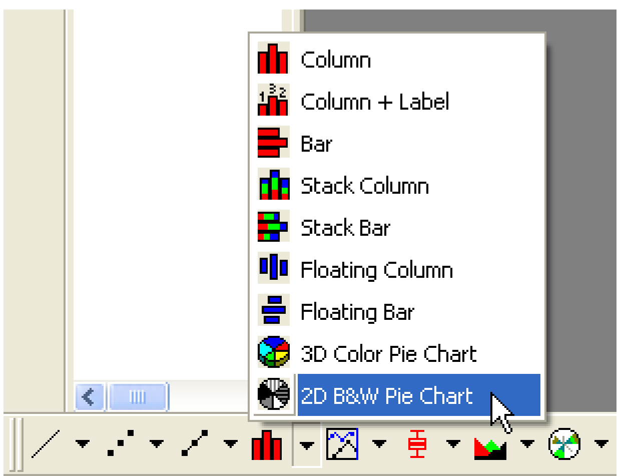
Steps

1. Create a new worksheet. Import the sample data (See the sample data here).

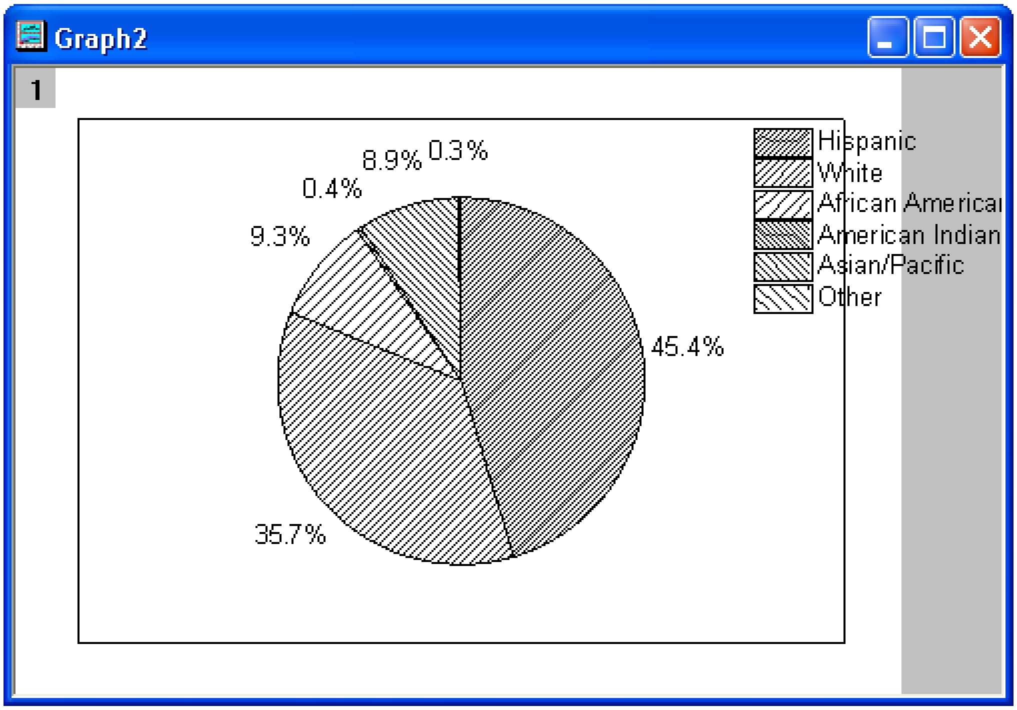


	A(X)	B(Y)	C(Y)
Long Name			
Units			
1	Hispanic	45.4	1
2	White	35.7	18
3	African American	9.3	19
4	American Indian	0.4	20
5	Asian/Pacific	8.9	24
6	Other	0.3	18
7			
8			
9			

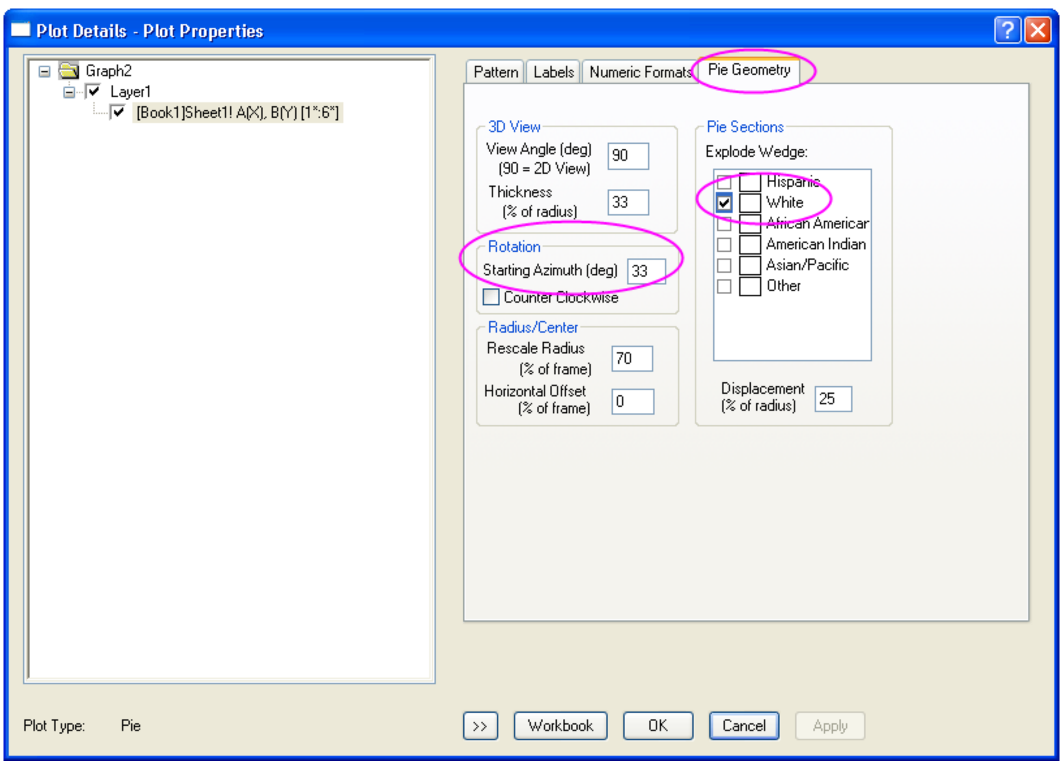
2. Highlight column **B** and select the **Pie Chart** button on the **2D Graphs** toolbar.



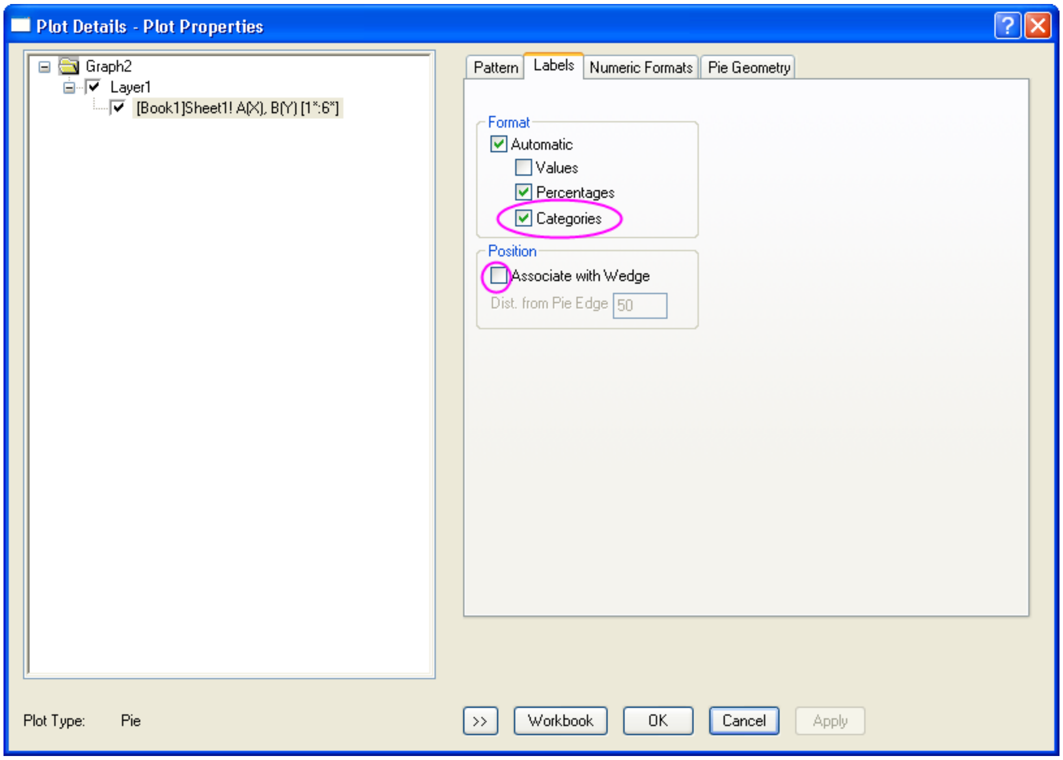
The graph should look like this:



3. Now we further customize the chart. Right-click on it and select **Plot Details** from the short-cut menu to open the **Plot Details** dialog. In the **Pie Geometry** tab, set the options as the screenshot below.

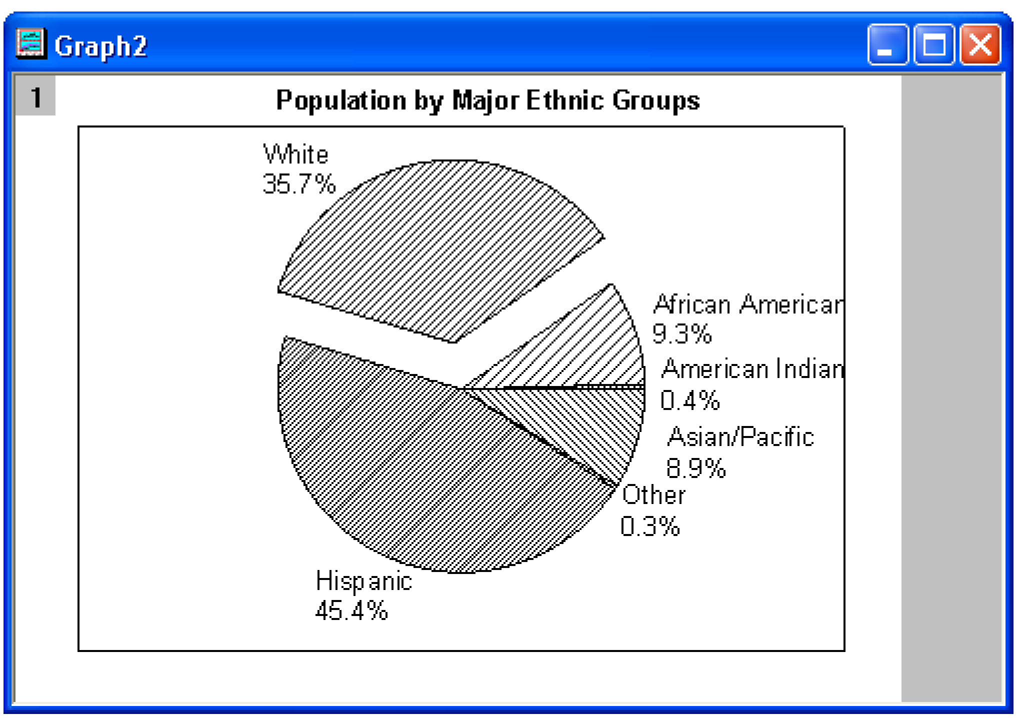


4. In the **Labels** tab, set the dialog as the screenshot below.



Click **OK** to close the **Plot Details** Dialog. Rearrangethe text label of each pie wedge.

5. Right click on the **Legend** and select **Delete** from the short-cut menu.
6. Select the **Text** button from the **Tools** toolbar. Then click near the top of the layer. Enter **Population by Major Ethnic Groups** to add a title for the graph. Then you can use the tools in the **Format** toolbar to customize the text. The graph should look like:



Sample Data

Download the **2D_Pie_Chart.txt** file from http://www.originlab.com/ftp/graph_gallery/data/2D_Pie_Chart.txt. Click the **Import Single ASCII button** . Select the file to import it into Origin.

The following table contains part of the sample data.

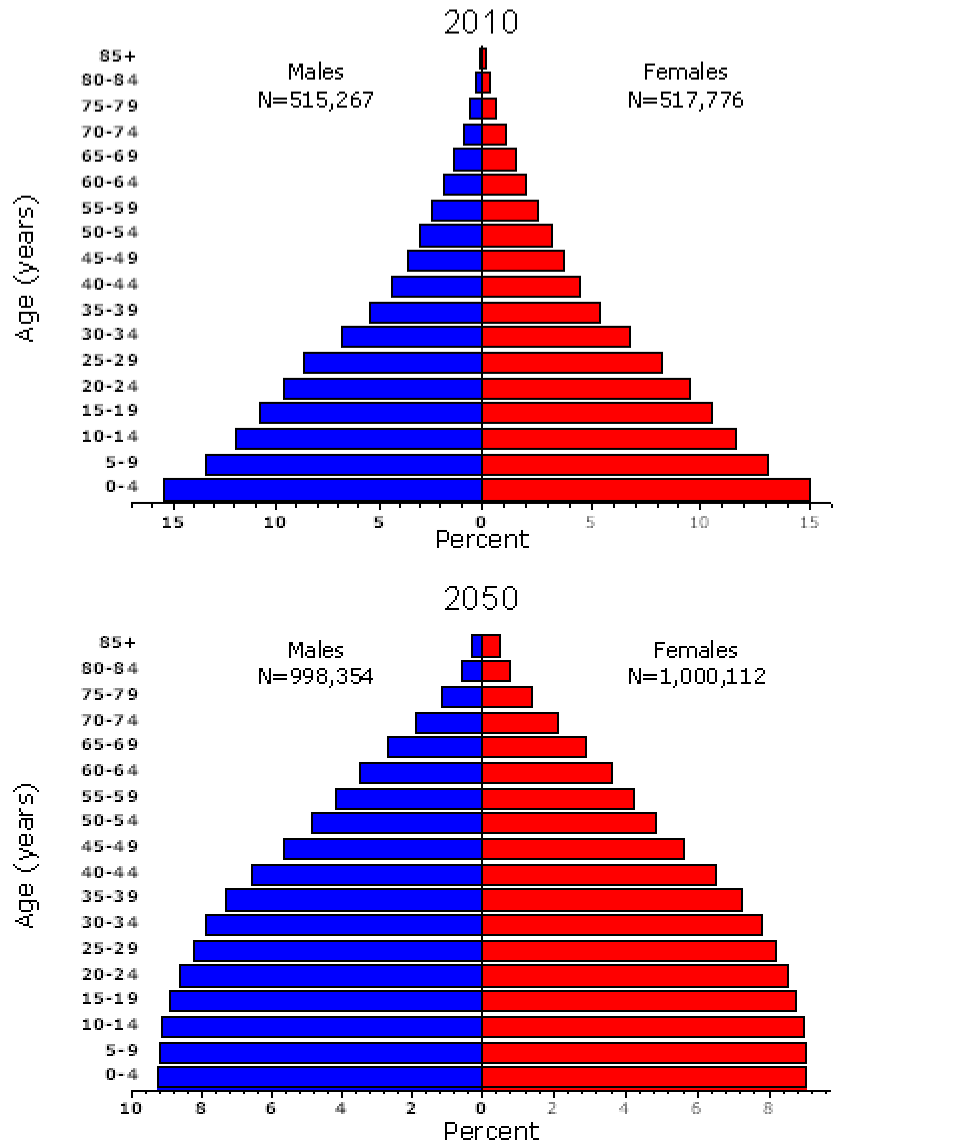
A(X)	B(Y)	C(Y)
Hispanic	45.4	1
White	35.7	18
African American	9.3	19
American Indian	0.4	20
Asian/Pacific	8.9	24
Other	0.3	18

5.3.2 Bar Graph of African Population

Summary

In this tutorial, a graph will be created to show the African population distribution, 2010 versus 2050.

Figure 1. African populaion by five-year age groups and sex, 2010 versus 2050



Source: Population Division of the Department of Economic and Social Affairs of the United Nations Secretariat, World Population Prospects: The 2008 Revision, http://esa.un.org/unpp, Monday,April?2,?010; 3:19:17PM.

Minimum Origin Version Required: Origin 8 SR0

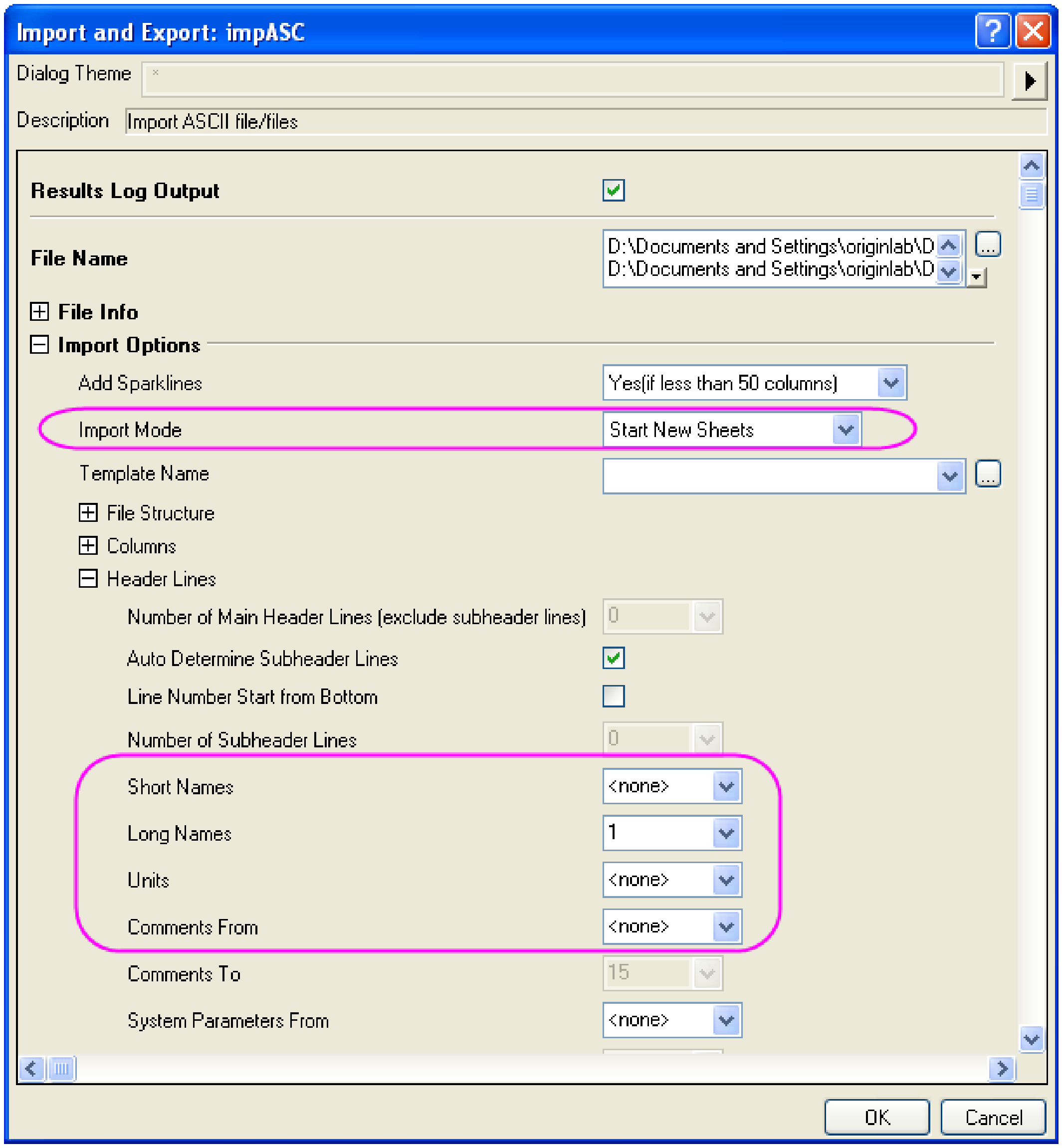
What you will learn

- Set axis scale to reverse bar chart
- Merge graphs

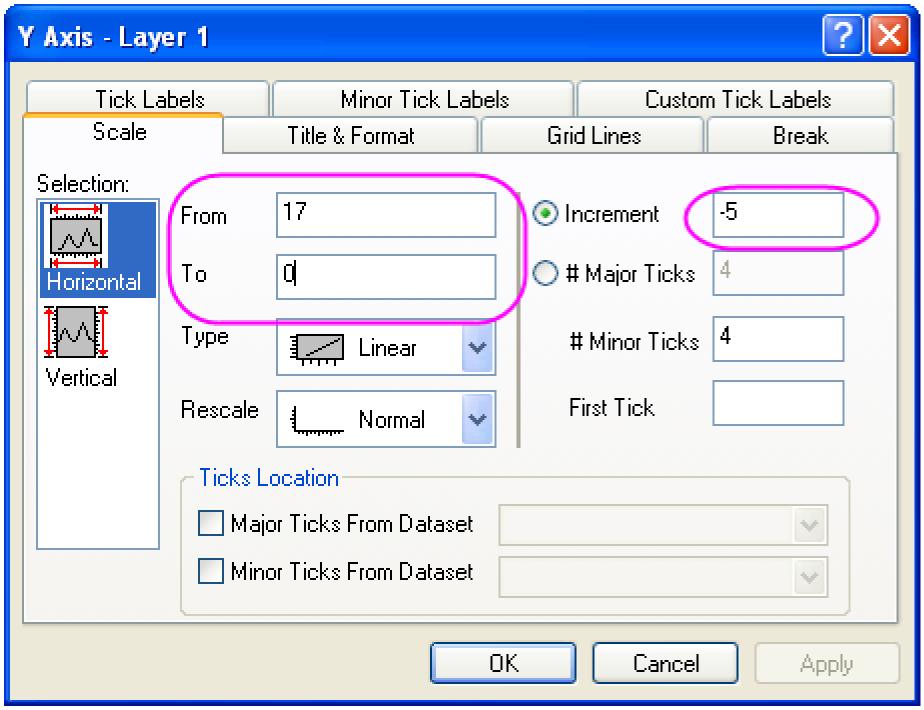
Steps

1. Download the source data files bar_graph_of_African_population_01.txt and bar_graph_of_African_population_02.txt from our ftp. Import these two files into an Origin

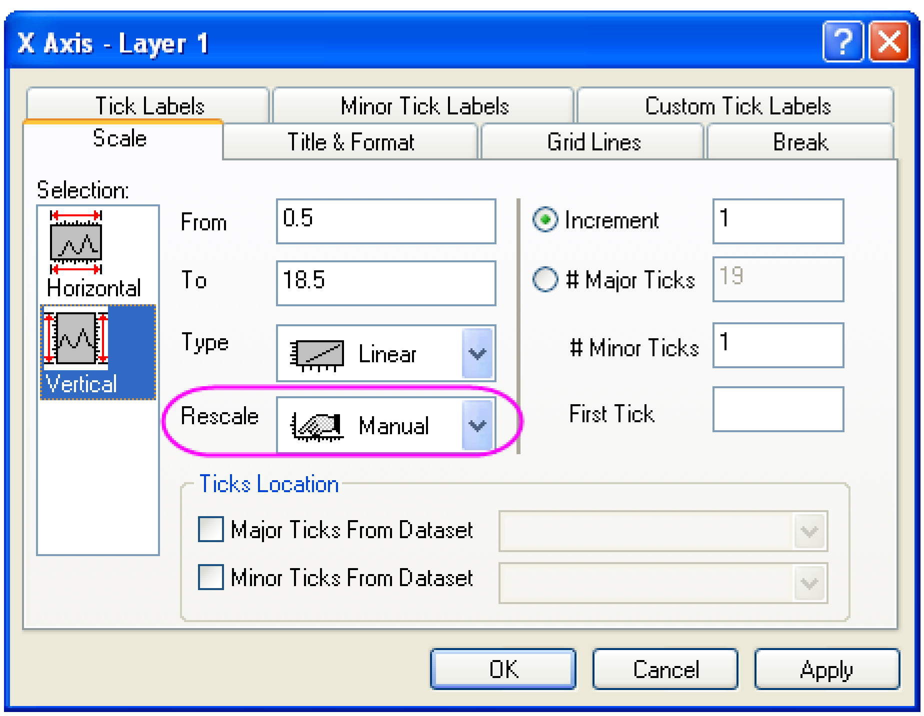
workbook with two worksheets, each for one .txt file. The **Import Multiple ASCII** dialog can be set as follow:



2. Highlight the "male" column in the "bar_graph_of_African_population1" sheet and click the Bar button on the 2D Graphs toolbar to plot a bar graph.
3. In the bar graph you just created, double-click X-axis to open the **X-Axis** dialog box. Go to the **Grid Lines** tab. Select **Horizontal** in the **Selection** box and then uncheck **Minor Grids** to hide the grid lines.
4. Go to the Scale tab. Select **Horizontal** in the **Selection** box. Change From and To to 17 to 0, respectively. Set increment to -5.

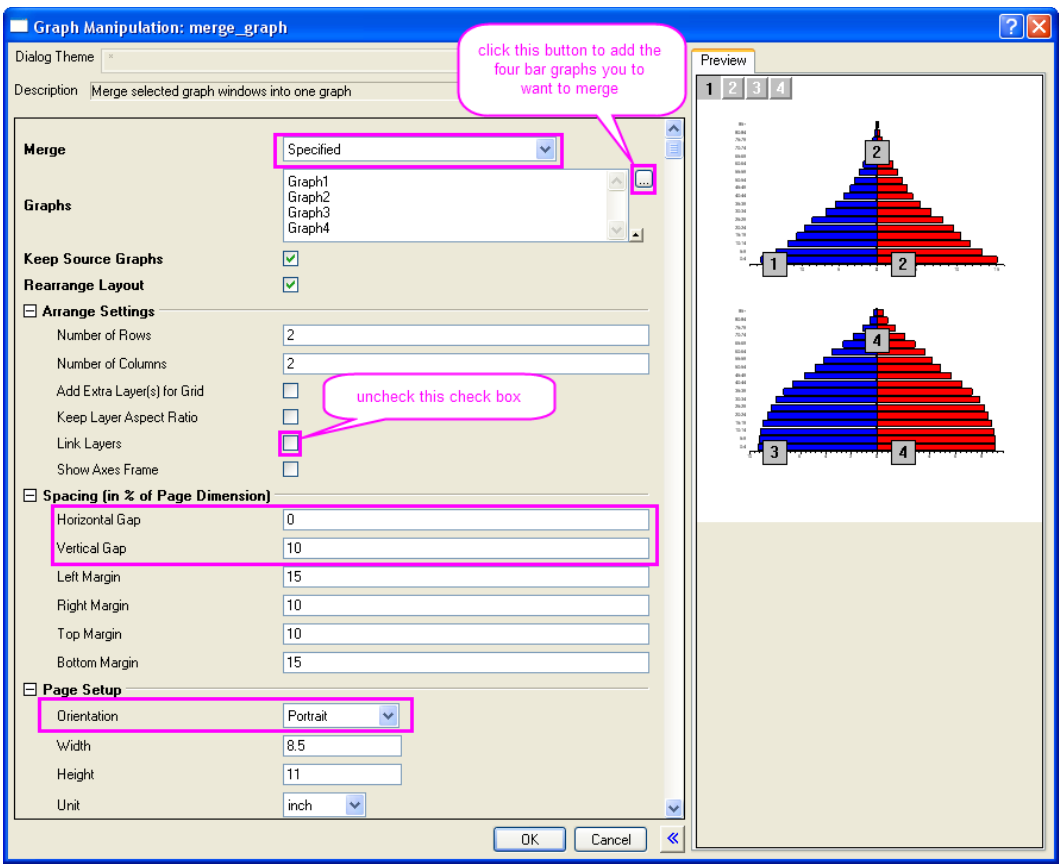


Then select **Vertical** in the **Selection** box and select **Manual** from the **Rescale** drop-down list. Click **OK** to apply.



5. Double-click on the bar graph to open the **Plot Details** dialog. Go to the **Pattern** tab and change **Color** (in the Fill box) to Blue. Click **OK** to change the bar fill color to blue.
6. Delete the XY axis titles and the Legend.

7. Highlight the "female" column in the "bar_graph_of_African_population1" worksheet and click the Bar button to plot another bar graph.
8. Repeat step 3 to turn off the grid lines. Then do the same things as above to reset the axis scale. However this time, change the Horizontal axis? From and To values to 0 to 16, respectively, and set Increment to 5.
9. Delete the XY axis Tick Labels, Title and Legend.
10. Repeat steps 2 through 9 to plot two more bar graphs from the worksheet "bar_graph_of_African_population_". At this time, set the X-Axis scale in the male bar graph to go From 10.0116 To 0, by an Increment of -2. Leave the Axis scale in the female bar graph as the default and change only the Y-axis to Manual.
11. Now you have four bar graphs. To merge these four graphs, activate one of the four graph windows and select **Graph: Merge Graph Windows...** from the main menu. Doing so opens the merge_graph dialog box. Change the settings as follows to merge these four bar graphs into one graph window.

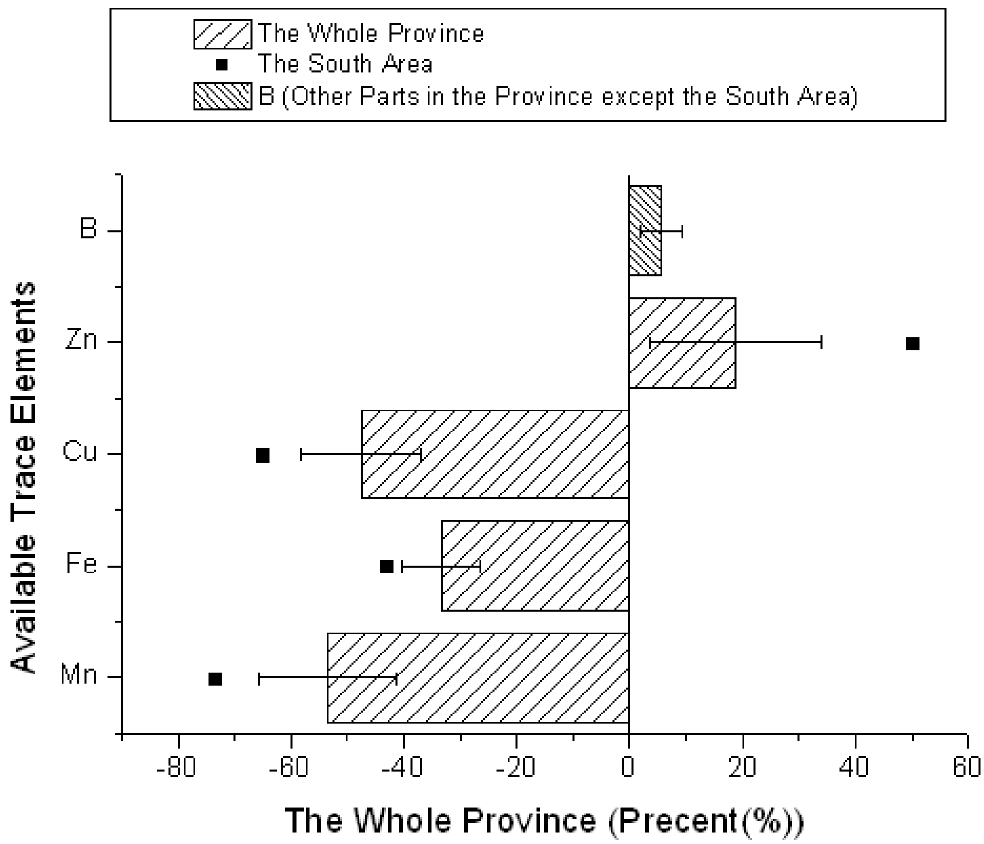


12. In the merged graph, add text boxes: "Percent" and "Age (years)", as axis titles of the upper and lower graphs. Also add some other text boxes, such as a graph title: "Figure 1. African population by five ? year age groups and sex, 2010 versus 2050", to better describe this graph.

5.3.3 Bar and Scatter Plot with Error Bar

Summary

Below graph is made of bar chart and scatter plot.



Minimum Origin Version Required: Origin 8.0 SR0

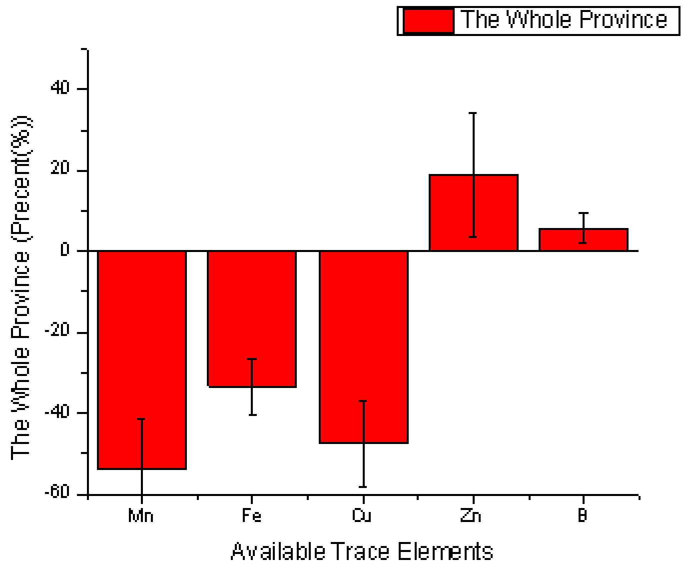
What will you learn

- How to add a scatter plot to a bar graph
- How to set the Plus and Minus error bar

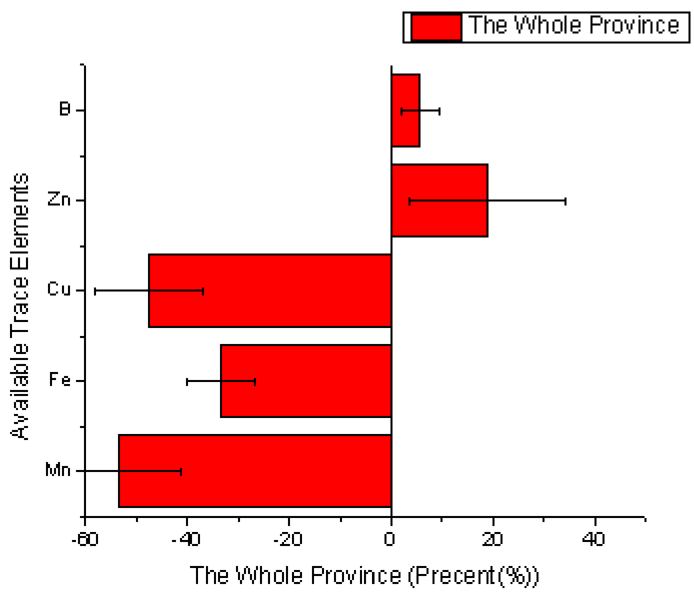
Steps

This tutorial is associated with the **2D and Contour Graphs: Column, Bar?Bar Plot with Errors** folder in the **2D and Contour Graphs** project (\Samples\2D and Contour Graphs.opj) which can be opened by selecting **File: Open Sample Projects? 2D and Contour Graphs** from the main menu.

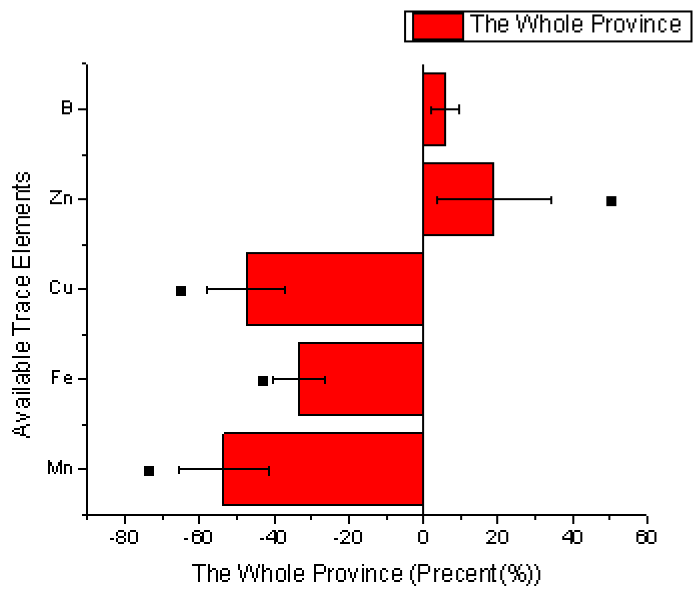
1. Active the workbook **Book2N**, right-click the col(C) to select **Set as: Y Error** from the context menu.
2. Highlight Col(A), col(B) and col(C), select **Plot: Column / Bar / Pie: Column** from the main menu to plot a column graph with Y error bar.



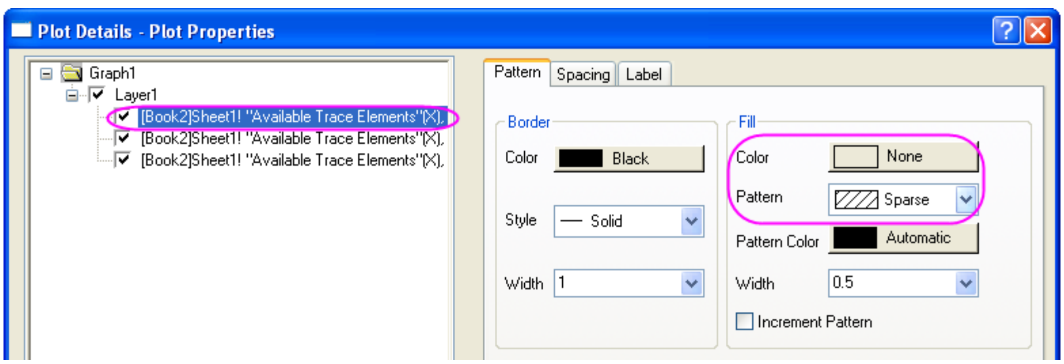
3. Active the graph window, and then select **Graph: Exchange X-Y Axes**.



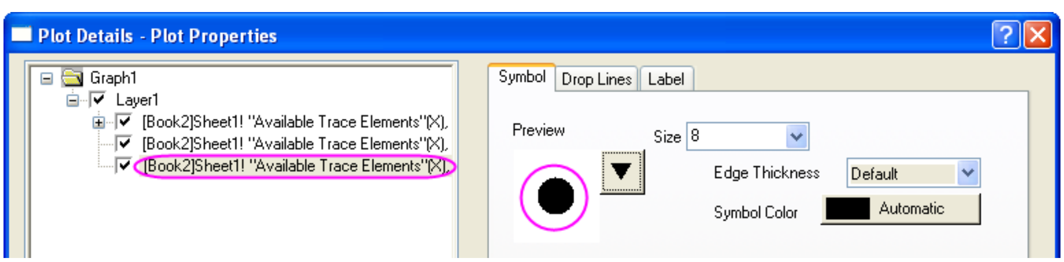
4. Highlight the col(D) in the worksheet and then active the graph window again. Select Graph: Add Plot to Layer: Scatter from the main menu to add the col(D) as scatter to the column graph.



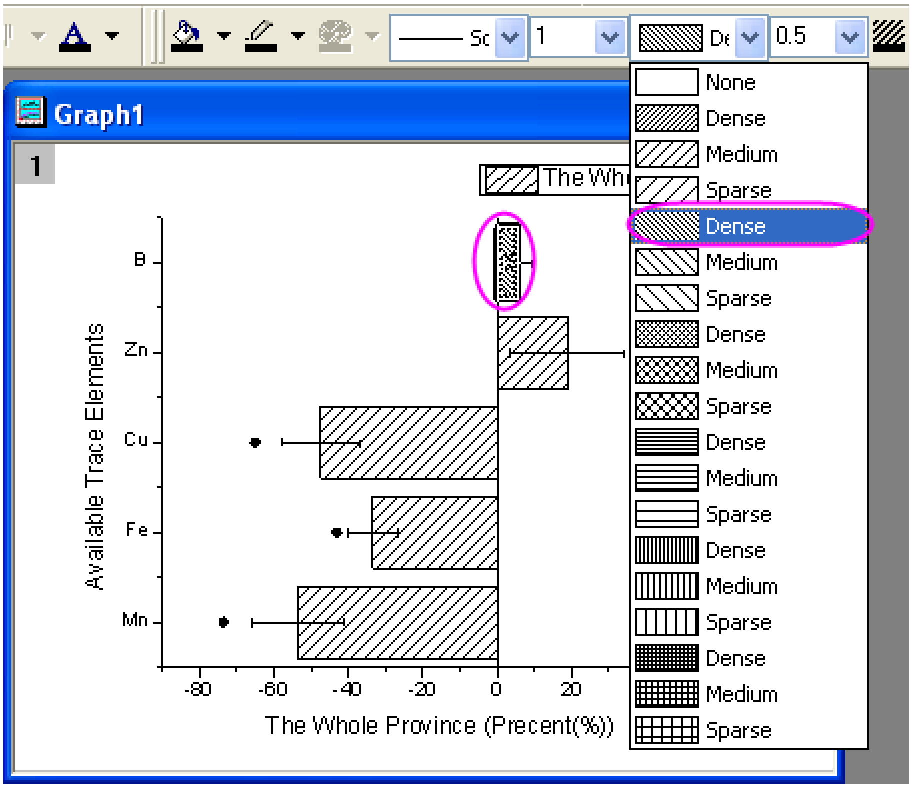
5. Double-click the column graph to open the Plot Details dialog. Set the **Color** and **Pattern** options in the **Fill** group of **Pattern** Tab as below.



6. Select the scatter plot in the left panel, then set the Symbol as below. Then click **OK** button to close the dialog.

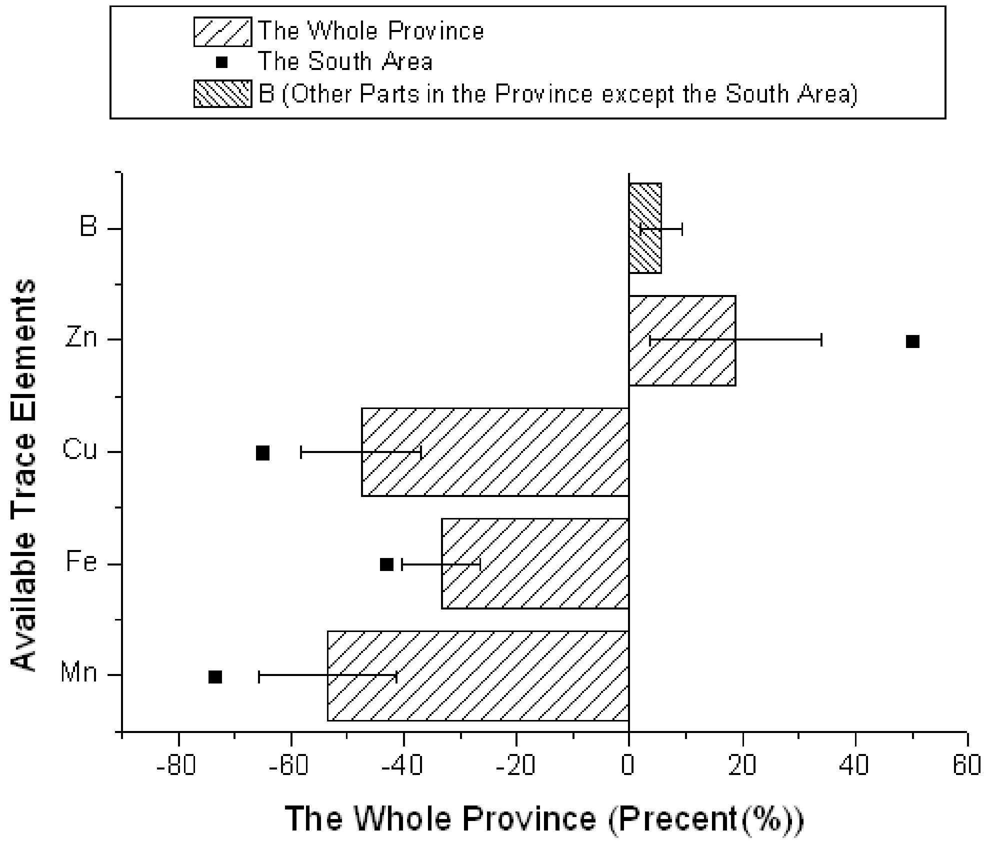


7. Active the graph window, hold the **Ctrl** key and select the first bar to set its pattern as **Dense** as below from the **Style** toolbar.



8. Delete the legend, then select Graph: New Legend from the main menu to add a new legend for graph.

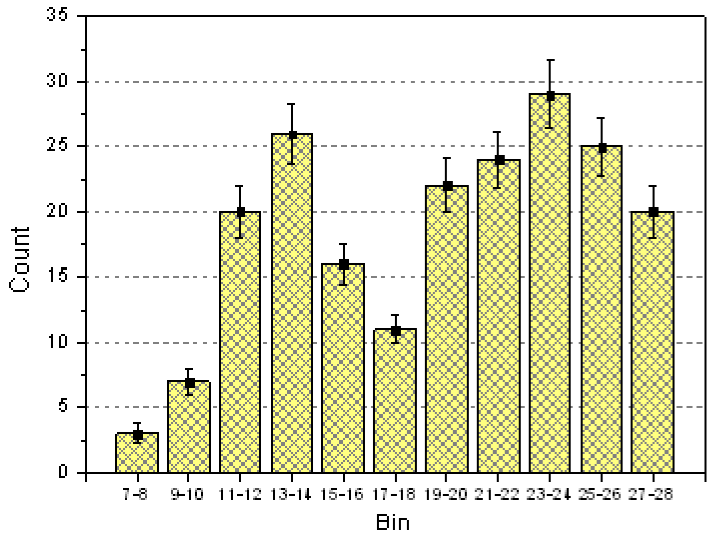
9. Right-click the legend to select Properties from the context menu to open the **Objects Properties** dialog. Set **Background** as **Black Line**. Then edit and move the legend as below.



5.3.4 Column Graph with Error Bars

Summary

This custom graph illustrates how Origin can include error bars along with its data plots. The same Y data is plotted twice, once as a scatter plot and again as a column plot. The error bars are attached to the scatter plot in this case. Error bars can be included in the graph in both the Y and X directions.



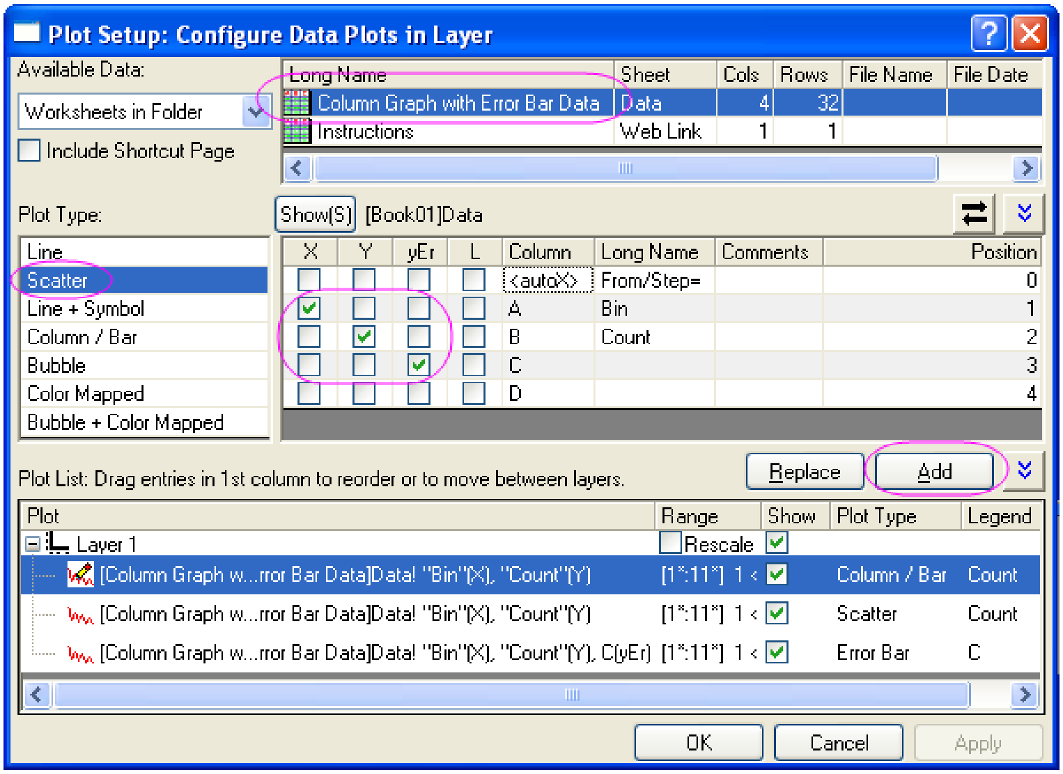
What you will learn

- Create and customize column graph
- Use Plot Setup dialog to add a new data plot into your graph

Steps

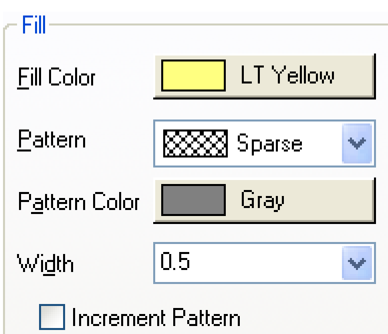
This tutorial is associated with the 2D and Contour Graphs project: *\Samples\2D and Contour Graphs.opj*. (If you don't have the Project file, please download the data file from here)

1. Open the Project file, and browse to the folder *2D and Contour Graphs: Column,bar: Column with Error Bar*. Active the worksheet and make sure the column type as **X**, **Y**, **Y Error** and **Label** accordingly.
2. Highlight column 2 and select **Plot: Column / Bar / Pie: Column** to create a column graph.
3. With the graph window active, select **Graph: Plot Setup** to bring up the Plot Setup dialog. We will add the scatter and error bars from this dialog as below:



Click the **Add** button to add scatter data to column plot. Then click **OK** to go back to column graph window.

4. Double-click the columns to bring up the **Plot Details** dialog to customize the graph in **Pattern** tab as below:

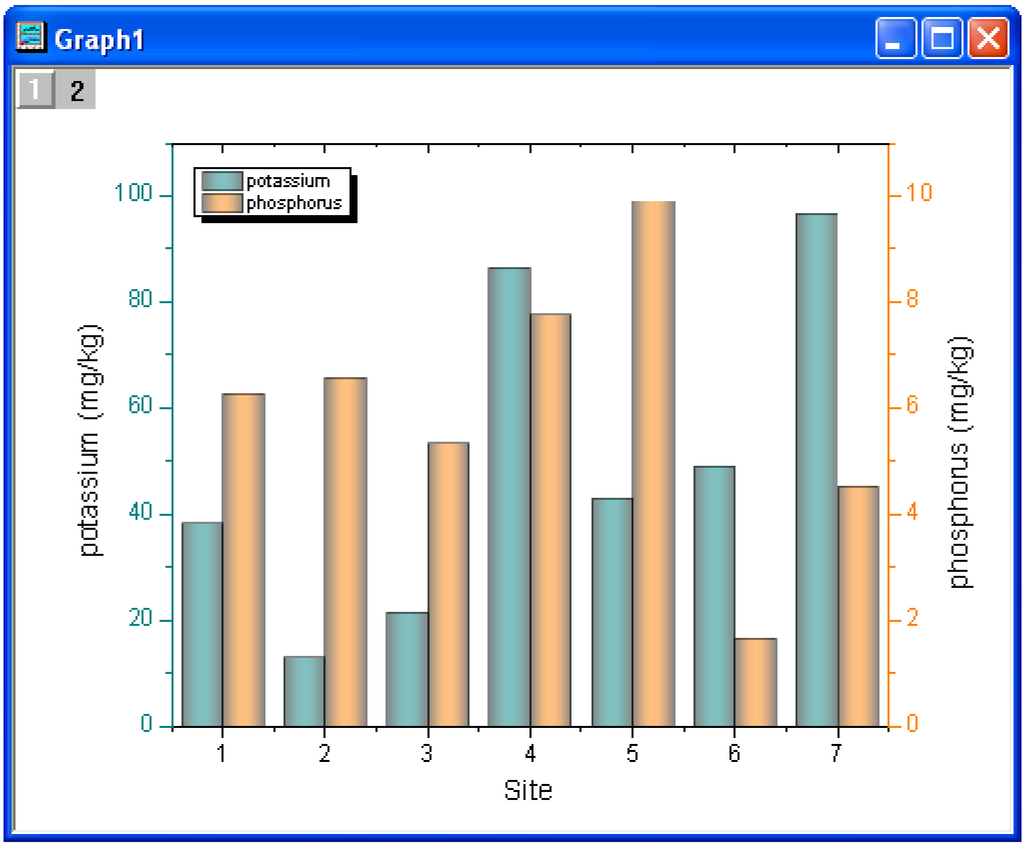


5. Double-click the Y axis, and set the Vertical axis scale **From** 0 **To** 35. Then active the **Grid Lines** tab, enable **Horizontal Major Grid** with **Dash** line. Then check the **Opposite** checkbox for both Horizontal and Vertical axis.Go to the the **Tick Labels** tab, choose the **Bottom** icon from the left panel. Choose **Text from dataset** from the **Type** drop-down list and choose **[Book01]Data!D** from the Dataset drop-down list. Click OK to apply these settings.
6. Delete or customize the legend and axis title as you need.

5.3.5 Column / Bar Gap / Offset Across Layers

Summary

This tutorial will show you how to create a multiple-layer column graph with gap across layers.



Minimum Origin Version Required: Origin 8.5.1 SR0

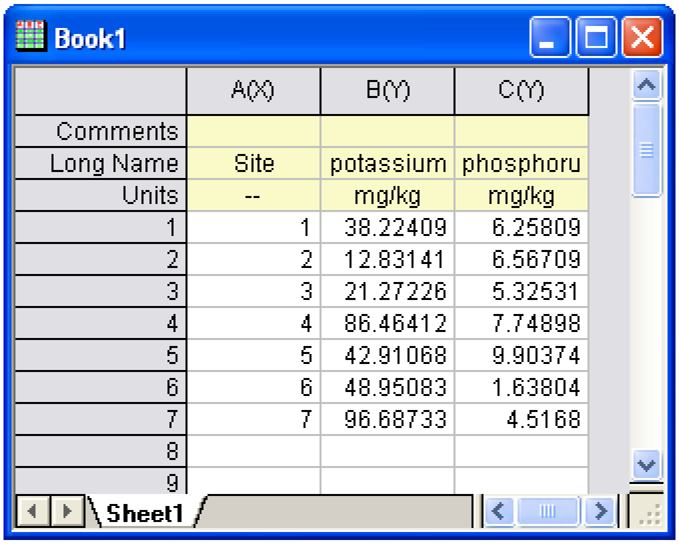
What will you learn

This tutorial will show you how to

- Create double-Y column graph
- Add gap for the column plots in different layers

Steps

1. Copy and paste the sample data to an new Origin worksheet. Set the first row as Long Name and the second row as Units.

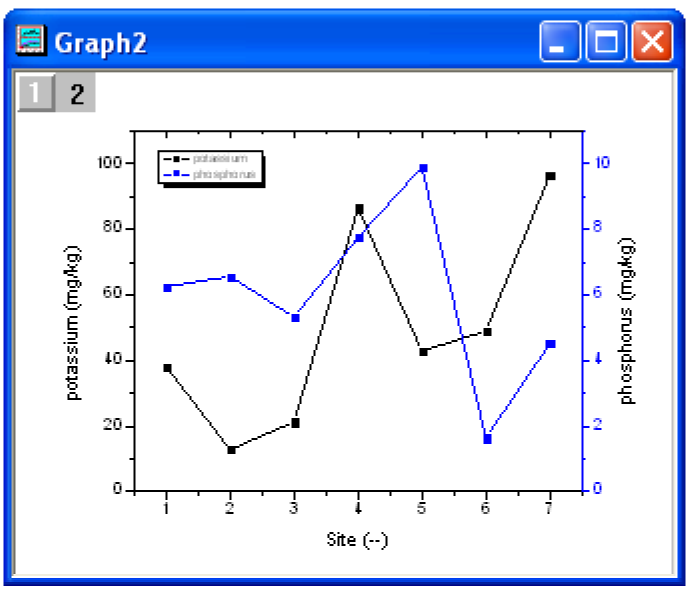


Book1

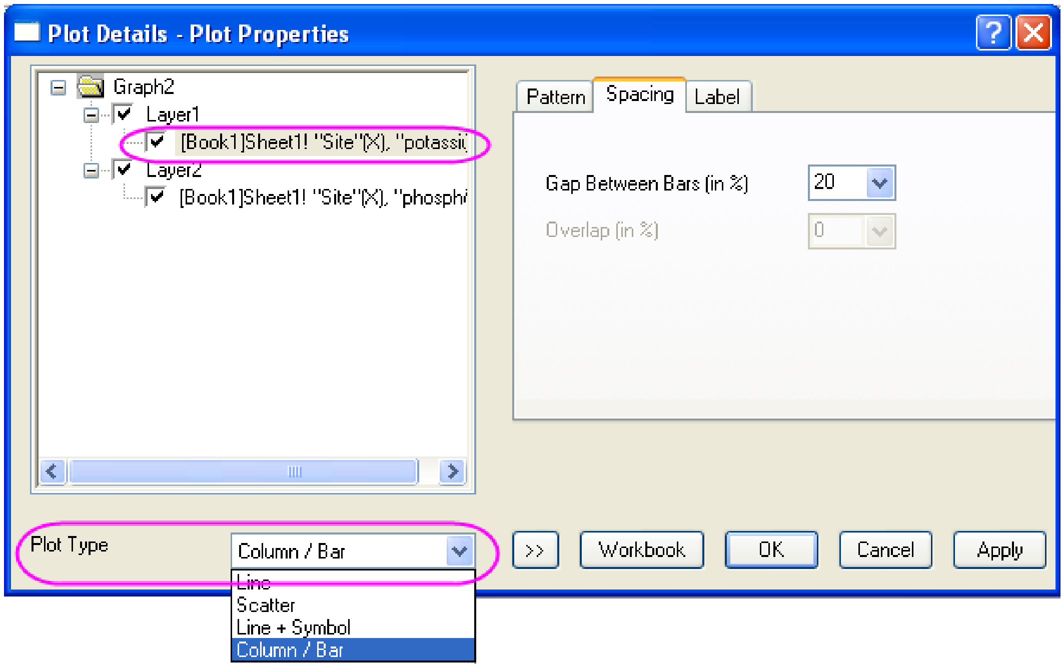
	A(X)	B(Y)	C(Y)
Comments			
Long Name	Site	potassium	phosphoru
Units	--	mg/kg	mg/kg
1	1	38.22409	6.25809
2	2	12.83141	6.56709
3	3	21.27226	5.32531
4	4	86.46412	7.74898
5	5	42.91068	9.90374
6	6	48.95083	1.63804
7	7	96.68733	4.5168
8			
9			

Sheet1

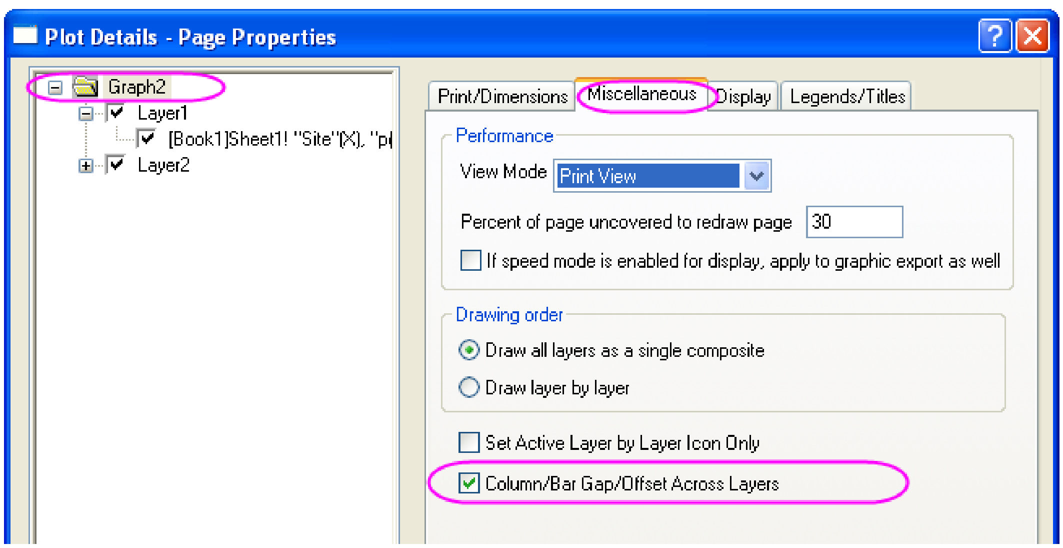
2. Plot a double-Y graph: Highlight all columns. In Origin?s main menu, click Plot, point to Multi-Curve, and then click Double-Y. Alternatively, you can simply click the Double-Y button on the 2D Graphs toolbar.



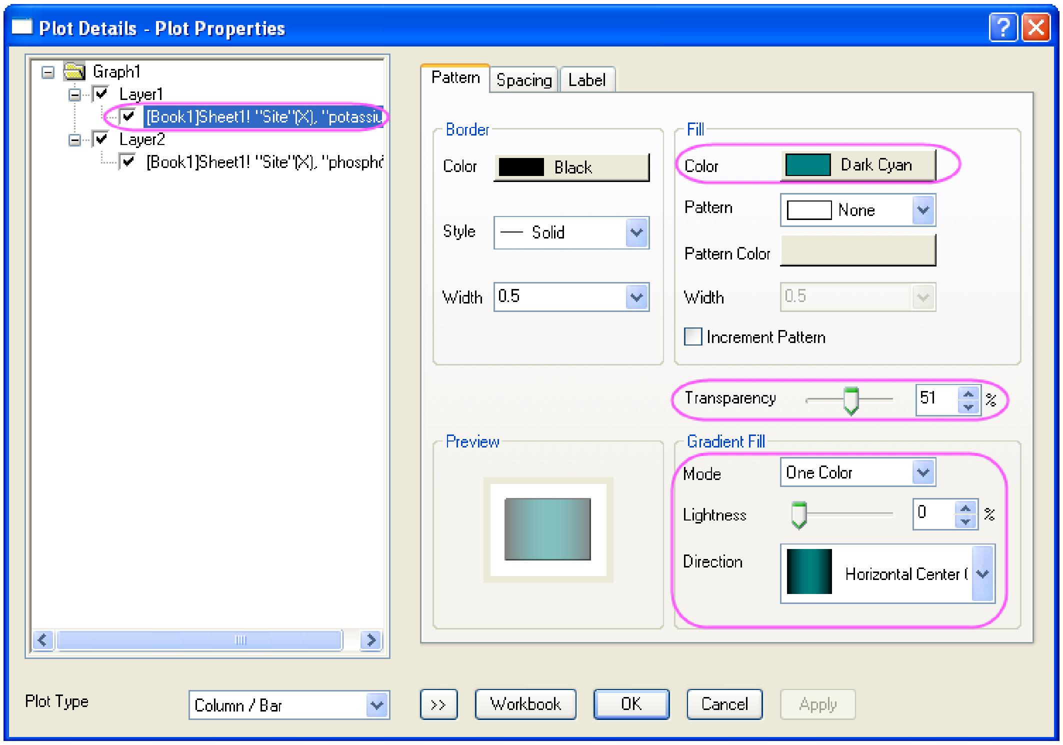
3. Double-click then graph to open the Plot Details dialog. In this dialog?s left menu, select a plot (as below) and change the plot type to Column/Bar. Do this for both plots.

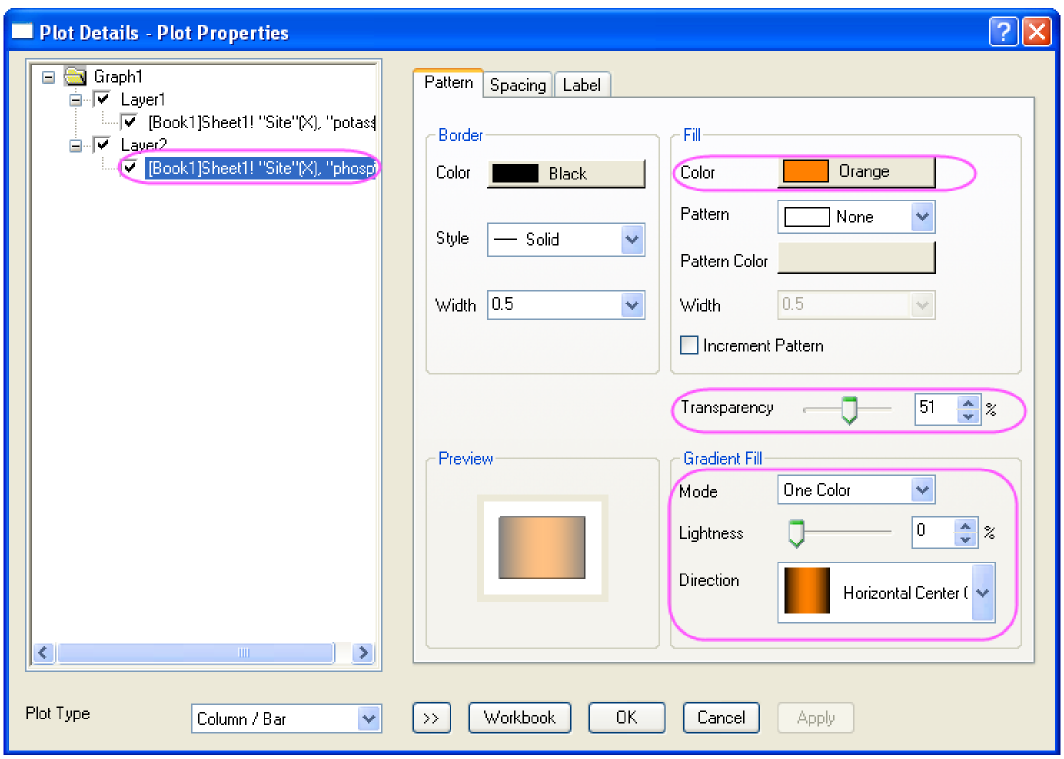


4. In the Plot Details dialog?s left menu, select the top level (Graph2 in the example below). On the Miscellaneous tab, select the Column/Bar Gap/Offset Across Layers check box.

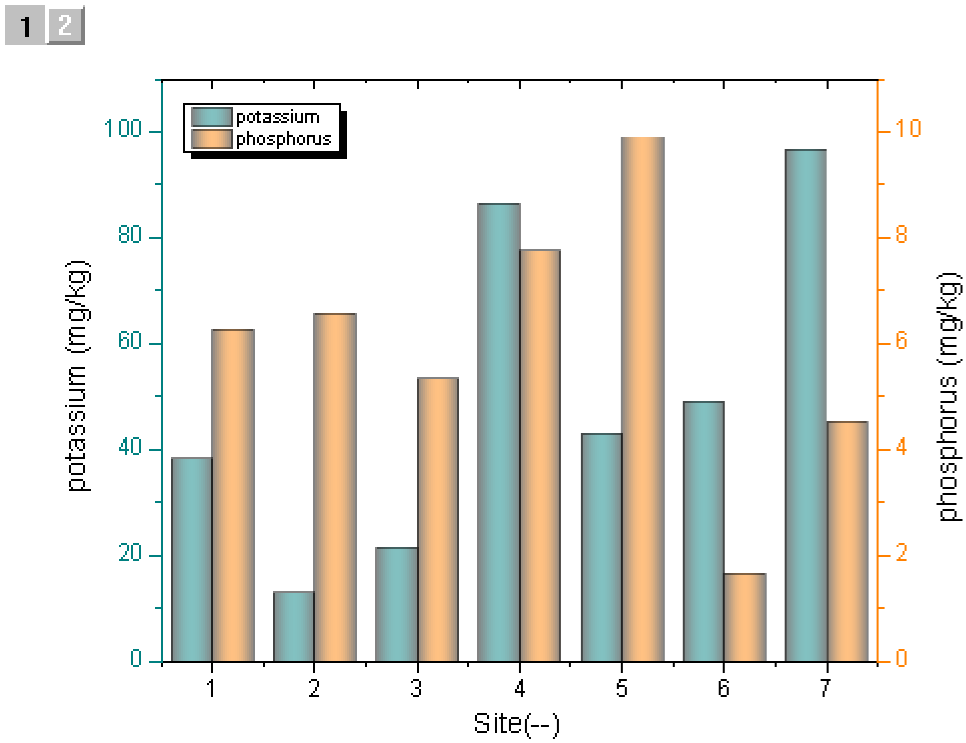


5. In the left panel, select the first plot and set the appearance of its columns as shown below. Do the same for the second plot, using the second image below as reference.





6. Click OK to close the dialog. Your final graph should look like this:



Sample Data

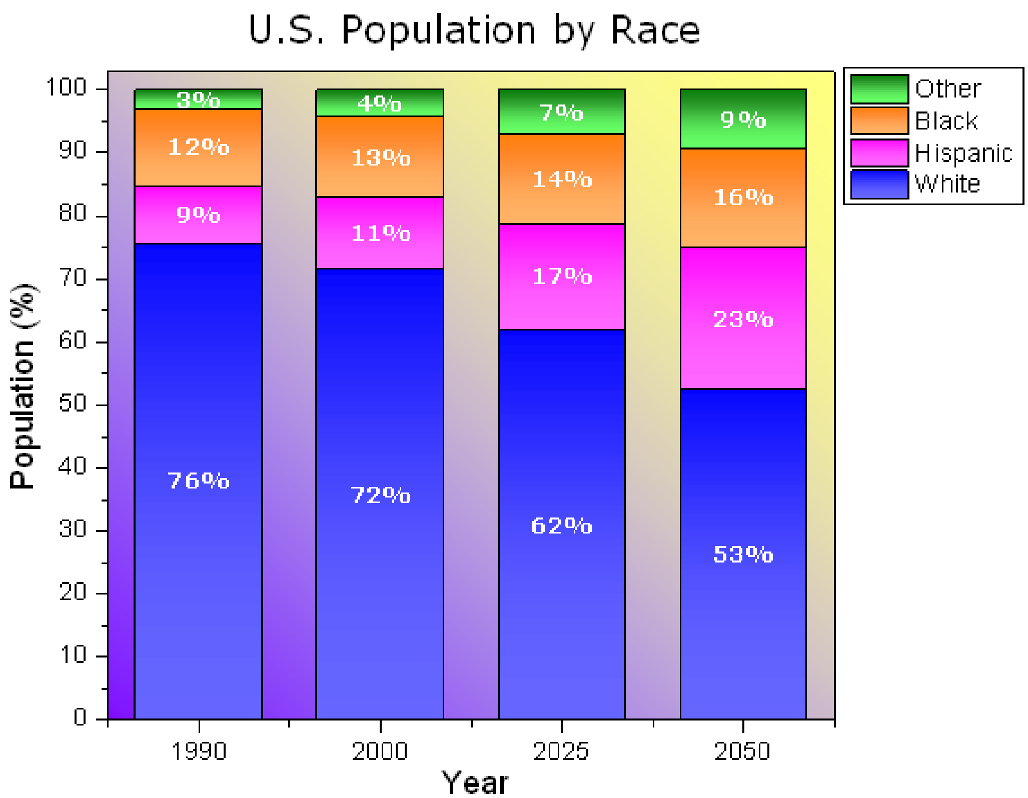
The following table contains the sample data.

Site	potassium	phosphorus
--	mg / kg	mg / kg
1	38.22409	6.25809
2	12.83141	6.56709
3	21.27226	5.32531
4	86.46412	7.74898
5	42.91068	9.90374
6	48.95083	1.63804
7	96.68733	4.5168

5.3.6 Stack Column With Labels

Summary

This graph displays a stacked column plot. Each data point in each column has been labelled using the data value with custom formatting.



Minimum Origin Version Required: 8.5 SR0

What will you learn

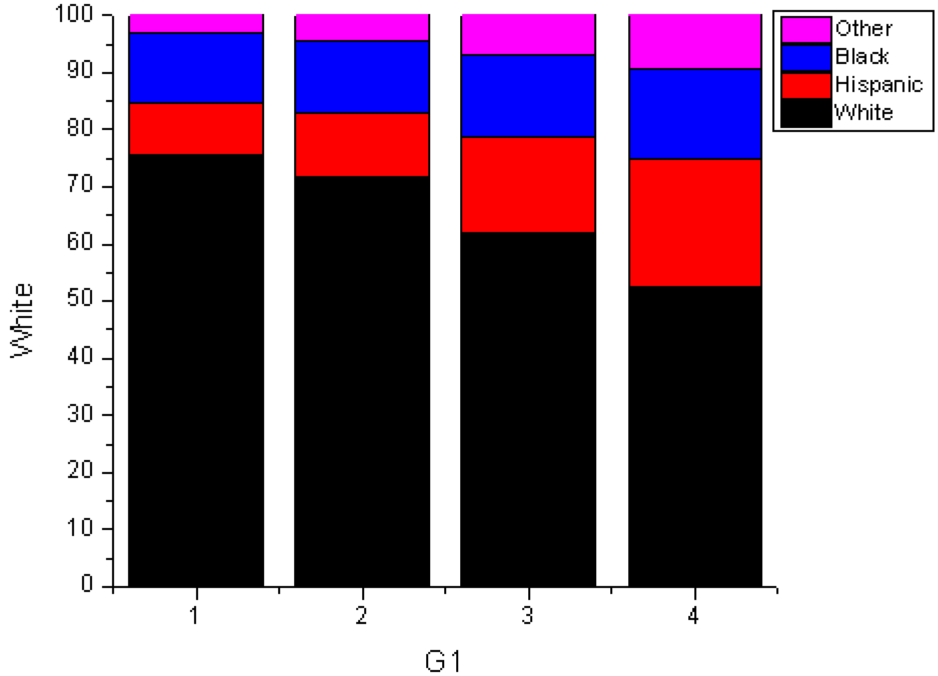
This tutorial will show you how to

- Create a stack column
- Add labels for columns
- Customize the column graph

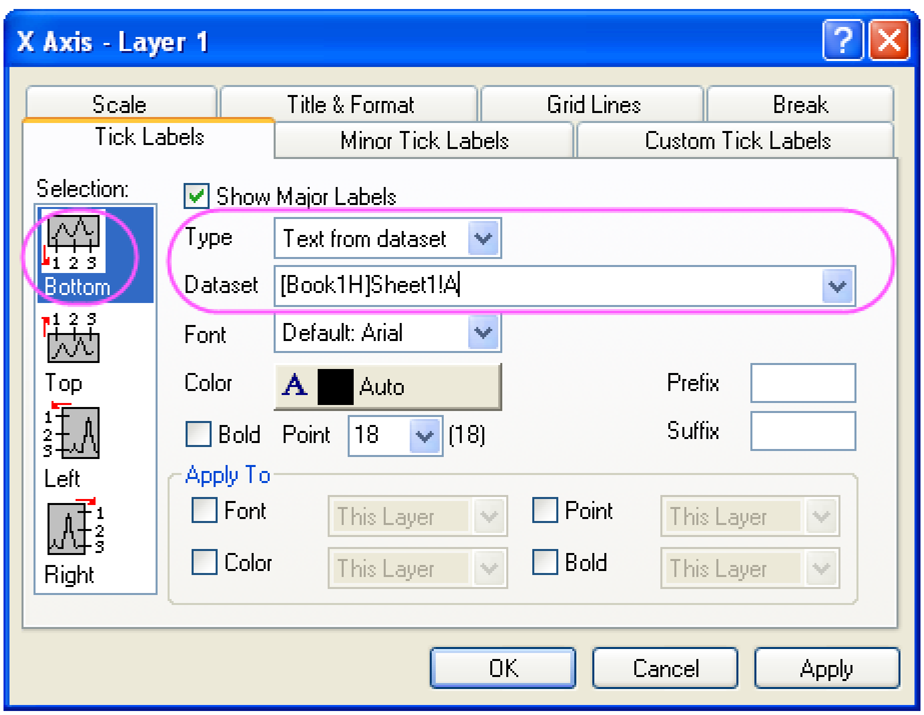
Steps

This tutorial is associated with the 2D and Contour Graphs project: *\Samples\2D and Contour Graphs.opj*.

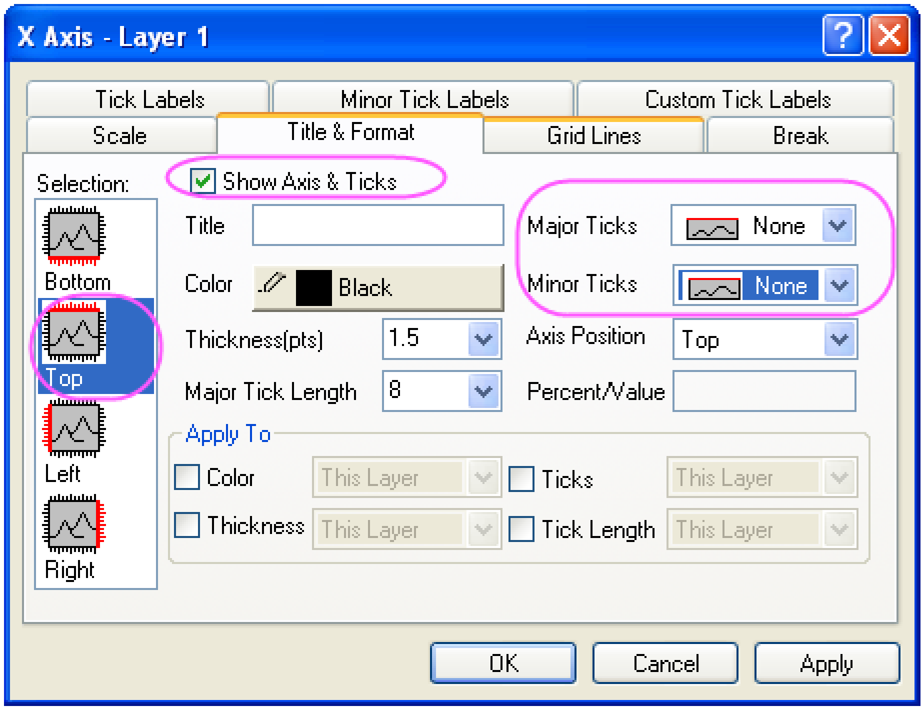
1. Open the Project file, and browse to the folder *2D and Contour Graphs\Column, Bar\Stack Column With Labels*.
2. Activate the worksheet and select columns B through E. In the main menu, click **Plot**, then point to **Column/Bar/Pie**, and then click **Stack Column**. Alternatively, you can simply click the **Stack Column** button on the 2D Graph toolbar.



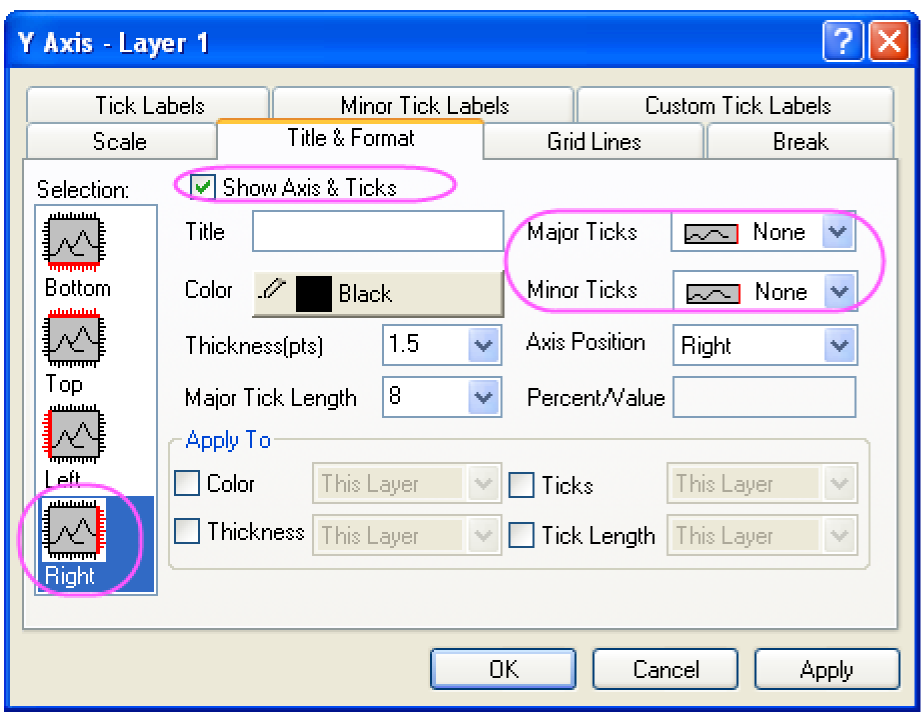
3. Double-click the X axis to open the Axis dialog. On the **Tick Labels** tab, select **Text from dataset** from the **Type** menu, and select **[Book1H]Sheet1!A** from the Dataset menu.



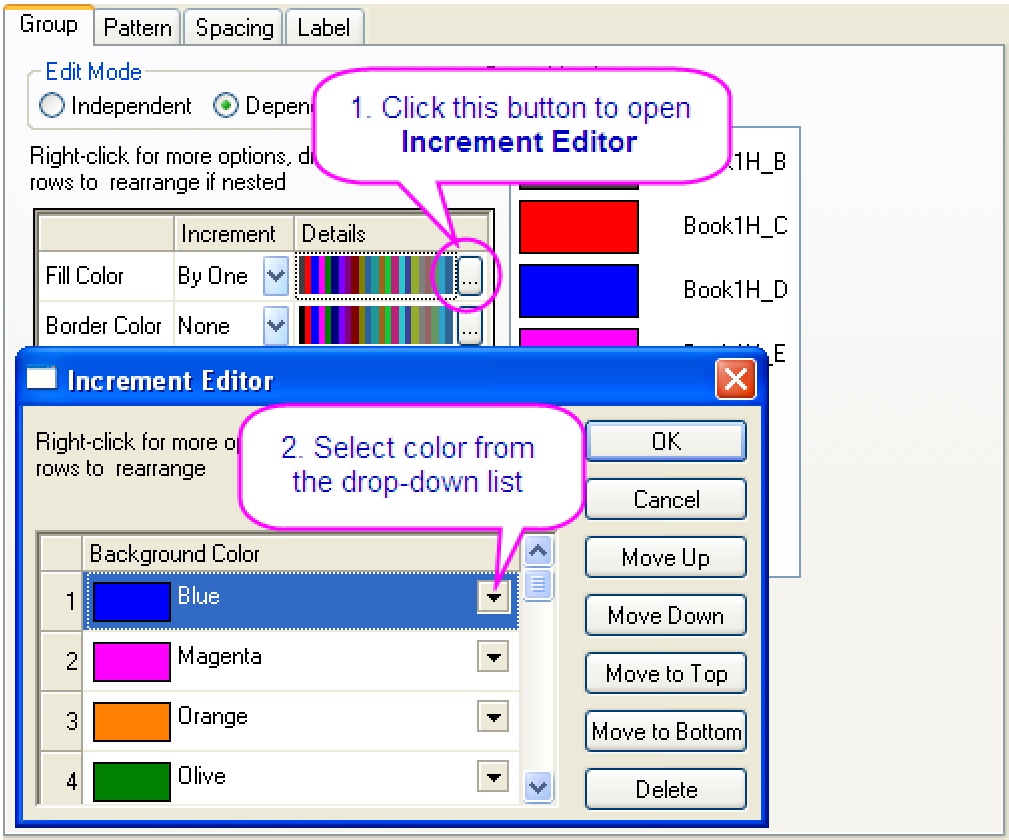
4. On the **Title & Format** tab, select the **Top** icon in the **Selection** box. Select **Show Axis & Ticks**, and then set **Major Ticks** and **Minor Ticks** to **None**.



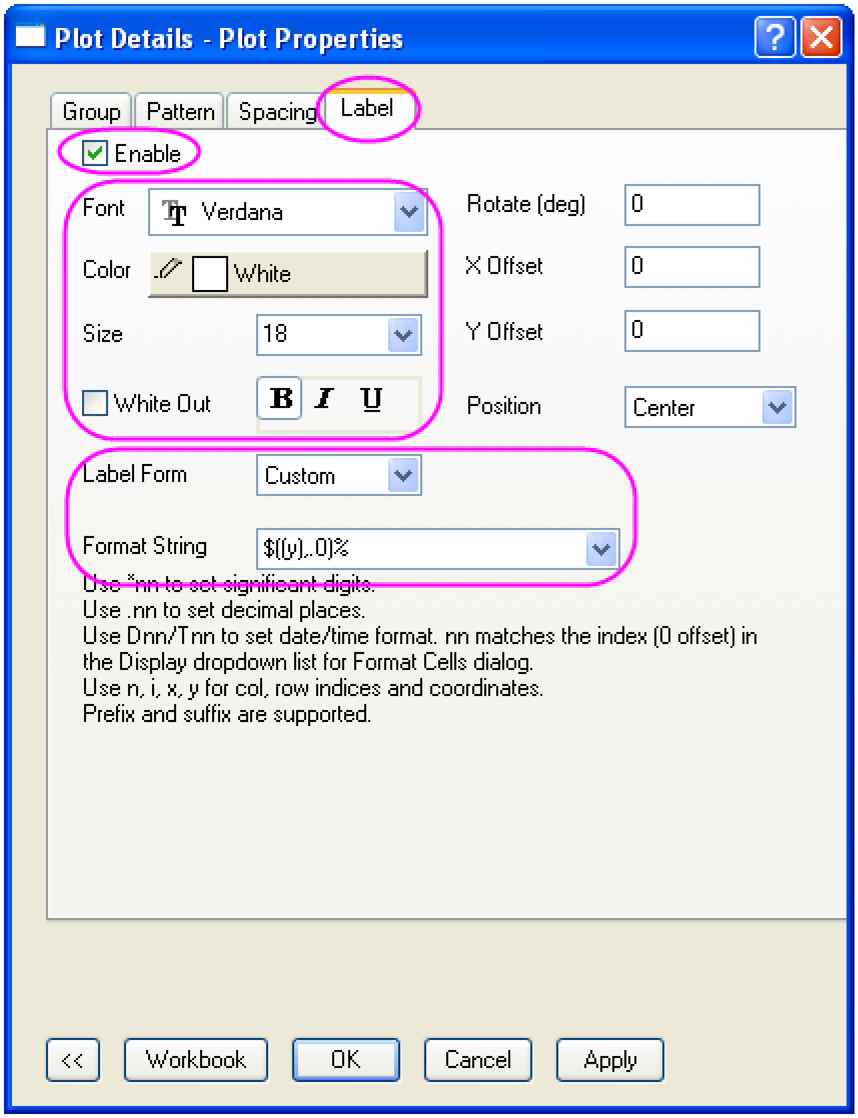
Select **Right** in the Selection box. Select **Show Axis & Ticks**, and then set **Major Ticks** and **Minor Ticks** to **None**.



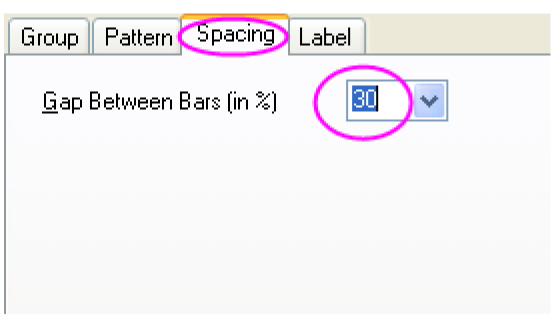
5. On the **Scale** tab, select **Vertical** in the Selection box. Set **To** to **102** and click **OK**.
6. Double-click the plot to open the **Plot Details** dialog. On the **Group** tab, set the **Fill Color** as below:



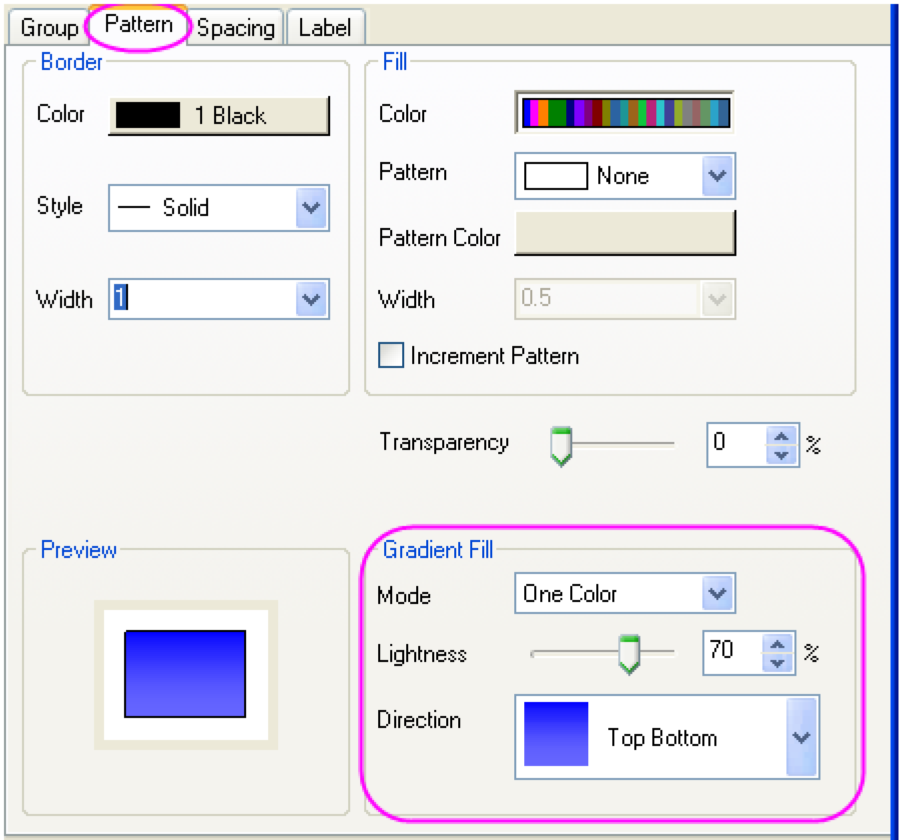
7. On the **Label** tab, select **Enable**. Set the **Font**, **Color**, and **Size** to **Verdana**, **white**, and **18**. Set **Label Form** to **Custom**, and then enter "$((y),.0)%" for the **Format String**.



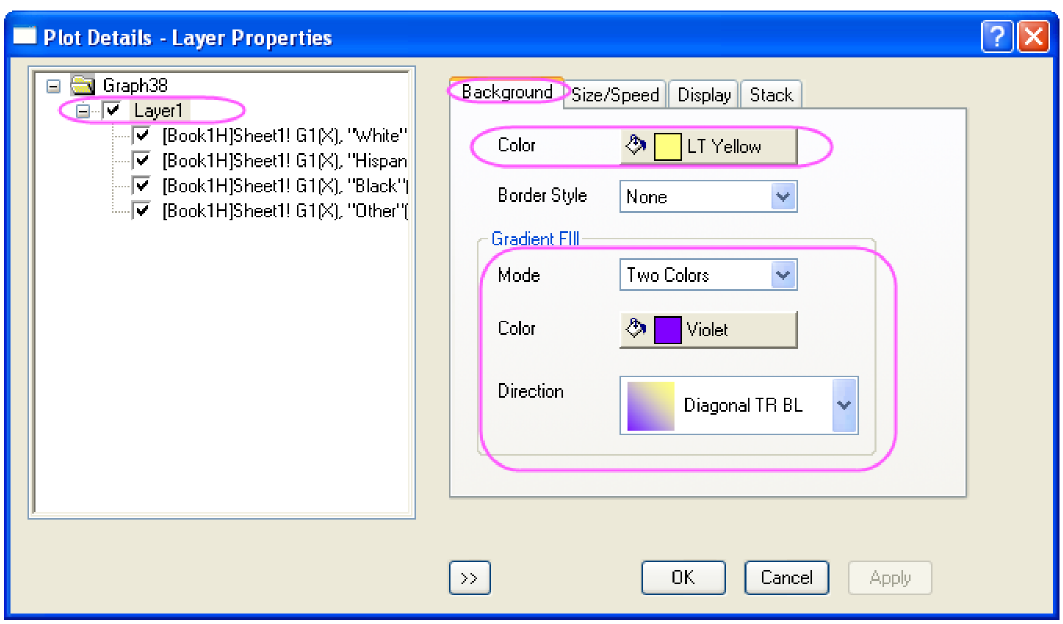
8. On the **Spacing** tab, set **Gap Between Bars(in %)** to **30**.



9. On the **Pattern** tab, set **Gradient Fill** as below:



10. In the left panel, select **Layer1**. On the **Background** tab, set **Color** and **Gradient Fill** as below:

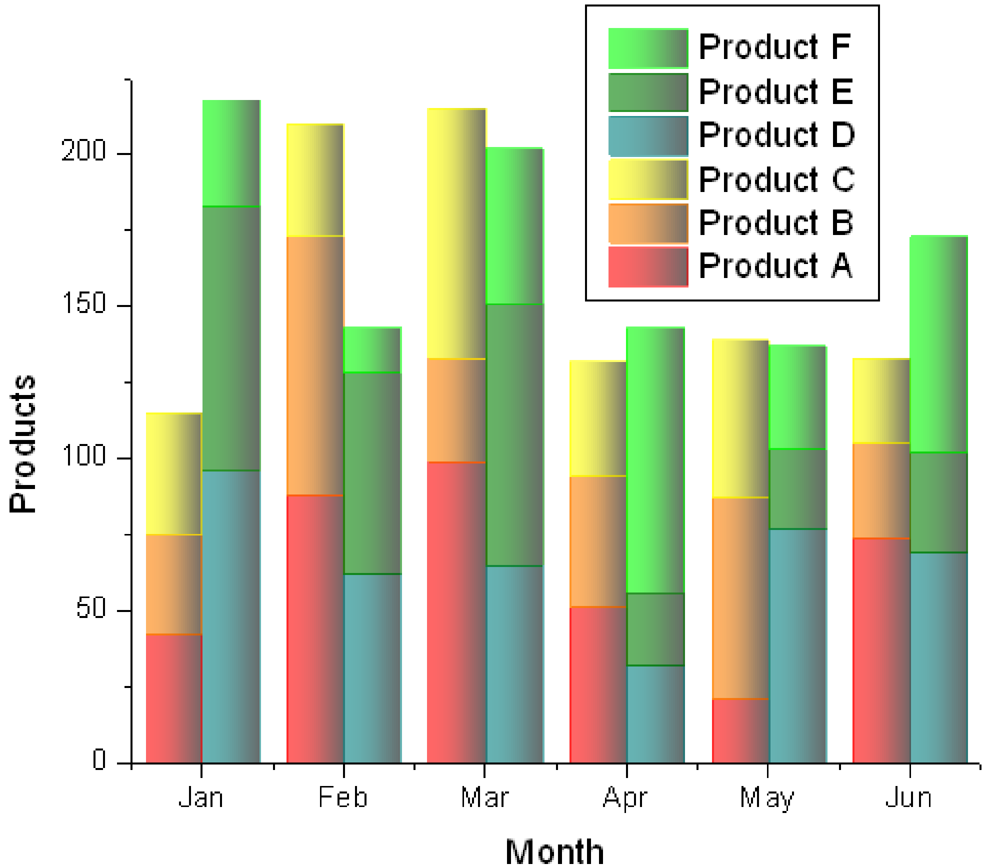


11. Click **OK** to close the dialog. To complete the graph, change the X and Y axis labels to "Year" and "Population (%)", and then add the graph title ?U.S. Population by Race.?

5.3.7 Clustered-Stacked Column Chart

Summary

This tutorial will show you how to create a clustered-stacked column chart in Origin. This graph consists of two graph layers, each of which has a stacked column.



Minimum Origin Version Required: Origin 8.5.1 SR0

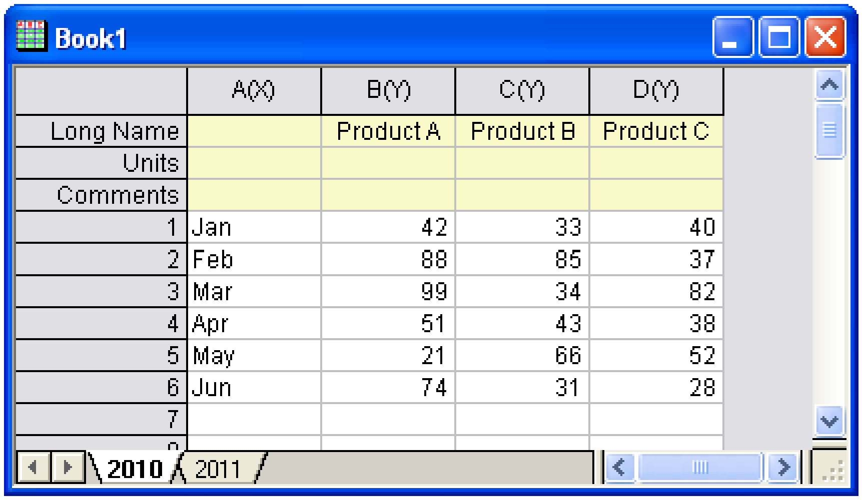
What will you learn

This tutorial will show you how to

- Create a stacked column graph.
- Create a graph that has two stacked column graphs.
- Customize a column chart.

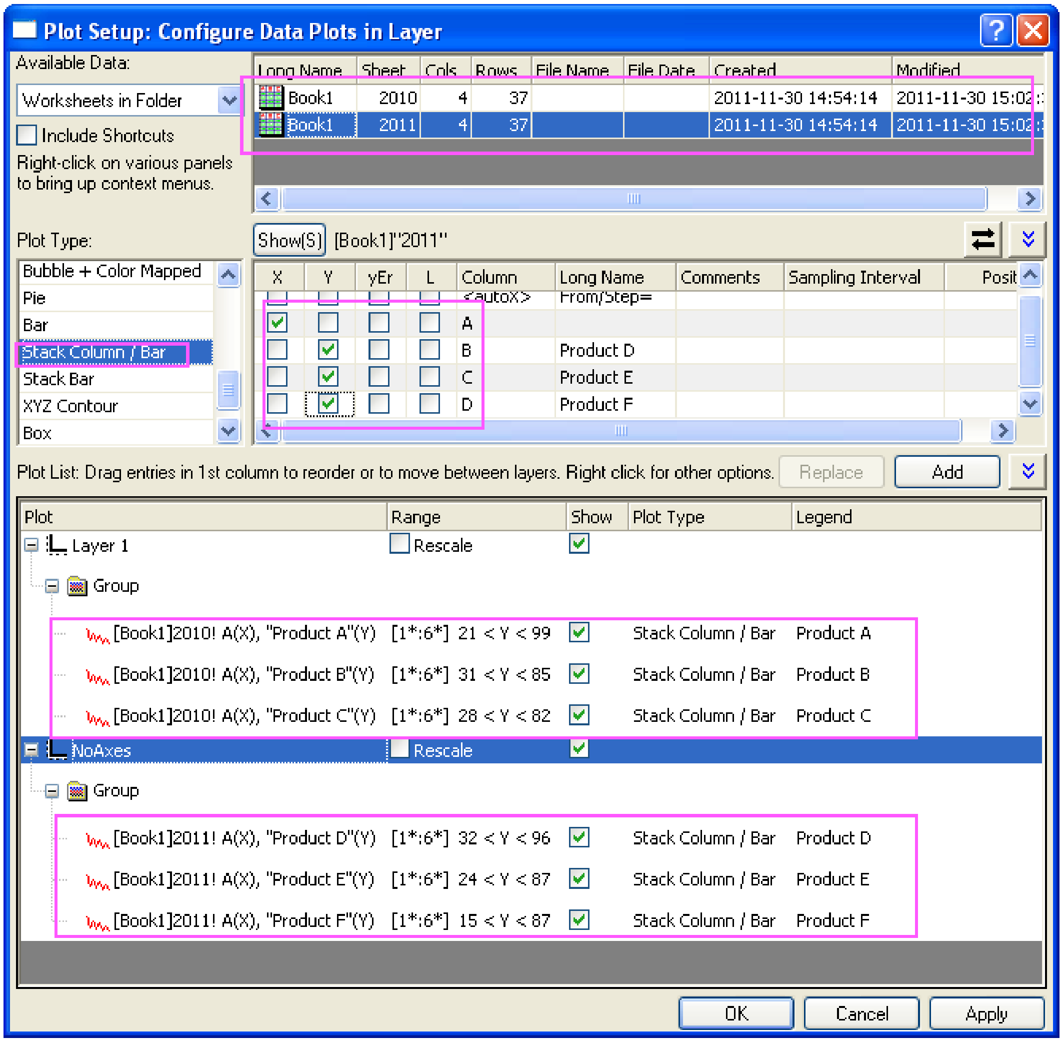
Steps

1. Import the sample data Data 1 and Data 2 into different worksheets in Origin. Set the first line as Long Name for them and set the sheet names as 2010 and 2011 respectively.

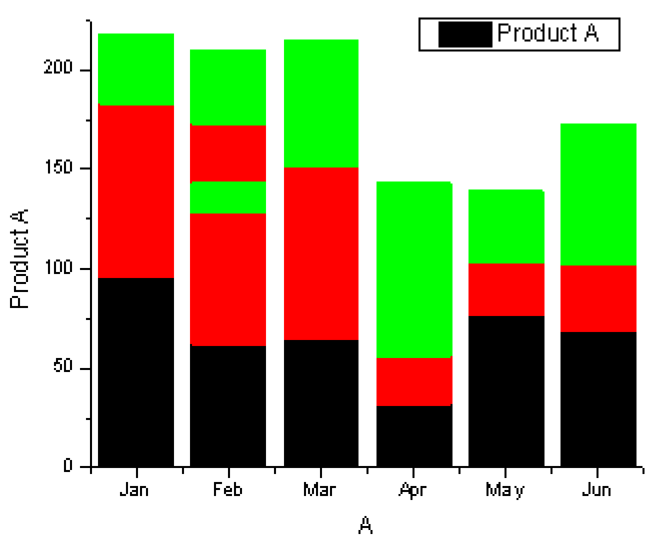


2. Create a new graph layer by selecting **File : New : Graph** from the main menu. And then select **Graph: New Layer: No Axes(Link XY Scale and Dimension)** from the main menu to add another layer to the graph window you just created.

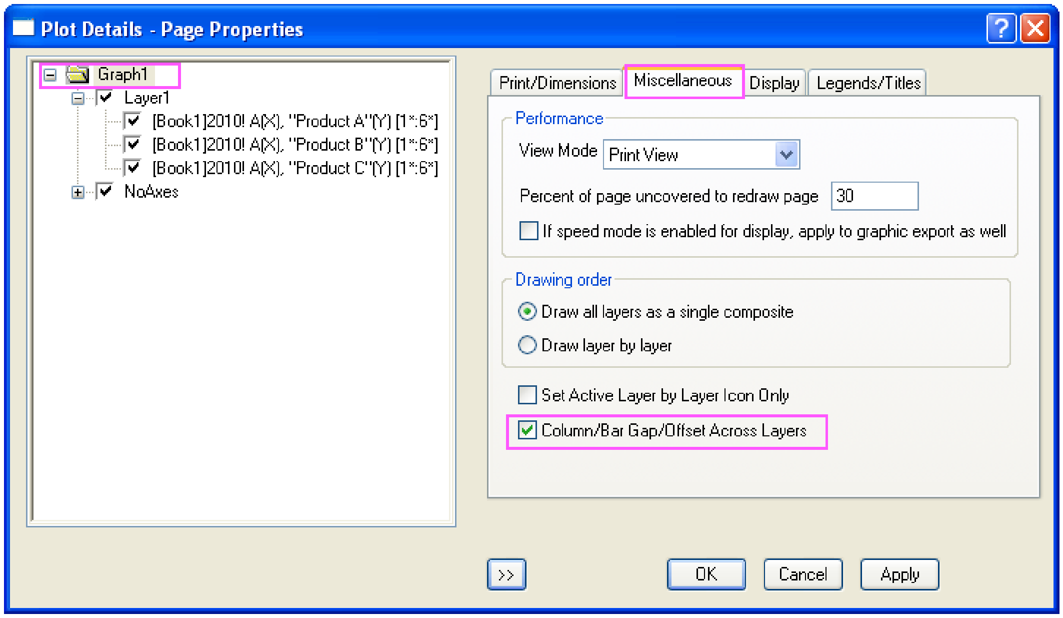
3. Right-click on the graph to select **Plot Setup...** from the context menu to open the **Plot Setup** dialog. In this dialog, select **Stacked Column/Bar** in the **Plot Type** box, select worksheet *2010* and *2011* in the top panel respectively, assign col(A) as *X* and col(B)~col(D) as *Y* in the middle panel, and then click the **Add** button to add them to **Layer 1** and **NoAxes** layer respectively.



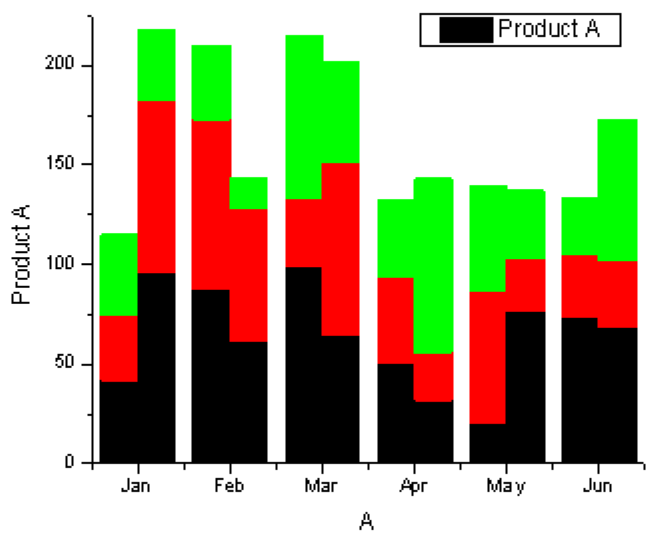
4. Click **OK** button to close the **Plot Setup** dialog. Click the **Rescale** button, you will get a graph with two stacked column graphs overlapped as shown below:



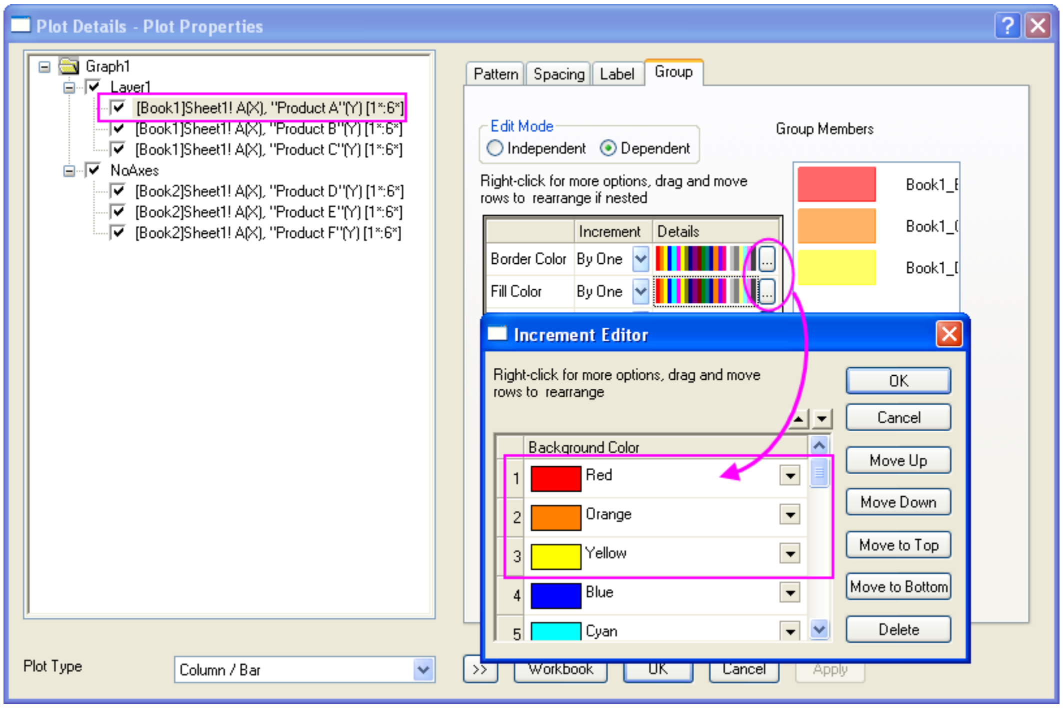
5. Double-click on the graph to open the **Plot Details** dialog. Select **Graph1** level in the left panel, go to the **Miscellaneous** tab in the right panel, and check the **Column/Bar Gap/Offset Across Layers** check box.

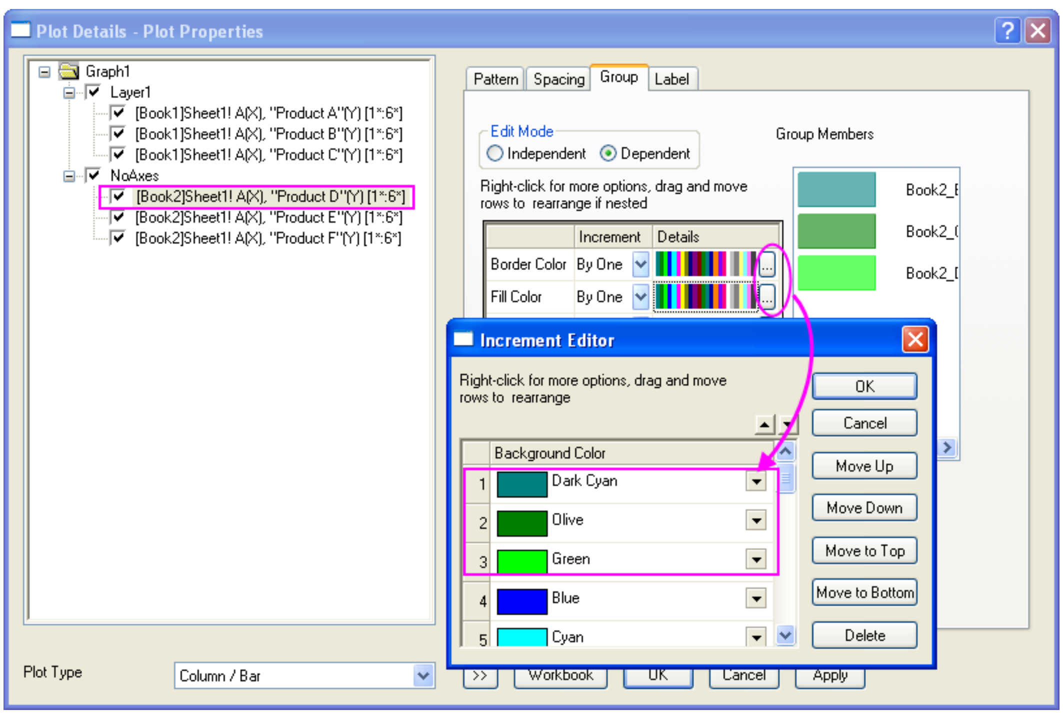


6. Click **OK** button, you will see the plot with two stacked columns will show side by side with an auto gap.

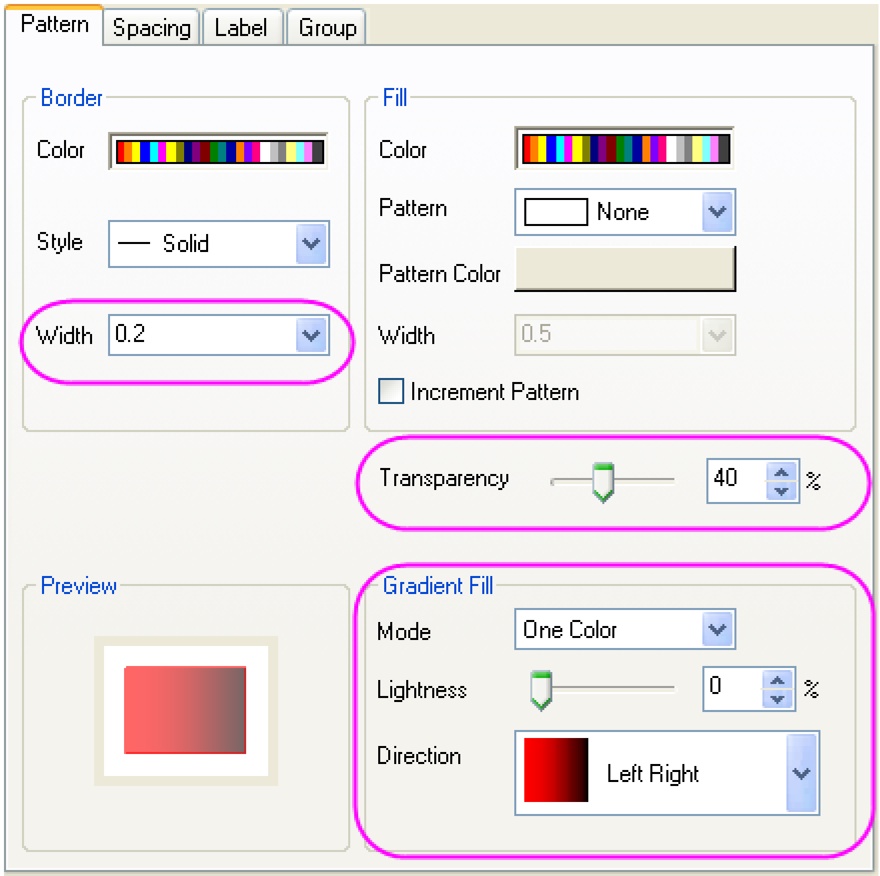


7. Double-click on the graph to open the **Plot Details** dialog again. Set the **Border Color** and **Fill Color** respectively, as shown below:

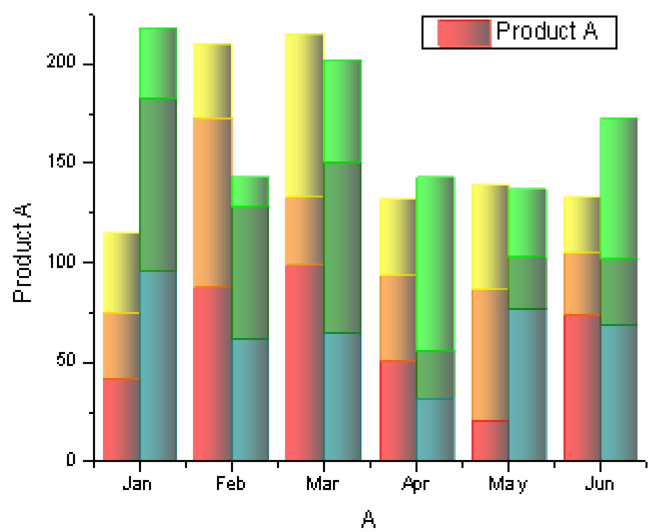




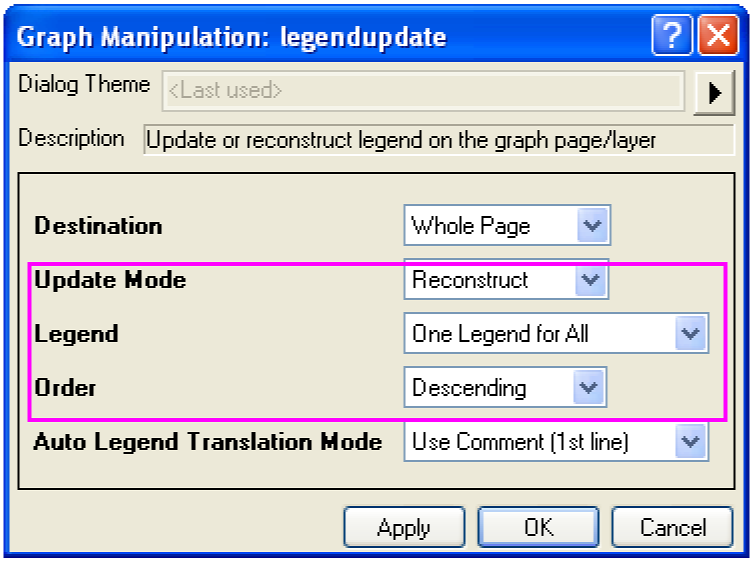
8. Go to the **Pattern** tab to set the **Transparency** as *40*, **Border Width** as *0.2*, and **Gradient Fill** as shown for both graph layers:



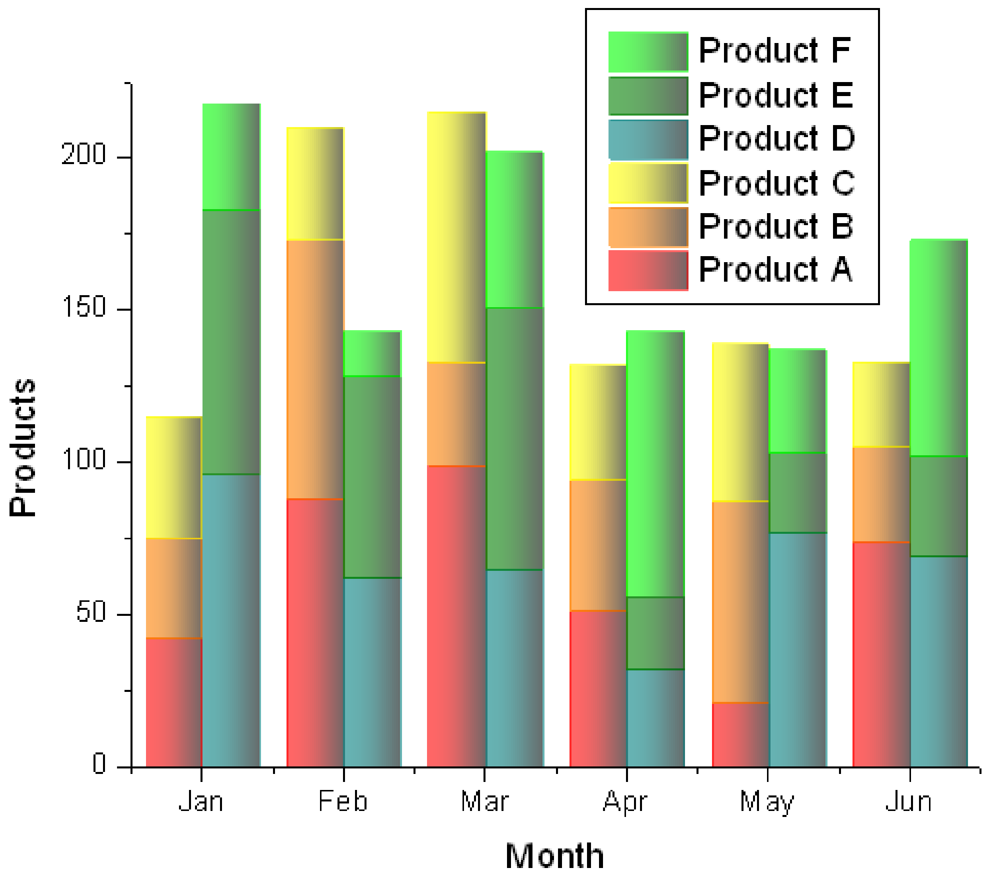
9. Click **OK** button to close the **Plot Details** dialog.



10. Right-click on the graph legend, select **Update Legend** from the context menu to open the **LegendUpdate** dialog. Set the **Update Mode** as **Reconstruct**, **Legend** as **One Legend for All**, and **Order** as **Descending**.



11. Click **OK** button to close the dialog and then update the axis labels, you will get the final graph shown below:



<u>Sample Data</u>

Data 1

	Product A	Product B	Product C
Jan	42	33	40
Feb	88	85	37
Mar	99	34	82
Apr	51	43	38
May	21	66	52
Jun	74	31	28

Data 2

	Product D	Product E	Product F
Jan	96	87	35
Feb	62	66	15
Mar	65	86	51
Apr	32	24	87
May	77	26	34
Jun	69	33	71

5.4 Multi-Axis Multi-Panel

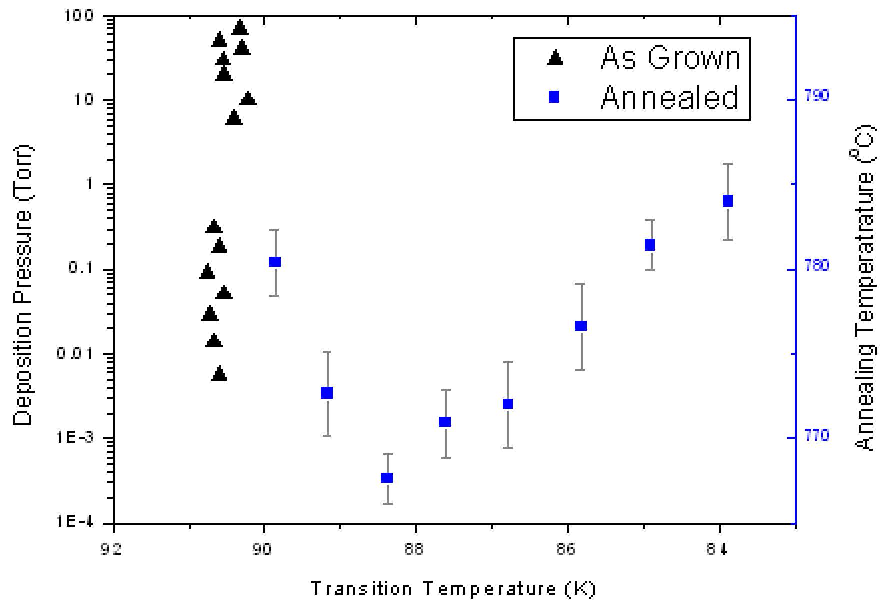
Topics covered in this section:

1. Double Y
2. 3Ys Y-YY
3. Multiple Layers with Linked Axis
4. Stack Lines by Y Offsets
5. Vertical 2 Panel Line
6. Multiple Axis Plot
7. Energy-Level Structure of the Er3+ Ion in A YAG Crystal
8. Multi-panel plot template

5.4.1 Double Y

Summary

This tutorial will show how to create a graph with double Y axes.



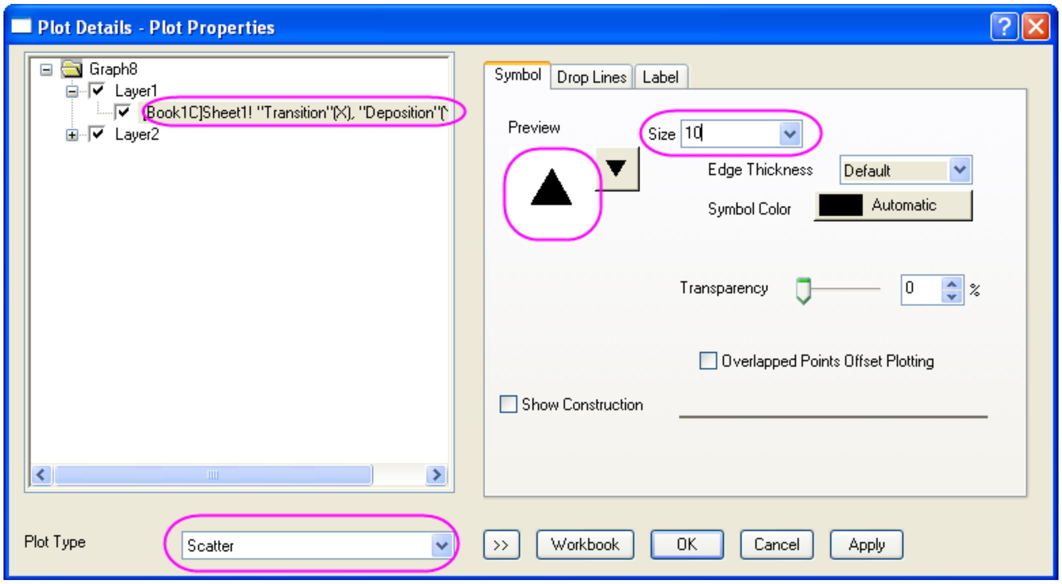
What you will learn

- Create a graph with double Y axes
- Customize settings of data scatters
- Change axis scale, type, title, etc.
- Update legend

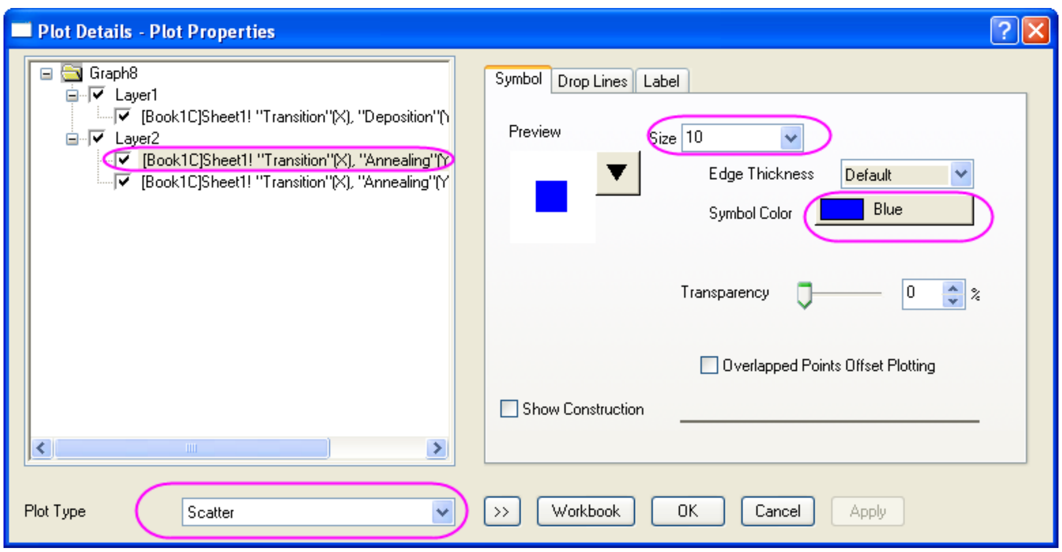
Steps

This tutorial is associated with the project: *\Samples\Graphing\Double Y.opj*.

1. Open the project *<Origin Installation Directory>\Samples\Graphing\Double Y.opj* and active the workbook.
2. Highlight four columns in the data worksheet, then choose menu **Plot: Multi-Curve: Double-Y** to create a graph with double Y axes.
3. Customize settings of data scatters.
   - Double click on the plot to open the **Plot Details** dialog. In the left panel, select the plot in Layer1, and then set the plot type as **Scatter**, symbol as **Triangle** and size as **10**.

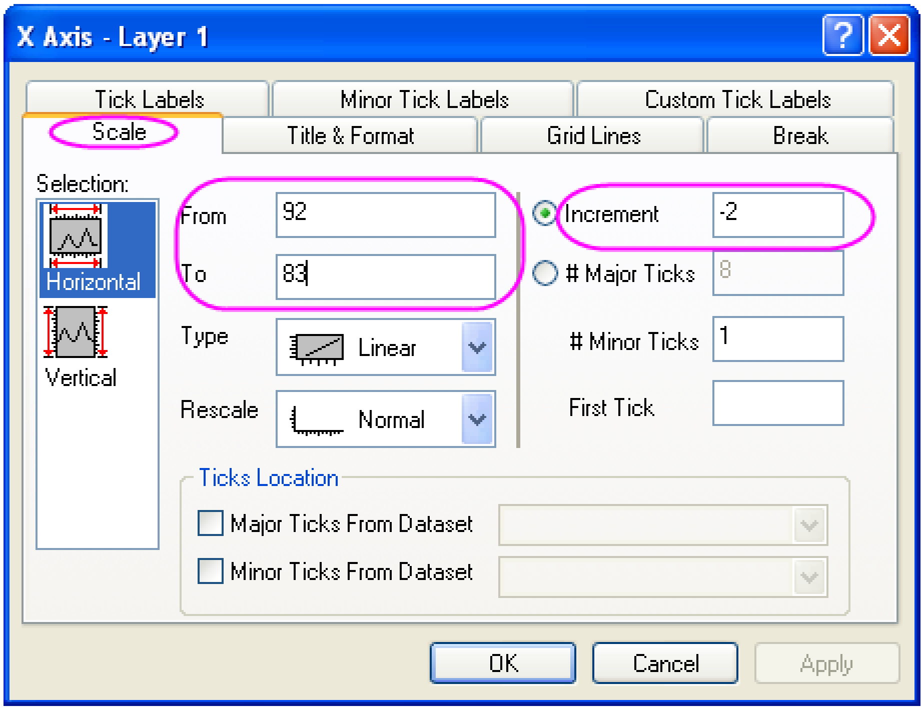


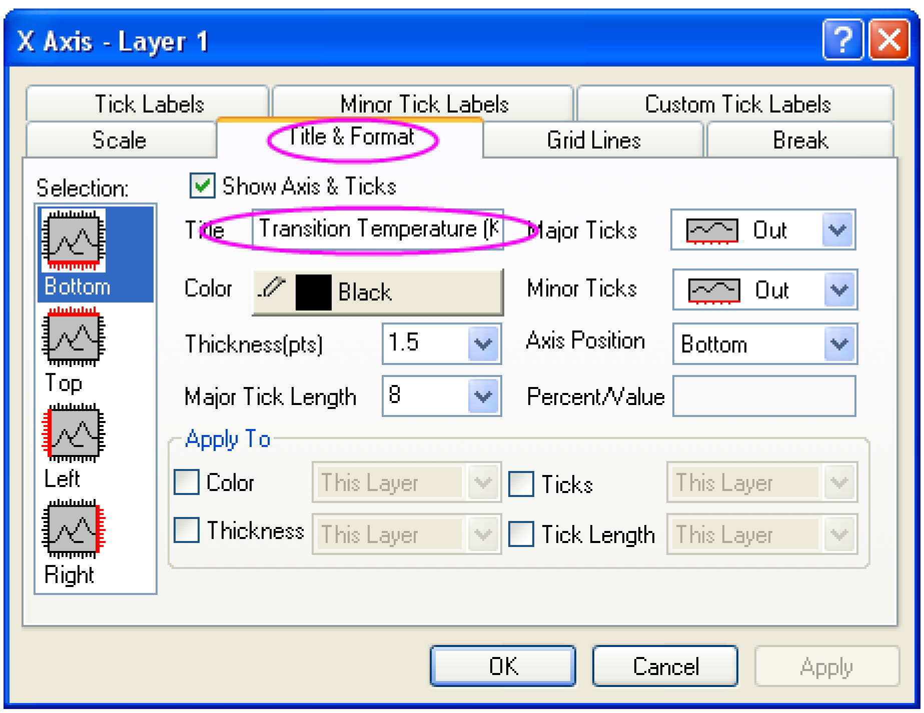
- o Choose the Annealing plot in Layer2, then change the **Plot Type**, **Size** and **Color** as below.



- o Choose the Err plot in Layer2, then change **Color** to **Gray** in the **Error Bar** tab.
- o Click the **OK** button to finish scatter settings.

4. Double click on X axis, in the opened dialog, select the **Scale** tab and change **From** *92* **To** *83* with **Increment** of *-2*. Then choose the **Title & Format** tab and set **Title** to *Transition Temperature (K)*. Click the **OK** button.



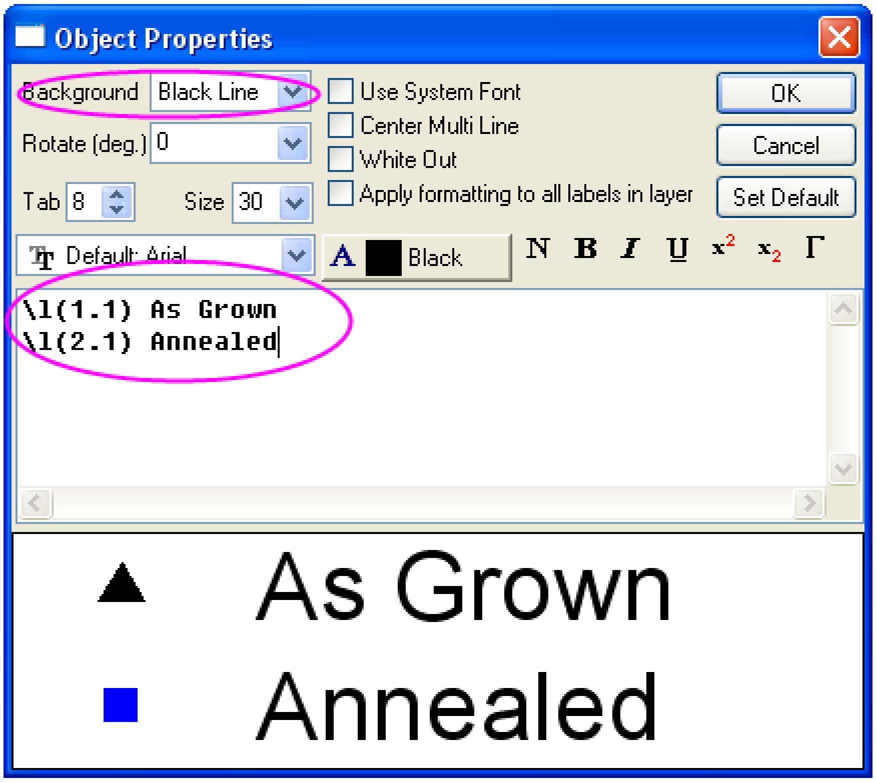


5. Double click on the left Y axis, in the opened dialog, do the same thing as last step (step 4), but with scale type of *Log10*, scale **From** *1E-4* **To** *100*, **Increment** of *1*, and **Title** to *Deposition Pressure (Torr)*.
6. Double click on the right Y axis, in the opened dialog, do the same thing as step 4, but with scale **From** *765* **To** *795*, **Increment** of *10*, **Title** to *Annealing Temperatrature (\+(0)C)*, and both **Major Ticks** and **Minor Ticks** to *In*.

7. Click the legend to select it and then choose the **Properties** from the right-click menu to open the **Object Properties** dialog. Change **Background** to *Black Line*, and in the text field, enter the following text.

\l(1.1) As Grown

\l(2.1) Annealed

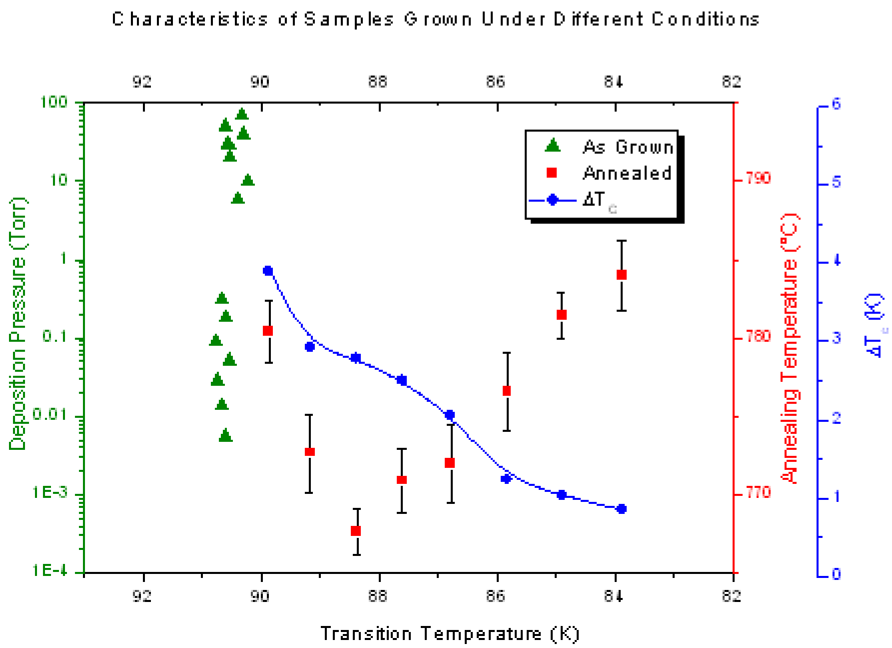


8. Move the legend to the proper position.

5.4.2 3Ys Y-YY

Summary

This tutorial will show how to create a graph with three Y axis, one left Y axes and double right Y axis.



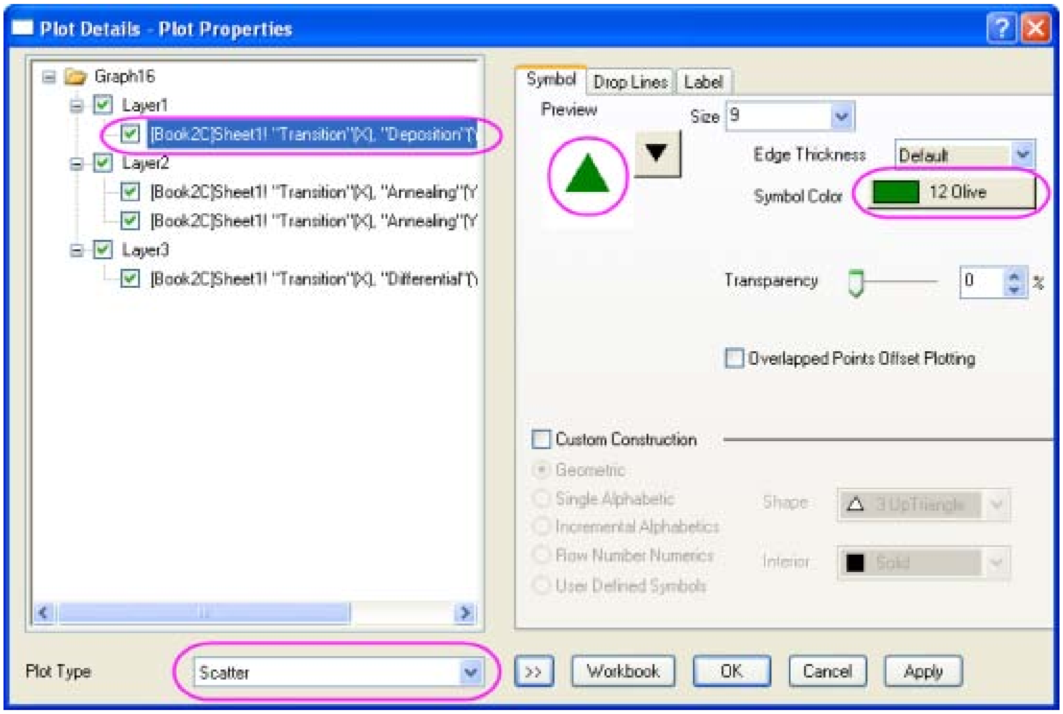
What you will learn

- Create a graph with three Y axis, one left and double right
- Customize settings of data scatters
- Change axis scale, type, title, etc.
- Show a axis
- Update legend

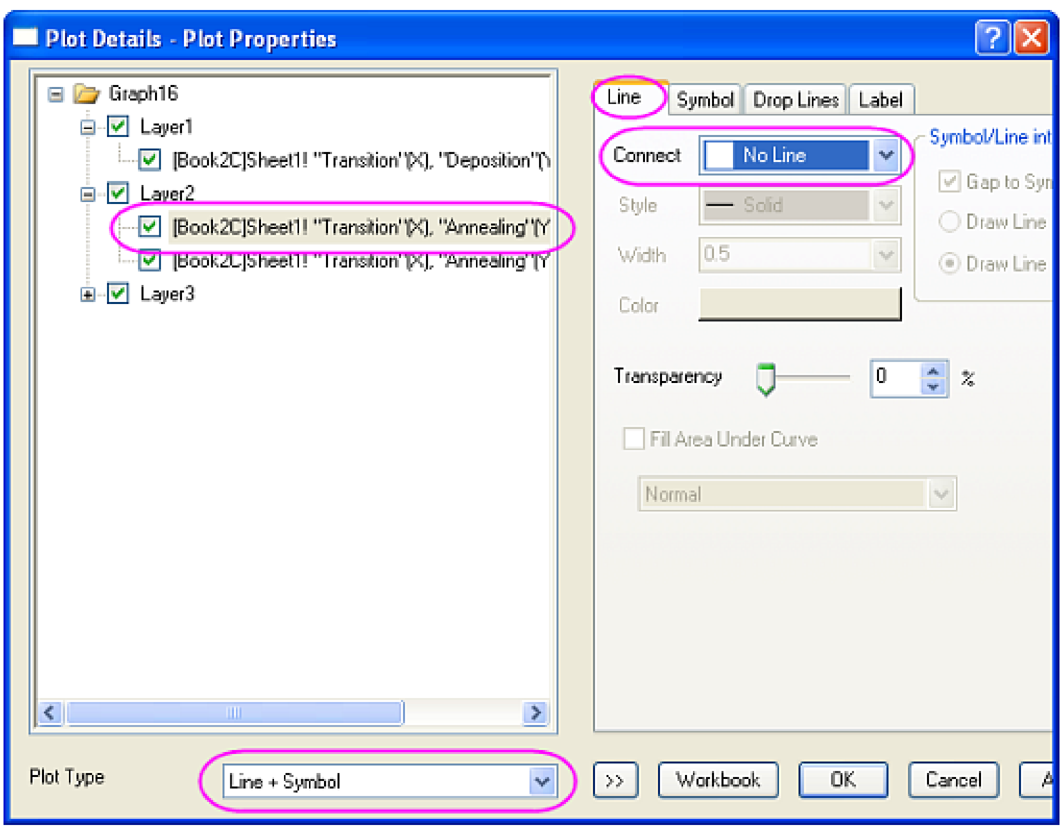
Steps

This tutorial is associated with the 2D and Contour Graphs project: *\Samples\2D and Contour Graphs.opj*.

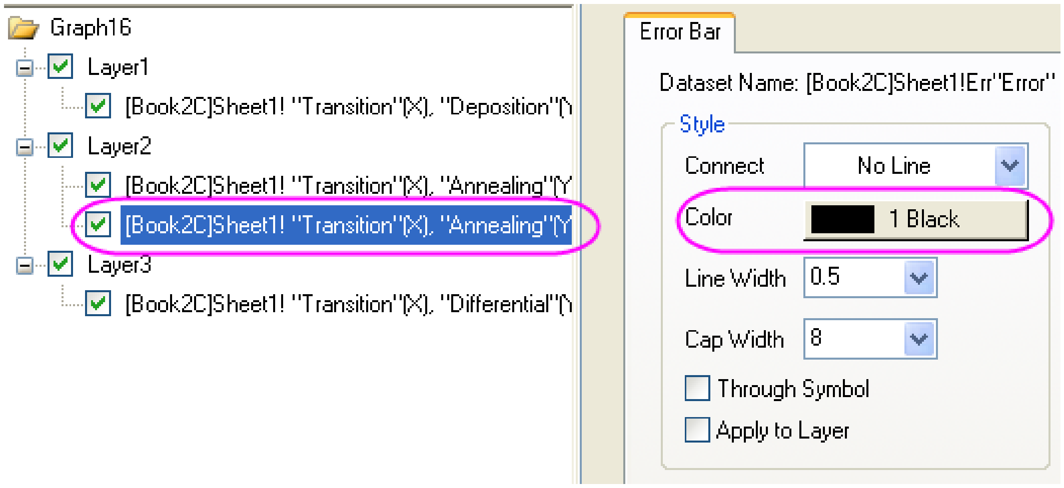
1. Open the 2D and Contour Graphs project *<Origin Installation Directory>\Samples\2D and Contour Graphs.opj*, and then browse to the *2D and Contour Graphs: Multi Axis and Multi Panel: 3Ys Y-YY* folder in **Project Explorer**.
2. Highlight all columns in the data worksheet, then choose menu **Plot: Multi-Curve: 3Ys Y-YY** to create a graph with three Y axes, one left Y and two right Y.
3. Customize data plots.
   - Double click on the plot to open the **Plot Details** dialog. In the left panel, select the plot in Layer1, then change the plot type, symbol and color as below.



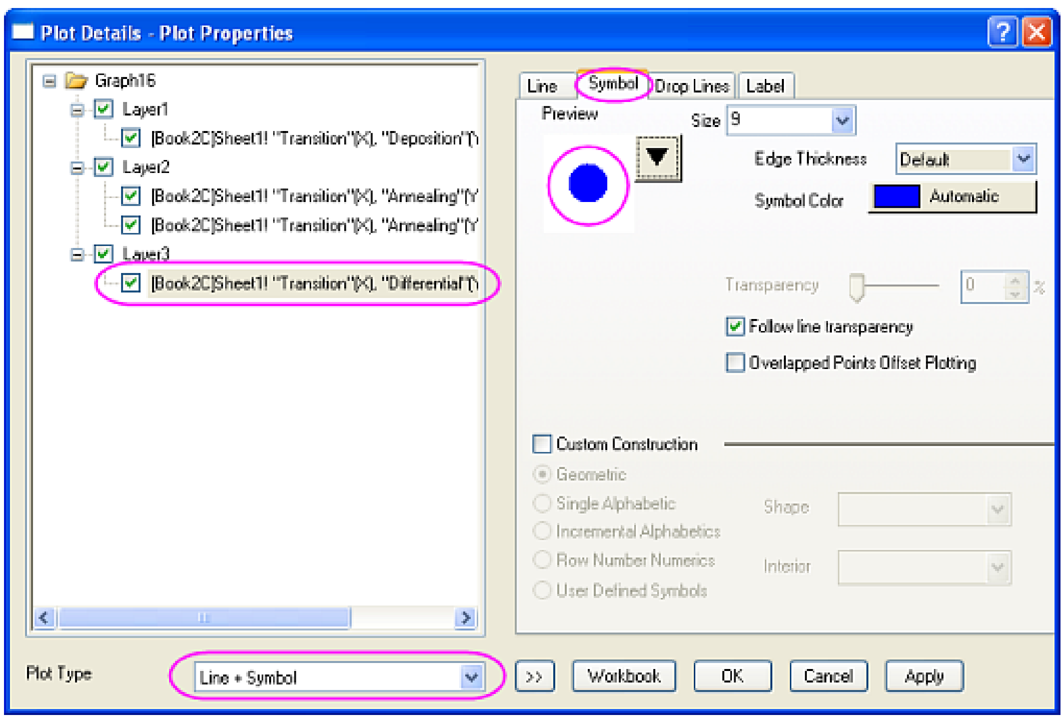
o Choose the Annealing plot in Layer2, then change the plot type to **Line + Symbol**. In the right panel, go to the **Line** tab, set **Connect** to **No Line**.



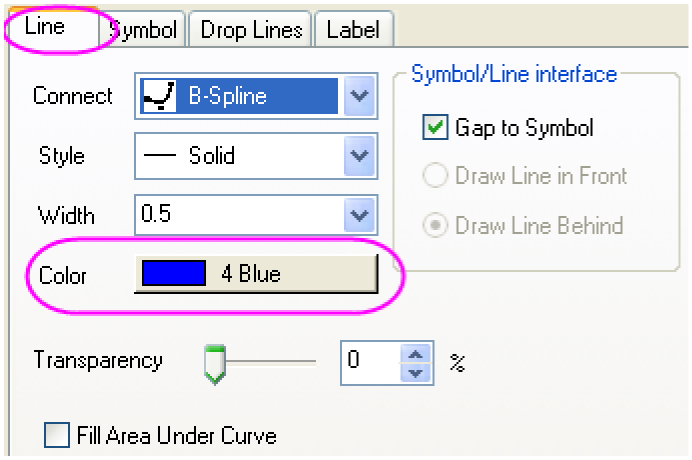
o Choose the Err plot in Layer2, then change color to Black.



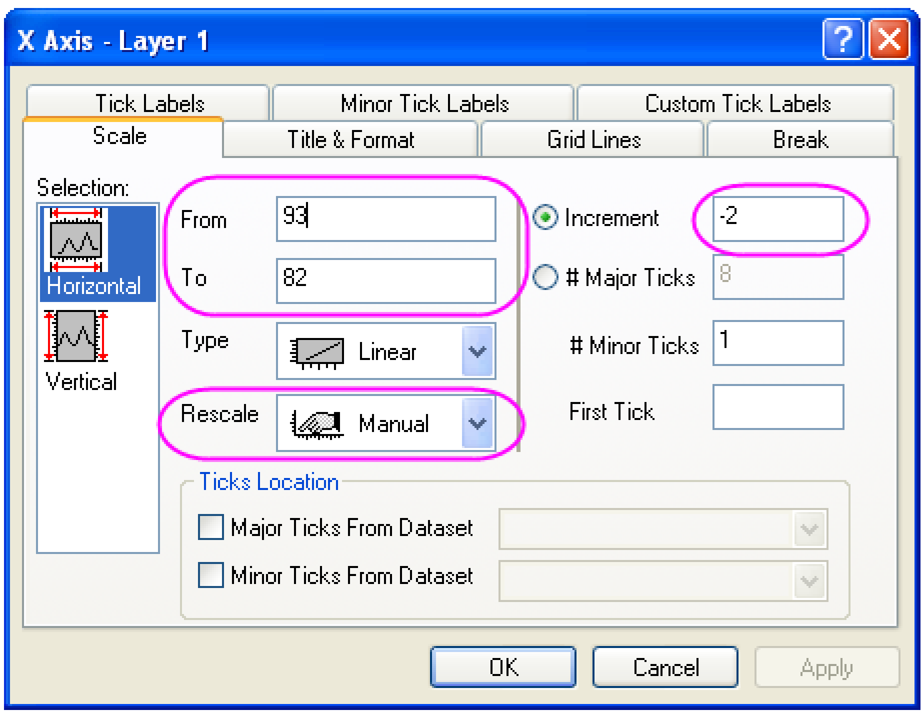
- Choose the plot in Layer3, then in the **Symbol** tab of the right panel, change the plot type to **Line + Symbol** and symbol to solid circle.

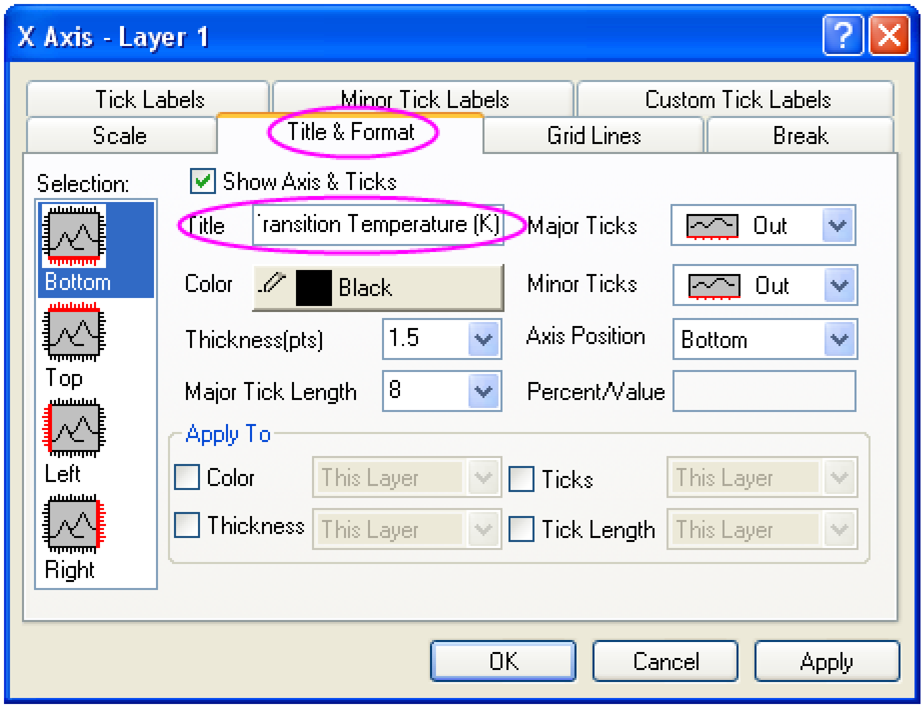


- Select the **Line** tab when the plot in Layer 3 is selected and change **Connect** to **B-Spline**.

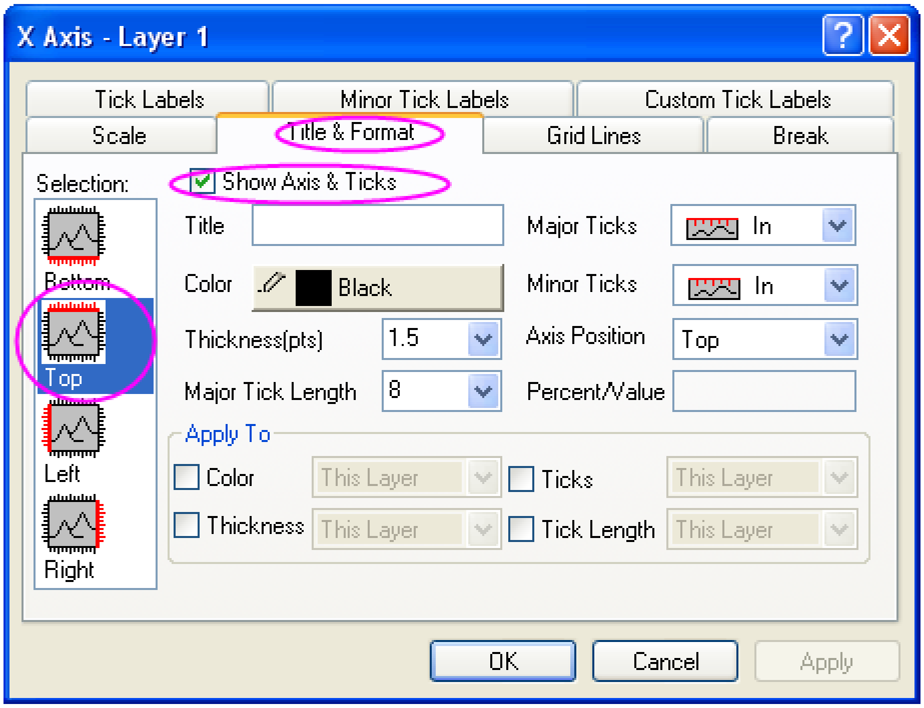


4. Double click on X axis, in the opened dialog, select the **Scale** tab and change **From** *93* **To** *82* with **Increment** of *-2*, and select **Manual** for **Rescale**. Then choose the **Title & Format** tab and set **Title** to *Transition Temperature (K)*. Click the **OK** button.

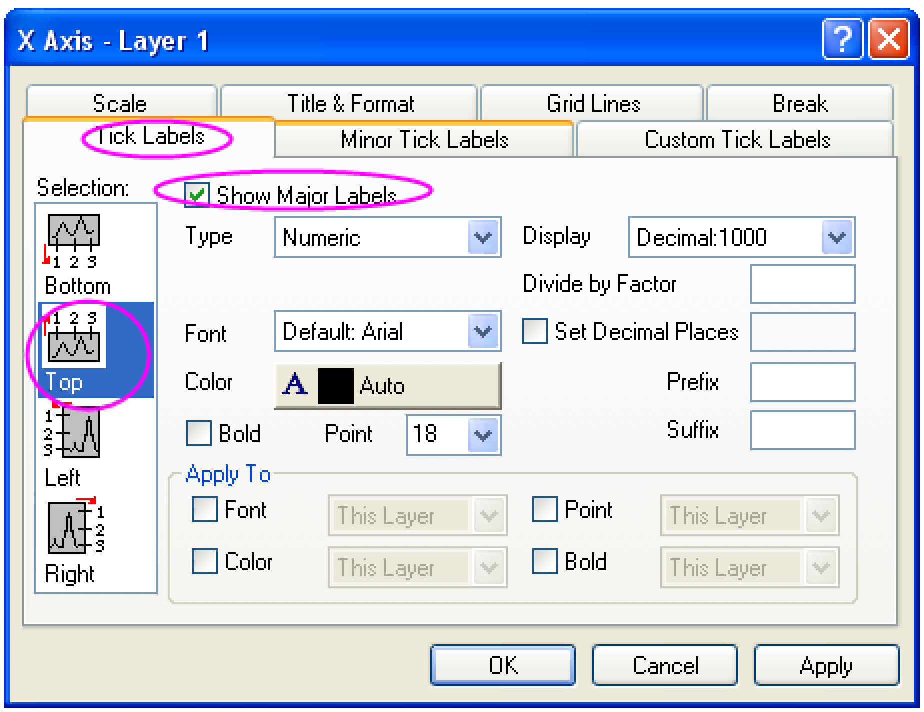




5. Double click on the left Y axis, in the opened dialog, do the same thing as last step (step 4), but with scale type of **Log10**, scale **From** *1E-4* **To** *100*, **Increment** of *1*, and **Title** to *Deposition Pressure (Torr)*.
6. Double click on the right red Y axis, in the opened dialog, do the same thing as step 4, but with scale **From** *765* **To** *795*, **Increment** of *10*, **Title** to *Annealing Temperatrature (\+(0)C)*.
7. Double click on the right blue Y axis, in the opened dialog, do the same thing as step 4, but with scale **From** *0* **To** *6*, **Increment** of *1*, **Title** to *\g(D)T\-(c) (K)*.
8. Show the top axis.
   - Double click on the bottom X axis, in the opened dialog, select the **Title & Format** tab and then choose **Top** from the **Selection** box. Check the **Show Axis & Ticks** check-box.



o Switch to the **Tick Labels** tab, check the **Show Major Labels** check-box.

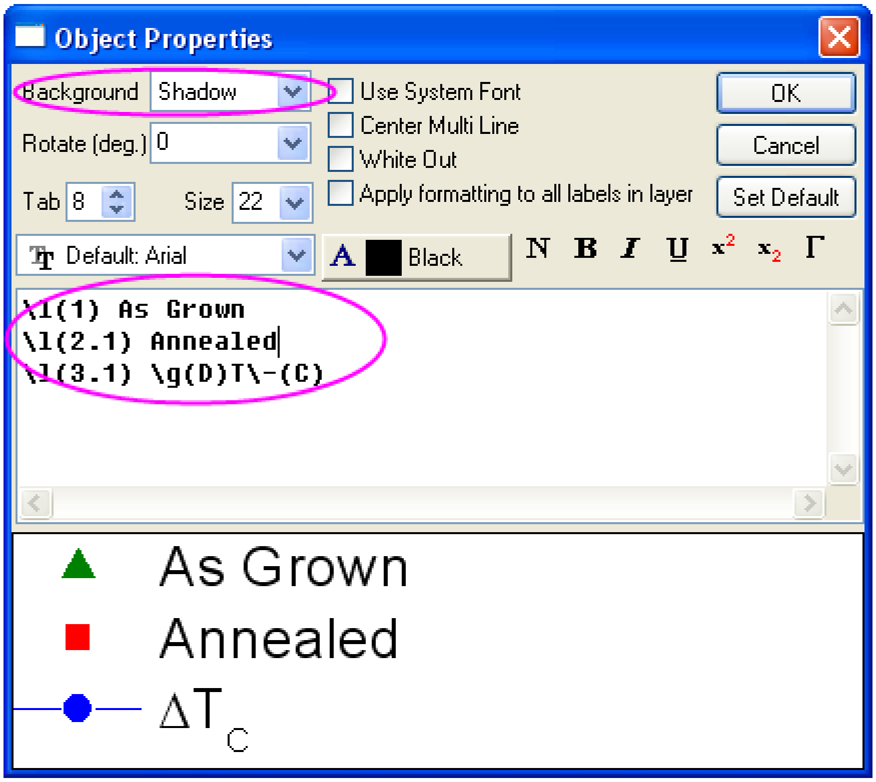


9. Click the legend to select it and then choose the **Properties** from the right-click menu to open the **Object Properties** dialog. Change **Background** to **Shadow**, and in the text field, enter the following text. Then move the legend to the proper position.

\l(1) As Grown

\l(2.1) Annealed

\l(3.1) \g(D)T\-(C)

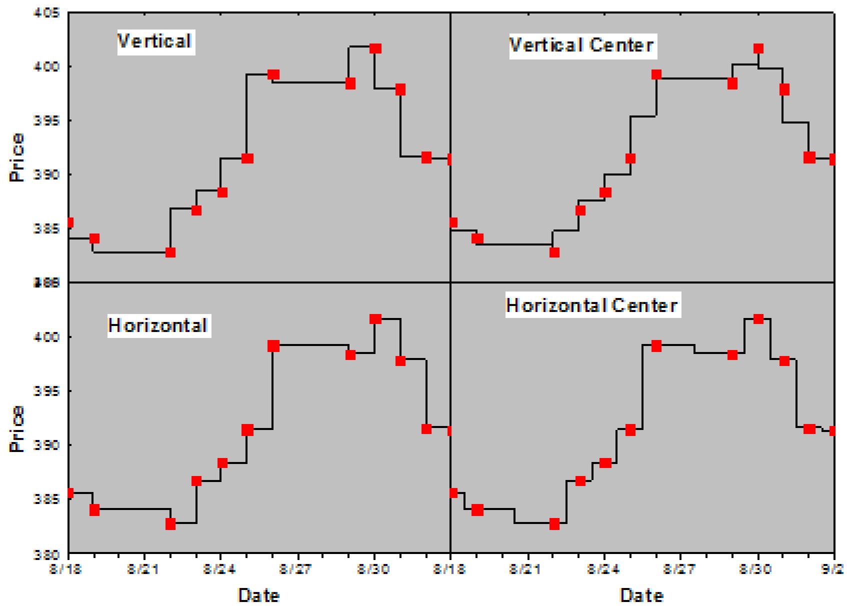


10. Add a text with *Characteristics of Samples Grown Under Different Conditions* as the graph title.

5.4.3 Multiple Layers with Linked Axis

Summary

This tutorial will show how to merge four graphs to one graph, which contains multiple layers with linked axis.



What you will learn

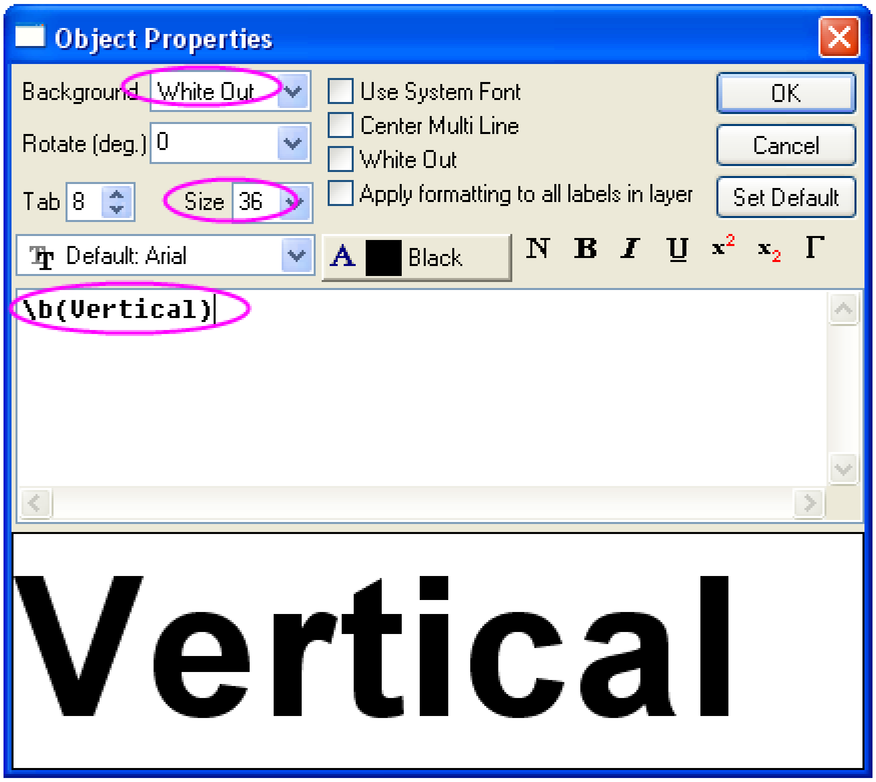
- How to make a Line + Symbol plot
- How to merge graphs
- How to link axis of layers
- How to customize axis

Steps

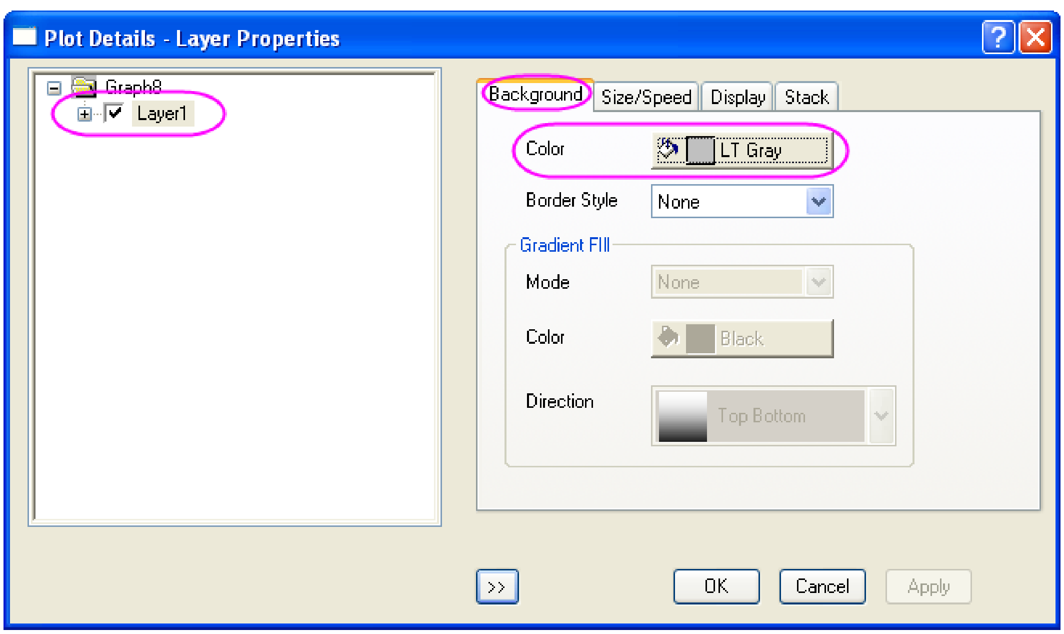
This tutorial is associated with the 2D and Contour Graphs project: *\Samples\2D and Contour Graphs.opj*.

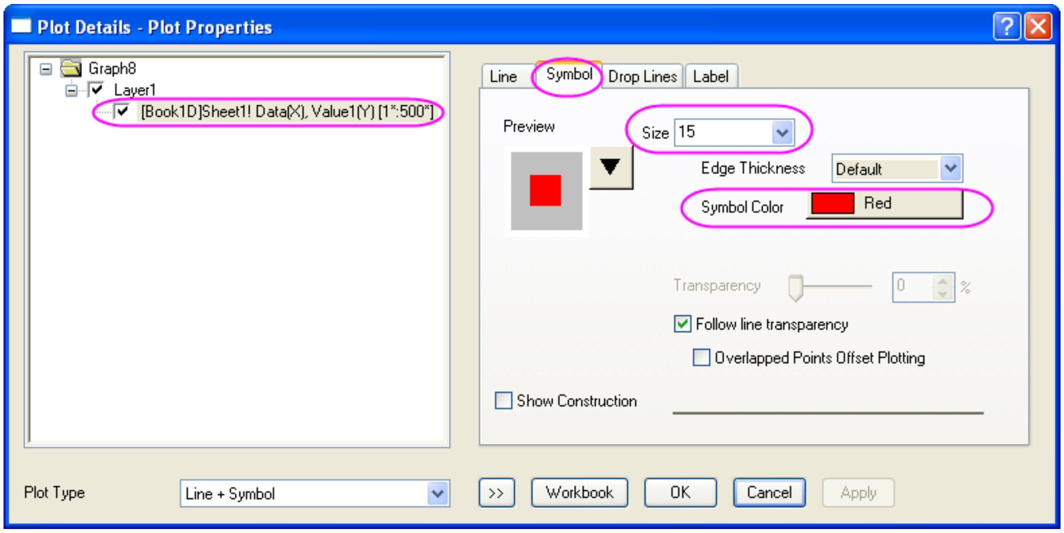
1. Open the 2D and Contour Graphs project, open the folder 2D and Contour Graphs: Multi Axis and Multi Panel: Multiple Layers with Step Plot in the **Project Explorer**.
2. Activate the workbook and highlight column Value1 and make a graph by selecting Origin menu **Plot: Line + Symbol: Line + Symbol**.
3. Activate the graph, right-click the legend and choose **Properties** from the context menu. In the **Object Properties** dialog, change the **Background** to **White Out**, **Size** to *36* and put the following text in the text field: *\b(Vertical)*.

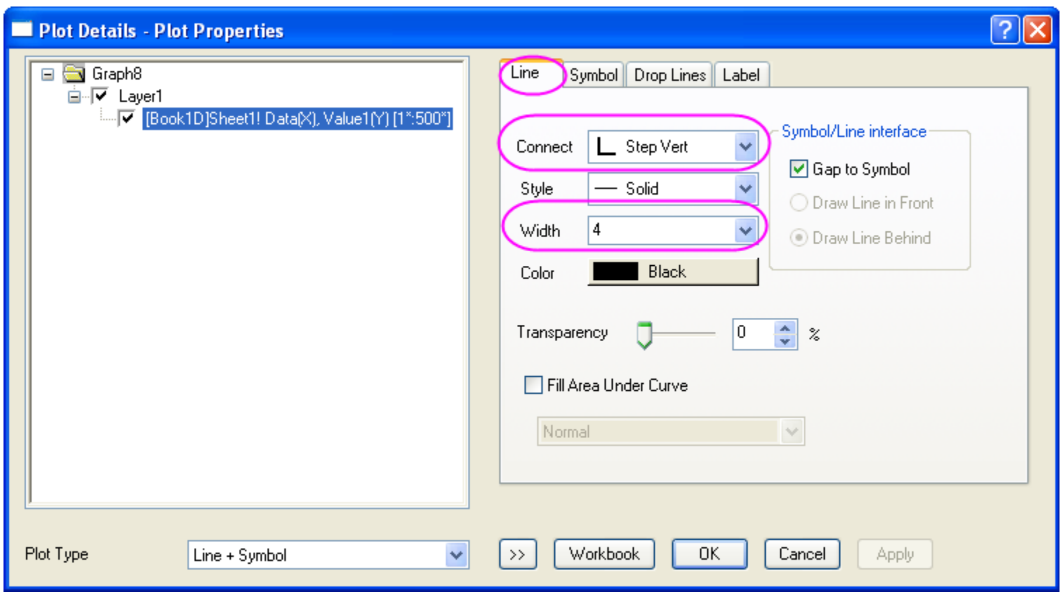
Click the **OK** button. Move the legend to the correct position.



4. Double click on the open white space in layer of the graph to open the **Plot Details** dialog. In the left panel, select **Layer1**. Activate the **Background** tab in the right panel, choose *LT Gray* for the **Color**. Then select the plot in **Layer1** from the left panel, under the **Symbol** tab in the right panel, set **Size** to *15* and **Symbol Color** to red. Switch to the **Line** tab, choose **Step Vert** from the **Connect** drop-down list and set **Width** to *4*.







5. Repeat step 2 to step 4 for column Value2, Value3 and Value4 respectively. For each column, the legend text and **Connect** drop-down list will be different, listed below.

   For column **Value2**:

legend text = \b(Vertical Center)

Connect = Step V Center

   For column **Value3**:

legend text = \b(Horizontal)

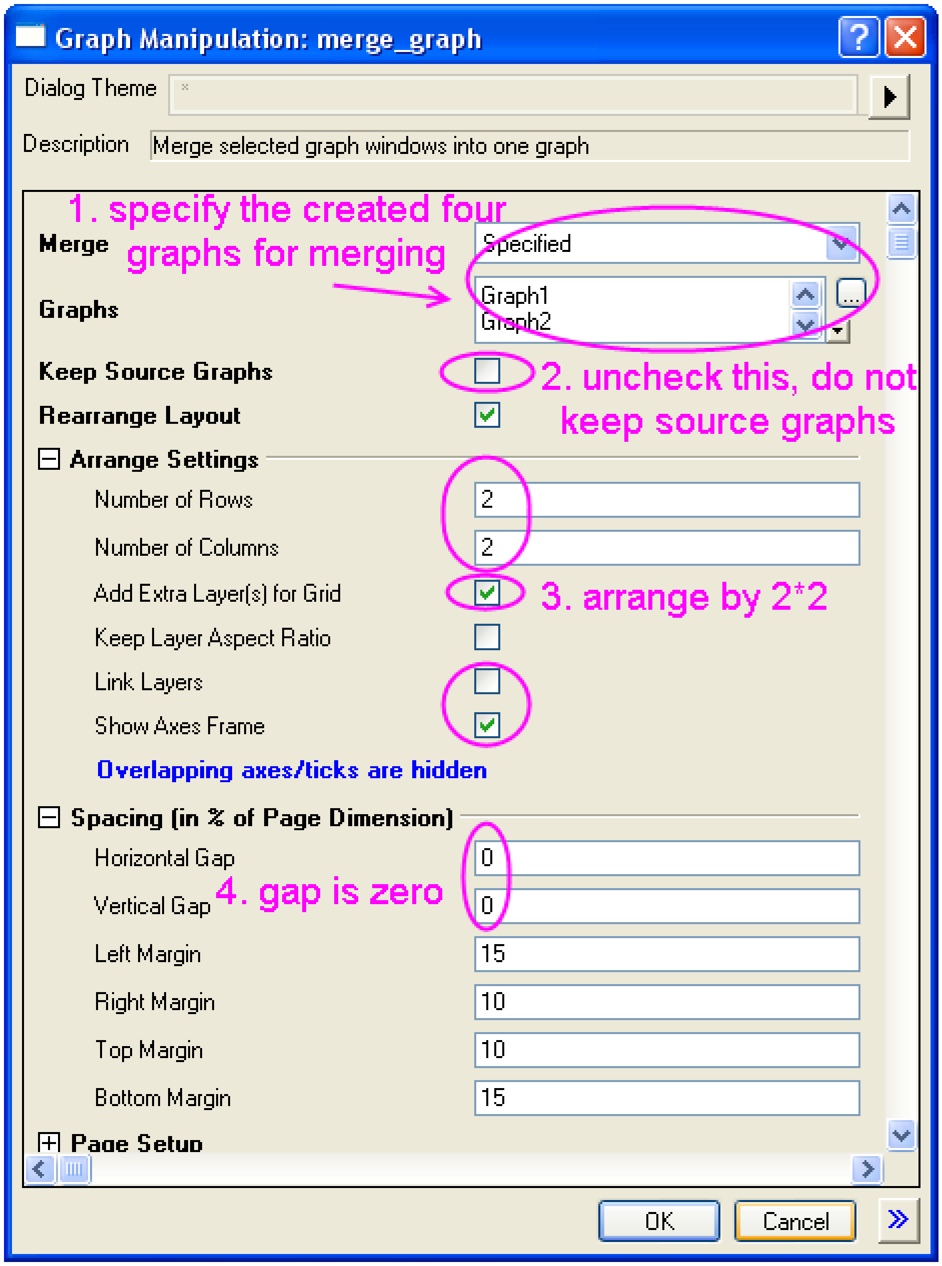
Connect = Step Horz

   For column **Value4**:

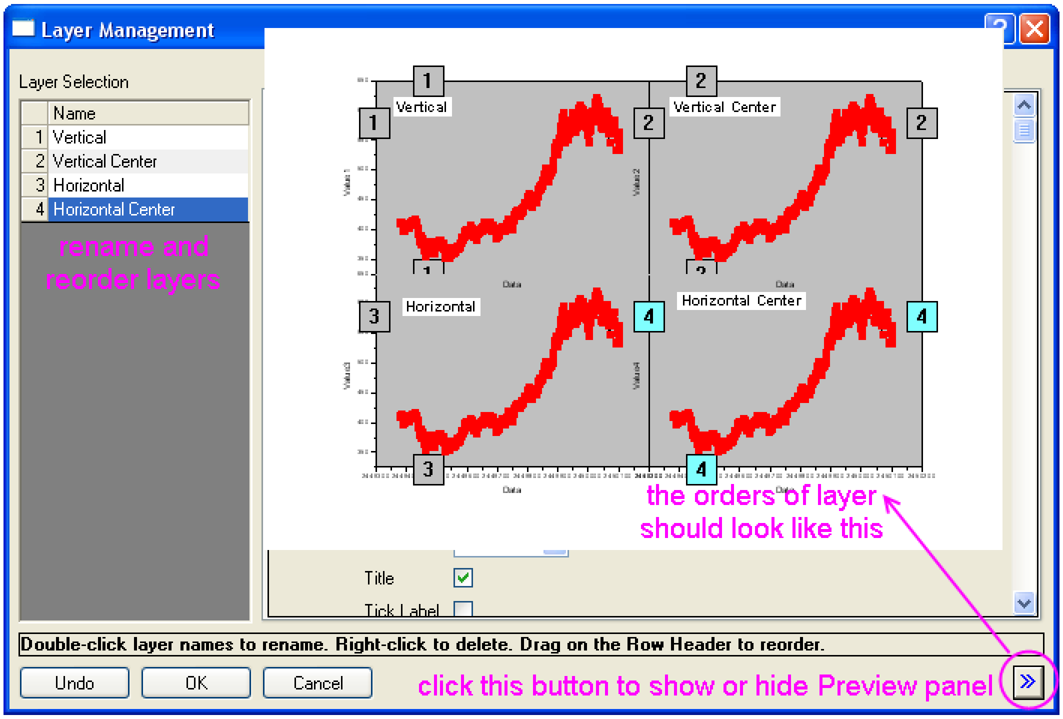
legend text = \b(Horizontal Center)

Connect = Step H Center

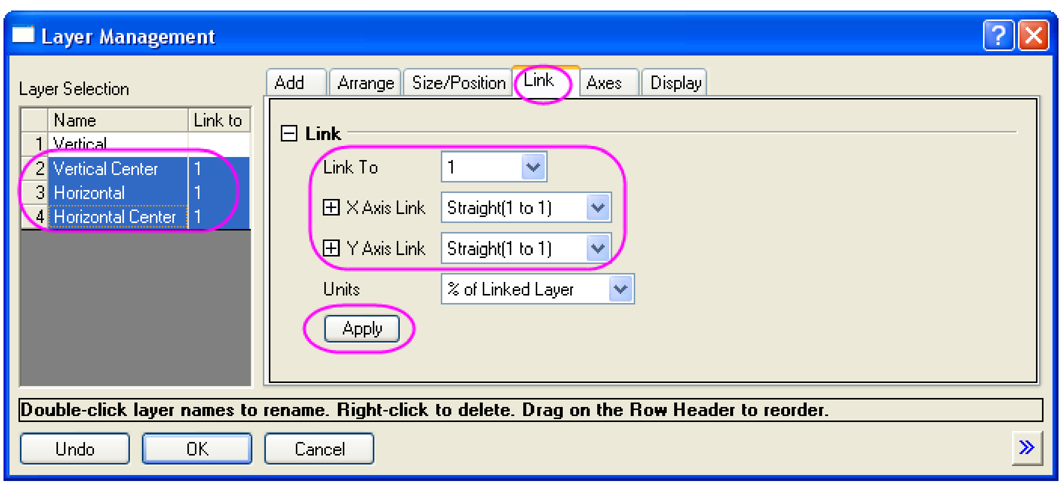
6. Activate a graph and then select Origin menu **Graph: Merge Graph Windows** to open the **merge_graph** dialog. Change the settings as the following image shows. Then click the **OK** button to merge these graphs.



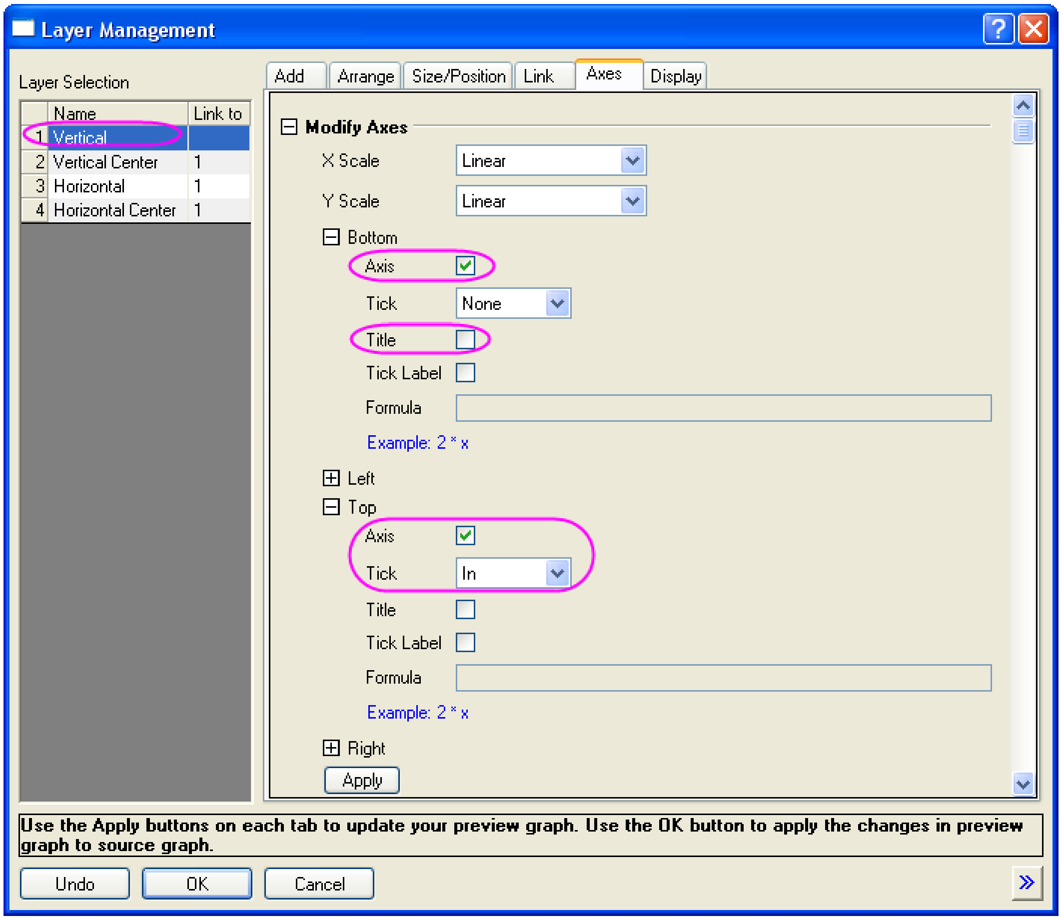
7. Activate the new merged graph, choose Origin menu **Graph: Layer Management** to open the **Layer Management** dialog. In the **Layer Selection** panel, rename the layers by double clicking on the name and reorder the layers by clicking on the layer index and dragging up and down. Make sure the final layers' names and order are the same as the image shows below (In the **Preview** panel, the layers' indices and positions can be shown).



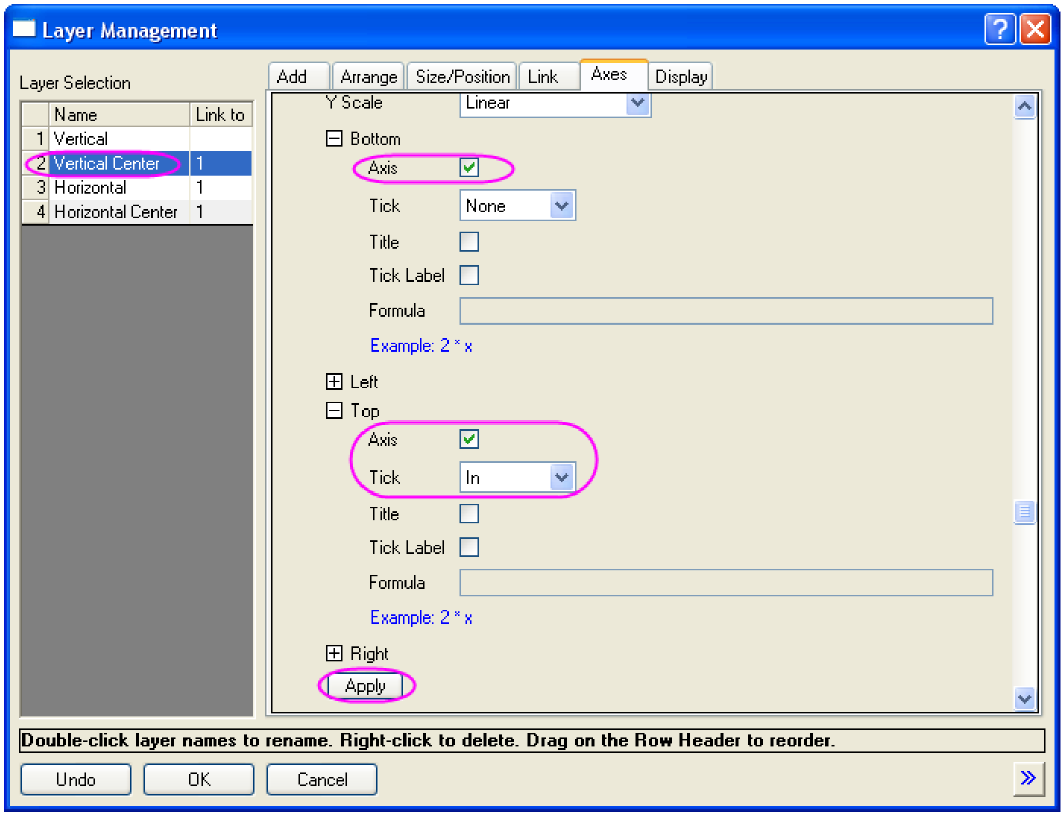
8. In the **Layer Selection** panel, press **Ctrl** key on the keyboard to select the following layers: *Vertical Center*, *Horizontal* and *Horizontal Center*. Go to the **Link** tab, select *1* from the **Link To** drop-down list. Then both **X Axis Link** and **Y Axis Link** are set to **Straight(1 to 1)**. Then click the **Apply** button.



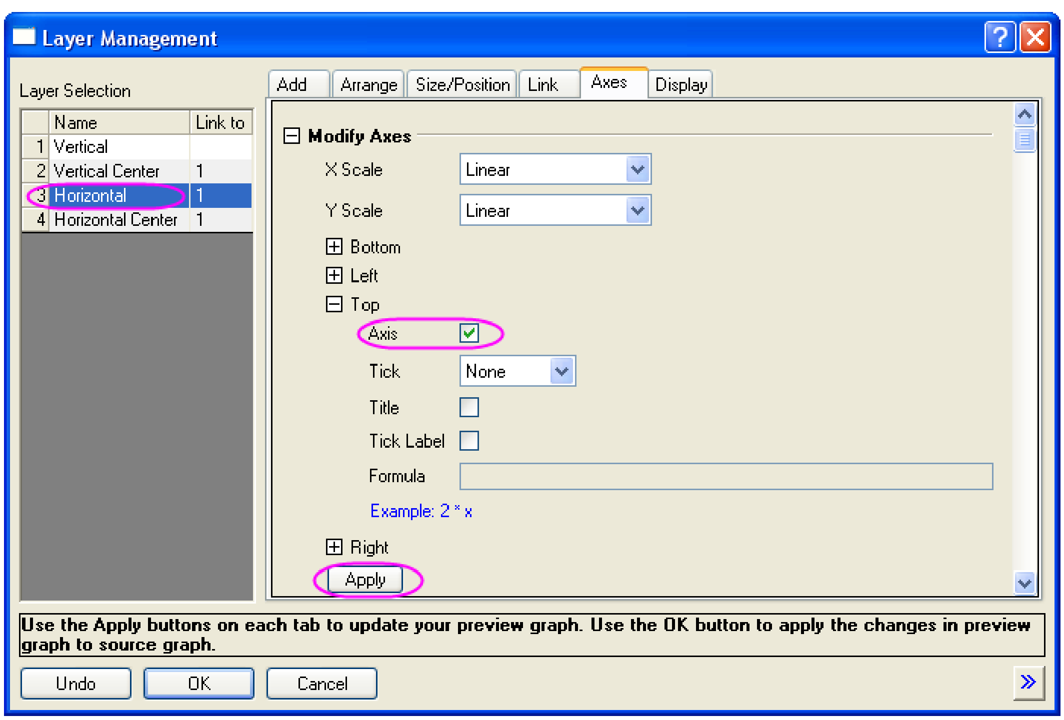
9. Go to the **Axes** tab, select *Vertical* layer from the left panel, then check all **Axis** check-boxes under **Bottom** branch, **Left** branch, **Top** branch and **Right** branch. Also, uncheck the **Title** check-box under **Bottom** branch and select *In* for **Tick** drop-down list under **Top** branch. Click the **Apply** button.



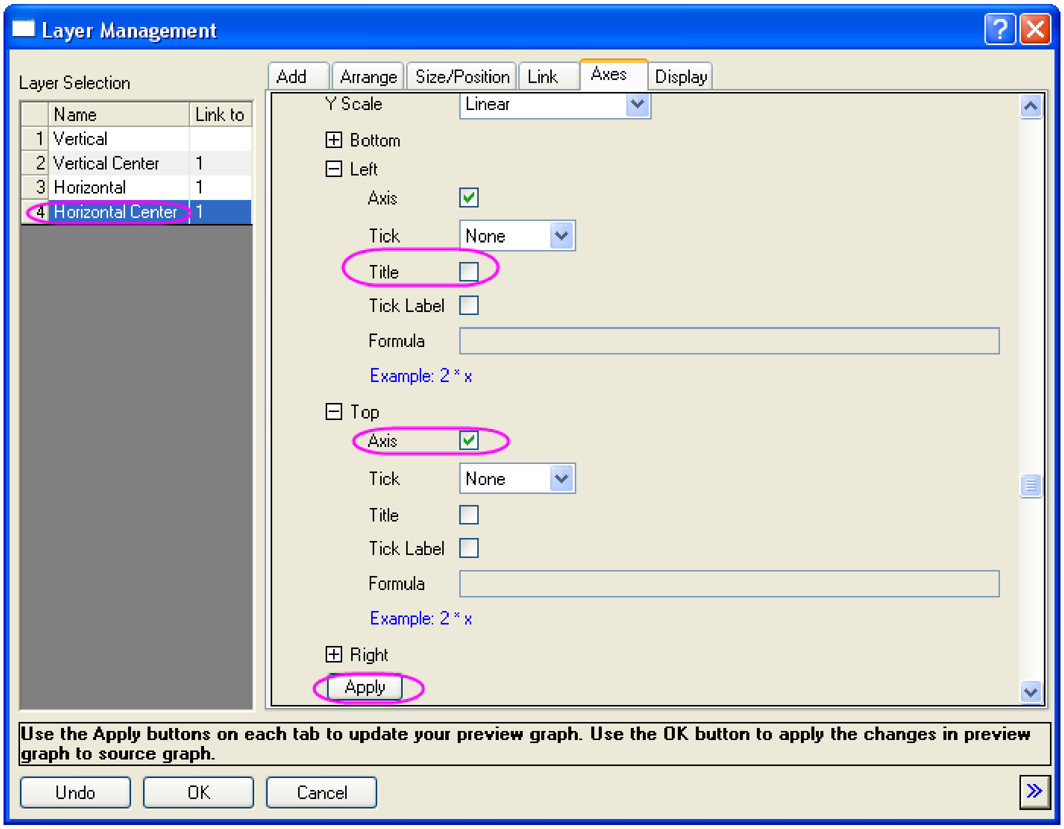
10. Select *Vertical Center* layer, in the **Axes** tab, **Bottom** branch, **Left** branch and **Right** branch share the same settings. And settings for **Top** branch shows in the image below. Click the **Apply** button.



11. Select *Horizontal* layer, in the **Axes** tab, check the **Axis** check-boxes under both **Top** branch and **Right** branch. Click the **Apply** button.

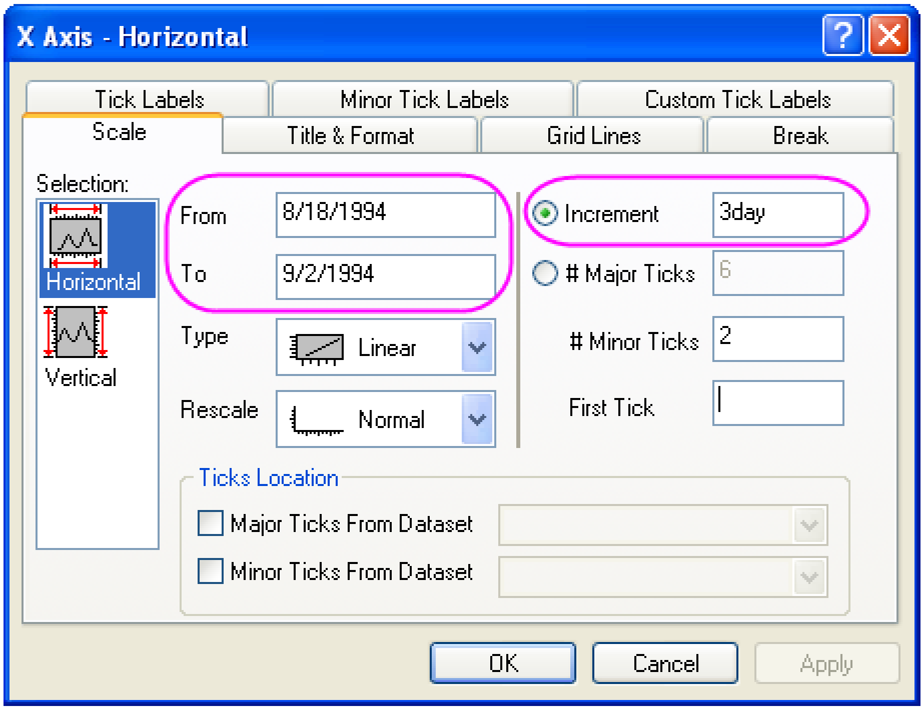


12. Select *Horizontal Center* layer, in the **Axes** tab, uncheck the **Title** check-box under **Left** branch and check the **Axis** check-box under **Top** branch. Click the **Apply** button.

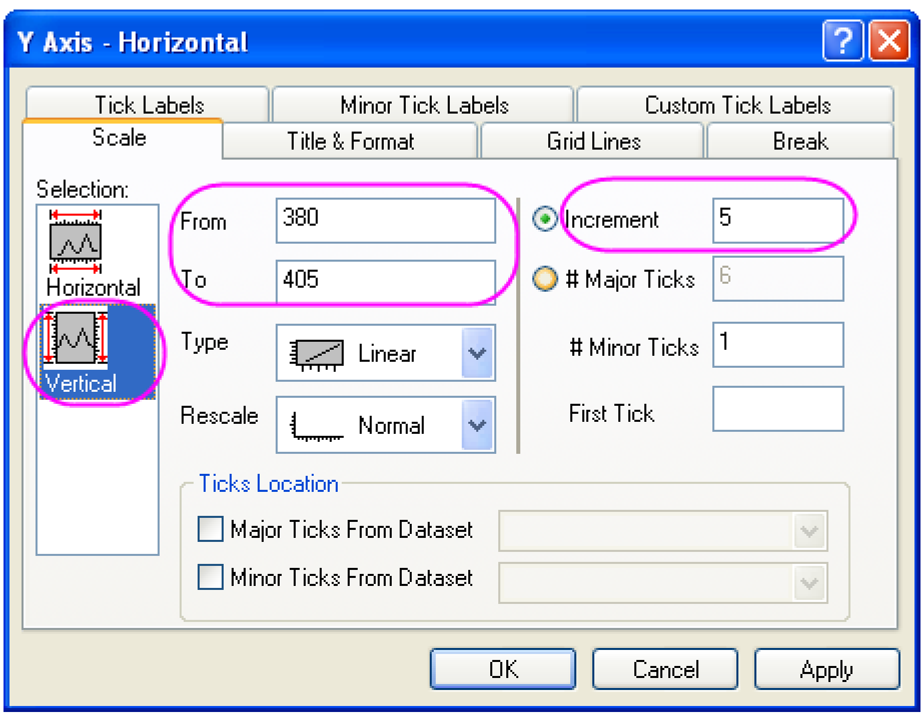


13. Click the **OK** button to close the **Layer Management** dialog.

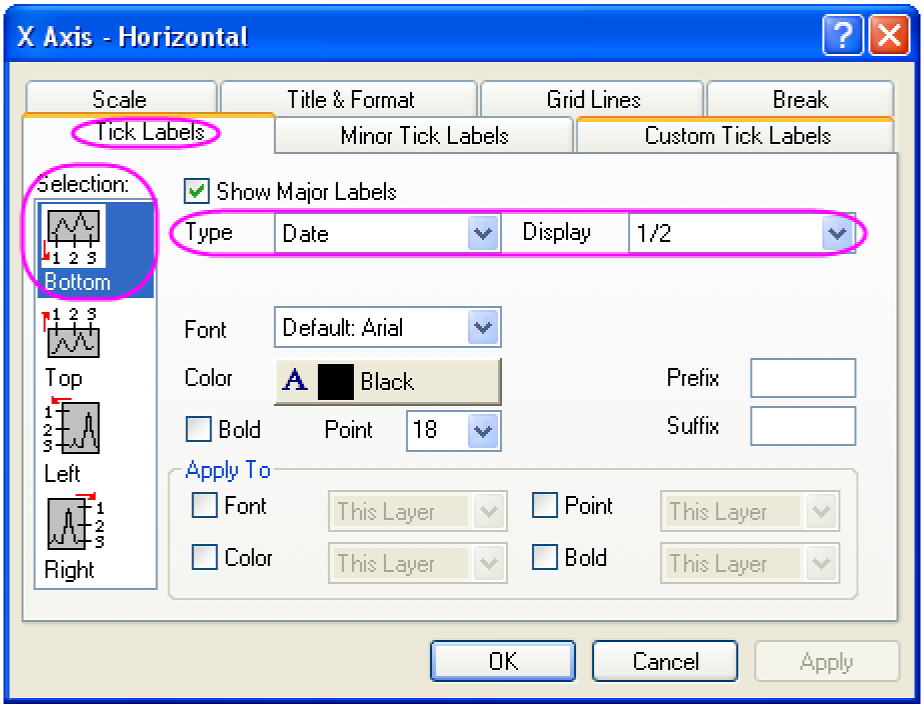
14. Double click on the X axis of the lower-left layer to open the **X Axis** dialog. In the **Scale** tab, select **Horizontal** from the **Selection** panel, change **From**, **To**, **Increment** and **Minor Ticks** as the following image shows.



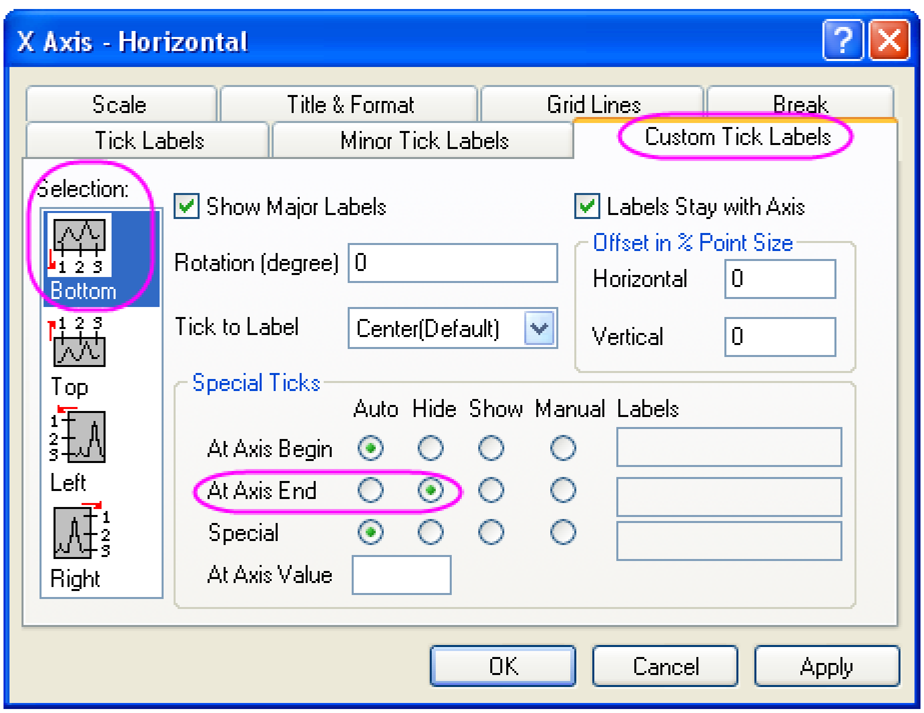
15. Select **Vertical** from **Selection** panel, change the settings as the following image shows.



16. Go to the **Tick Labels** tab, select **Bottom** from the **Selection** panel, set **Type** to *Date* and **Display** to *1/2*.



17. Go to the **Custom Tick Labels** tab, select **Bottom** from the **Selection** panel, choose **Hide** for the **At Axis End**.

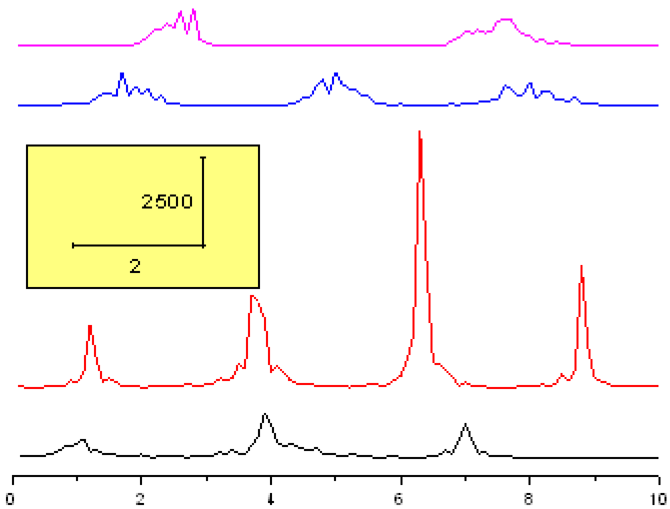


18. Repeat step 14 to step 17 for the X axis of the lower-right layer.
19. Select the labels of X axis and Y axis, then in the **Format** toolbar, change the size to 30 and click down the **Bold** button.
20. Change the Y title to **Price** and then set size of X and Y title to 36.

5.4.4 Stack Lines by Y Offsets

Summary

This tutorial will show how to create a graph with stack lines by Y offsets.



What you will learn

- How to create a graph with stack lines by Y offsets
- How to customize a rectangle object
- How to add new XY scaler

Steps

This tutorial is associated with the 2D and Contour Graphs project: *\Samples\2D and Contour Graphs.opj*.

1. Open the 2D and Contour Graphs project *<Origin Installation Directory>\Samples\2D and Contour Graphs.opj*, and then browse to the *2D and Contour Graphs: Multi Axis and Multi Panel: Stack Lines by Y Offsets* folder in **Project Explorer**.
2. Highlight all columns in the data worksheet of **Book6A**, then choose menu **Plot: Multi-Curve: Stack Lines by Y Offsets** to create a graph with stack lines by Y offsets.
3. Delete the following objects by selecting them and then pressing the **Delete** key on your keyboard:

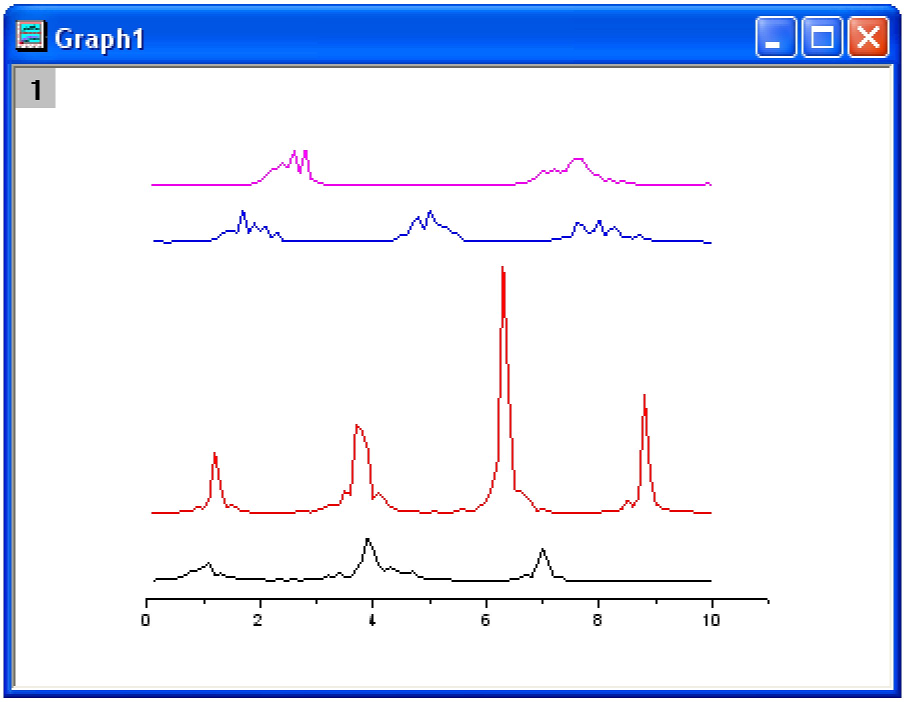
Legend

Y axis

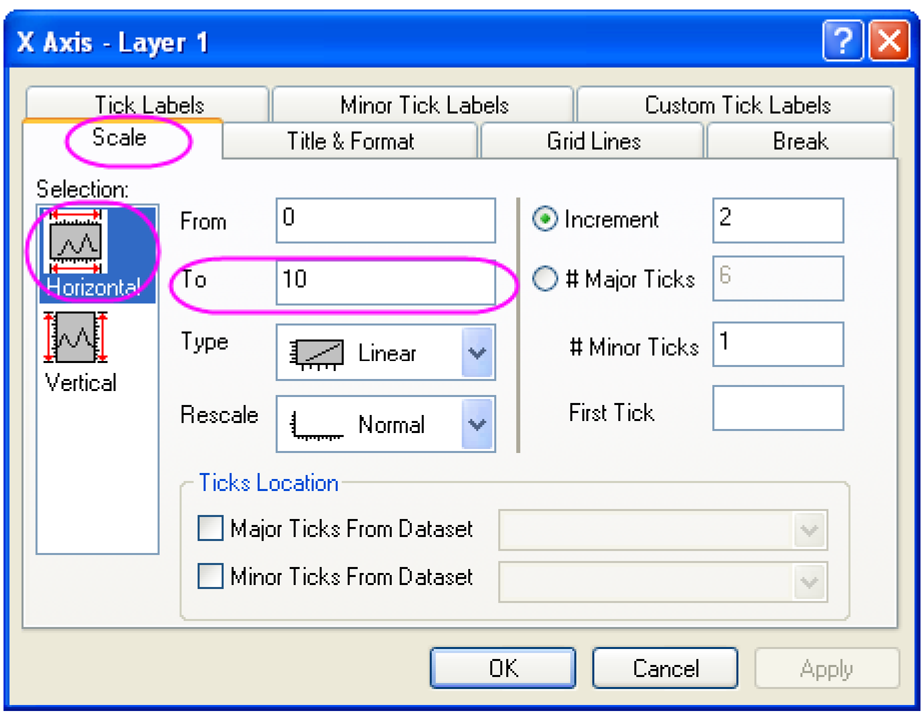
Y axis labels

Y axis title

X axis title



4. Double click on the X axis to open the Axis Properties dialog. In the **Scale** tab, make sure that **Horizontal** is selected in the **Selection** list. Change **To** to *10*. Click **OK** to close the dialog.

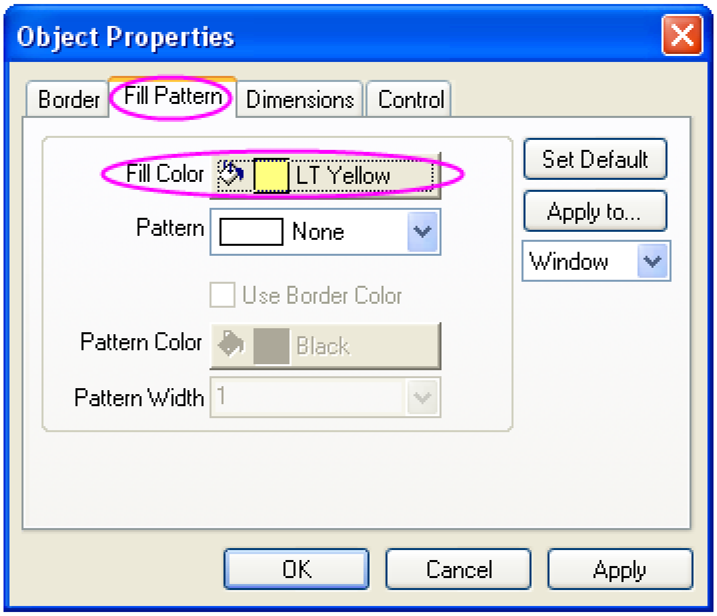


5. Click the **Rectangle** button on the **Tools** toolbar and then create a rectangle on the graph.

6. Double click on the rectangle to open its properties dialog. Change the following settings.

   Fill Pattern tab

Fill Color = LT Yellow



Dimensions tab

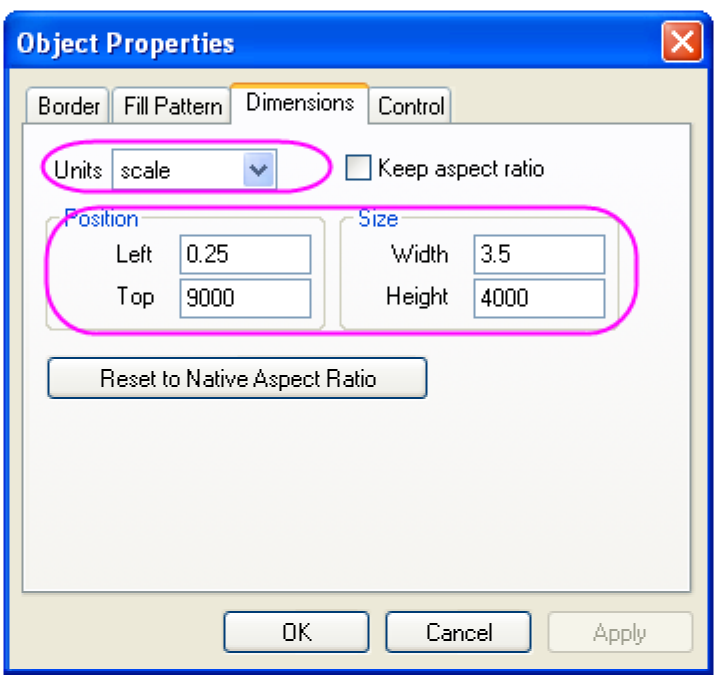
Units = scale

Left = .25

Top = 9000

Width = 3.5

Height = 4000



7. Click OK to close the **Object Properties** dialog.

8. Select **Graph: New XY Scaler** from the Origin menu to create a new XY scaler.
9. Double click on the scaler to open the **Scaler Properties** dialog. Set the following properties in the dialog:

   Font Settings branch

   Font Size = 24

   X branch

   Length(Scale) = 2

   Title = 2

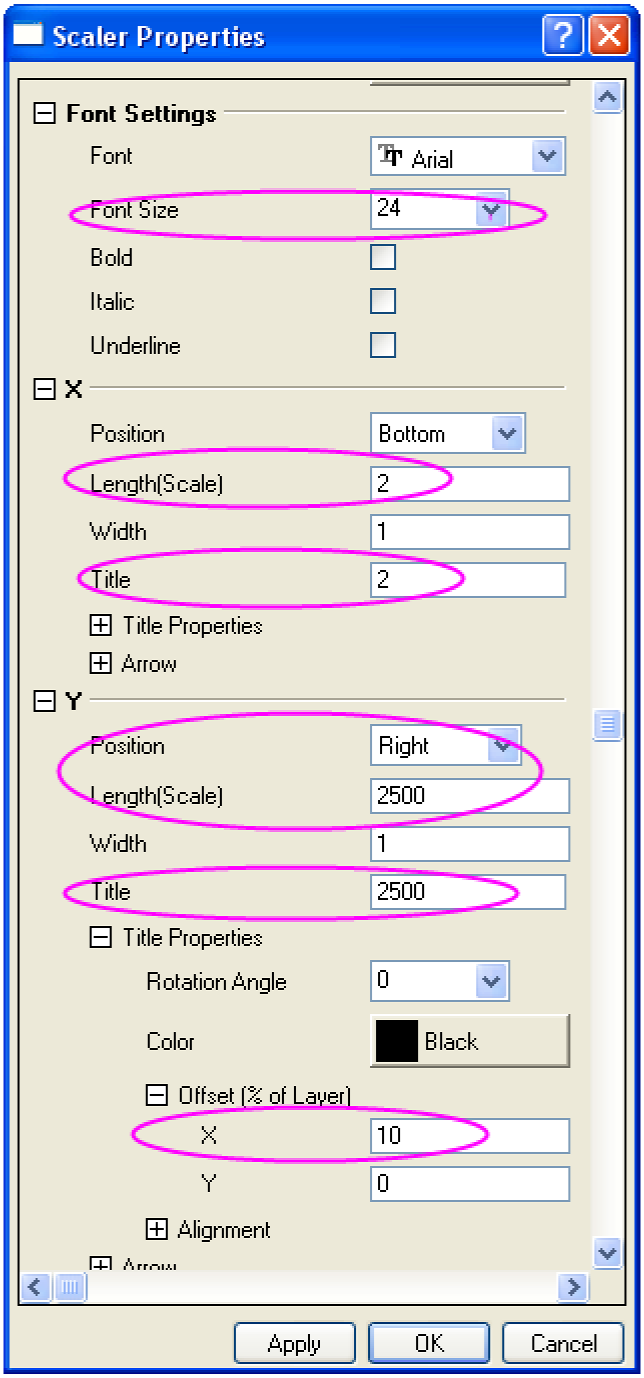
   Y branch

   Position = Right

   Length(Scale) = 2500

   Title = 2500

   X (in the **Offset** branch under the **Title Properties** branch) = 10

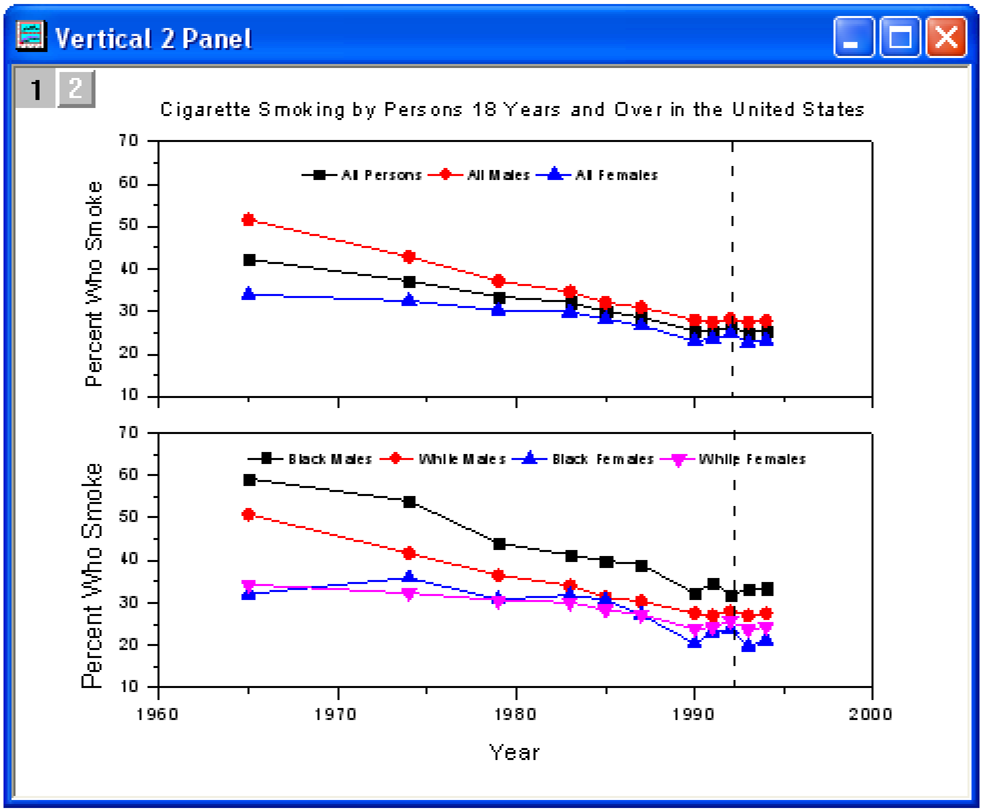


10. Click OK to close the **Scaler Properties** dialog.
11. Move the Scaler object onto the rectangle you created.

5.4.5 Vertical 2 Panel Line

Summary

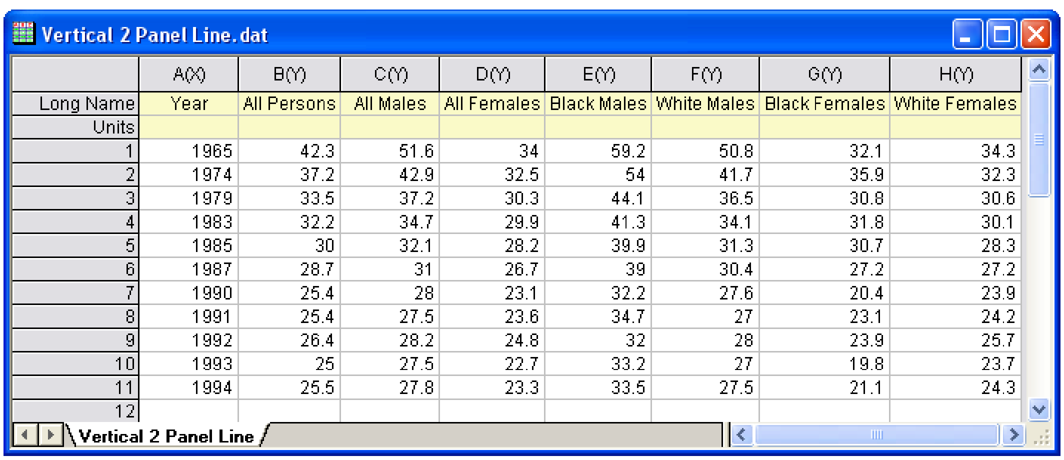
This tutorial will show you how to merge two graphs and how to customzie them.



Minimum Origin Version Required: Origin 8.0 SR6

Steps

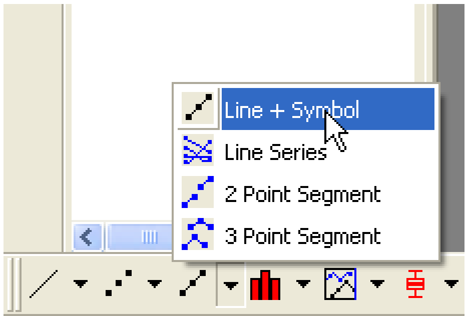
1. Create a new worksheet. Import the data.


Vertical 2 Panel Line.dat

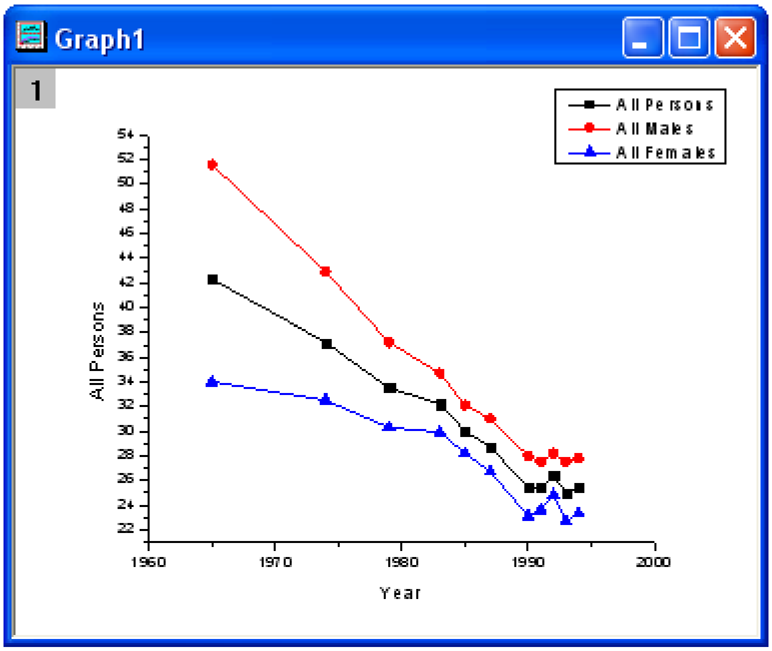
	A(X)	B(Y)	C(Y)	D(Y)	E(Y)	F(Y)	G(Y)	H(Y)
Long Name	Year	All Persons	All Males	All Females	Black Males	White Males	Black Females	White Females
Units								
1	1965	42.3	51.6	34	59.2	50.8	32.1	34.3
2	1974	37.2	42.9	32.5	54	41.7	35.9	32.3
3	1979	33.5	37.2	30.3	44.1	36.5	30.8	30.6
4	1983	32.2	34.7	29.9	41.3	34.1	31.8	30.1
5	1985	30	32.1	28.2	39.9	31.3	30.7	28.3
6	1987	28.7	31	26.7	39	30.4	27.2	27.2
7	1990	25.4	28	23.1	32.2	27.6	20.4	23.9
8	1991	25.4	27.5	23.6	34.7	27	23.1	24.2
9	1992	26.4	28.2	24.8	32	28	23.9	25.7
10	1993	25	27.5	22.7	33.2	27	19.8	23.7
11	1994	25.5	27.8	23.3	33.5	27.5	21.1	24.3
12								

Vertical 2 Panel Line

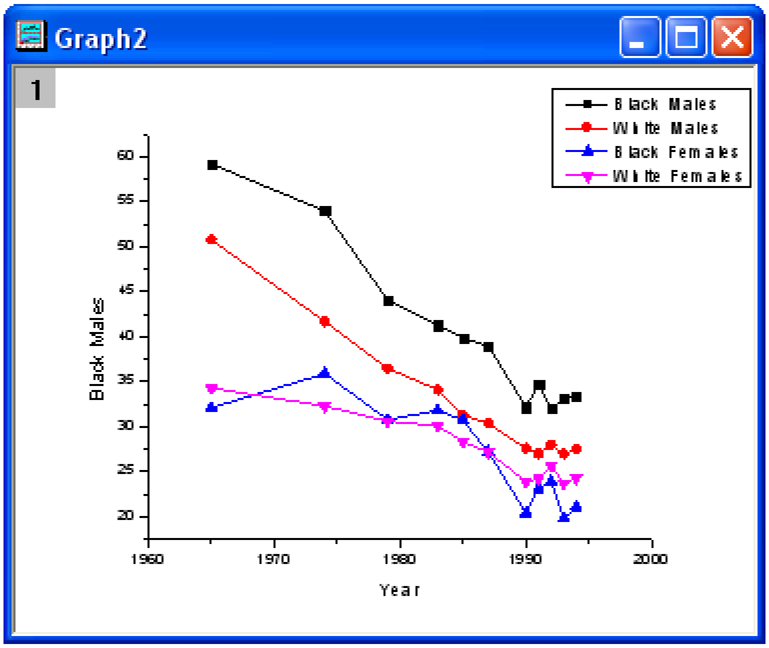
2. Select column **2** to column **4**. Click the **Line + Symbol** button on the **2D Graphs** toolbar.



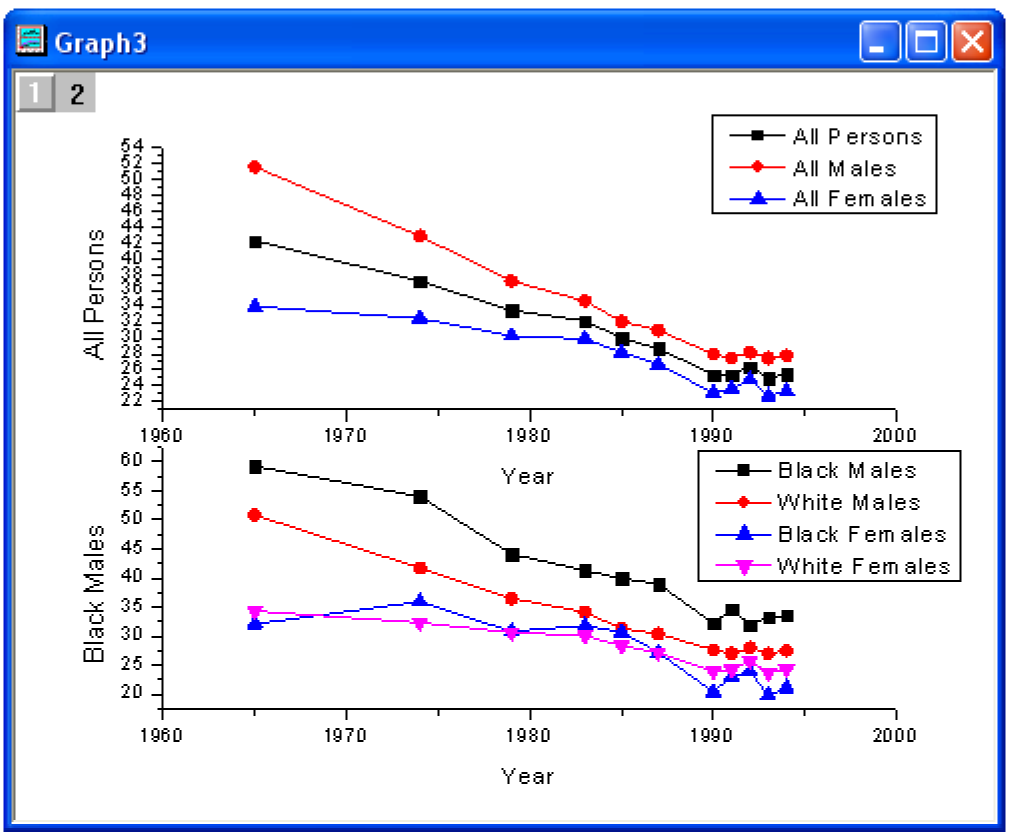
The first graph should look like



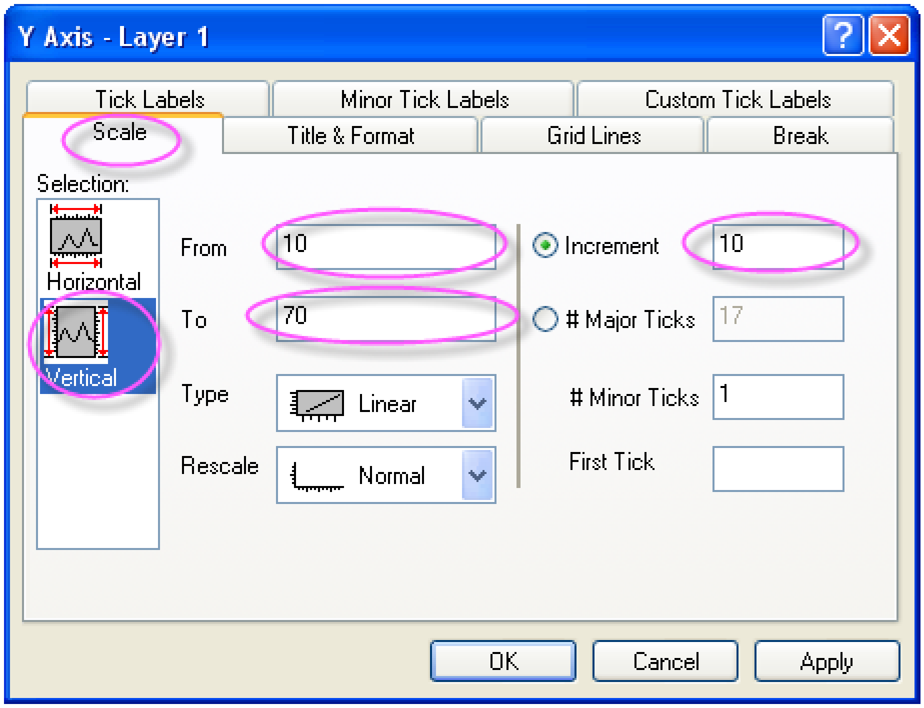
Go back to the worksheet. Select column **5** to column **8** and then click the **Line + Symbol** button on the **2D Graphs** toolbar to create a new graph. The second graph should look like



3. To merge two graphs, select **Graph: Merge Graph Windows** from the menu. Accept the default settings. Click **OK** to merge two graphs. The merged graph should look like

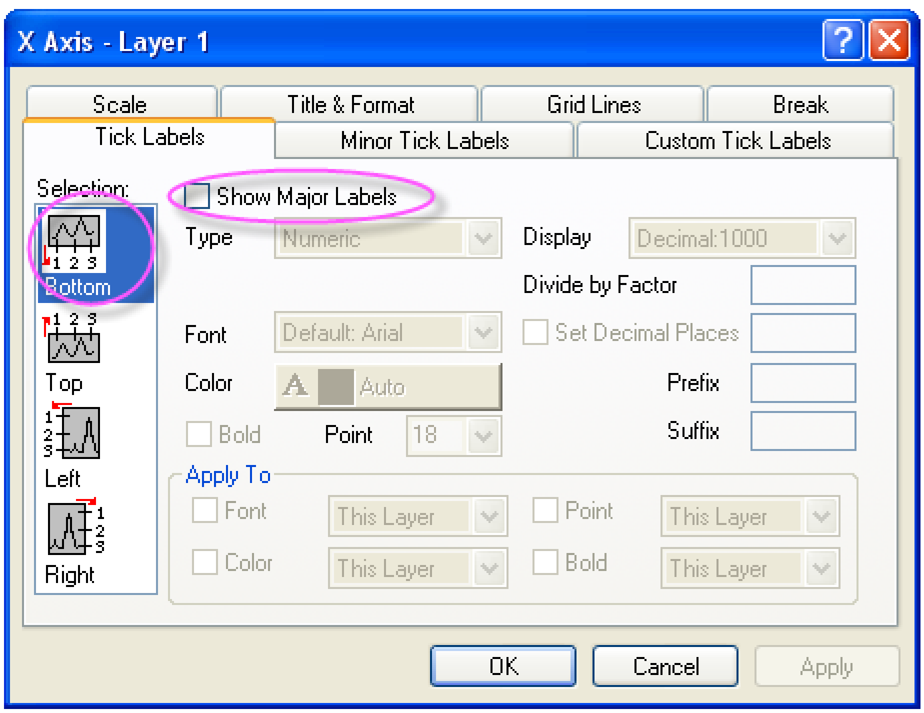


4. Double-click on the **Y** axis of Layer 1 to open the **Axis Properteis** dialog. In the **Scale** tab, set the dialog options as the screenshot below.

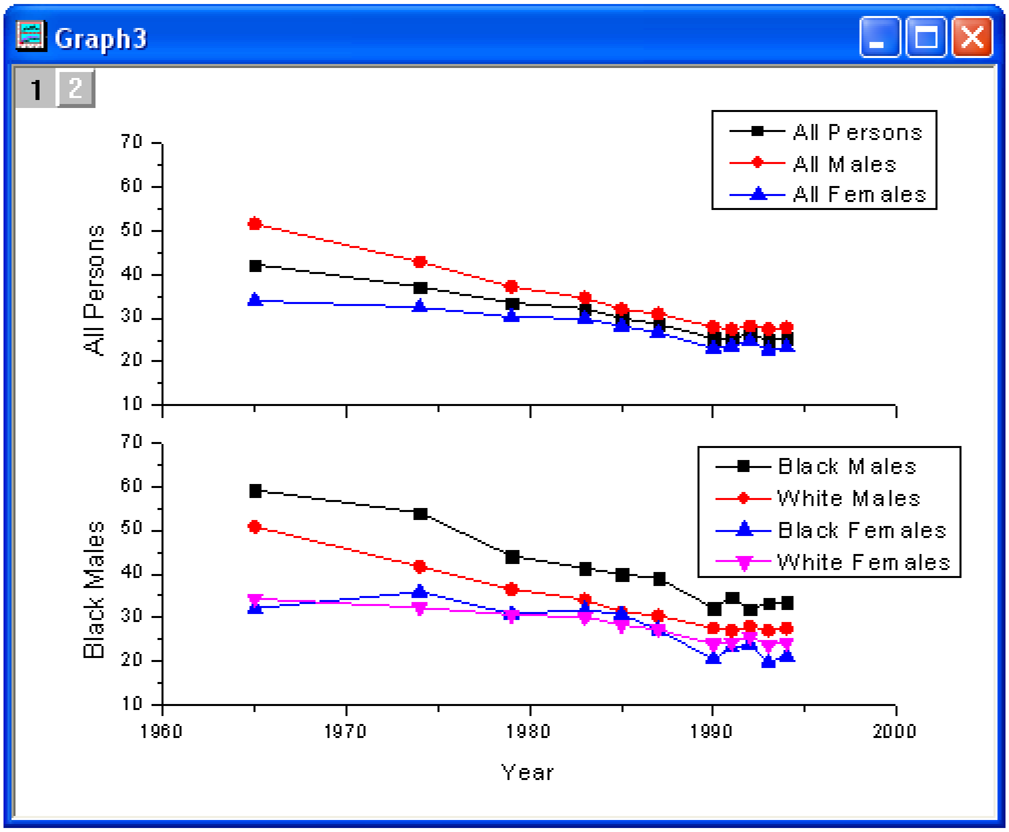


In **Tick Labels** tab, set the dialog options as the screenshot below. Click OK to finish.

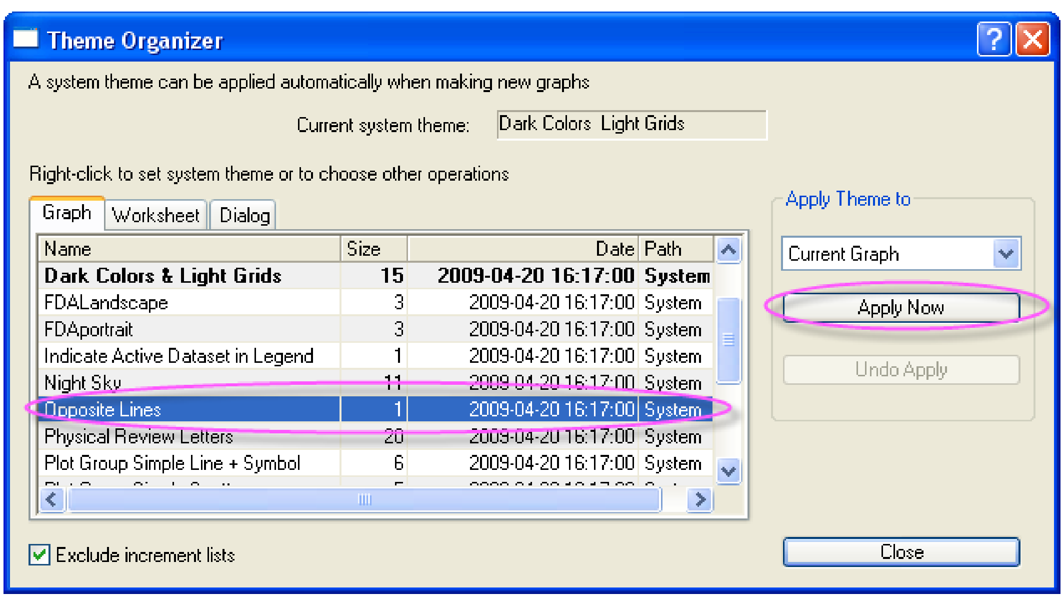
5. Delete the **X** title in Layer 1.



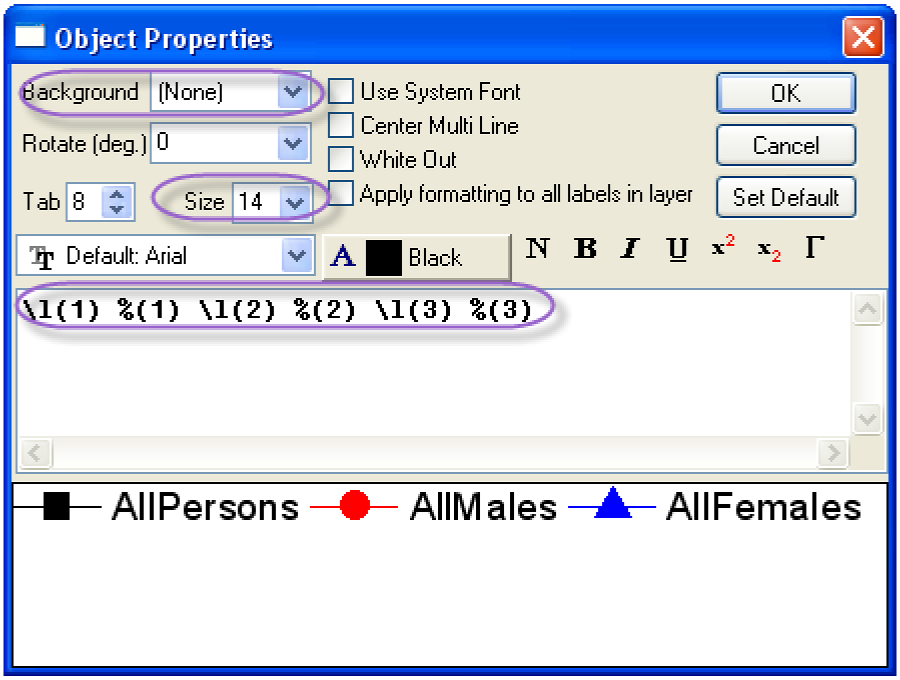
Set the same scale for the Y axis of Layer 2. After all of these operations, the graph should look like



6. Now, apply a theme to add a top **X** axis and a right **Y** axis. Select **Tool: Theme Organizer** from the main menu to open the **Theme Organizer** dialog. Activate the **Graph** tab and select the **Opposite Lines** from the table. Then click the **Apply Now** button. Click the **Close** button to close the dialog.



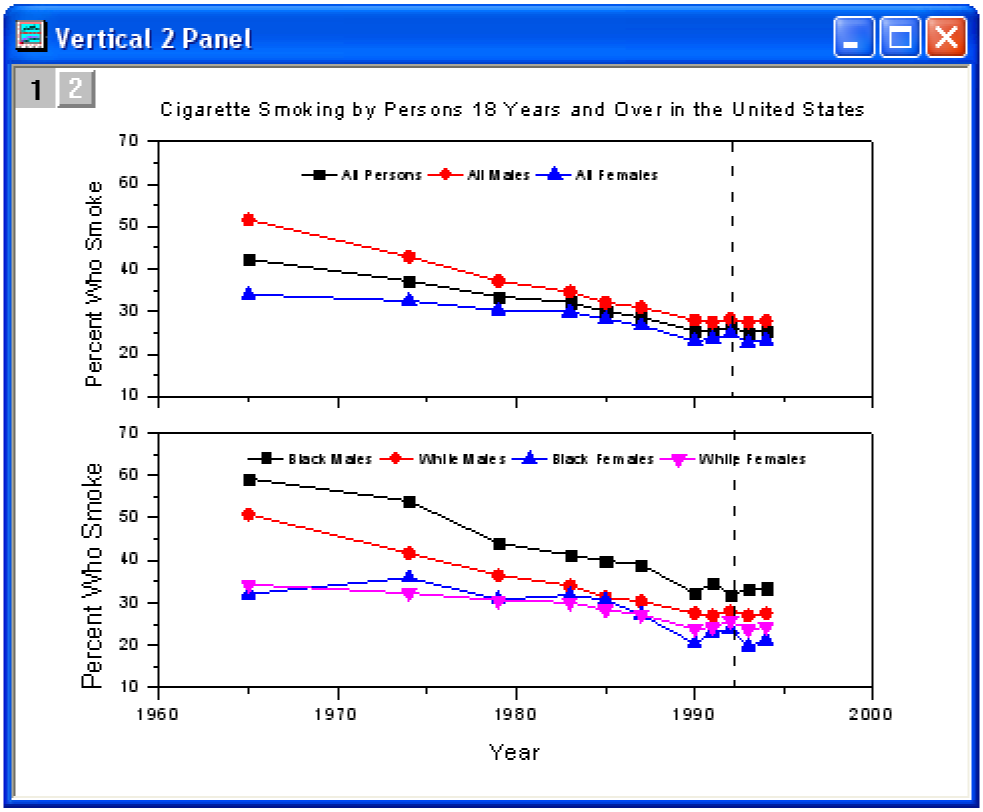
7. Select the **Legend** of Layer1 and right-click on it to select **Properties** from the short-cut menu. Then set the dialog options as following image shows:



Move the legend to the position as the sample image shows.

8. Click the **Line** button on the **Tools** toolbar and draw a line on the Layer 1 as the sample image shows. Hold down the **SHIFT** key while drawing to ensure that the line is vertical. Double-click on the line to open the properties dialog of the line object. In the **Line** tab, select **Dash** with the **Type** drop-down list. And enter **2** in the **Size** edit box. Activate the **Control** tab. Select both the **Horizontal Movement** and the **Vertical Movement** check boxes. Then click the **OK** button.
9. Repeat steps **6-7** for Layer **2**.
10. Click the **Text** button in the **Tools** toolbar. Then click in the middle of the graph and enter **Cigarette Smoking by Persons 18 Years and Over in the United States** to add a title for the graph. Double-click on the **Y** axis title of Layer 1 and Layer 2, then enter **Percent Who Smoke** in them.

The graph should look like



Sample Data

Download the **Vertical_2_Panel_Line.txt** file from http://www.originlab.com/ftp/graph_gallery/data/Vertical_2_Panel_Line.txt. Click the **Import Single ASCII button**  and select the file to import it into Origin.

The following table contains part of the sample data.

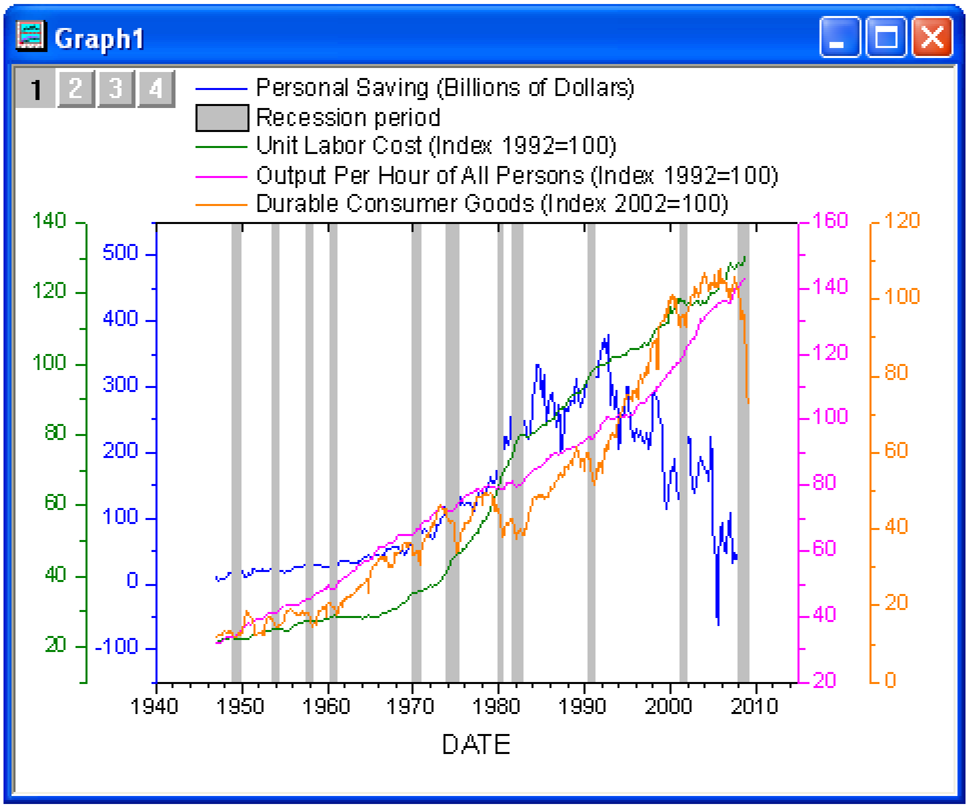
Year(X)	All Persons(Y)	All Males(Y)	All Females(Y)	Black Males(Y)	White Males(Y)	Black Females(Y)	White Females(Y)
1965	42.3	51.6	34	59.2	50.8	32.1	34.3
1974	37.2	42.9	32.5	54	41.7	35.9	32.3
1979	33.5	37.2	30.3	44.1	36.5	30.8	30.6
1983	32.2	34.7	29.9	41.3	34.1	31.8	30.1
1985	30	32.1	28.2	39.9	31.3	30.7	28.3
1987	28.7	31	26.7	39	30.4	27.2	27.2
1990	25.4	28	23.1	32.2	27.6	20.4	23.9
1991	25.4	27.5	23.6	34.7	27	23.1	24.2
1992	26.4	28.2	24.8	32	28	23.9	25.7
1993	25	27.5	22.7	33.2	27	19.8	23.7
1994	25.5	27.8	23.3	33.5	27.5	21.1	24.3

5.4.6 Multiple Axis Plot

Contents

Summary

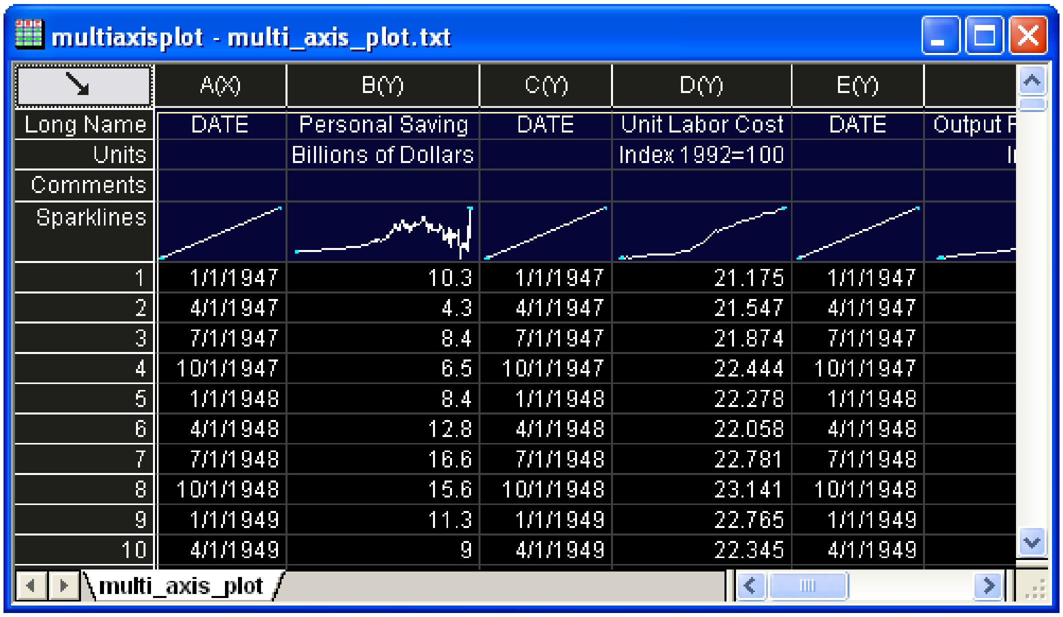
This tutorial will show you how to create a 4-Y plot and add recession bars.



Minimum Origin Version Required: Origin 8.5.1

Steps

1. Create a new workbook and import the sample data into it.
2. Place the mouse cursor near the top left corner of the worksheet. When the cursor changes to a down-right arrow, click the left button to select the whole worksheet.

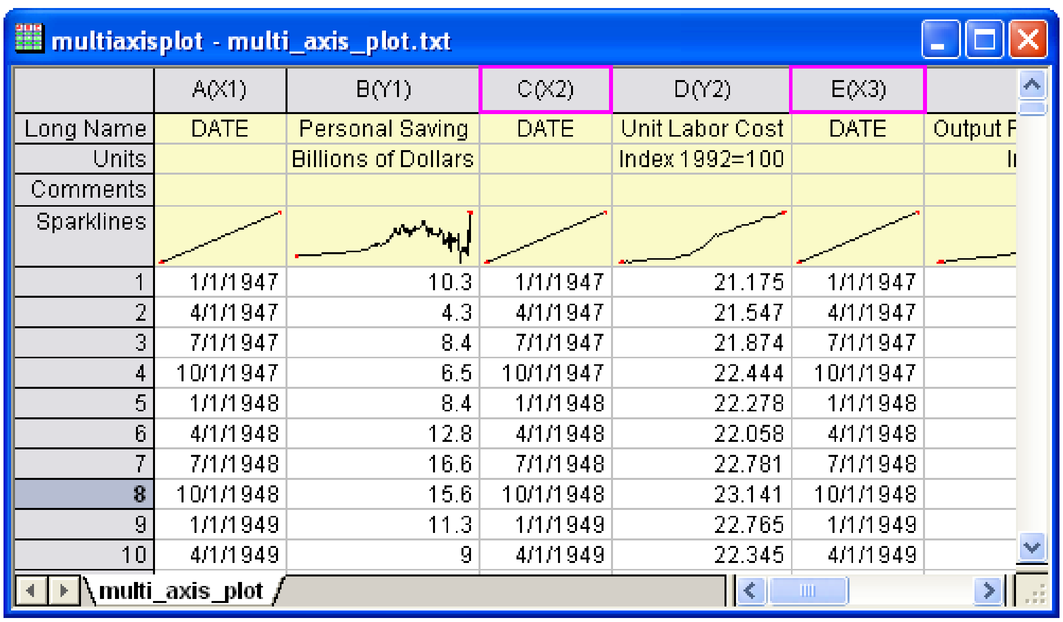


multiaxisplot - multi_axis_plot.txt

	A(X)	B(Y)	C(Y)	D(Y)	E(Y)	
Long Name	DATE	Personal Saving	DATE	Unit Labor Cost	DATE	Output F
Units		Billions of Dollars		Index 1992=100		I
Comments						
Sparklines						
1	1/1/1947	10.3	1/1/1947	21.175	1/1/1947	
2	4/1/1947	4.3	4/1/1947	21.547	4/1/1947	
3	7/1/1947	8.4	7/1/1947	21.874	7/1/1947	
4	10/1/1947	6.5	10/1/1947	22.444	10/1/1947	
5	1/1/1948	8.4	1/1/1948	22.278	1/1/1948	
6	4/1/1948	12.8	4/1/1948	22.058	4/1/1948	
7	7/1/1948	16.6	7/1/1948	22.781	7/1/1948	
8	10/1/1948	15.6	10/1/1948	23.141	10/1/1948	
9	1/1/1949	11.3	1/1/1949	22.765	1/1/1949	
10	4/1/1949	9	4/1/1949	22.345	4/1/1949	

multi_axis_plot

3. Right-click and select **Set as: XY XY** from the short-cut menu to set proper plotting designations for the worksheet columns.

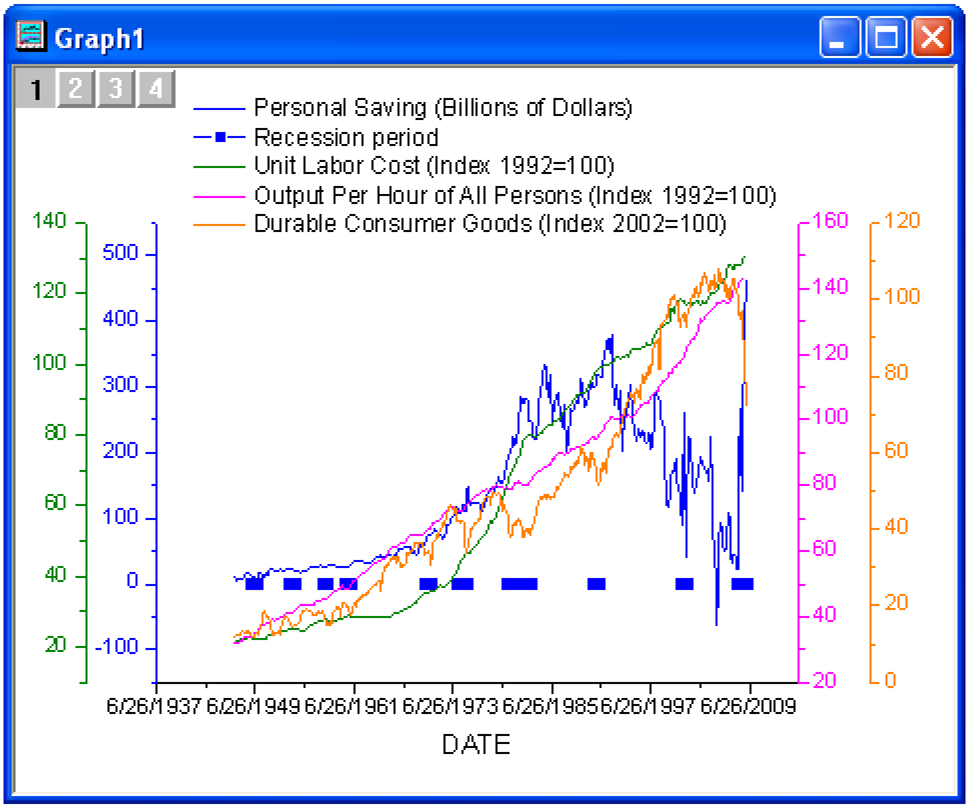


multiaxisplot - multi_axis_plot.txt

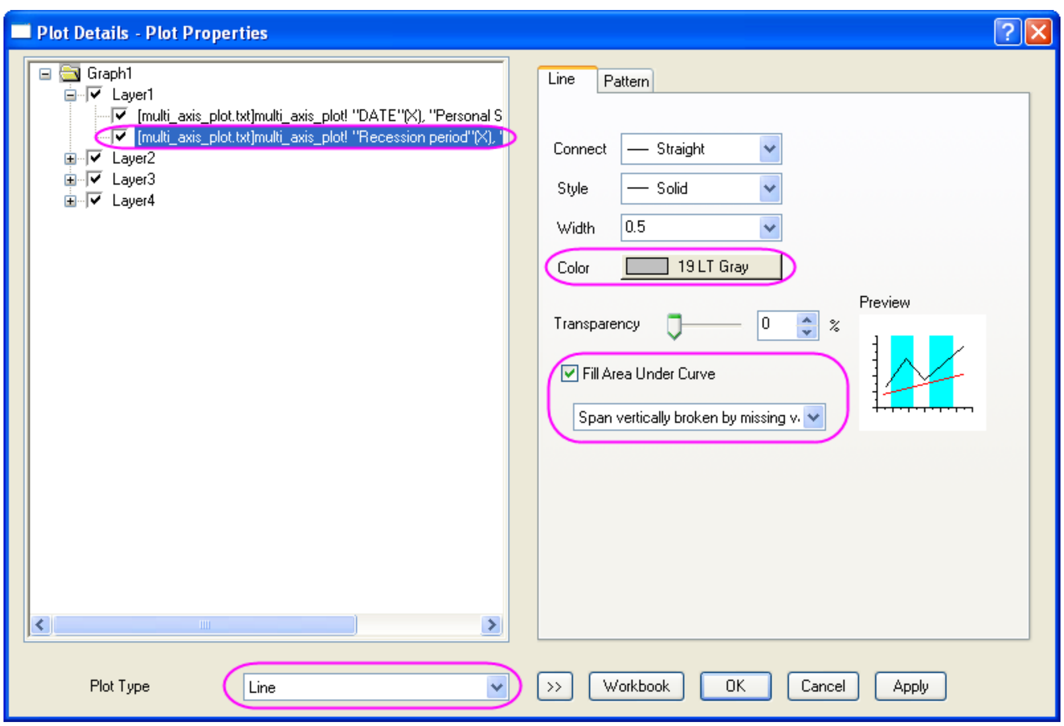
	A(X1)	B(Y1)	C(X2)	D(Y2)	E(X3)	
Long Name	DATE	Personal Saving	DATE	Unit Labor Cost	DATE	Output F
Units		Billions of Dollars		Index 1992=100		I
Comments						
Sparklines						
1	1/1/1947	10.3	1/1/1947	21.175	1/1/1947	
2	4/1/1947	4.3	4/1/1947	21.547	4/1/1947	
3	7/1/1947	8.4	7/1/1947	21.874	7/1/1947	
4	10/1/1947	6.5	10/1/1947	22.444	10/1/1947	
5	1/1/1948	8.4	1/1/1948	22.278	1/1/1948	
6	4/1/1948	12.8	4/1/1948	22.058	4/1/1948	
7	7/1/1948	16.6	7/1/1948	22.781	7/1/1948	
8	10/1/1948	15.6	10/1/1948	23.141	10/1/1948	
9	1/1/1949	11.3	1/1/1949	22.765	1/1/1949	
10	4/1/1949	9	4/1/1949	22.345	4/1/1949	

multi_axis_plot

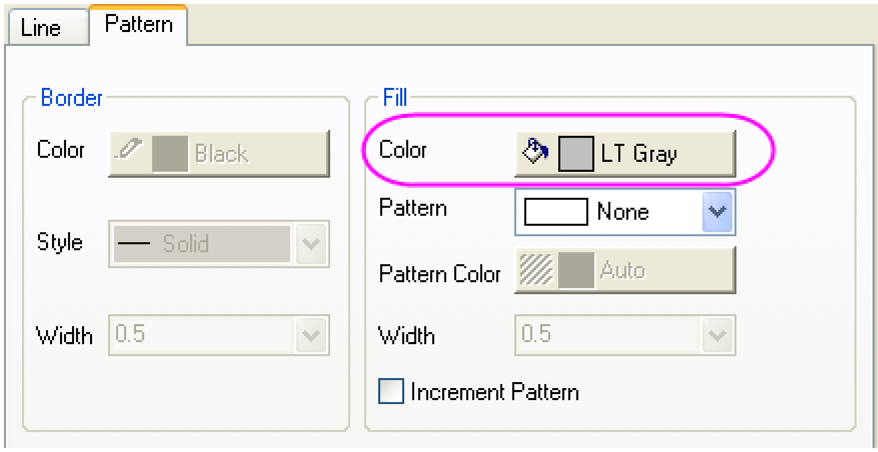
4. Highlight all columns. From the main menu, select **Plot: Multi-curve: 4Ys YY-YY** to create a plot.



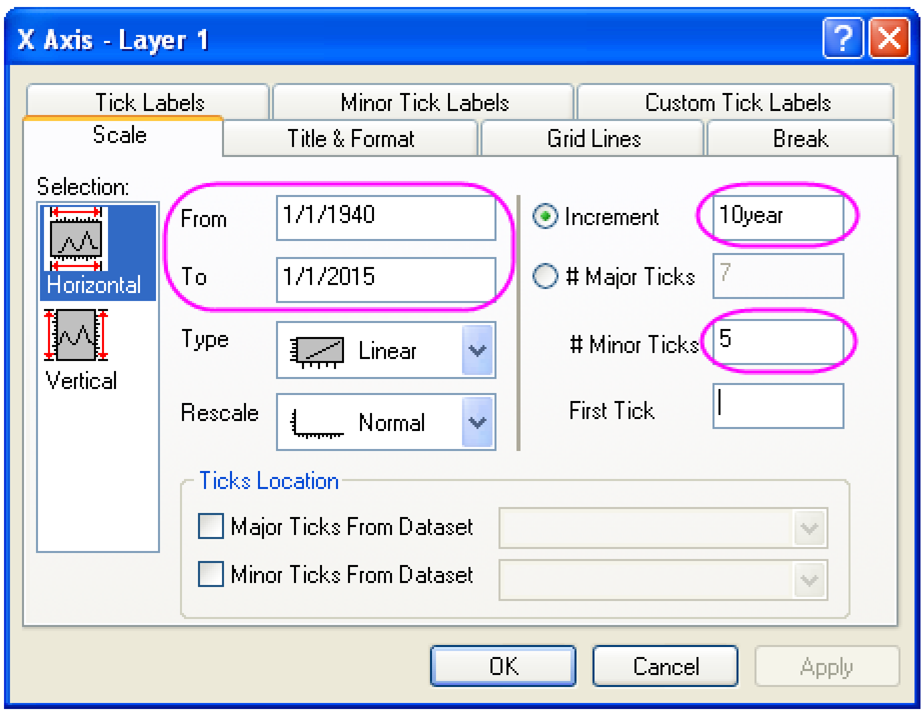
5. Double-click on the graph to open the **Plot Details** dialog. In the left panel, select the second plot in Layer 1 and set the plot type as line and click Apply button. Then go to the **Line** tab in the right panel as the following screenshot and then click **Apply**.



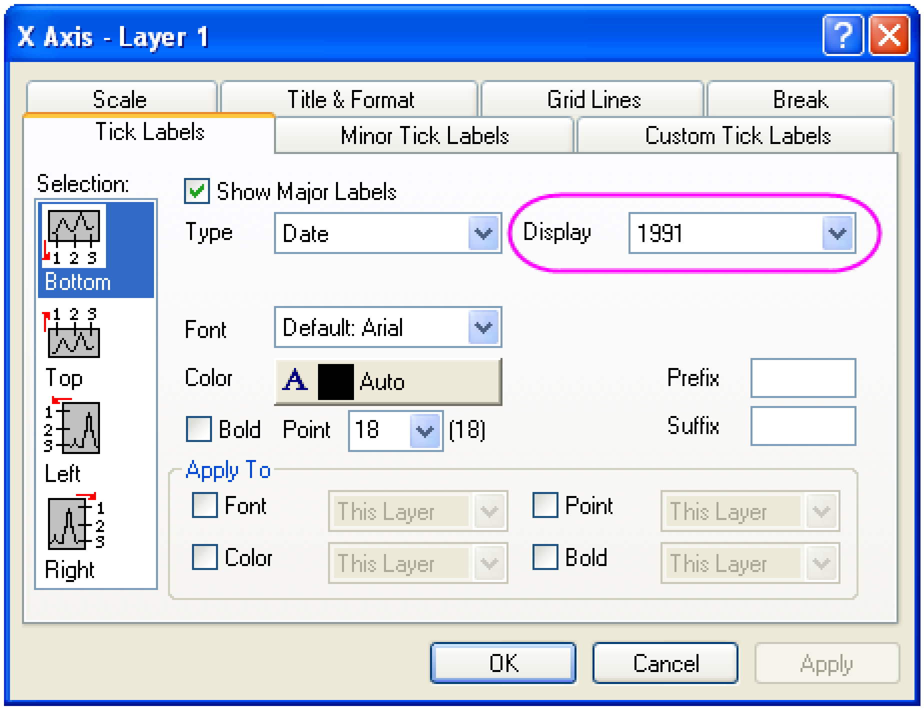
6. Go to the **Pattern** tab. Set the dialog options as follows and then click **OK**.



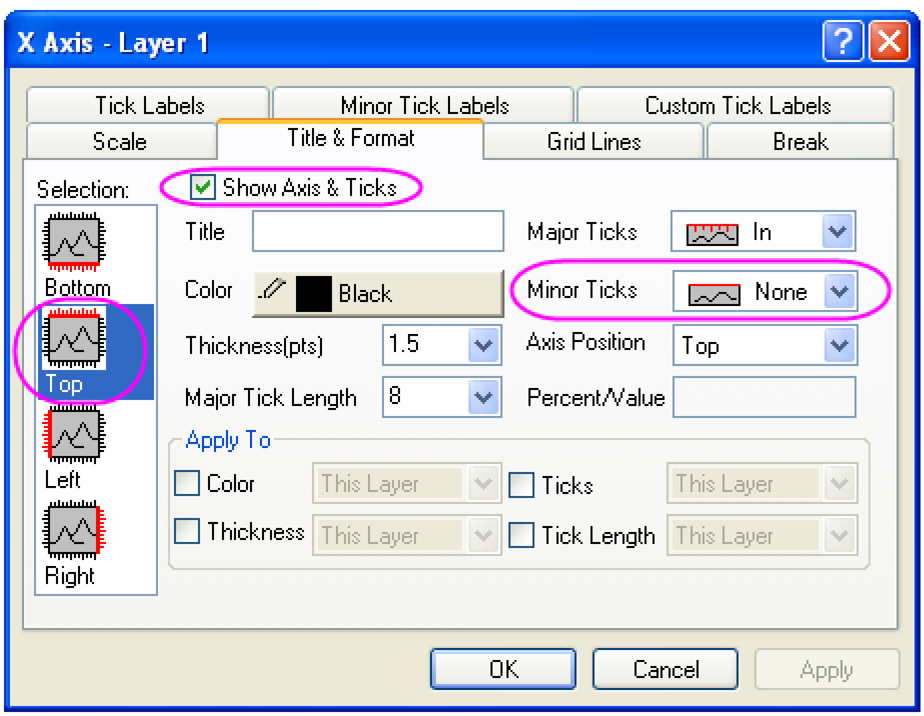
7. Double-click on the bottom X axis to open the **Axis Properties** dialog. In the **Scale** tab, change the dialog options as follows and then click **Apply**.



8. Go to the **Tick Labels** tab, change the dialog options as follows and then click **Apply**.

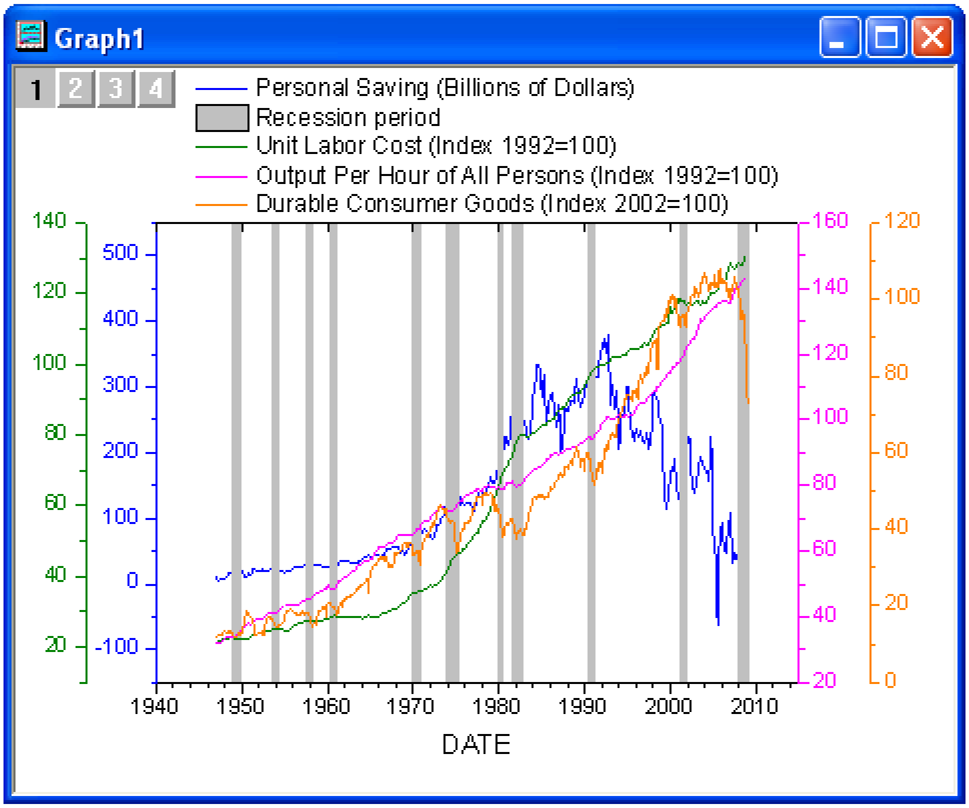


9. Activate the **Title & Format** tab. Select **Top** in the **Selection** list. Then change the dialog options as follows and then click **Apply**.

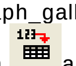


10. From the Origin menu, select **Graph: New Legend** to update the legend. Click on the legend to select it and then use the mouse to drag it to a better position.

The final graph should be like:



Sample Data

Download the sample data file from http://www.originlab.com/ftp/graph_gallery/data/multi_axis_plot.txt. Create a new worksheet. Click the **Import Single ASCII** button  and select the file to import it into Origin.

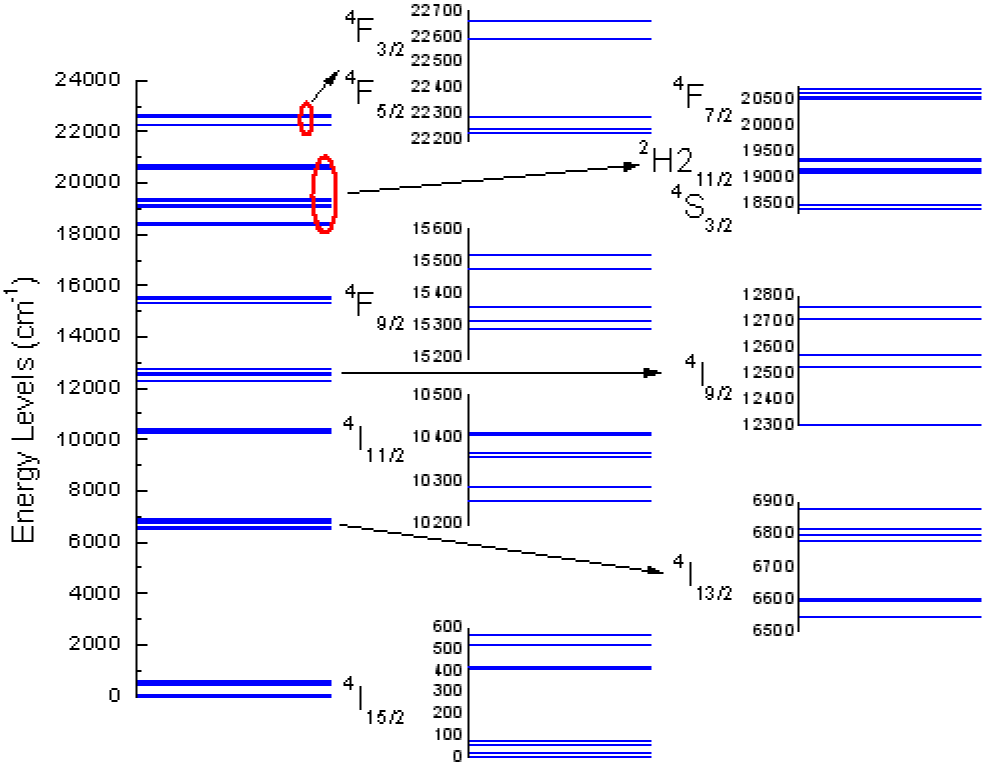
The following table contains part of the sample data.

DATE	Personal Saving	DATE	Unit Labor Cost	DATE	Output Per Hour of All Persons	DATE	Durable Consumer Goods Recession period	Recession Period	Recession Period
1/1/1947	10.3	1/1/1947	21.175	1/1/1947	32.11	1/1/1947	11.8583	11/1/1948	1
4/1/1947	4.3	4/1/1947	21.547	4/1/1947	32.304	2/1/1947	12.222	10/1/1949	1
7/1/1947	8.4	7/1/1947	21.874	7/1/1947	32.069	3/1/1947	12.4888	--	1

5.4.7 Energy-Level Structure of the Er3+ Ion in A YAG Crystal

Summary

In this tutorial, a graph will be created to show the energy-level structure of the Er3+ ion in a YAG crystal.



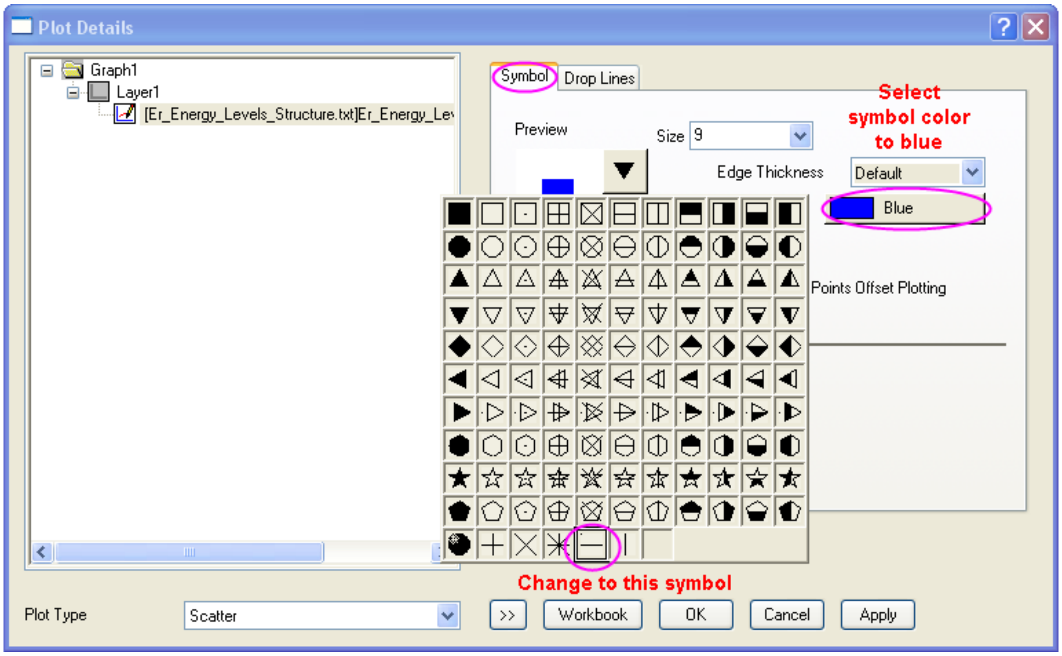
Minimum Origin Version Required: Origin 8 SR0

What you will learn

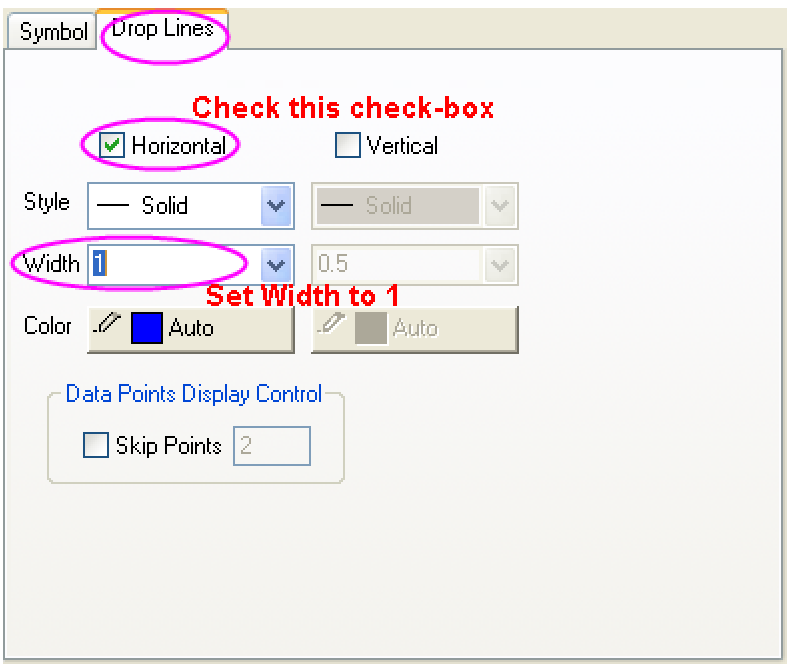
- Create a graph by selecting a part of data from a column
- Set symbol properties
- Merge graphs
- Change the size and position of a layer

Steps

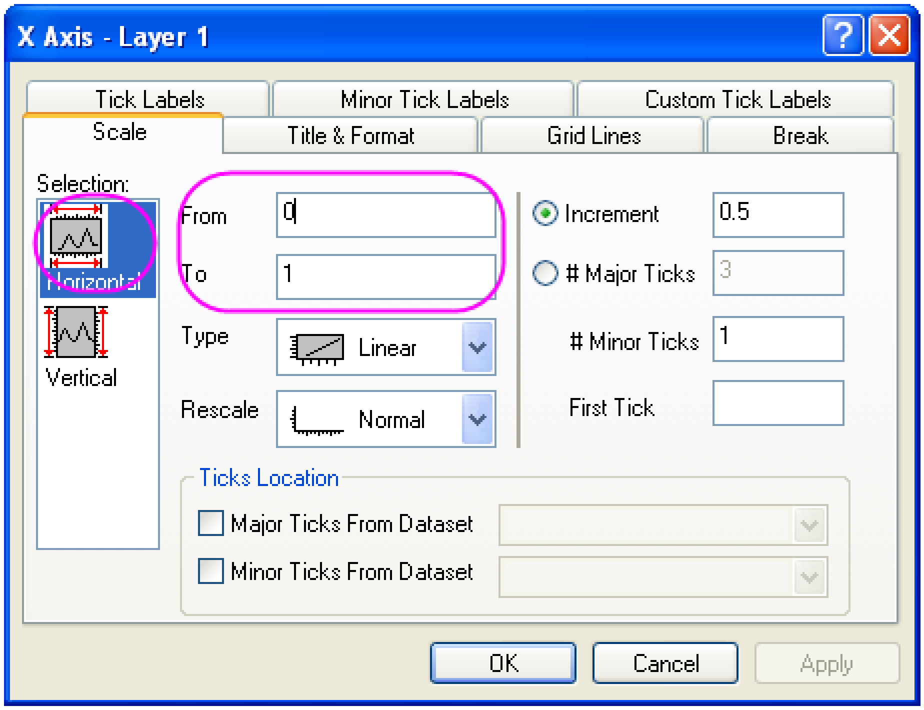
1. To start this tutorial, please download the data file from ftp.
2. Create an empty workbook, click the **Single Import ASCII**  to import the data file downloaded in step 1.
3. Highlight column B and create a symbol plot by selecting menu **Plot: Symbol: Scatter**.
4. Double click on the scatter to open the **Plot Details** dialog. In the **Symbol** tab of the right panel, change the symbol and **Symbol Color** as following image shows.



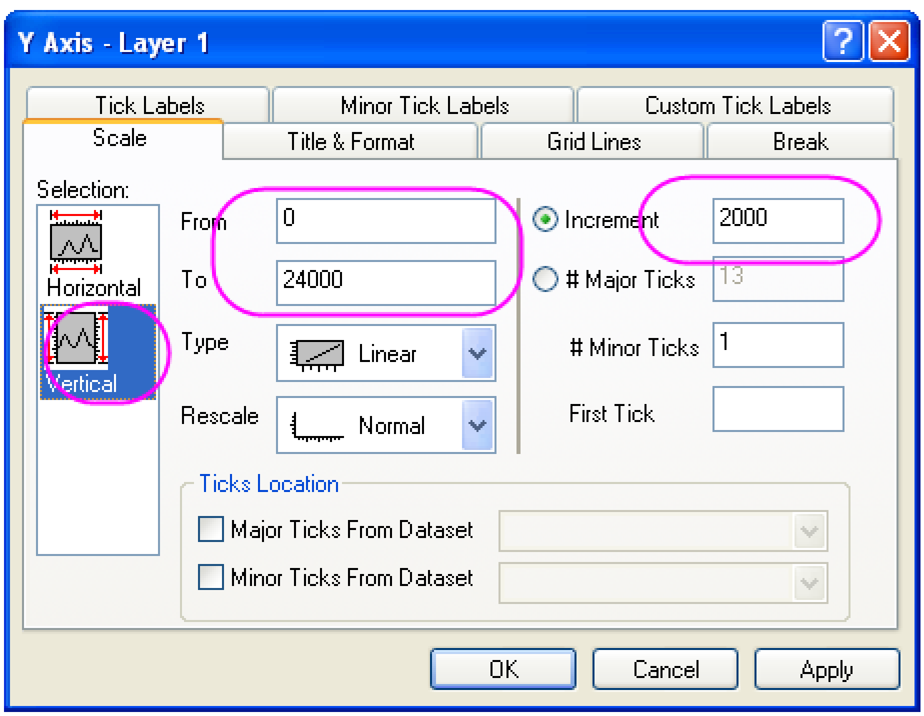
5. In the **Drop Lines** tab, check the **Horizontal** check-box and set **Width** to 1. Then click the **OK** button.



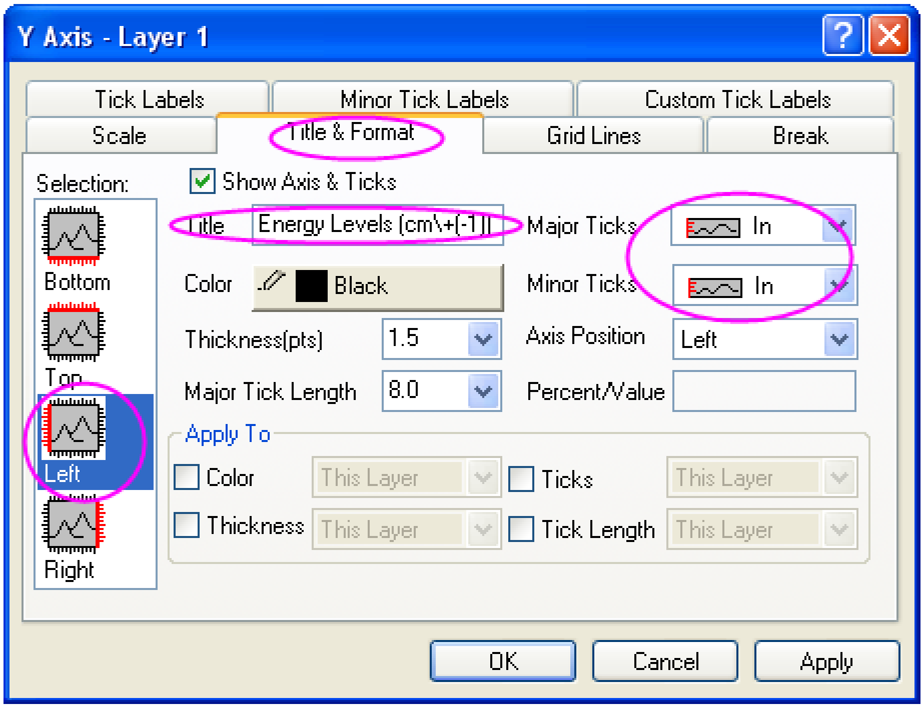
6. Double click on the X axis to open the axis setting dialog. Set X axis **From** 0 **To** 1.



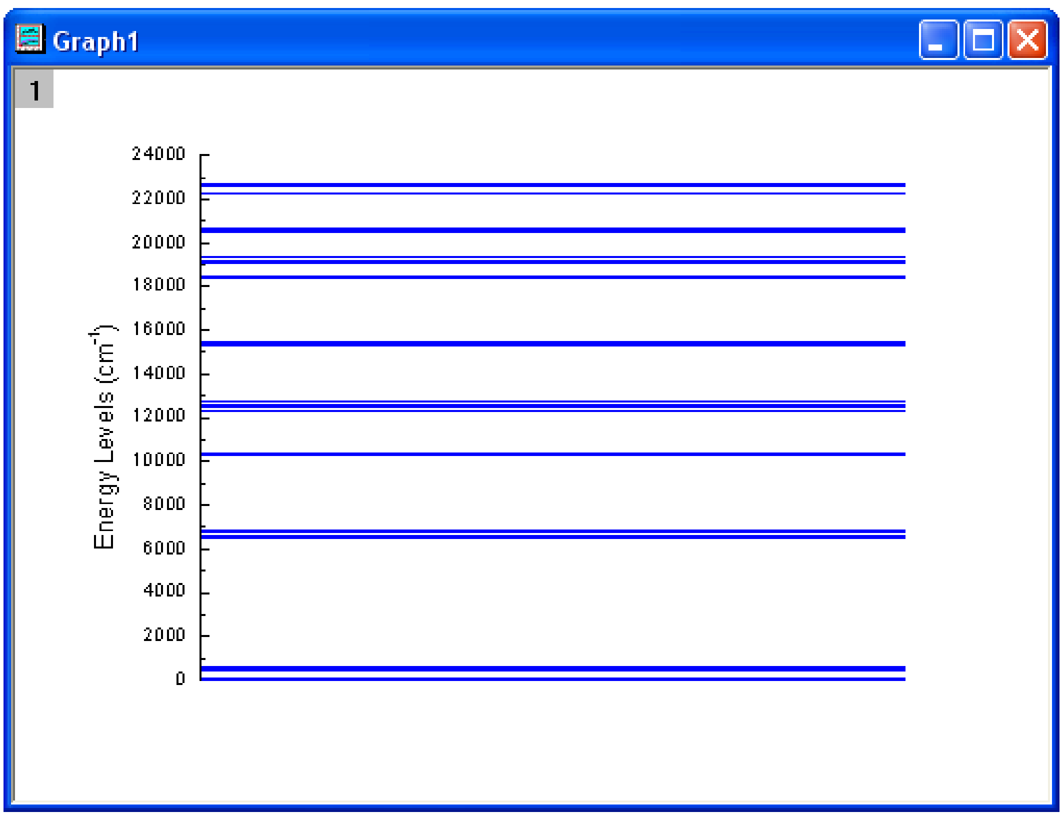
7. Select **Vertical** from the left **Selection** panel. Set Y axis **From** 0 **To** 24000, and **Increment** is 2000.



8. Switch to the **Title&Format** tab, make sure **Left** is selected in the left **Selection** panel. Change the **Title** to *Energy Levels (cm\+(-1))*, **Major Ticks** and **Minor Ticks** both are *In*. Then click the **OK** button.



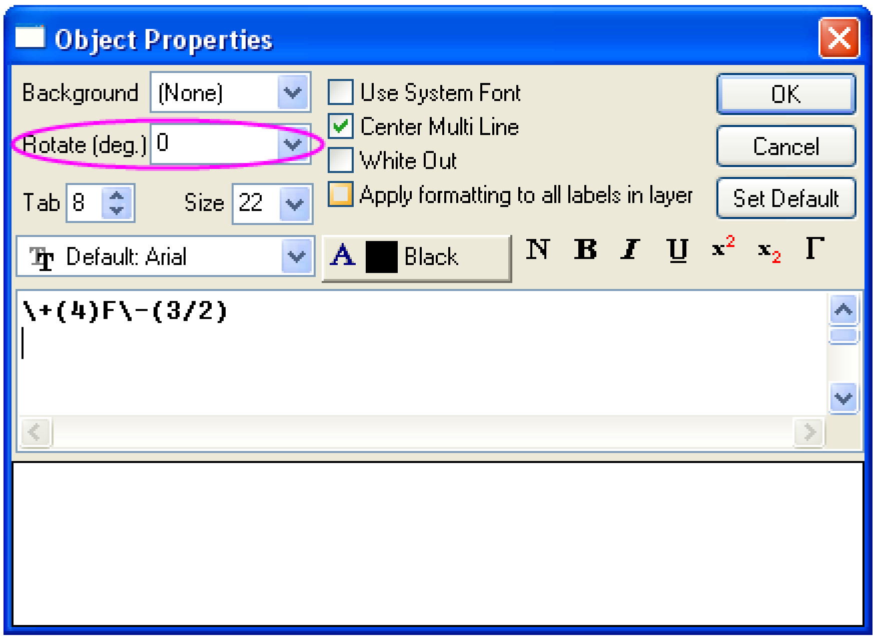
9. Delete the legend, X axis, title of X axis and tick labels of X axis.



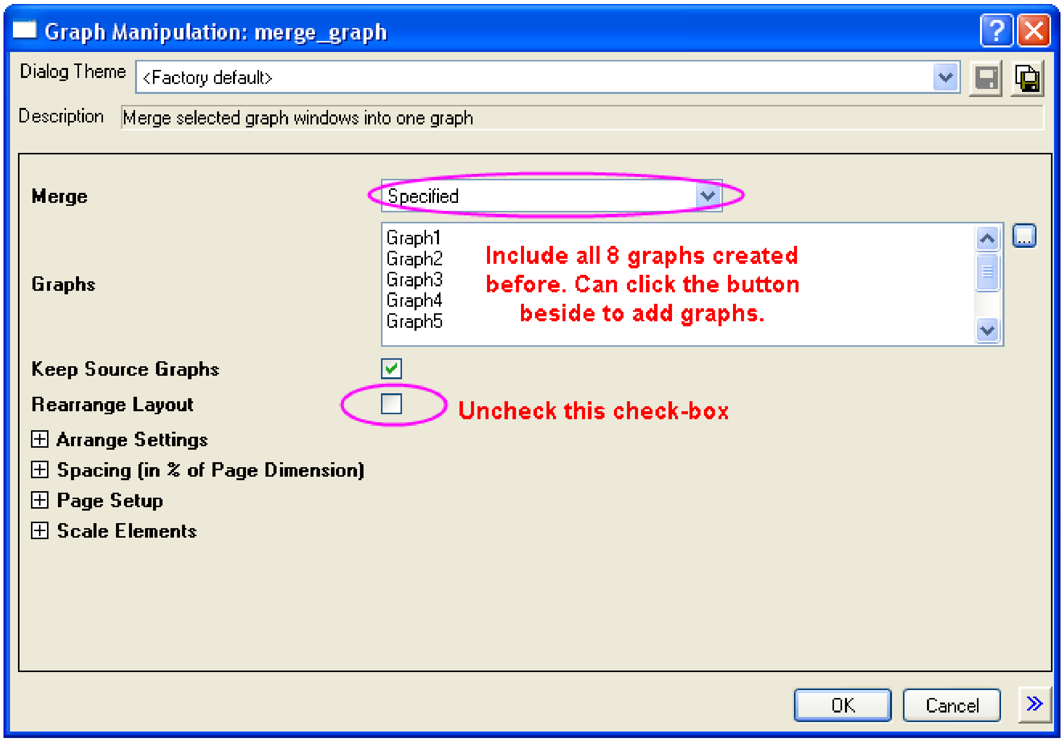
10. Highlight cell 1 to cell 8, cell 9 to cell 15, cell 16 to cell 21, cell 22 to cell 26, cell 27 to cell 31, cell 32 to cell 43, and cell 44 to 48 respectively, and then repeat step 3 through step 9 to make another 7 graphs. The **Increment** of Y axis (step 7) all are set to 100, and **From** and **To** are 0 to

600, 6500 to 6900, 10200 to 10500, 12300 to 12800, 15200 to 15600, 18300 to 20800, 22200 to 22700, respectively. Also, the **Title** of Y axis (step 8) are different, as *\+(4)I\-(15/2)*, *\+(4)I\-(13/2)*, *\+(4)I\-(11/2)*, *\+(4)I\-(9/2)*, *\+(4)F\-(9/2)*, *\+(4)F\-(7/2) \+(2)H2\-(11/2) \+(4)S\-(3/2)*, and *\+(4)F\-(3/2) \+(4)F\-(5/2)*, respectively.

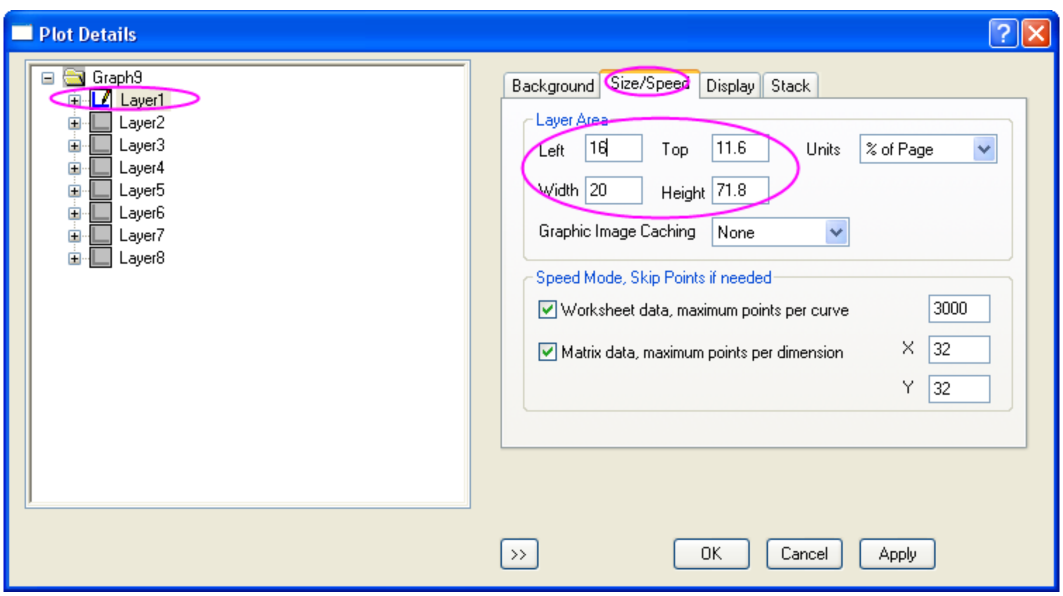
11. Change the Y title's rotate angle of graphs created in step 10. Just select the Y title and choose **Properties** from the right-click menu and set **Rotate(deg.)** to 0 in the open **Object Properties** dialog.



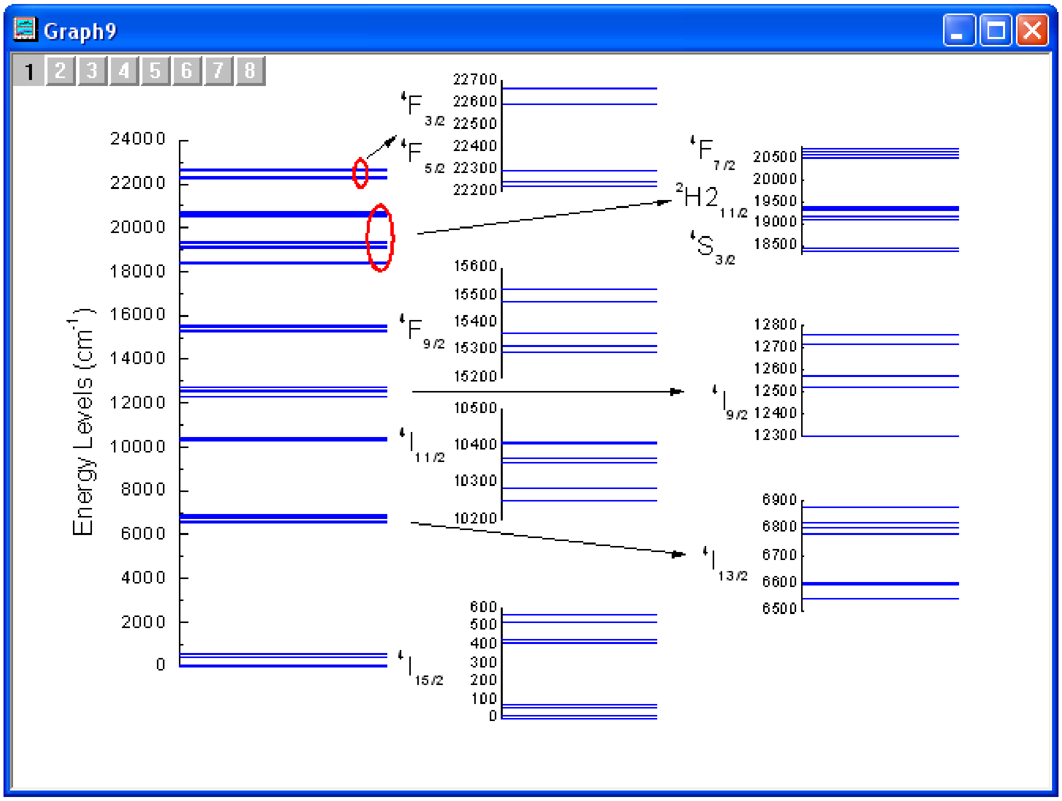
12. Activate one of the graphs, select menu **Graph: Merge Graph Window**. Change the settings as following and click the **OK** button to merge 8 graphs created before.



13. Double click on the merged graph to open the **Plot Details** dialog. In the **Size / Speed** tab of layer 1, change the **Layer Area** as following.



14. Change other layers' size as 15 by 15 (Width and Height in step 13), then move them to the proper position, adjust the size of Y axis label and add the corresponding arrows and circle, finally get the graph as the following image shows.



5.4.8 Create an 8 layer multi-panel plot template

Summary

All child windows in Origin, with the exception of the Notes window, are created from template files. These template files describe how to construct the window. For a graph window, the template file determines all page and layer characteristics, such as page size, number of layers, inclusion of text labels, data plot style information, etc.

The template library lists all built-in as well as user-created templates.

Minimum Origin Version Required: Origin 8.0 SR6

What you will learn

- How to create an 8 layer multi-panel plot
- How to save the formatting as a template
- How to reuse the template with similar data

Steps

Choosing the Data Source

1. Click the **New Project** button on the Standard toolbar, to begin with a new project.

2. Click the **Import Wizard** button on the Standard toolbar. The Import Wizard opens. (Note that if this is the first time that you have started the Import Wizard, you will experience a slight delay as Origin compiles the necessary files.)
3. Verify that the **ASCII** radio button is selected in the **Data Type** group.
4. Click the browse button to the right of the File text box. Navigate to the Origin folder; browse to the Samples folder and then the Curve Fitting folder. Select Step01.dat from the list of files.
5. Click the **Add File(s)** button.
6. Click **OK**.
7. Leave the **Import Filters for current Data Type** as **Data Folder: step**. (This filter has the settings to use when importing the file.)
8. Click the **Finish** button. The data file imports into the worksheet.

Plotting the Data

1. Highlight the entire worksheet of data. (Note that you can select an entire worksheet by placing your cursor in the blank area in the upper left corner of the worksheet. When the cursor becomes a downward pointing arrow, click once to select the entire worksheet.)
2. Select **Plot:Multi-Curve:9 Panel**. A new 9 layer graph is created.
3. Select layer 9 by clicking inside it.
4. Press the **Delete** key on the keyboard. This will delete layer 9, leaving you with 8 layers.
5. Select **Graph:Layer Management**. The **Layer Management** dialog opens.
6. Select the **Arrange** tab.
7. Set **Column** to 2 and **Row** to 4.
8. Click the **Apply** button. The preview in the dialog redraws to show you a 2x4 arrangement.
9. Click **OK**.

Editing the Graph

The goal is to save this 8 panel graph as a template; i.e. a new plot type, so that it can be used again with new similar data. Since the template will also save plot style information, let's customize the graph a bit further.

1. Double-click on the X axis in layer 1. The **X-Axis** dialog opens.
2. Select the **Grid Lines** tab.
3. Check the **Major Grids** and **Minor Grids** checkboxes.
4. Set the **Line Color** for both Major and Minor grids to **LT Gray**.
5. Check the **Apply To Grid Lines** checkbox for **This Layer**.
6. Click OK.
7. Select layer 1 by clicking inside it.
8. Right-click inside the layer and select **Copy Format:All Style Formats**. This will copy the style formats of layer 1.
9. To apply formatting to all layers, right-click outside of any layer (make sure that no layer is selected...one easy way to do that is to right-click in the gray area of the window, outside the white printable part of page), and select **Paste Format**.

Saving as a new graph template

1. Select **File:Save Template As**. A dialog opens allowing you to choose the category that the template will be saved in as well as the name given to the new template.
2. Change the **Category** to **UserDefined**.
3. Change the **Template Name** from **PAN9** to **PAN8**. (Note that the **Template Name** that appears when the dialog opens is the name of the original template that was used to create the graph.)
4. Click OK.

Plotting into your new template

1. Click the **New Folder** button on the Standard toolbar.
2. Click the **Import Wizard** button on the Standard toolbar. The Import Wizard opens.
3. Click the browse button to the right of the File text box. Navigate to the Origin folder; browse to the Samples folder and then the Curve Fitting folder. Select Step02.dat from the list of files.
4. Click the **Add File(s)** button.
5. Click **OK**.
6. Leave the **Import Filters for current Data Type** as **Data Folder: step**. (This filter has the settings to use when importing the file.)
7. Click the **Finish** button. The data file imports into the worksheet.
8. Highlight the entire worksheet of data. (Note that you can select an entire worksheet by placing your cursor in the blank area in the upper left corner of the worksheet. When the cursor becomes a downward pointing arrow, click once to select the entire worksheet.)
9. Select **Plot:Template Library** or click the **Template Library** button on the **2D Graphs** toolbar.
10. Scroll down to the **UserDefined** category under Graph Template.
11. Select **PAN8**. (Note that the Preview window is not a preview of the new data that you are plotting. It is an image of the graph when you saved your template.)
12. Click **Plot**.

5.5 Contour

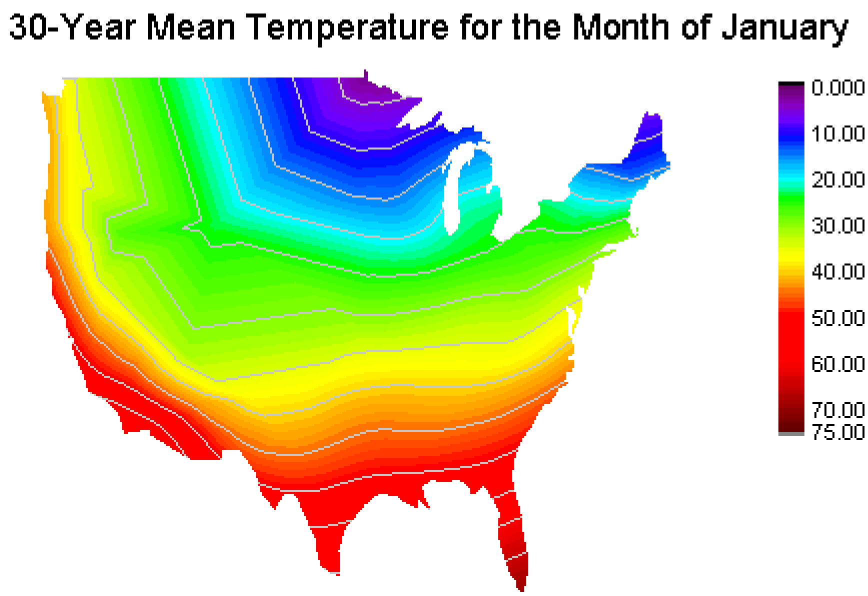
Topics covered in this section:

1. XYZ Contour
2. Contour Plot with Major and Minor Levels Filled by Using Color Palette
3. Contour Plots and Color Mapping
4. Polar Contour
5. Ternary Contour
6. Combining Line and Contour Plots
7. Contour Graph with XY Data Points and Z Labels
8. Flattened Colormap Surfaces with Increasing Z Offset
9. Contour Plot with Vector Overlay

5.5.1 XYZ Contour

Summary

This tutorial will show you how to create a contour from XYZ data.



What will you learn

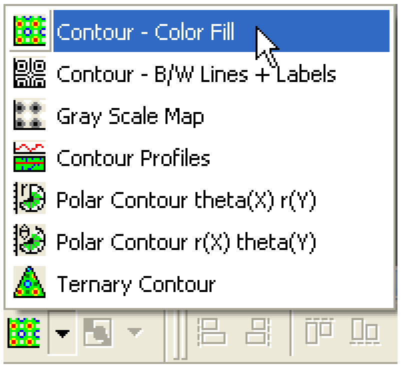
This tutorial will show you how to

- Create a Contour Plot from XYZ data
- Customize levels, lines, and color mapping
- Use a Custom Boundary
- Cutomize the Color Scale
- Customize the axes of the plot

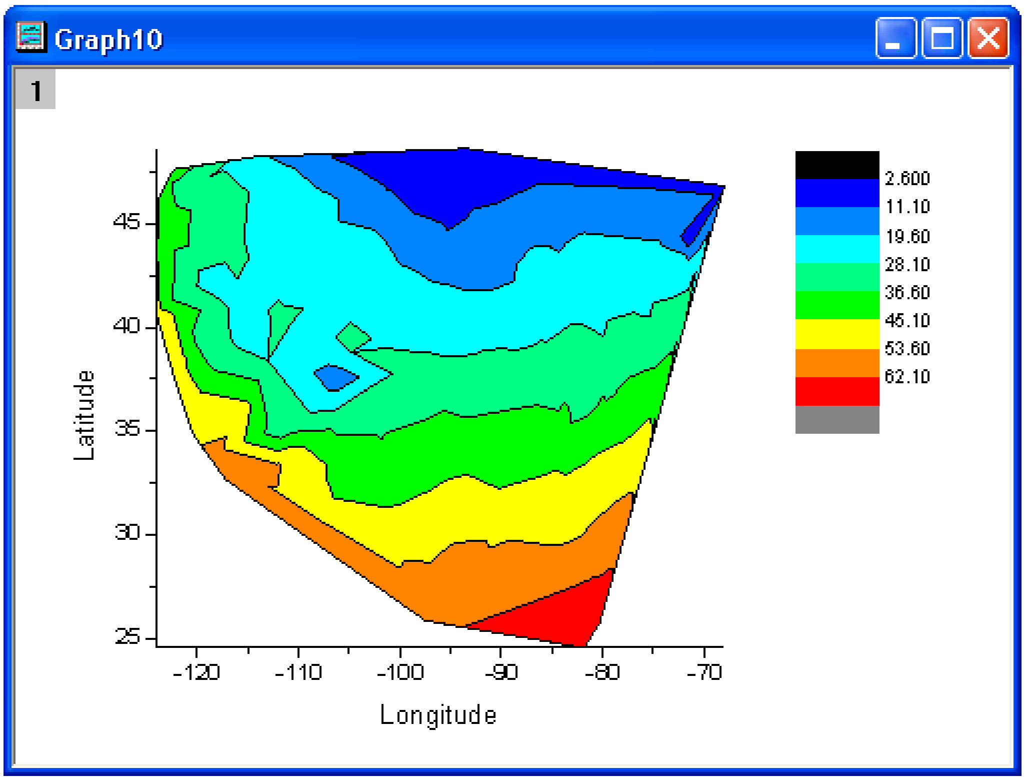
Steps

This tutorial is associated with the 2D and Contour Graphs project:*\Samples\2D and Contour Graphs.opj*.

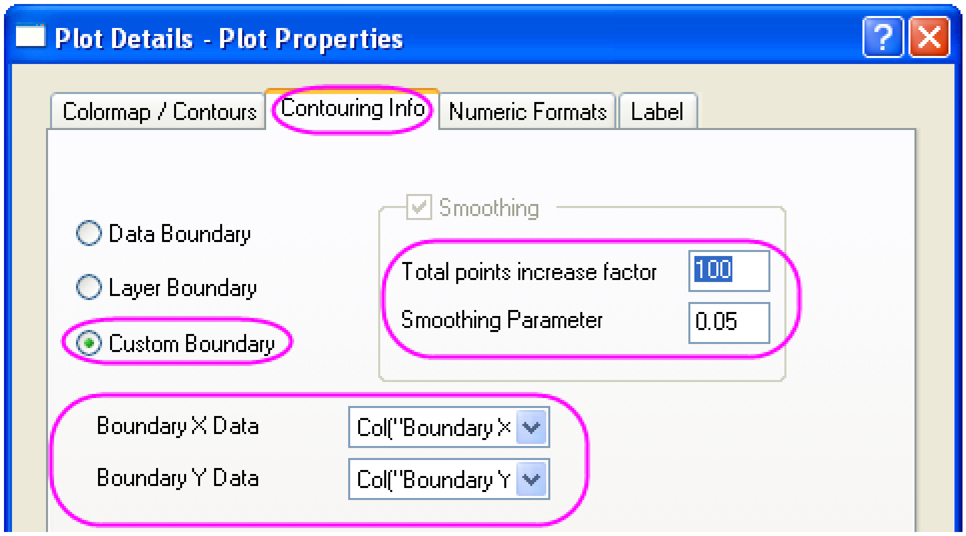
1. Open the *2D and Contour Graphs: Contour: XYZ Contour* folder in the **Project Explorer**. Activate **Book1** and highlight column **D** and then click **Contour-Color Fill** button from the **3D and Contour Graphs toolbar**.



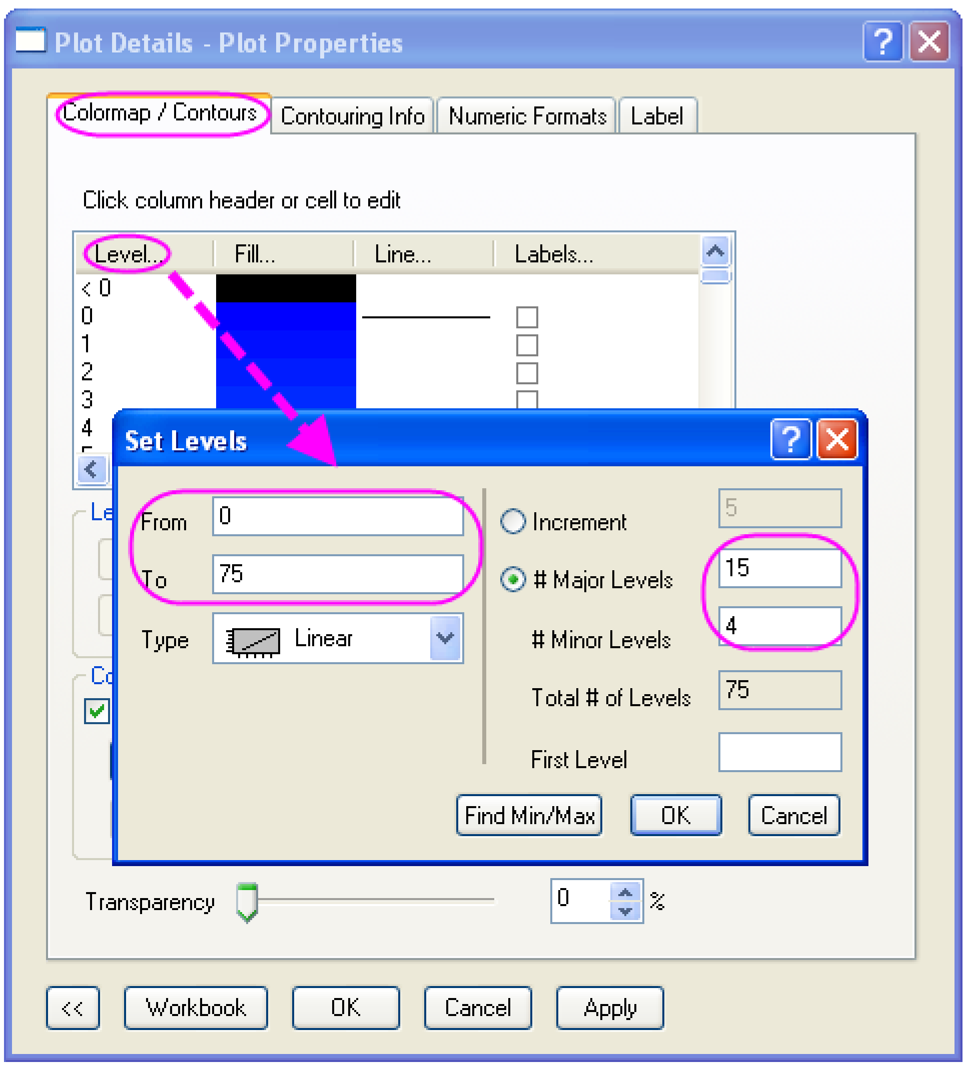
The graph should look like:



2. Double-click on the contour plot to bring up **Plot Details** dialog. Select the **Contouring Info** tab and set the options in the dialog as the screenshot below:

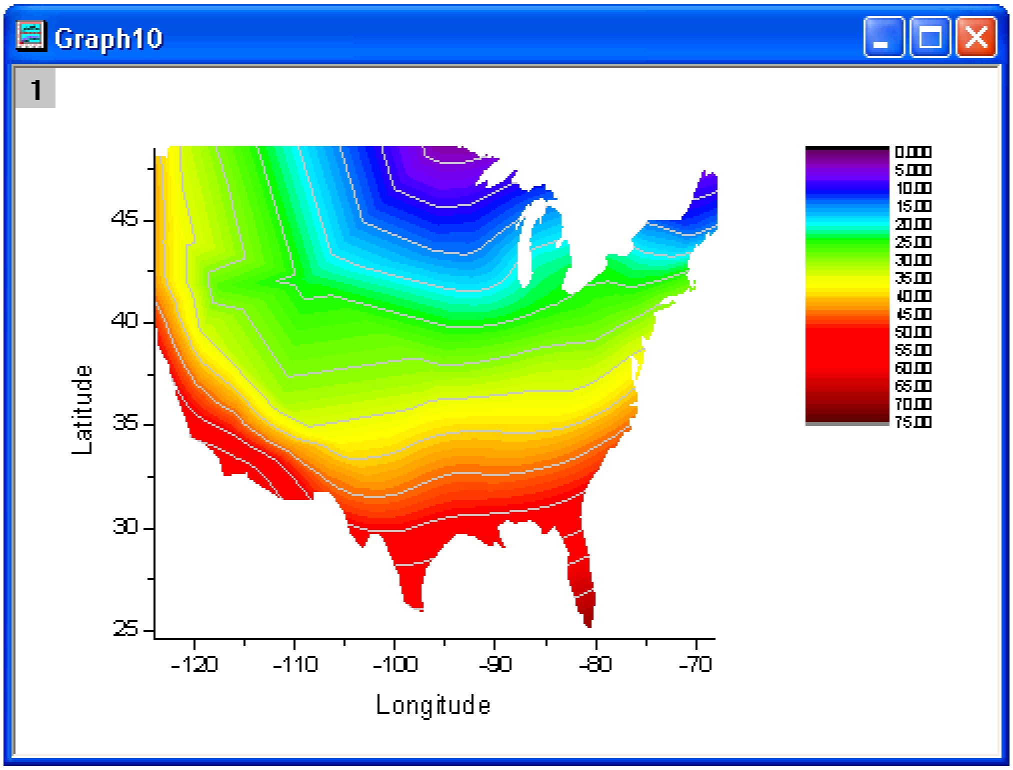


3. Select the **Color Map/Contours** tab, click the **Level** heading and set the dialog as the following screenshot shows:

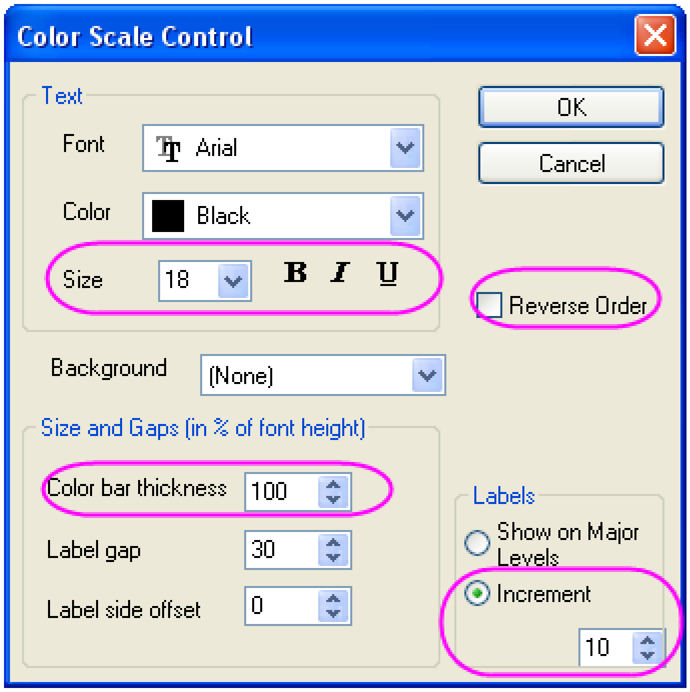


- Click the **Fill** heading, select **Load Palette** and then select **Rainbow** from the Palette list.
- Click the **Lines** heading, select **Show on Major Levels** only. Enable **Apply to All** checkbox and choose **LT Gray** for the **Color** dropdown list.

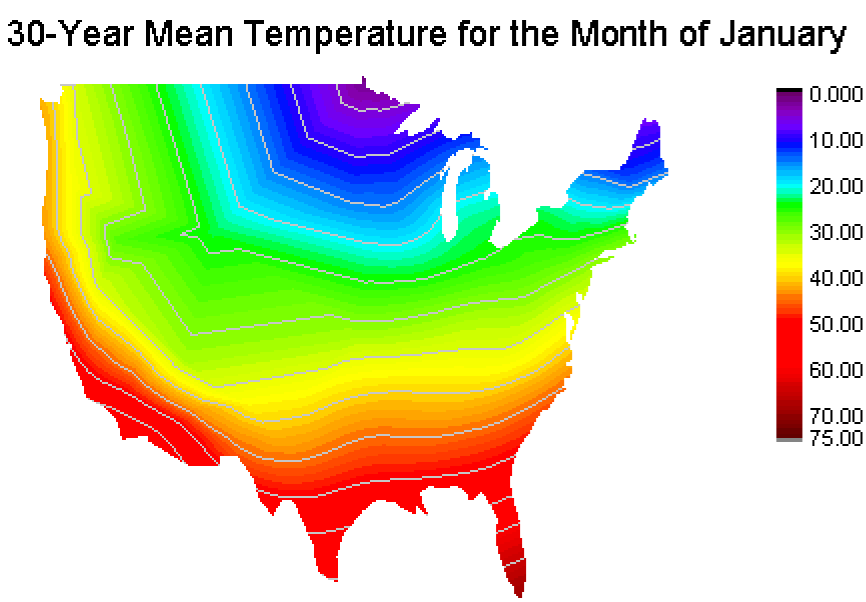
4. Click OK to close the Plot Details dialog, then the graph should look like



5. Select **Format: Axes: X Axes** to open the **Axis** dialog. Then do the following things.
   - Select the **Scale** tab and then select **Horizontal** icon in the **Selection** list box , set **X From**, **To** and **Increment** equal to **-127**, **-65** and **10** individually.
   - Select the **Vertical** icon in the **Selection** list box, set **Y From**=**23**, **To**=**50**, and **Increment**=**5**.
   - Select the **Title and Format** tab and then select **Left** icon in the **Selection** list box, disable the **Show Axis & Ticks** check box.
   - Select the **Bottom** icon in the **Selection** list box,disable the **Show Axis & Ticks** check box.
   - Select the **Tick Labels** tab and then select **Bottom** icon in the **Selection** list box,disable the **Show Major Labels** check box.
   - Select the **Left** icon in the **Selection** list box, disable the **Show Major Labels** check box.
6. Click OK to close the dialog. Double-click on the color scale legend to bring up its properties dialog. And set the dialog as the following screenshot shows:



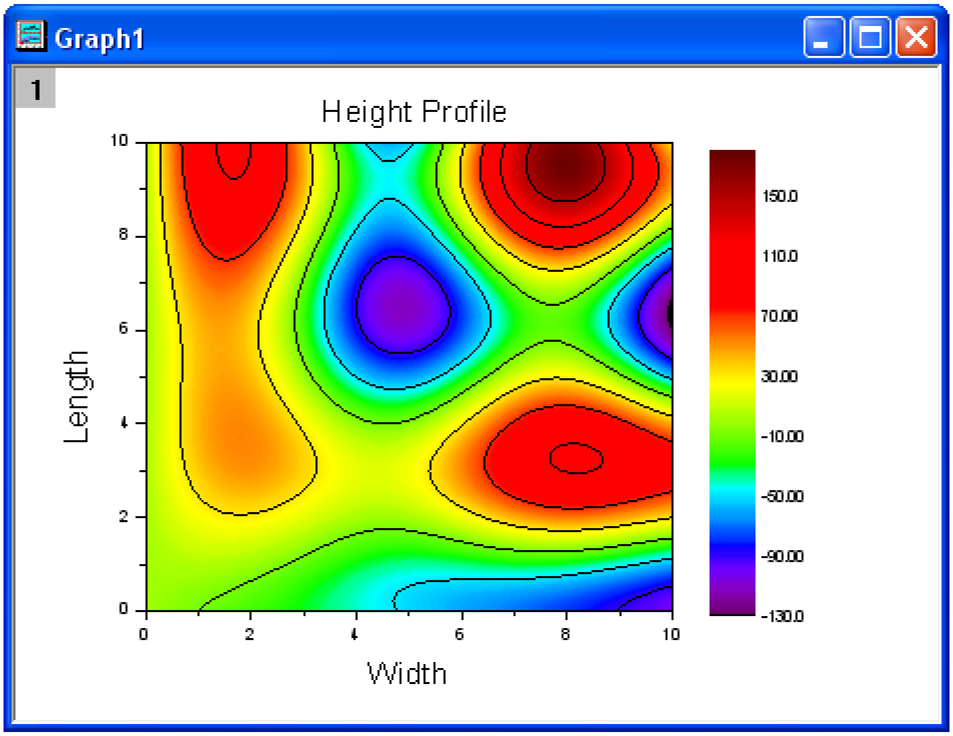
7. Right-click above the contour plot and select **Add Text** to add a graph title. Use the **Format** toolbar and/or double-click on the completed text label to update the font, font size, etc. The graph should look like



5.5.2 Contour Plot with Major and Minor Levels Filled by Using Color Palette

Summary

The levels in a contour plot can be divided into many levels from the minimum to the maximum, including major levels and minor levels. Each level can then be assigned a specific color, or a color palette could be used to assign colors to the levels. Contour line for each level can be customized as well. In this tutorial, a contour plot is created from a matrix, and then Rainbow palette will be used to assign colors.



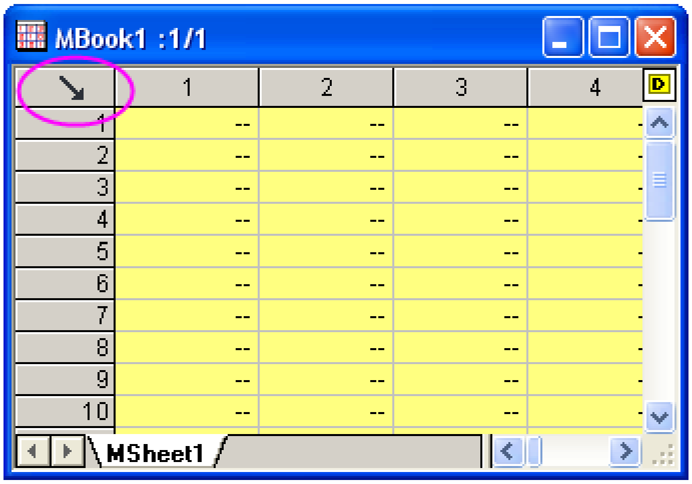
Minimum Origin Version Required: Origin 8.1 SR0

What you will learn

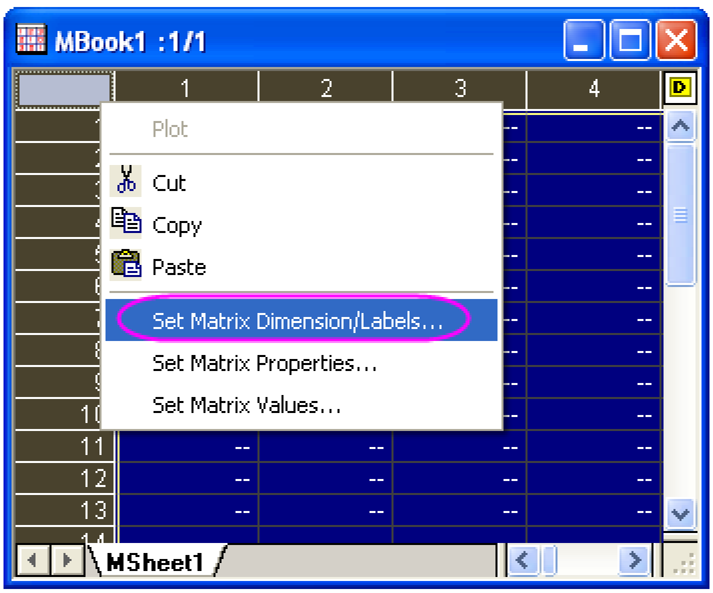
- Set dimensions for a matrix
- Set values for a matrix
- Turn on/off the **Speed Mode**
- Create a contour plot from a matrix
- Customize levels, contour lines and filled color for a contour plot
- Change the settings of the color scale

Steps

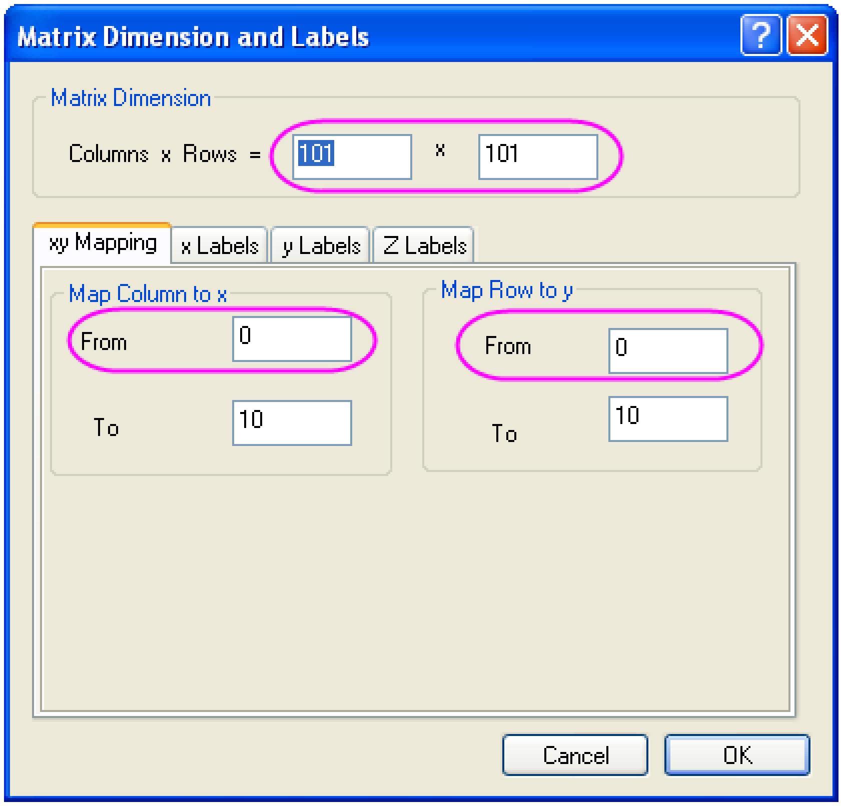
1. New a project, then click the **New Matrix**  button to create a new matrix. Then move the mouse to the upper left corner of the matrix until the shape of the pointer has been changed as the following image shows and click it to select the entire matrix.



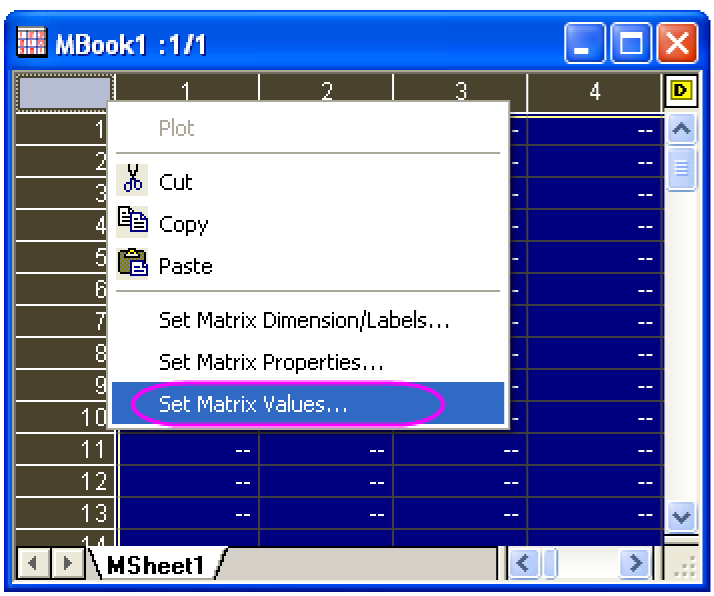
2. Right-click and select **Set Matrix Dimensions** from the context menu.



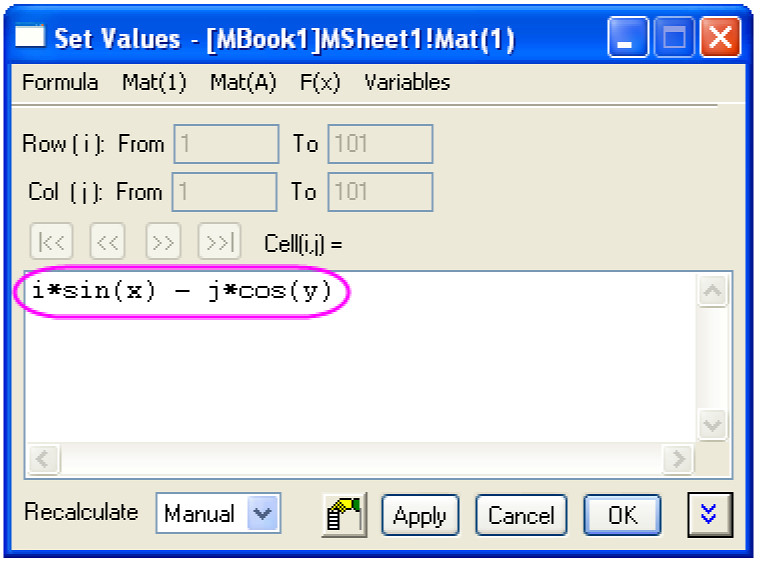
3. Set the pop-up dialog as the following image shows. Then click the **OK** button to finish setting dimensions.



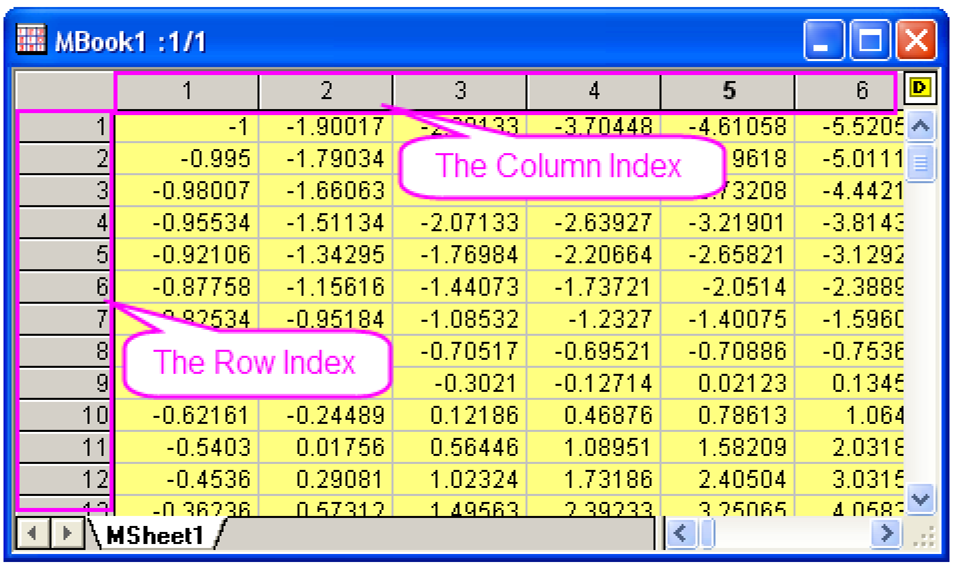
4. Highlight the matrix and right-click on it. Select **Set Matrix Values** to open the **Set Values** dialog.



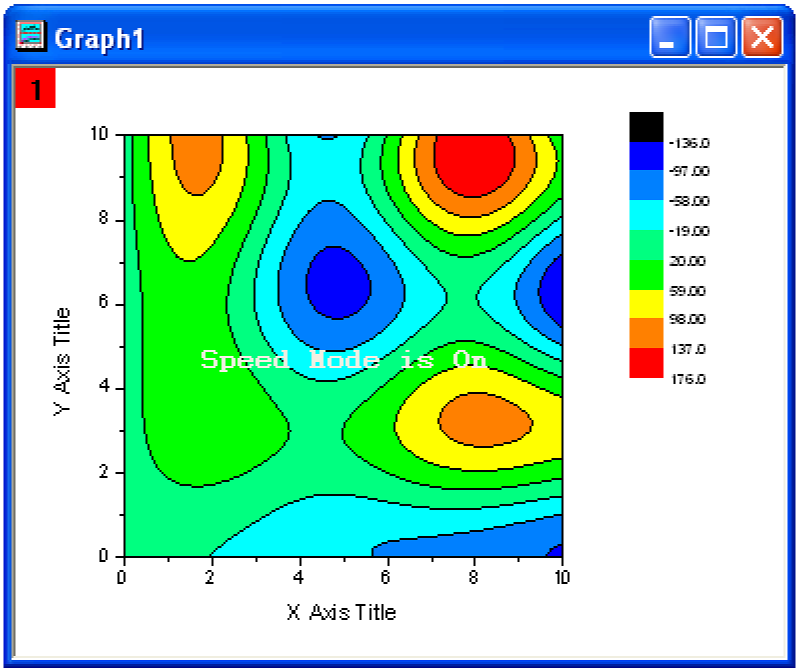
5. In the open dialog, input **i*sin(x) - j*cos(y)** in the **Formula** edit box.



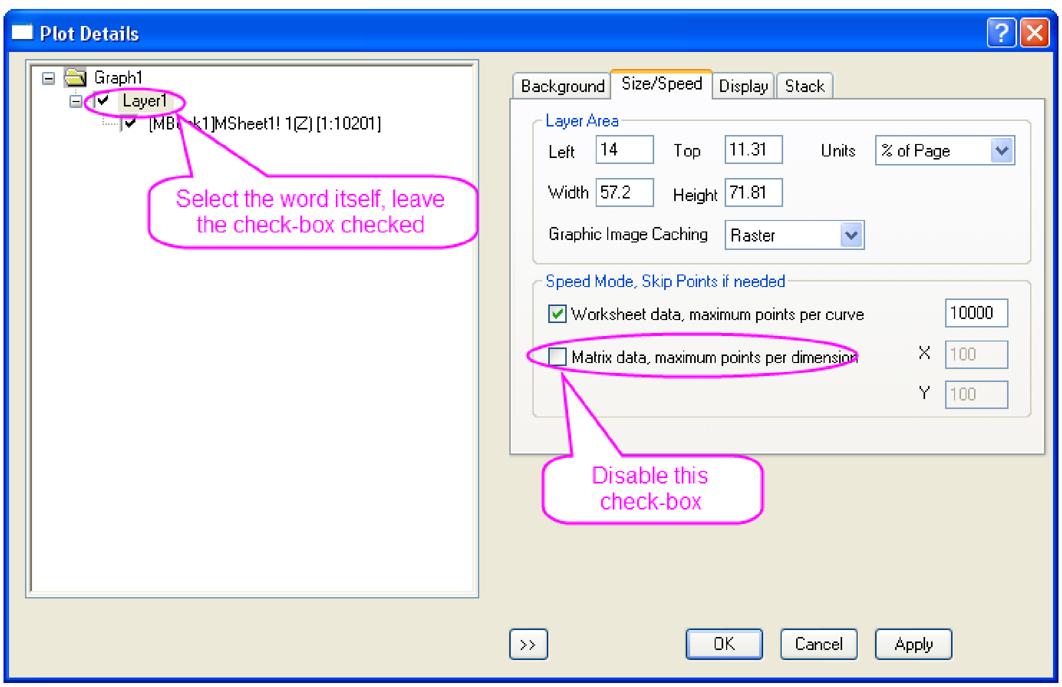
6. Click the **OK** button to generate data and the matrix look like this:



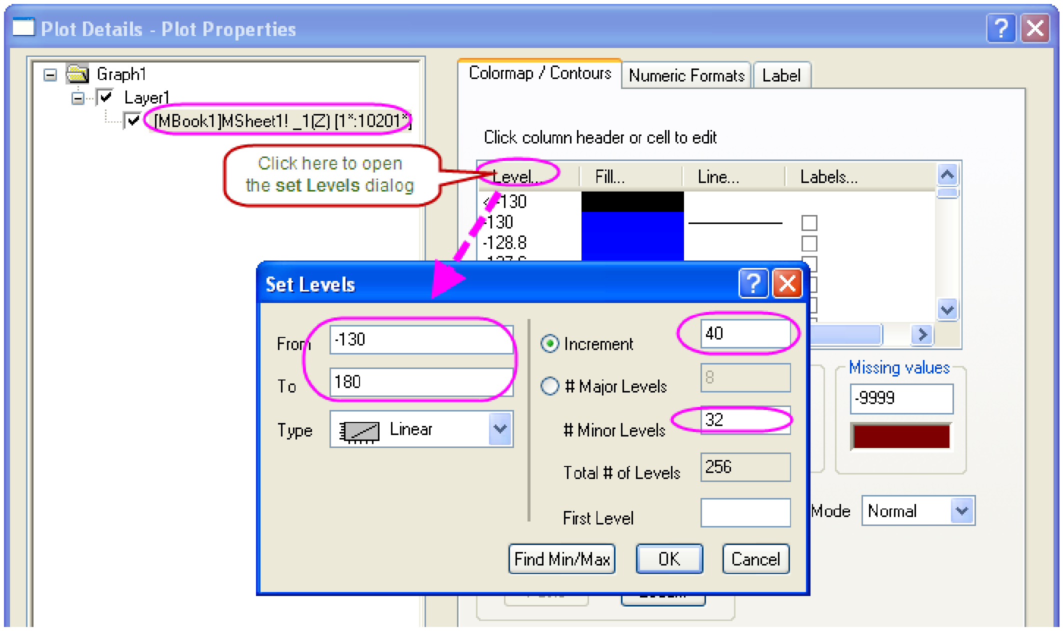
7. Activate this matrix and select **Plot: Contour: Color Fill** from the Origin main menu to create a contour plot. The graph looks like this:



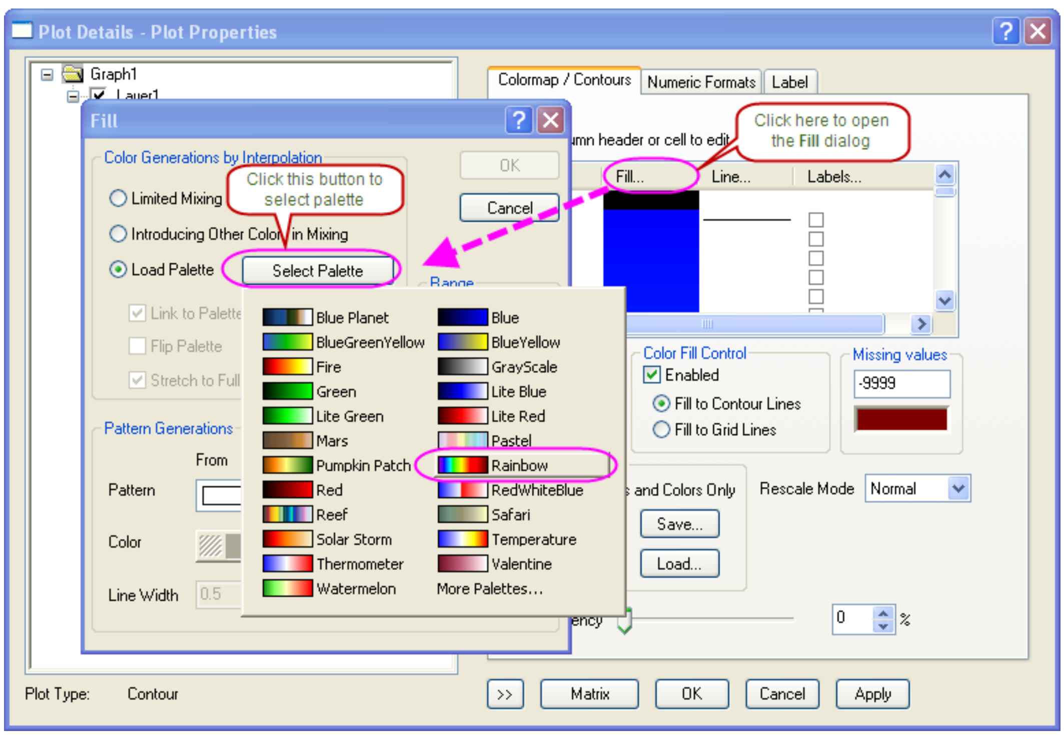
8. Double-click on the contour plot to open the **Plot Details** dialog. Select **Layer1** in the left panel and then go to the **Size/Speed** tab in the right panel, disable the **Matrix data, maximum points per dimension** to turn off the **Speed Mode**.



9. Select **[MBook1]MSheet1!1(Z)[1:10201]** to go to the matrix level. Click on the **Level** heading in the right panel to open the **Set Levels** dialog. Set values as the following image shows.



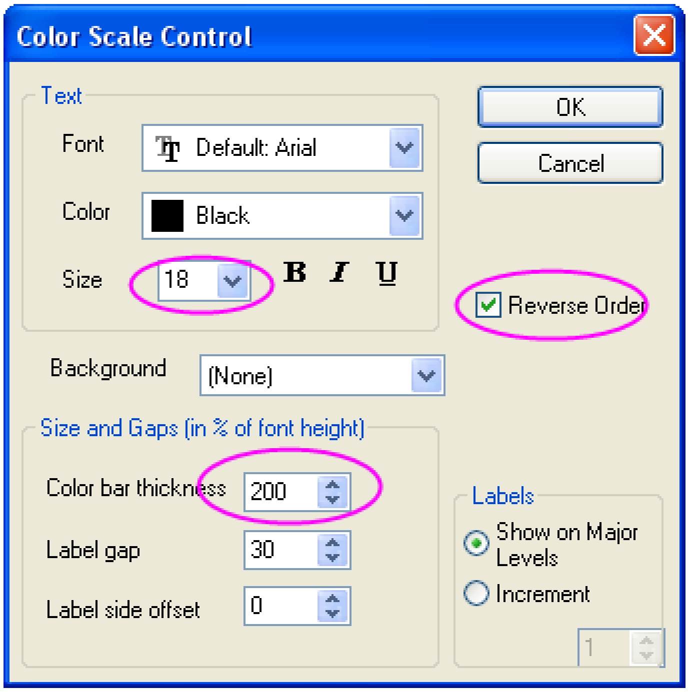
10. Click on the **Fill** heading to open the **Fill** dialog. Select **Load Palette** radio box and then click the **Select Palette** button to select the **Rainbow** palette. Click the **OK** button to close this dialog.



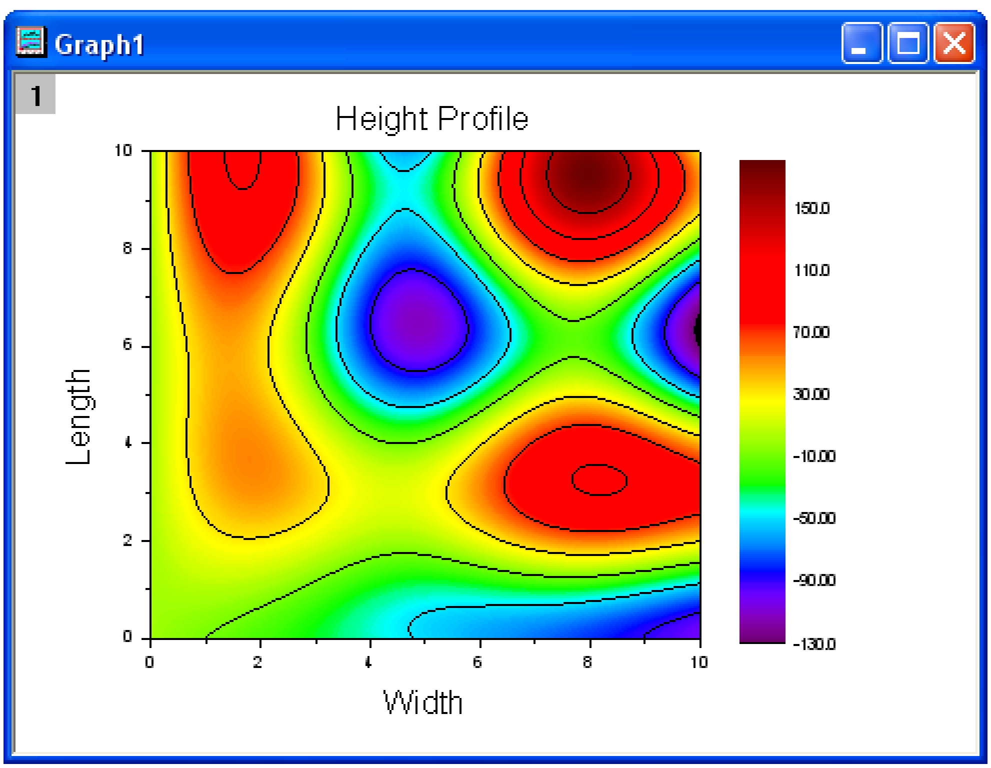
11. Click the **OK** button to close the **Plot Details** dialog.

12. Double click on the color scale to open the **Color Scale Control** dialog. Set **Size** to 18 and **Color bar thickness** to 200. Check the check-box before **Reverse Order**. Then click the

OK button to apply the settings to the color scale. Select the color scale and drag the handles around it to get a appropriate size.



13. Change the labels of axes, label of X axis is **Width** and label of Y axis is **Length**, both font size is 31. Add a text object, say **Height Profile**, to the top of layer, font size is 31. Finally get a contour graph as follow:



5.5.3 Contour Plots and Color Mapping

Summary

Origin offers rectangular, polar, and ternary contour plots. For **rectangular contour plots**, the data can be either in a matrix or in a worksheet in XYZ format. **Polar contour plots** can be generated from three columns of data in a worksheet, organized either as R Θ Z or Θ R Z. **Ternary contour plots** can be generated from worksheet data organized in X Y Z Z format where the 2nd Z-column contains the 4th parameter which is the height value at a given XYZ point in the ternary space.

Many options are available for customizing contour plots, such as setting different major and minor contour levels, displaying contour lines only at major levels, applying color palettes, and there is also a control for a custom boundary in the case of contour plots created directly from the worksheet.

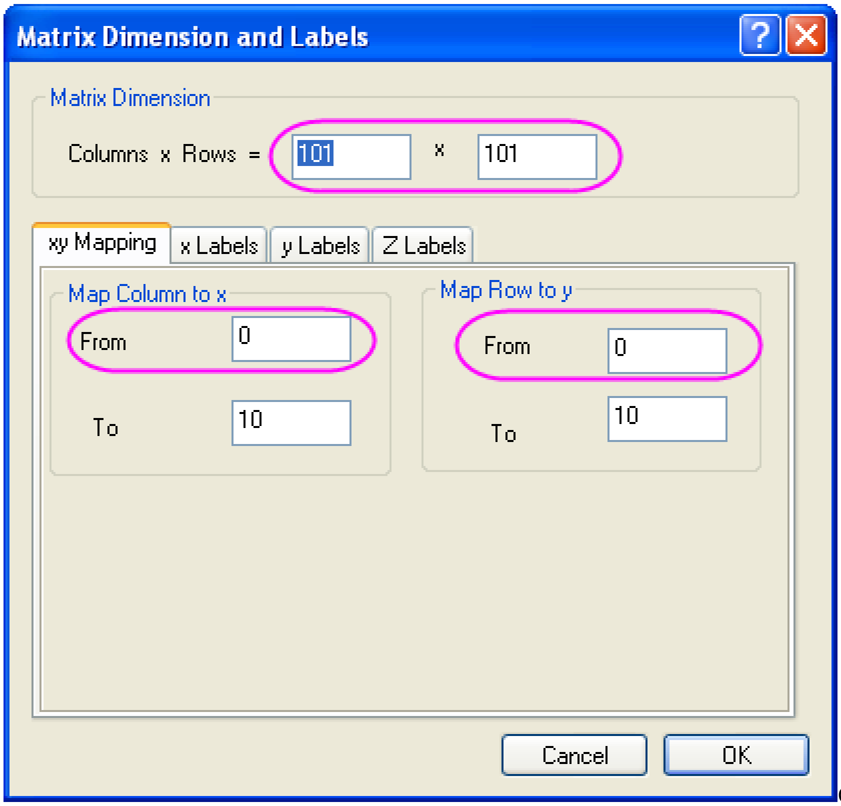
This tutorial will show you how to:

- Set values in a Matrix and create a Contour Plot
- Customize levels, lines, and color mapping
- Extract data from contour lines
- Create a Contour Plot directly from XYZ data
- Use a Custom Boundary

Create Contour Plot from Matrix

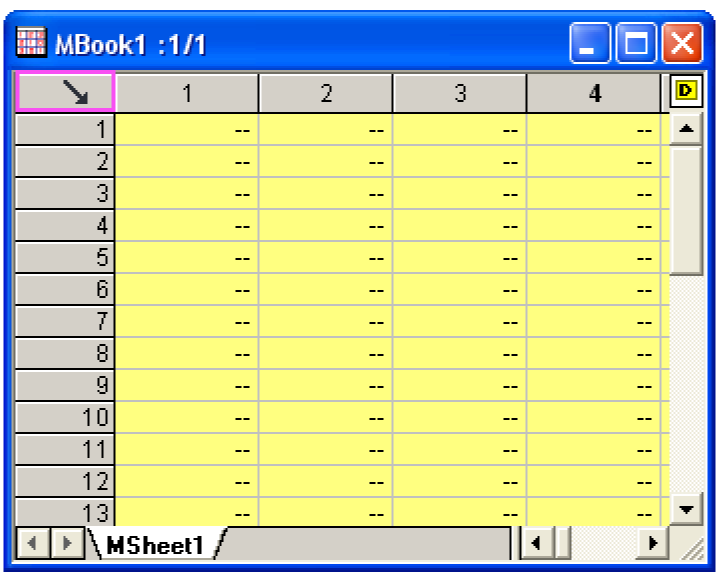
1. Click the **New Matrix button** to create a new matrix.

2. Select **Matrix: Set Matrix Dimensions/Labels** from the main menu and set the dialog as shown in the following image:



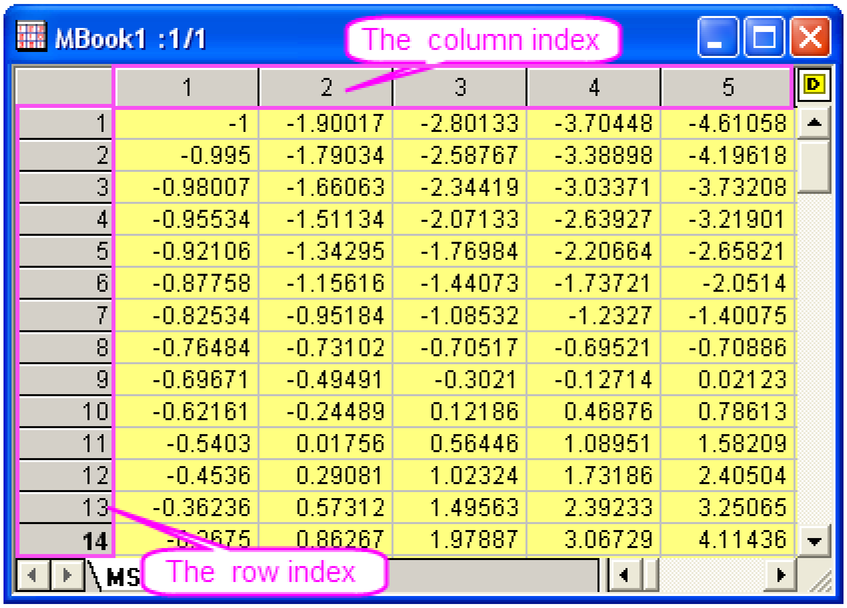
Click **OK**.

3. Then move the mouse to the upper left corner of the matrix until the shape of the pointer has been changed, as the following image shows, and click it to select the entire matrix.



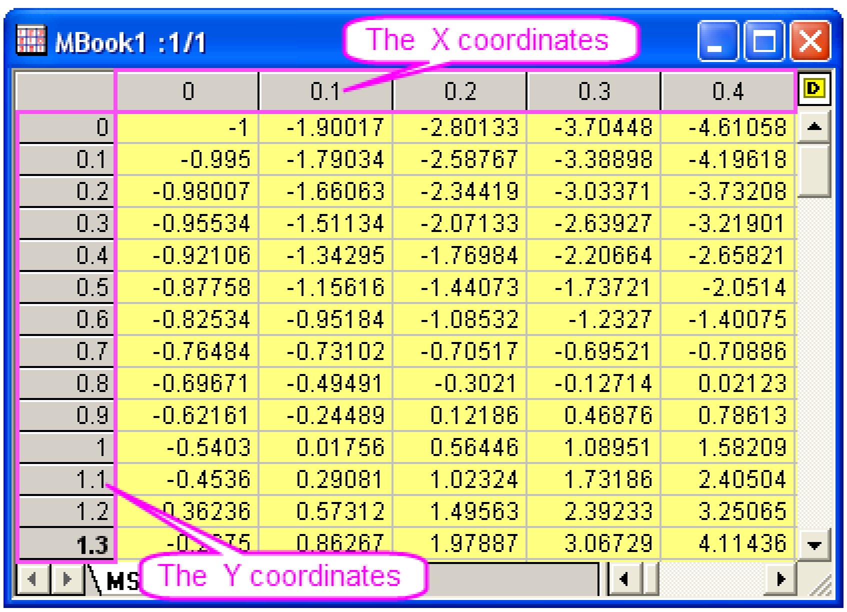
4. Input data into the matrix by using the **Set Values** dialog. Right-click on the matrix. Select **Set Matrix Values** to open the **Set Values** dialog. Input **i*sin(x) - j*cos(y)** in the **Formula** edit

box and click **OK** to generate data. The matrix should look like this:



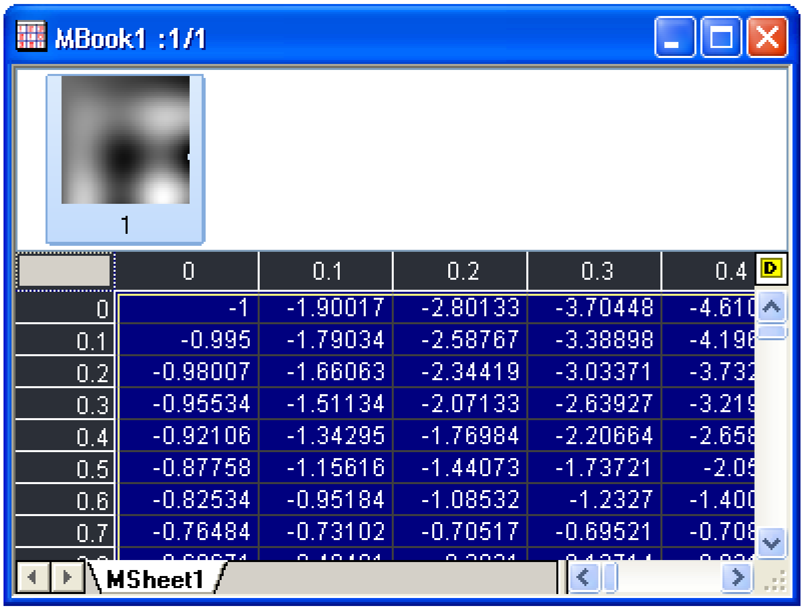
	1	2	3	4	5
1	-1	-1.90017	-2.80133	-3.70448	-4.61058
2	-0.995	-1.79034	-2.58767	-3.38898	-4.19618
3	-0.98007	-1.66063	-2.34419	-3.03371	-3.73208
4	-0.95534	-1.51134	-2.07133	-2.63927	-3.21901
5	-0.92106	-1.34295	-1.76984	-2.20664	-2.65821
6	-0.87758	-1.15616	-1.44073	-1.73721	-2.0514
7	-0.82534	-0.95184	-1.08532	-1.2327	-1.40075
8	-0.76484	-0.73102	-0.70517	-0.69521	-0.70886
9	-0.69671	-0.49491	-0.3021	-0.12714	0.02123
10	-0.62161	-0.24489	0.12186	0.46876	0.78613
11	-0.5403	0.01756	0.56446	1.08951	1.58209
12	-0.4536	0.29081	1.02324	1.73186	2.40504
13	-0.36236	0.57312	1.49563	2.39233	3.25065
14	-0.2675	0.86267	1.97887	3.06729	4.11436

5. Each data point in the matrix corresponds to two different kinds of indices. One is the column and row indices. The other is the X and Y coordinates. You can select **View: Show X/Y** to see the X, Y coordinates.

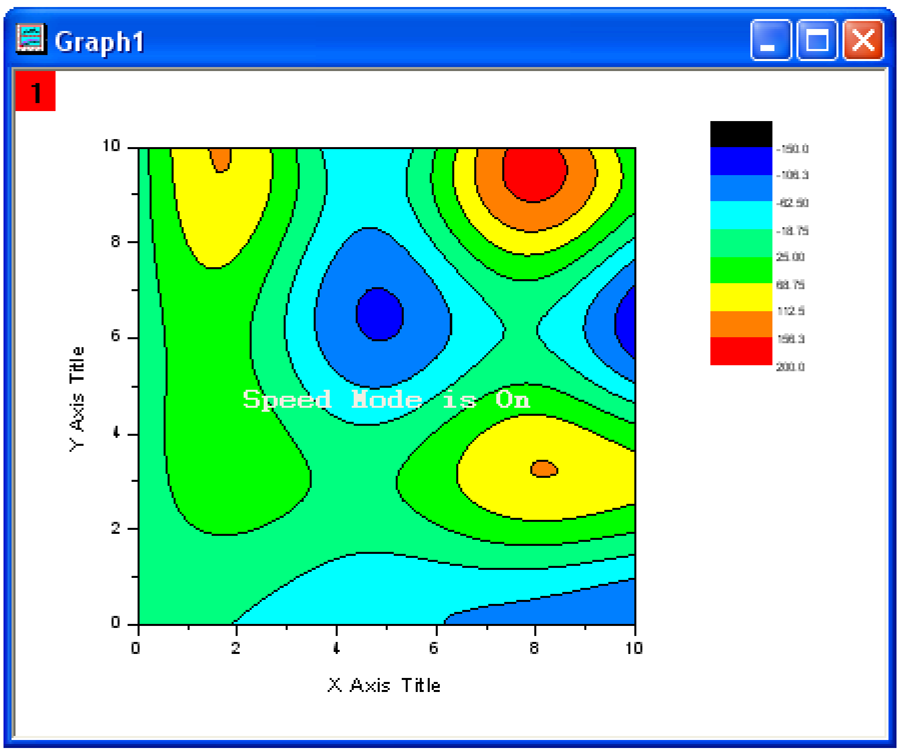


	0	0.1	0.2	0.3	0.4
0	-1	-1.90017	-2.80133	-3.70448	-4.61058
0.1	-0.995	-1.79034	-2.58767	-3.38898	-4.19618
0.2	-0.98007	-1.66063	-2.34419	-3.03371	-3.73208
0.3	-0.95534	-1.51134	-2.07133	-2.63927	-3.21901
0.4	-0.92106	-1.34295	-1.76984	-2.20664	-2.65821
0.5	-0.87758	-1.15616	-1.44073	-1.73721	-2.0514
0.6	-0.82534	-0.95184	-1.08532	-1.2327	-1.40075
0.7	-0.76484	-0.73102	-0.70517	-0.69521	-0.70886
0.8	-0.69671	-0.49491	-0.3021	-0.12714	0.02123
0.9	-0.62161	-0.24489	0.12186	0.46876	0.78613
1	-0.5403	0.01756	0.56446	1.08951	1.58209
1.1	-0.4536	0.29081	1.02324	1.73186	2.40504
1.2	-0.36236	0.57312	1.49563	2.39233	3.25065
1.3	-0.2675	0.86267	1.97887	3.06729	4.11436

6. Select **View: Show Image Thumbnails** to turn on thumbnails.



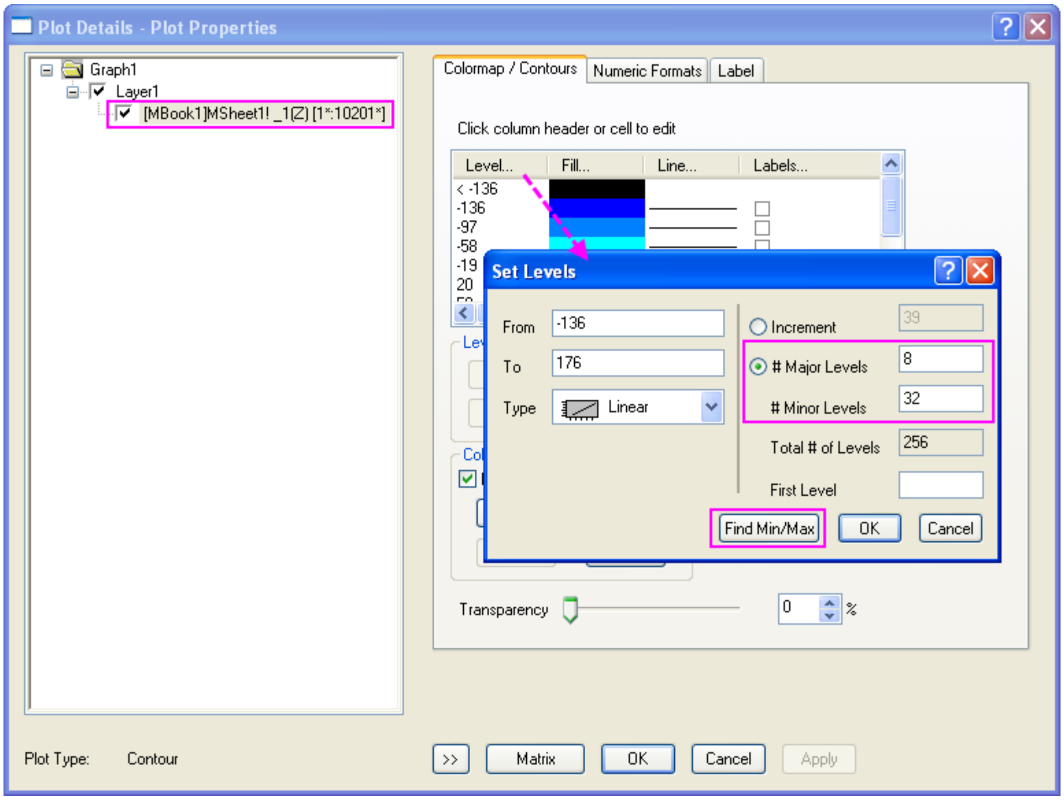
7. Now, create a contour plot. Activate the matrix and select **Plot: Contour: Contour - Color Fill** to create a contour. The graph should look like this:



Customize Levels, Lines and Color Map

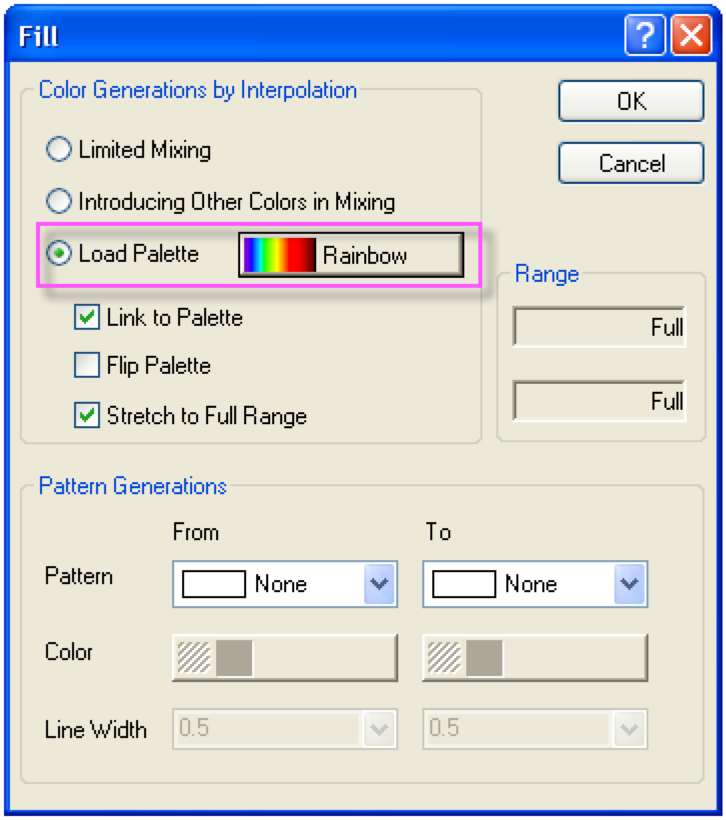
Origin makes it easy to customize a contour plot, including changing the color scale and adding labels. The following steps show you how.

1. Double-click on the contour plot to open the **Plot Details** dialog. Click on the word **Layer1** in the left panel (leave the check box checked, and click on the word itself), and then select the **Size/Speed** tab in the right panel, and disable **Matrix data, maximum points per dimension** to turn off Speed Mode.
2. Then select **[MBook1]MSheet1!1(Z)(1:10201)** to go to the matrix level. Click on the **Level** heading in the right panel to open the **Set Levels** dialog. Click **Find Min/Max** button and then set **Major Levels** to **8** and **Minor Levels** to **32**, and then click OK to close it.



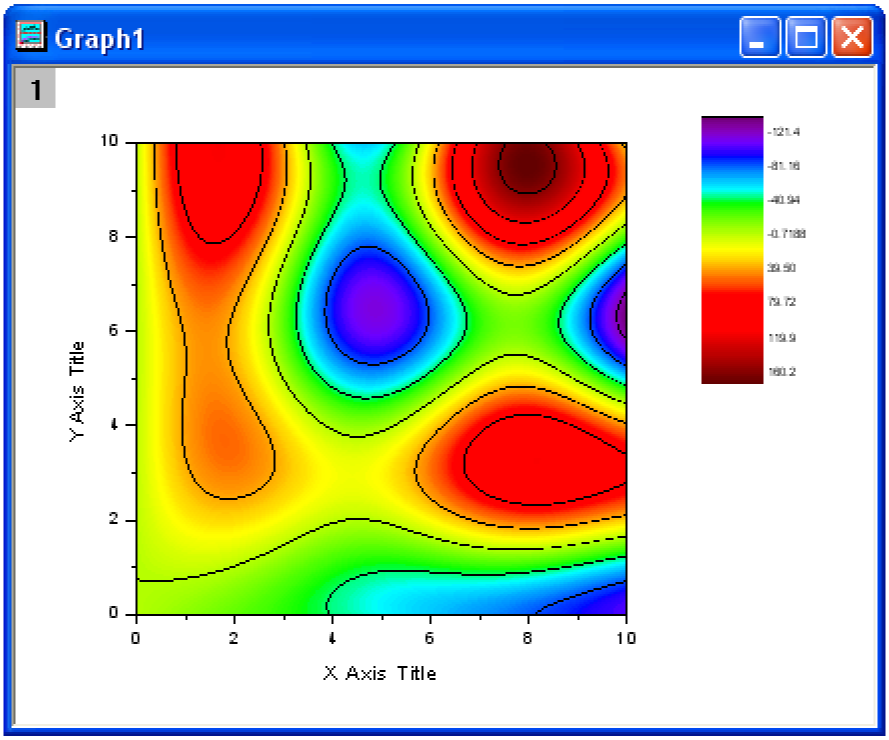
3. Now load a palette for the contour. Click on the **Fill** heading to open the **Fill** dialog, select the **Load Palette** radio box and then click the **Select Palette** button to select the **Rainbow** palette.

Click Ok to close the dialog.

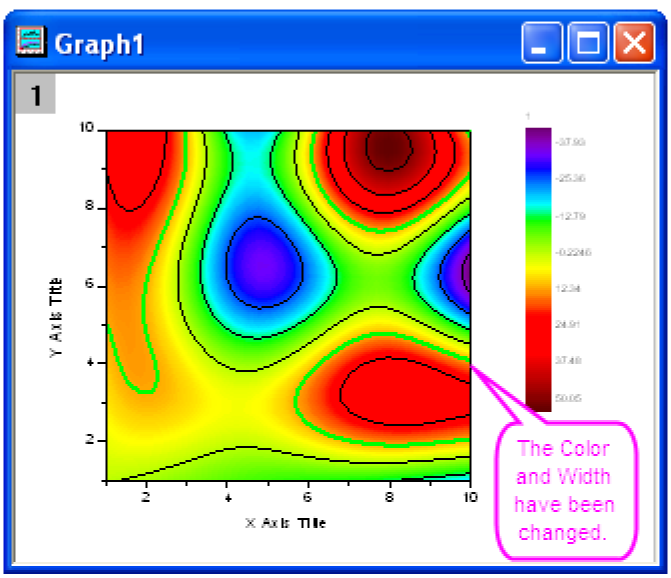


4. Click the **Line** heading to open the **Contour Lines** dialog. Then select **Show on Major Levels only** and click OK to close the dialog. Then click OK to close the Plot Details dialog.
5. Set the properties for the color scale. Right-click on the color scale and select **Properties** to open the **Color Scales Control** dialog. Select the **Show on Major Levels** check box and click OK to

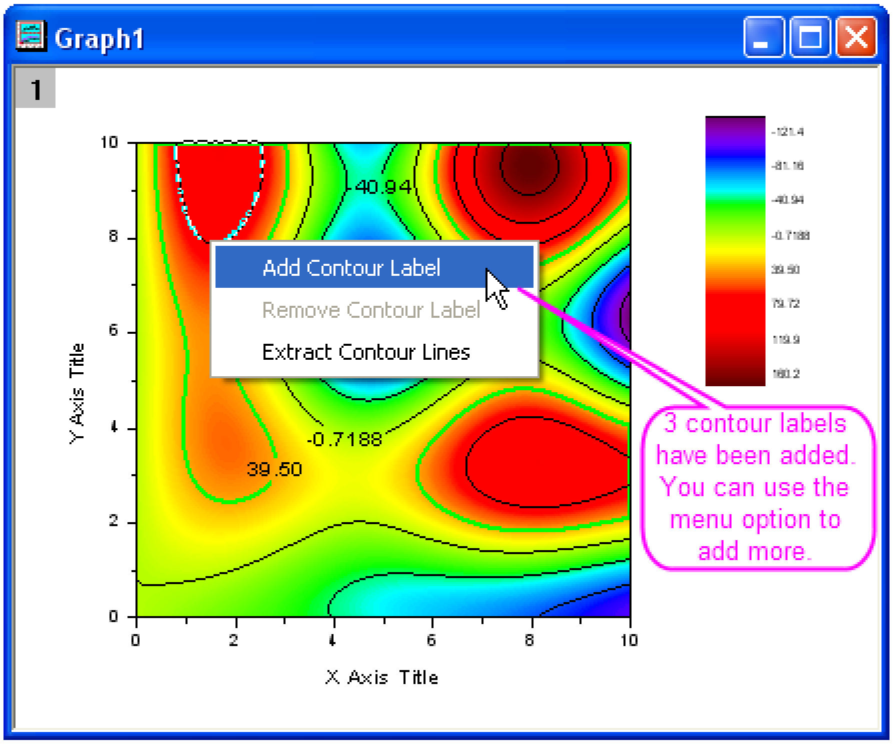
apply it to the color scale. Then the contour should look like



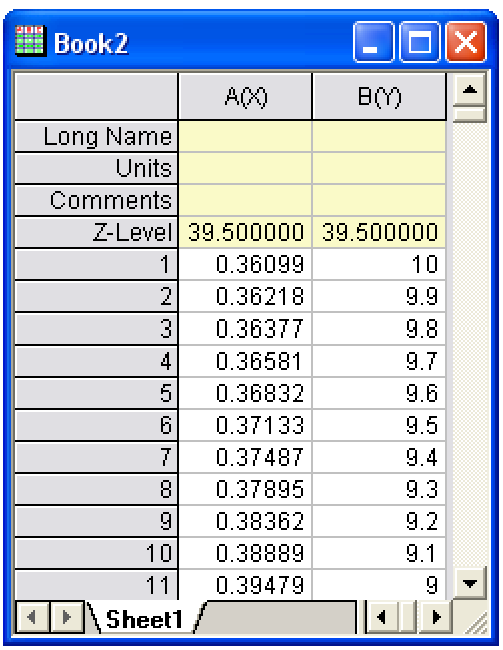
6. Click twice on a contour line (two single clicks, the first click selects the layer denoted by a red square at the plot center, the second click selects the contour line; if you mistakenly double-click, the Plot Details dialog will open) to select all lines at that level. Change the color to **Green** by using the Line/Border Color button  in the **Font** toolbar. Also change the width to **3** by using the **Line/Border Width** button  in the same toolbar.



Right-click on the selected contour line and select **Add Contour Label** to add a label.



7. Click three times on a contour line to select it, then right-click and select **Extract Contour Lines**. Origin will extract the data for that contour line to a worksheet. The following image shows part of the data in the worksheet.


Book2

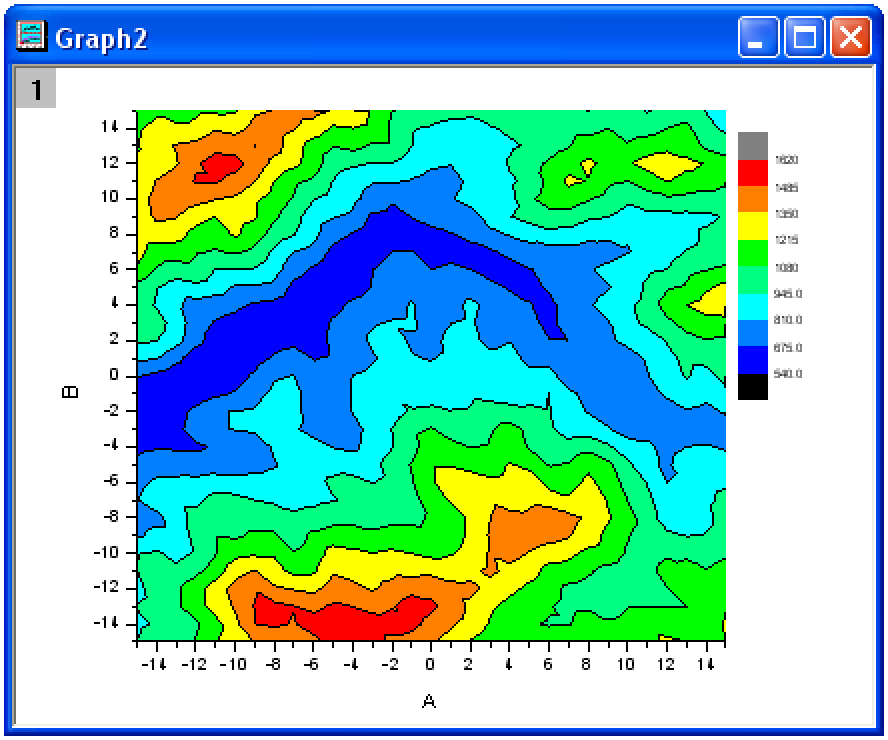
	A(X)	B(Y)
Long Name		
Units		
Comments		
Z-Level	39.500000	39.500000
1	0.36099	10
2	0.36218	9.9
3	0.36377	9.8
4	0.36581	9.7
5	0.36832	9.6
6	0.37133	9.5
7	0.37487	9.4
8	0.37895	9.3
9	0.38362	9.2
10	0.38889	9.1
11	0.39479	9

Sheet1

Create a Contour Plot from XYZ Data

Origin can create contour plots directly from XYZ data in a worksheet without the need for an intermediate matrix. Delaunay Triangulation is used to compute and draw the contour lines.

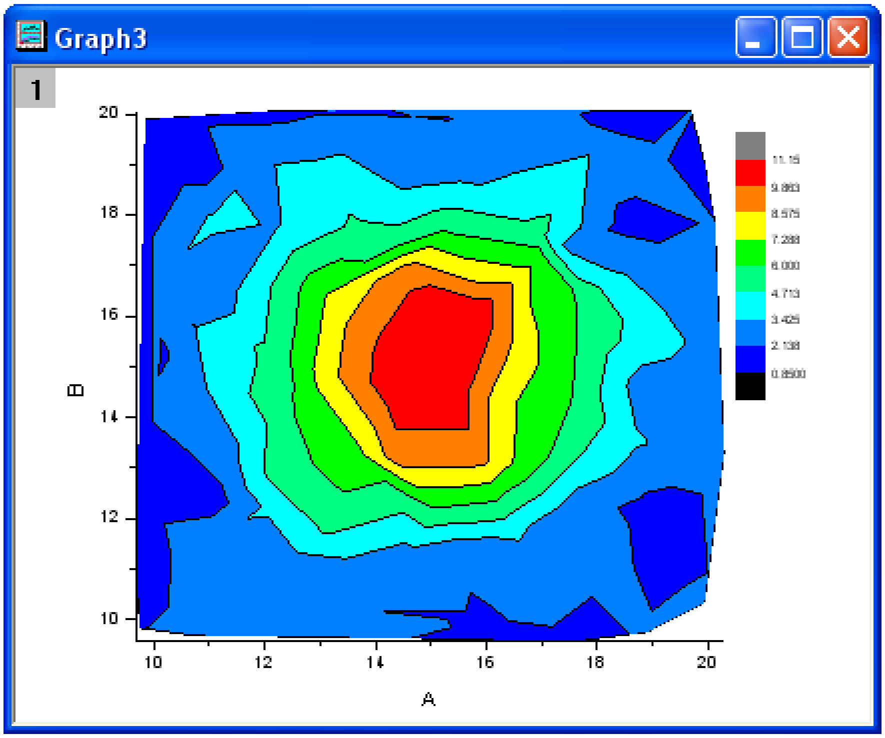
1. Create a new worksheet and import the **3D XYZ.dat** from the **\Samples\Matrix Conversion and Gridding** folder by using the **Import Single ASCII** button .
2. Highlight the third column and right-click on it to select **Set As: Z**. Then select **Plot: Contour: Color Fill** to create a graph, as the following image shows.



<u>Show the Triangulation Grid</u>

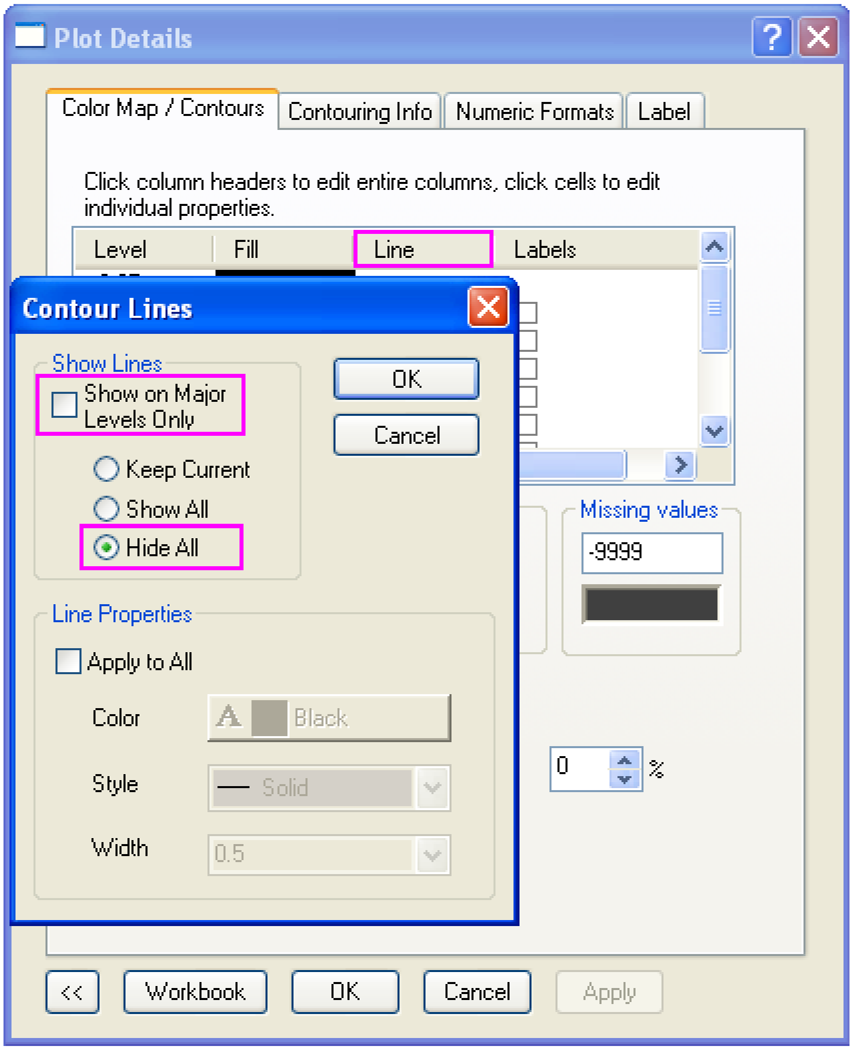
1. Show the triangulation grid that is used to create a contour plot from the data. The individual data points are located at the nodes or vertices of the grid. First, create a new worksheet and import the **XYZ Random Gaussian.dat** file from the **\Samples\Matrix Conversion and Gridding** folder by using the **Import Single ASCII** button .

2. Highlight the third column and right-click on it to select **Set As: Z**. Then select **Plot: Contour: Color Fill** to create a graph.



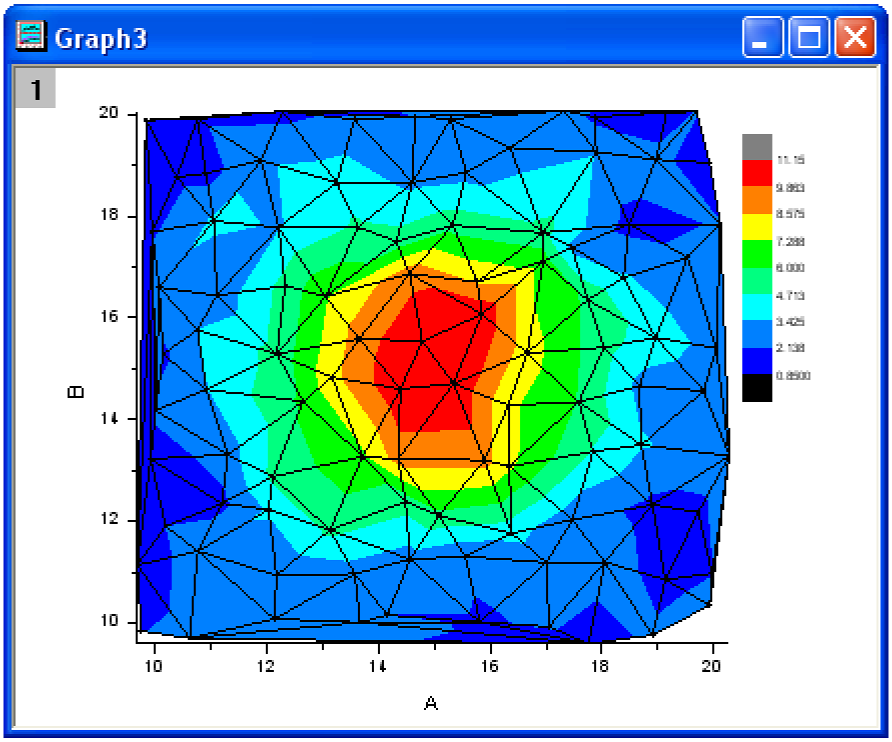
3. Double-click on the contour plot to open the **Plot Details** dialog. Go to the **Color Map/Contour** tab, click on the **Line** heading to open the **Contour Lines** dialog, and select **Hide All** to hide all

the contour lines. Click OK to close this dialog.



4. Then select the **Show Grid Lines** and **Show Data Points** check boxes in the **Color Map/Contours** tab. Click OK to close the Plot Details dialog.

The contour plot with its superimposed grid should look like this:



Apply Custom Boundary

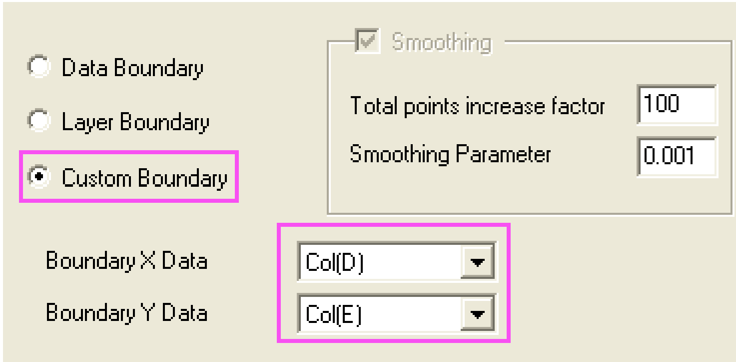
A custom boundary can be applied to contour plots created directly from XYZ data, where the user provides the data points for the boundary in additional columns on the same worksheet. This feature is useful when you have specific boundary data that defines the outline of an object, such as, say, the profile of an engine, and the contour shows engine temperature.

1. We will continue to use the worksheet and the contour plot created with **XYZ Random Gaussian.dat** in the previous example.

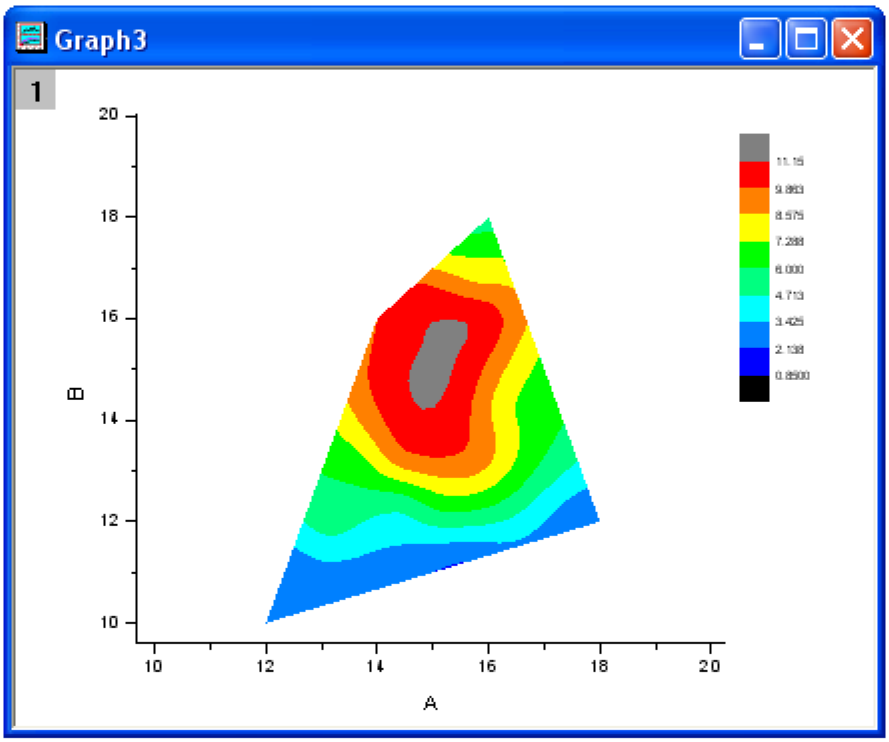
2. Go to the **XYZRandomGaus** workbook and click the **Add New Columns** button  twice to add two columns. Highlight the two columns and right-click to select **Set As: XY XY**. Enter four rows of data, like you see below:

12	10
18	12
16	18
14	16

3. Double-click on the contour plot to open the **Plot Details** dialog. Select the **Contour Info** tab in the right panel, and set the dialog like the following image shows, to customize the boundary of the contour plot. Remember to uncheck the **Show Grid Lines** and **Show Data Points** check boxes in the **Color Map/Contours** tab. Click OK to close the Plot Details dialog.



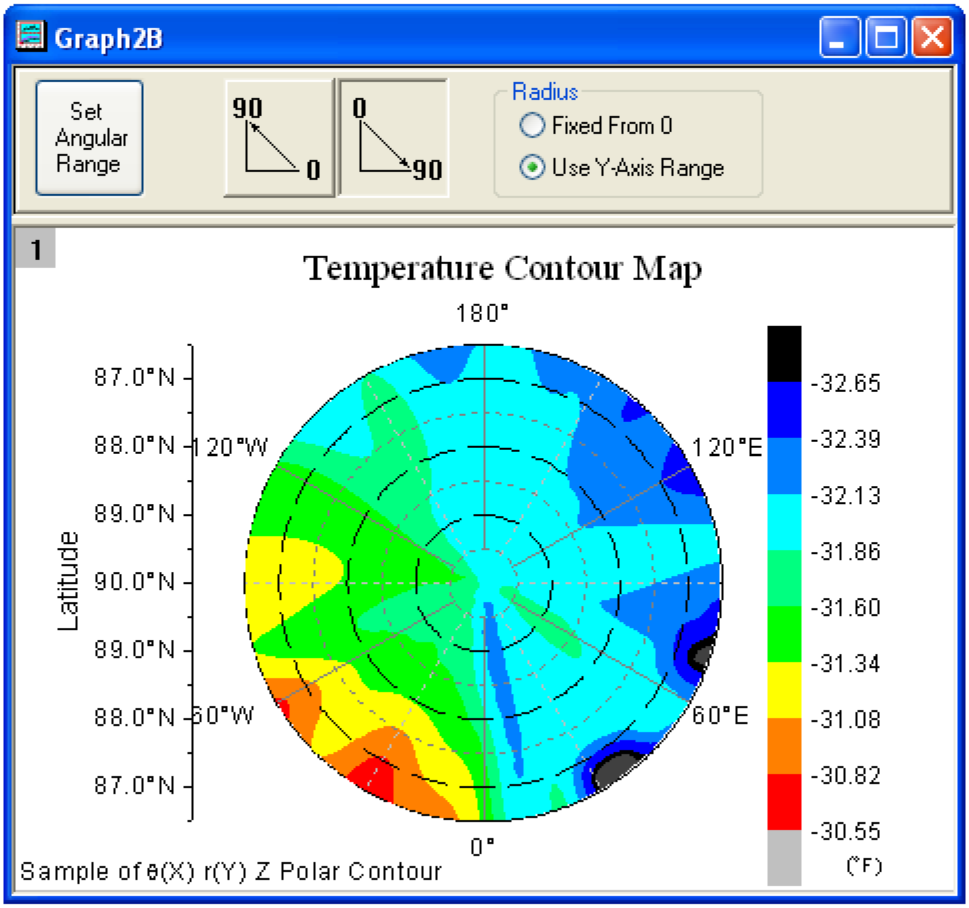
The graph should look like:



5.5.4 Polar Contour

Summary

This tutorial will show you how to create a Polar Contour graph.



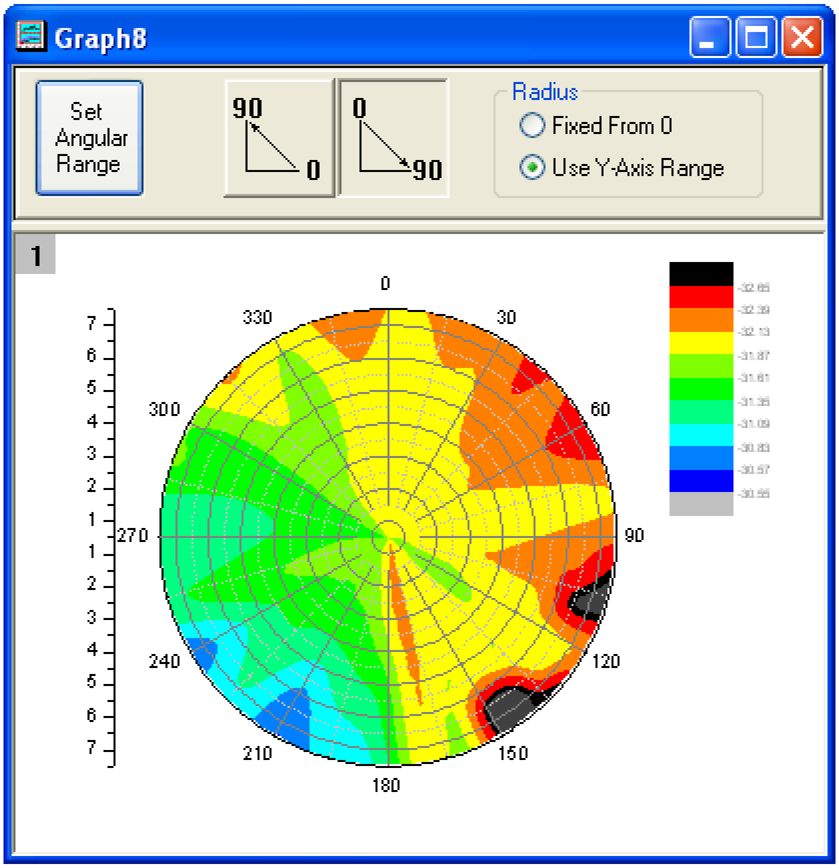
What will you learn

- Create a Polar Contour graph
- Customize the graph by using the Plot Details dialog

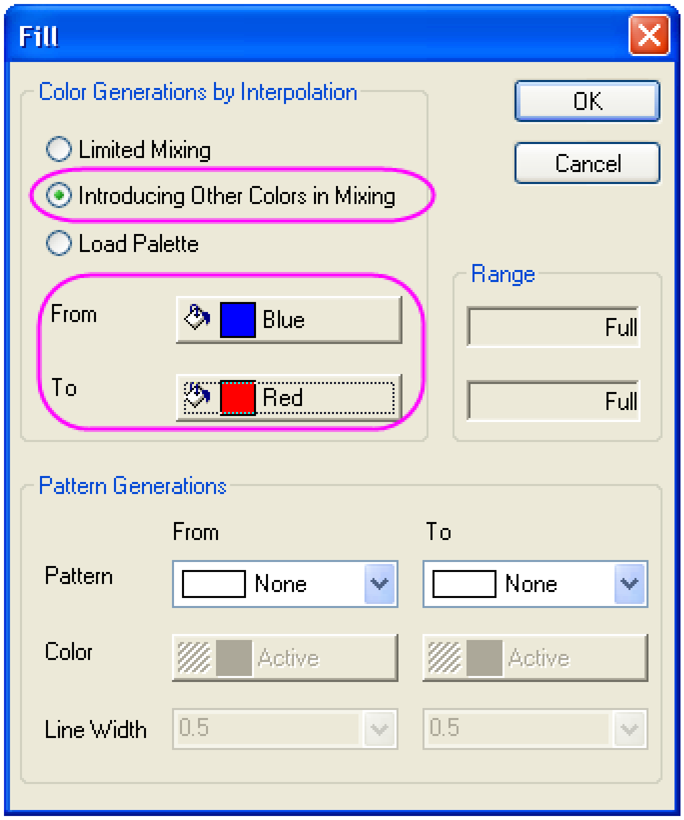
Steps

This tutorial is associated with the 2D and Contour Graphs project: *\Samples\2D and Contour Graphs.opj*.

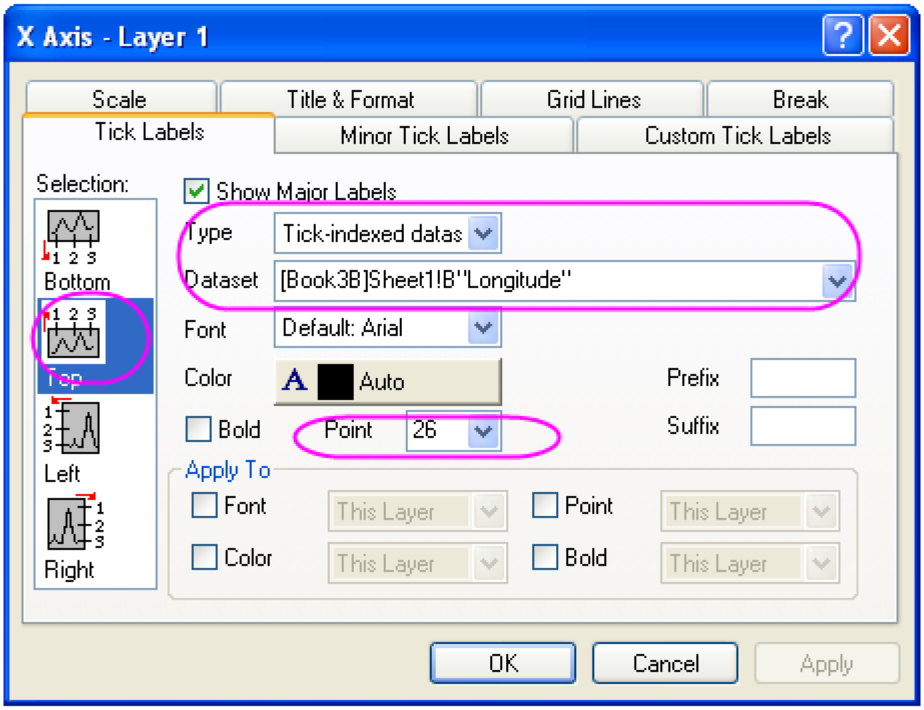
1. Open the *2D and Contour Graphs: Contour: Polar Contour* folder in the **Project Explorer**. Activate **Book2B** with the temperature and location data and select column **C**. Select **Plot: Contour: Polar Contour Theta(X) r(Y)** from the menu to create a polar contour graph. The graph should look like:



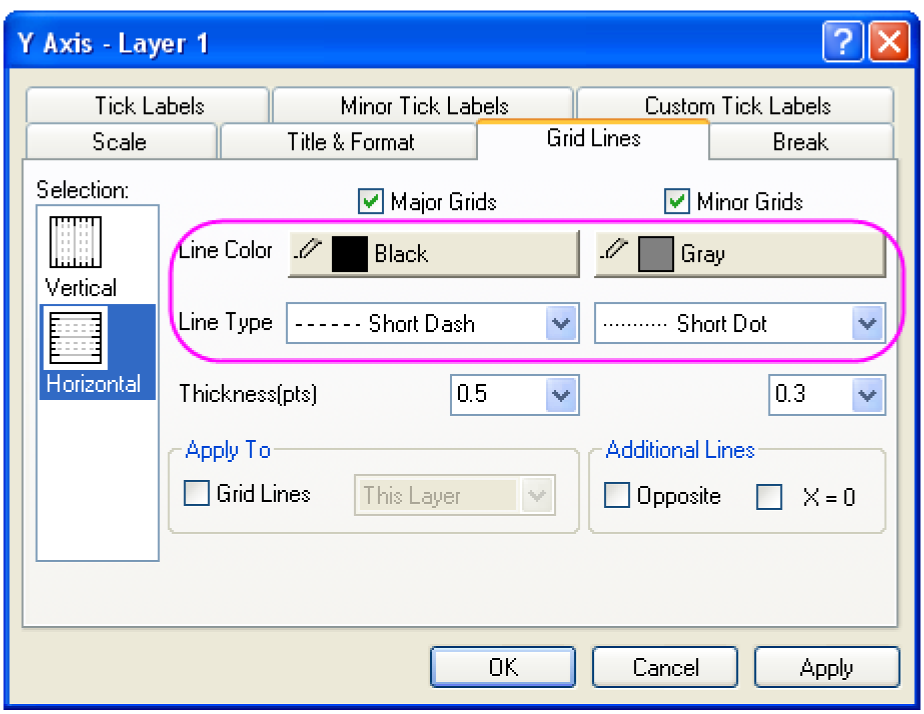
2. Then we wil customize the contour graph by using the **Plot Details** dialog. Double-click on the contour plot to bring up Plot Details dialog, select the **Color Map/Contours** tab and click the **Fill** heading, and then set the dialog as the follwing graph shows.



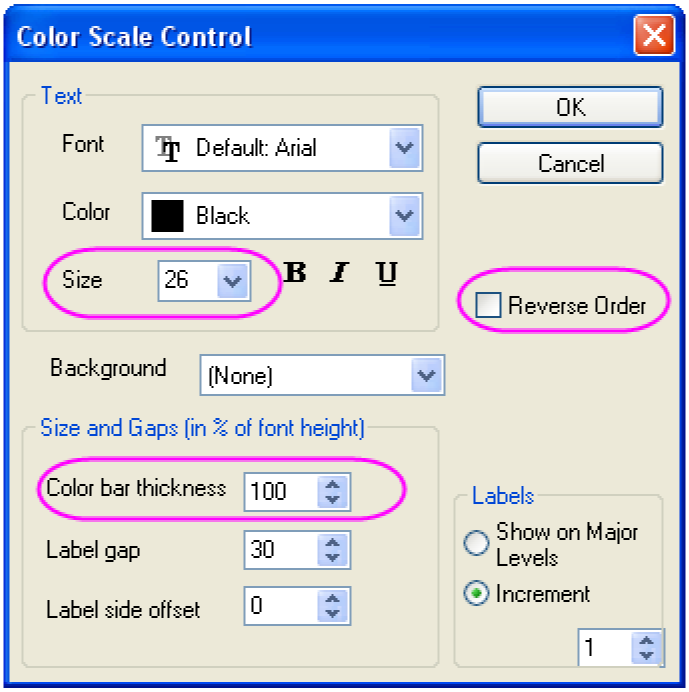
3. Click the **Contouring Info** Tab, select the **Layer Boundary** radio button.
4. Click OK button to close the **Plot Details** dialog.
5. Then we will customize the axes of the graph. Double-click on any tick label to open the **Axis** dialog.Then do the following things
    - Select the **Scale** tab, choose **Horizontal** in the **Selection** list, set **Increment** to 60.
    - Select **Vertical** in the **Selection** list, set **From** to **0**, set **To** to **7** and set **Increment** to **2**.
    - Select **Tick Labels** tab and select **Top** in the **Selection** list. Then select **Tick-Indexed Dataset** with the **Type** drop-down list. Choose **[Book3B]Sheet1!B** for **Dataset**. Change **Point** to **26**.



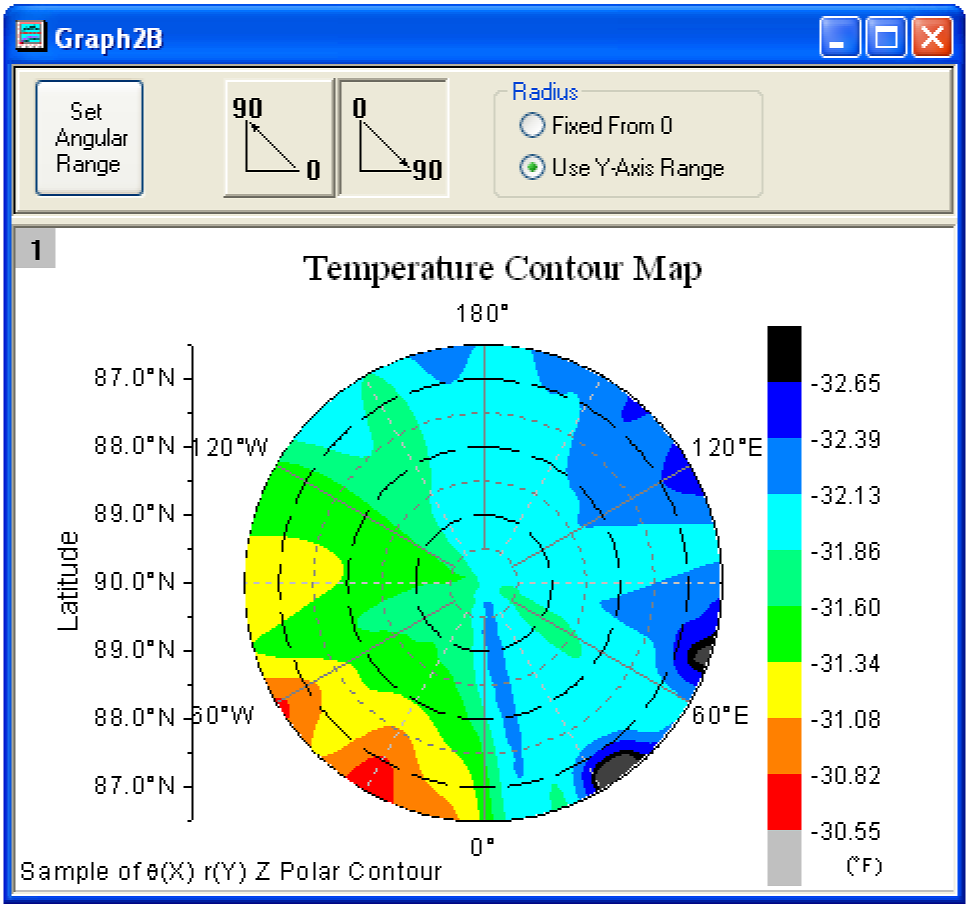
- Select **Left** in the **Selection** list. Then select **Tick-Indexed Dataset** with the **Type** drop-down list. Choose **[Book3B]Sheet1!A** for **Dataset**. Change **Point** to **26**.
- Select **Right** in the **Selection** list. Then select **Tick-Indexed Dataset** with the **Type** drop-down list. Choose **[Book3B]Sheet1!A** for **Dataset**. Change **Point** to **26**.
- Go to the **Grid Lines** tab and make sure **Horizontal** has been selected in the **Selection** list. Then set the dialog as the following image shows. Click **OK** to close **Axis** dialog.



6. Double click on the Color Scale to open the **Color Scale Control** dialog and set the dialog as the following image shows.



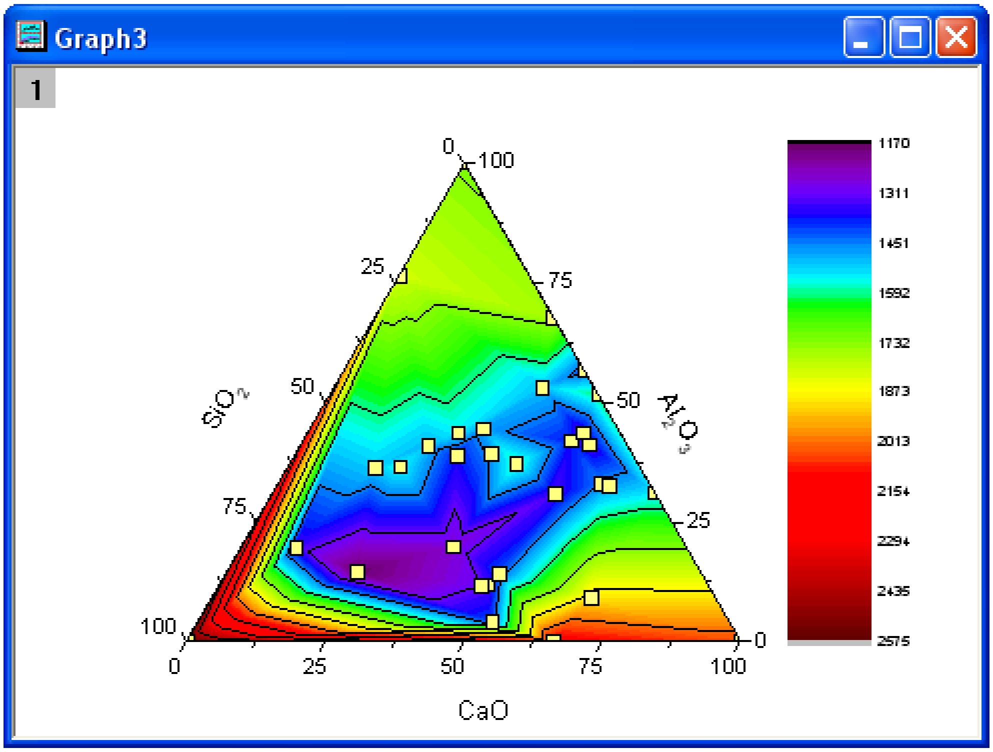
7. The graph should look like



5.5.5 Ternary Contour

Summary

This tutorial will show you how to create a Ternary Contour graph and add scatters overlay.



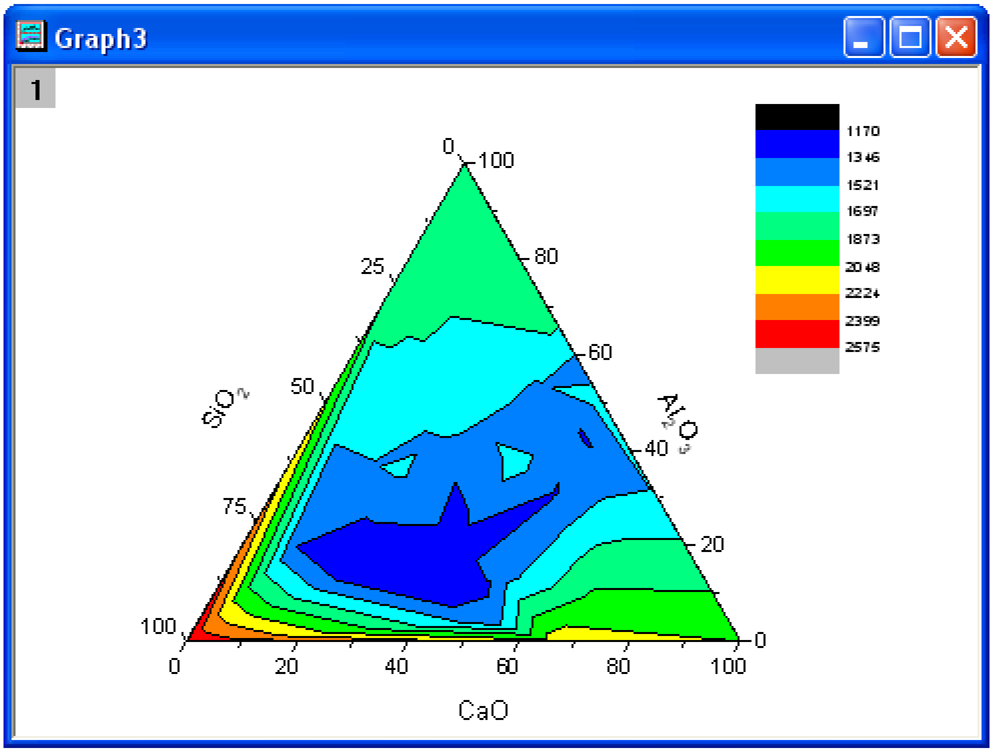
What will you learn

- Create the Ternary Contour graph
- Add scatters overlay with the Plot Setup dialog.
- Customize the graph by using the Plot Details dialog

Steps

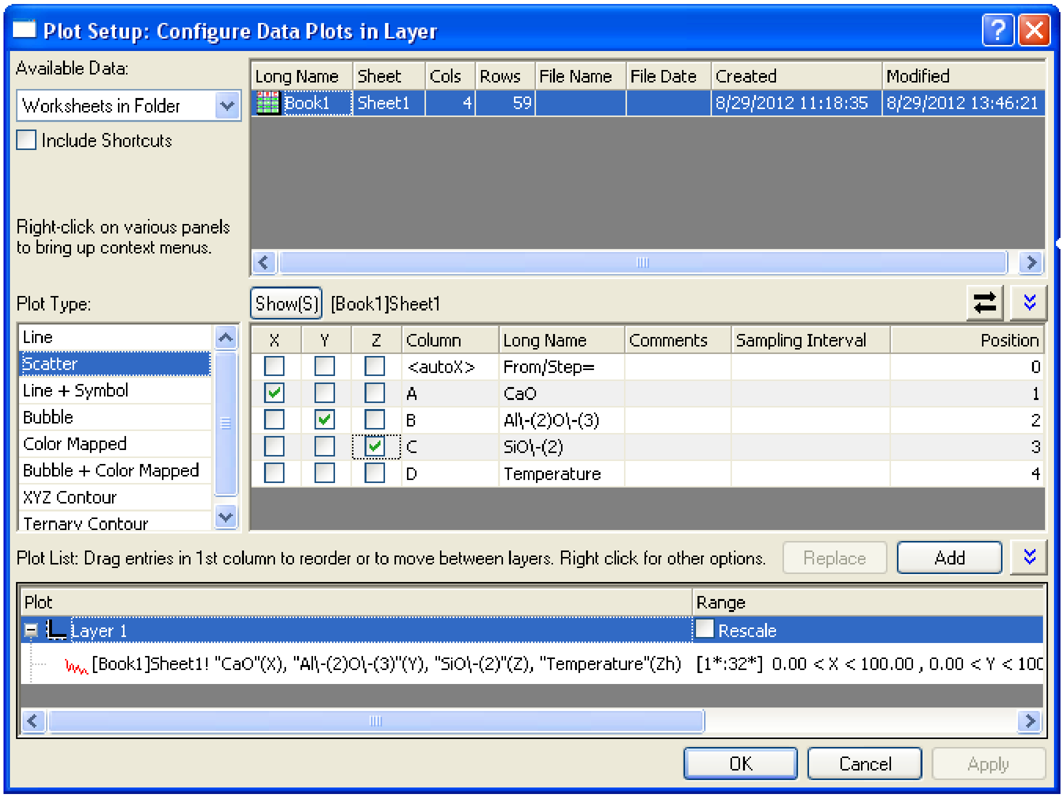
This tutorial is associated with the 2D and Contour Graphs project: *\Samples\2D and Contour Graphs.opj*.

1. Open the *2D and Contour Graphs: Contour: Ternary Contour* folder in the **Project Explorer**. Activate **Book1**,highlight entire worksheet and select **Plot: Contour: Ternary Contour** to create a Ternary Contour graph. The graph should look like:

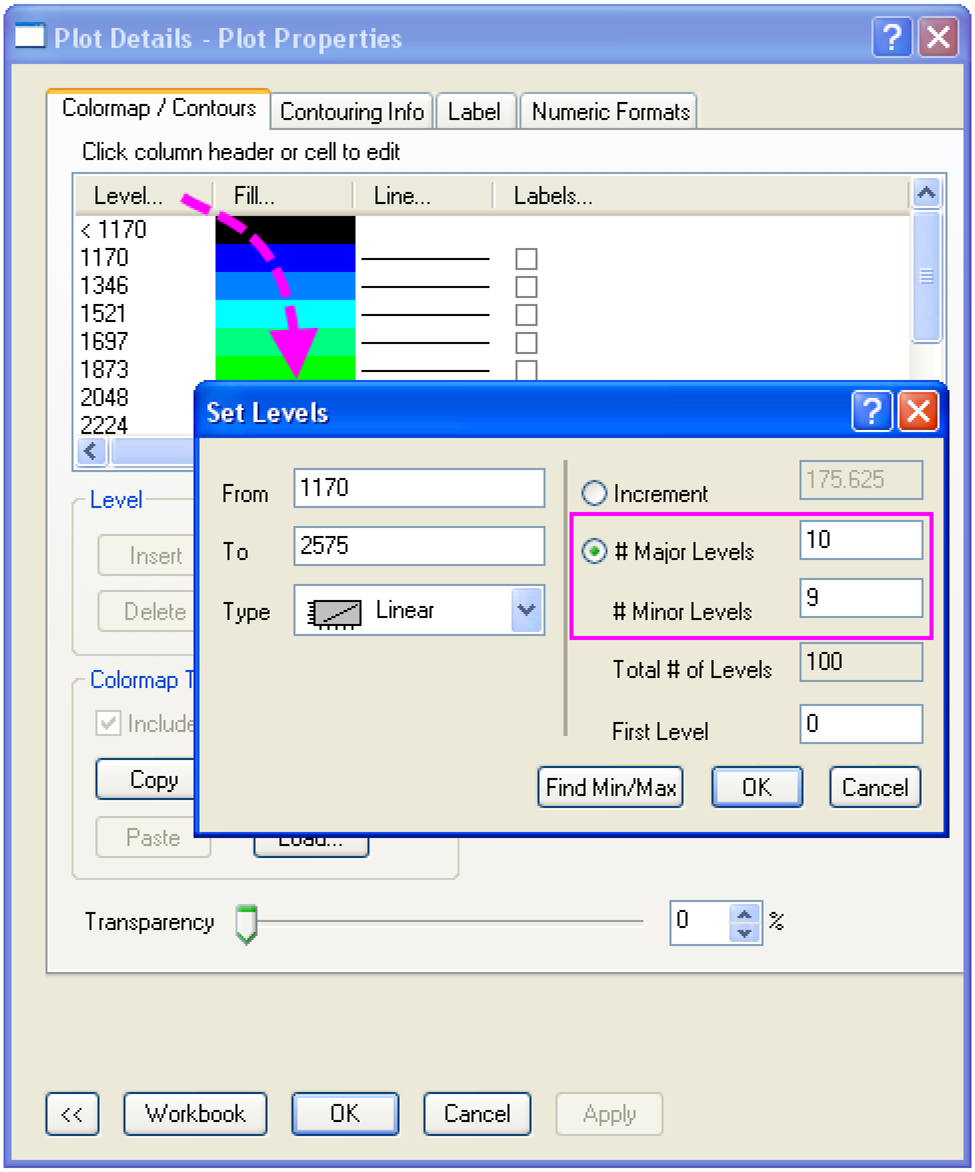


2. Now we will add overlay scatters to the plot. Right click on the layer icon and select **Plot Setup** in the context menu to open the **Plot Setup** dialog.

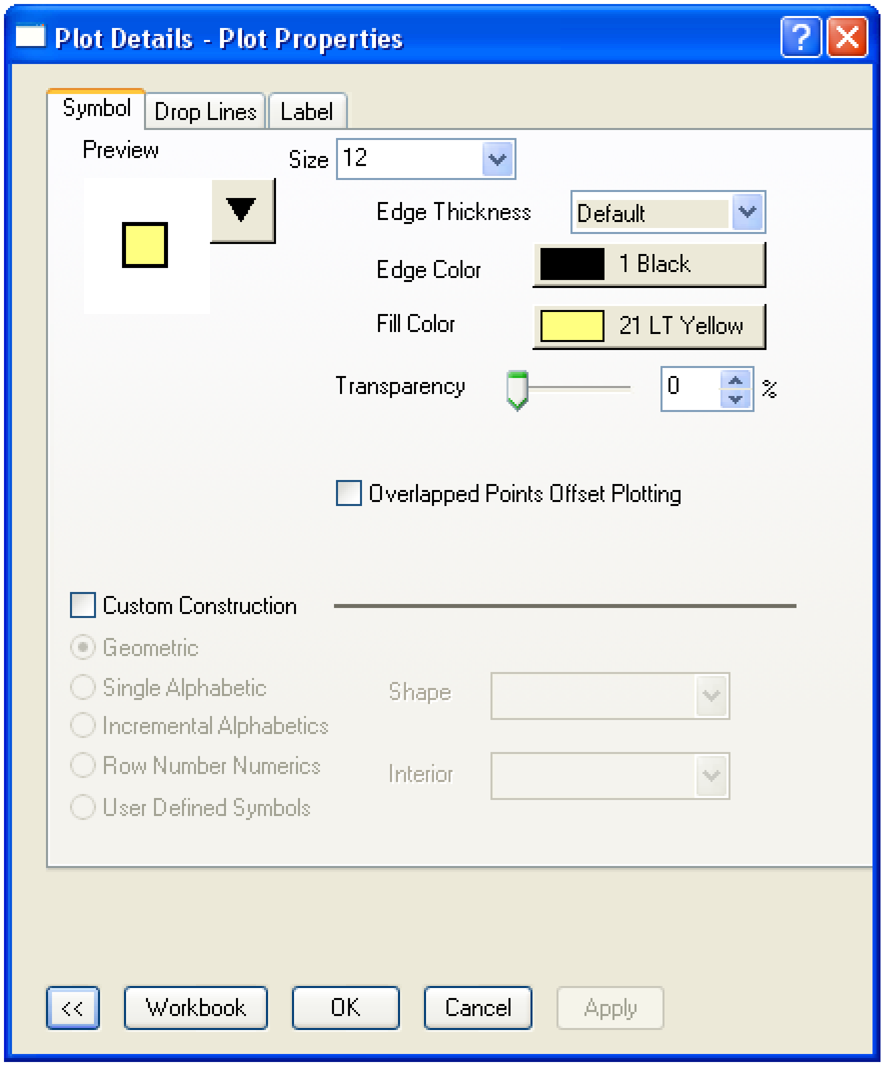
3. Set **Worksheets in Folder** as **Available Data**, select **Book1**, choose **Scatter** as **Plot Type** and column A, B, C as X, Y, Z respectively. Click **Add** to add the scatter plot to the ternary contour plot. The settings should be as following:



4. Now we will customize the contour graph and the scatter. Double-click on the contour plot to bring up the **Plot Details** dialog.
   - select the **Color Map/Contours** tab and click the **Level** heading, then set the dialog as the follwing graph shows.

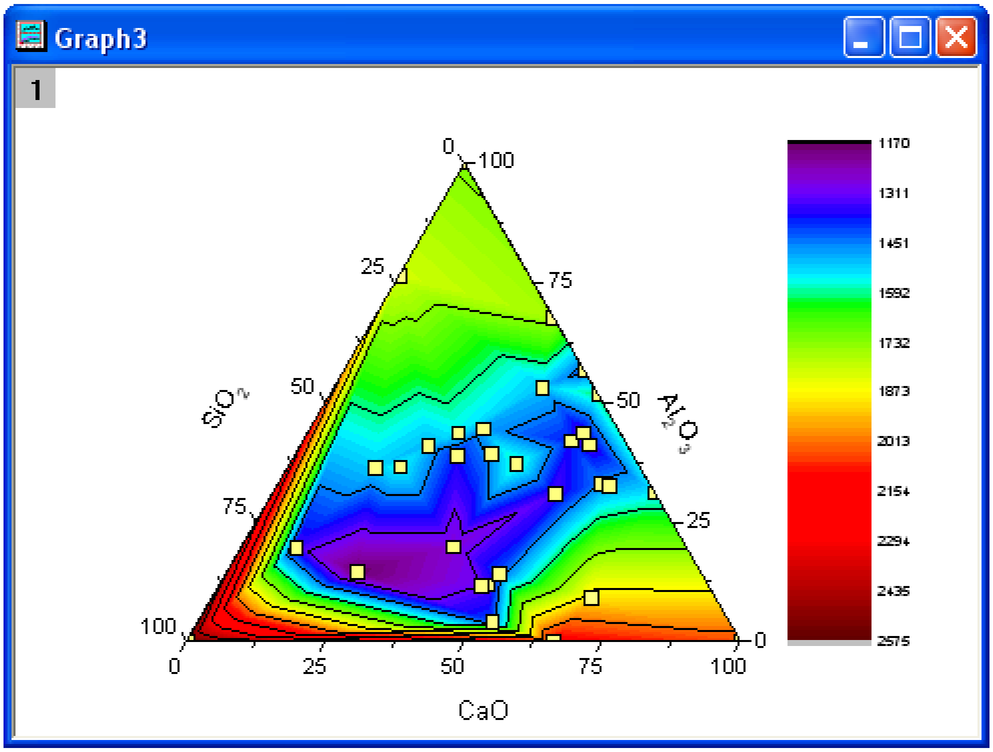


- Click OK to close the **Set Levels** dialog then click the **Fill** heading to open the **Fill** dialog. Select **Load Palette** and then select **Rainbow** from the **Palette** list and click OK.
- In the left panel of **Plot Details** dialog, select the scatter plot and customize the symbol as following:



- Click OK to close the **Plot Details** dialog.
- Drag and drop the color scale to resize and move it to a desired place

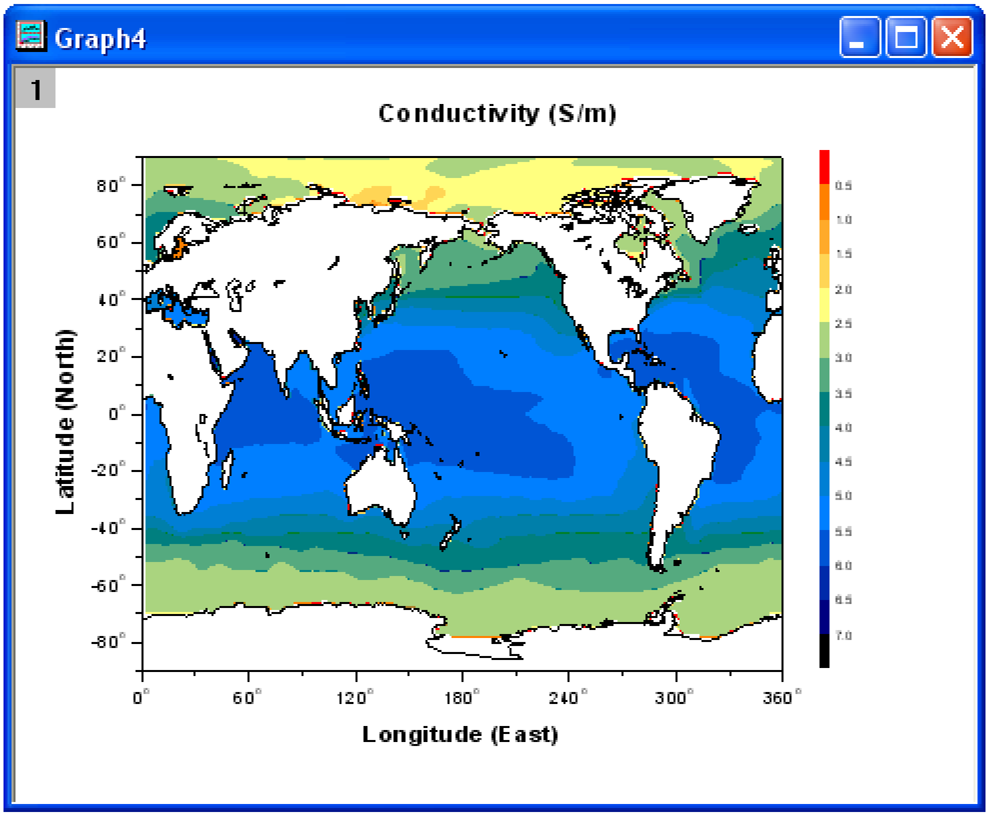
The graph should look like



5.5.6 Combining Line and Contour Plots

Summary

This tutorial will show you how to create a Word Map by combining Line Plot and Contour graph.



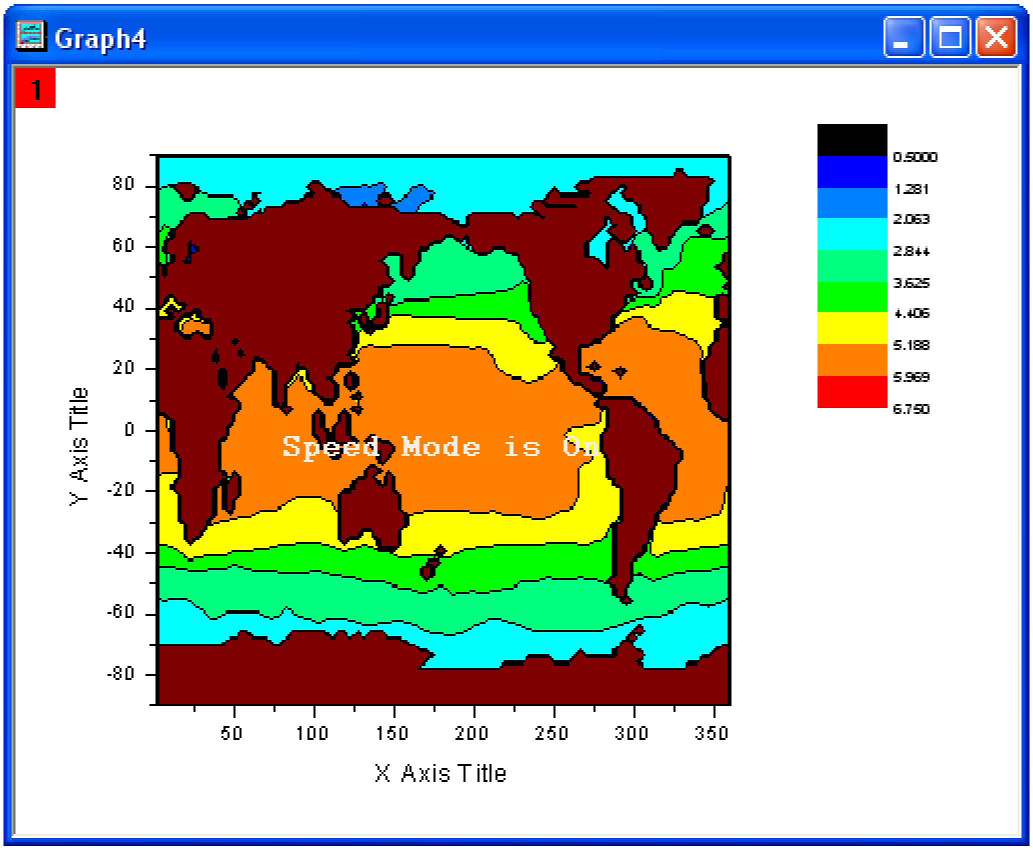
What will you learn

- Create a contour graph
- Combine a Line Plot and a Contour graph
- Customize the contour graph

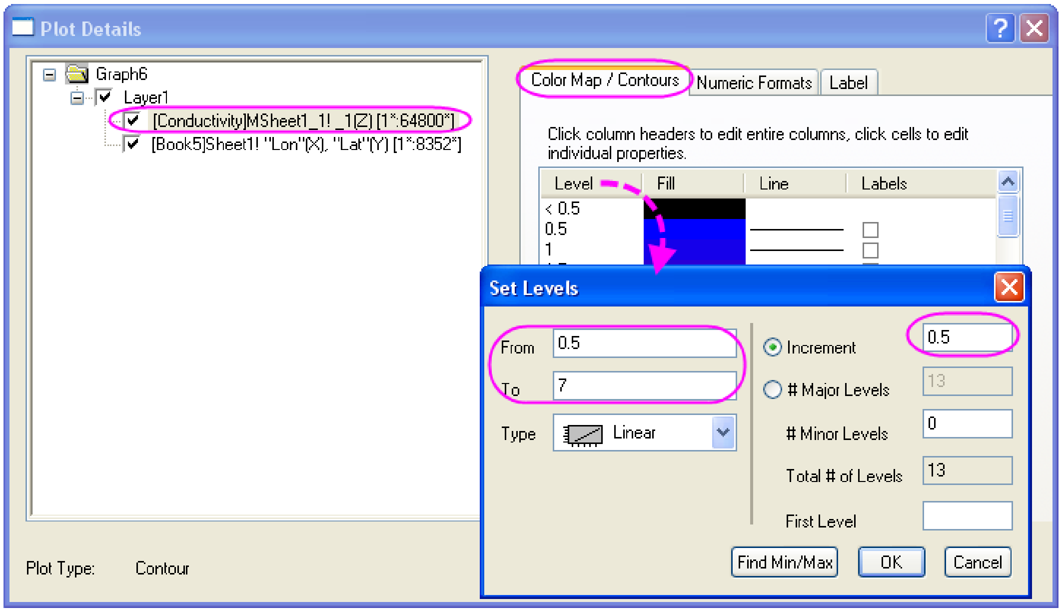
Steps

This tutorial is associated with the 2D and Contour Graphs project: *\Samples\2D and Contour Graphs.opj*.

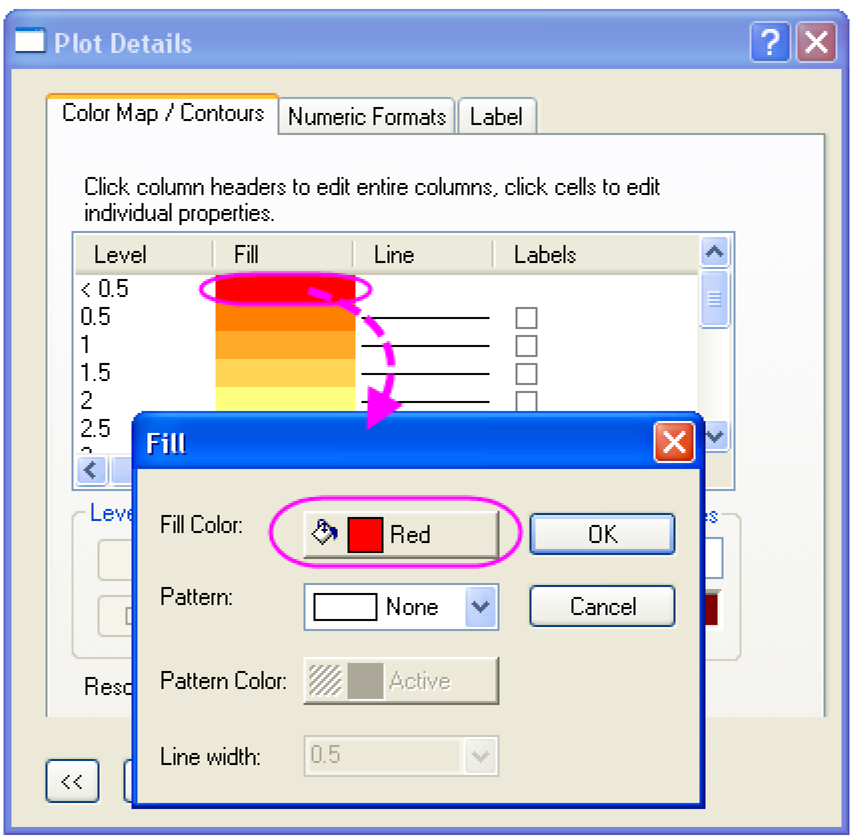
1. Open the *2D and Contour Graphs: Contour: Map Combining Line and Contour Plots* folder in the **Project Explorer**. Activate **Conductivity** matrix and highlight entire matrix. Select **Plot: Contour: Contour - Color Fill** to create a contour graph.



2. At this step, we will add a Line plot to the contour graph. Select all columns of **Book5** and then active the new created contour graph. Then select **Graph: Add Plot to Layer: Line** to create a new line plot in the same layer. Delete the legend for the added line plot.
3. In the following steps, we will customize the graph. Select **Format: Layer Properties** to open the Plot Details dialog. Select the **Size/Speed** tab, disable two checkboxes that in the **Speed Mode, Skip Points if needed** group.
4. Expand the **Layer1** branch and select contour plot in the left panel of the Plot Details dialog. Then do the following things:
    - Go to the **Color Map/ Contours** tab, click the **Level** heading to open the **Set Levels** dialog. Set the dialog as the following image shows:

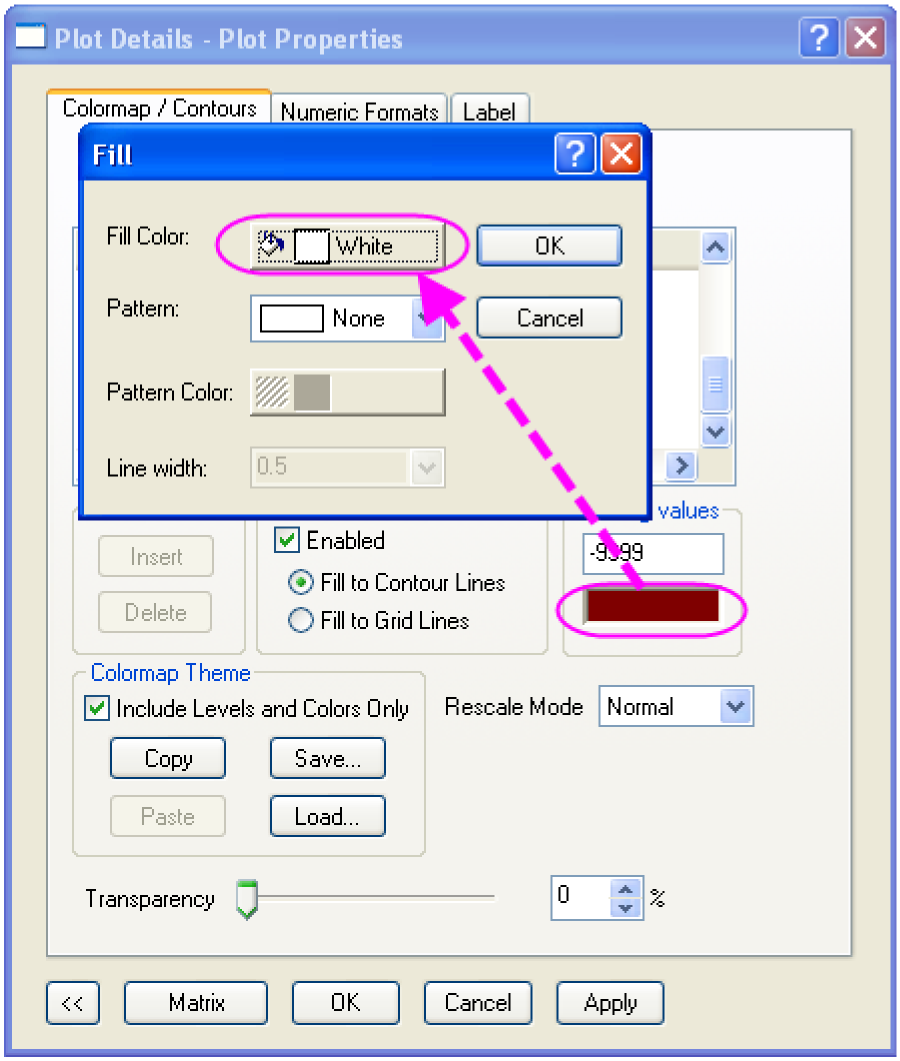


- o Click the **Fill** head to open the **Fill** dialog, select **Introduce Other Colors in Mixing** radio box. Then set **From** to **Orange** and **To** to **Navy**.
- o Click the cell the in the **Fill** column and **<0.5** row to set fill color to **Red**.

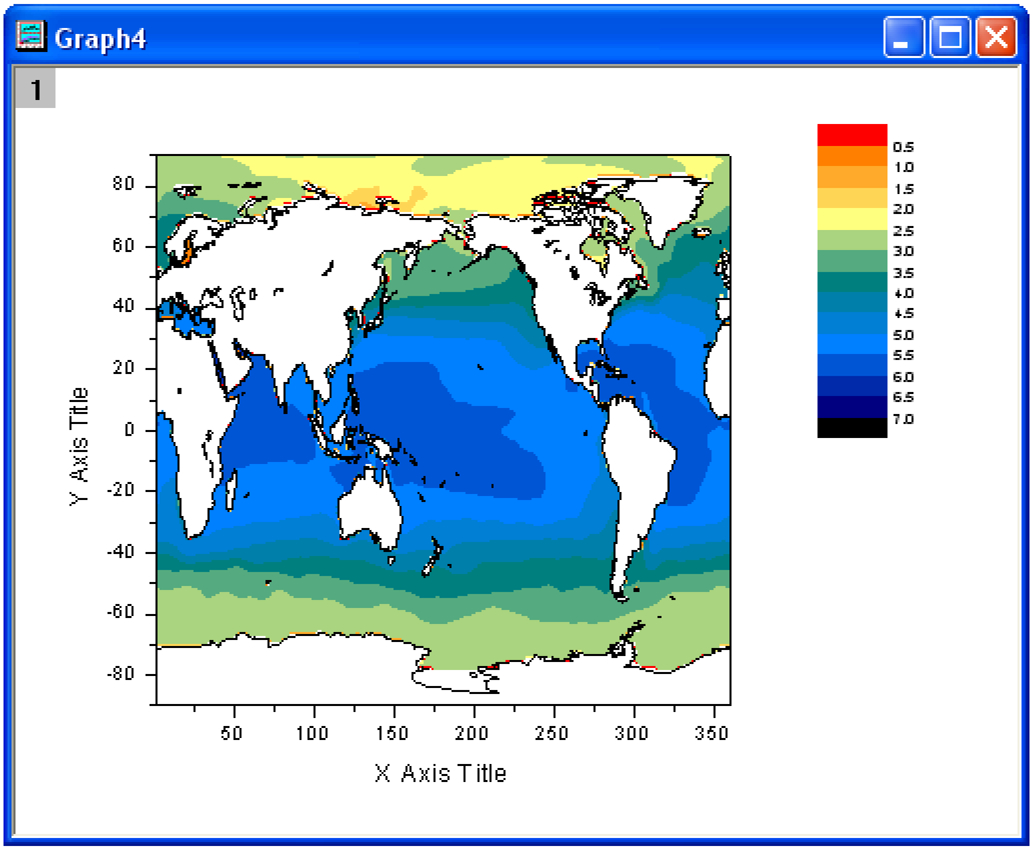


- o Click the cell the in the **Fill** column and **>7** row to set fill color to **Black**.

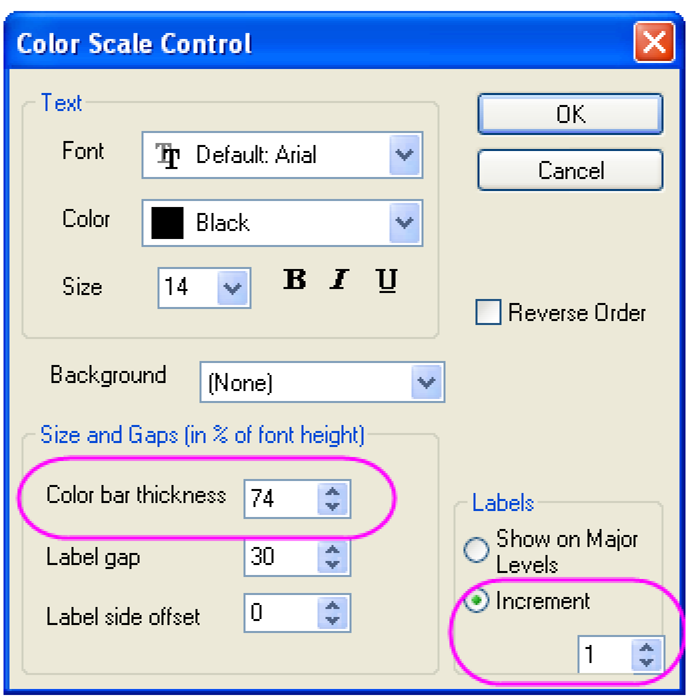
- Click the **Lines** heading to open the Contour Lines dialog, uncheck the **Show on Major Levels only** check box and then select the **Hide All** option.
- Click the **Color** box that in the **Missing Value** group, set fill color to **White**. Click OK button.



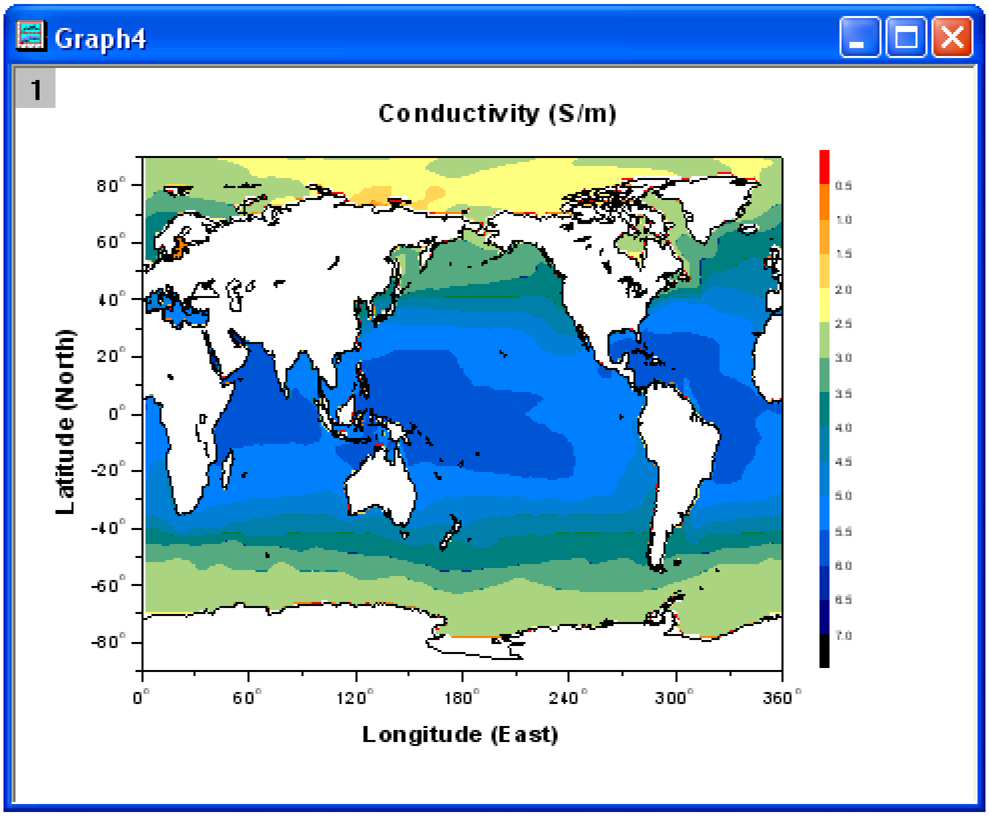
5. Go to **Numeric Formats** tab of **Plot Details** dialog, select **Decimal Places** radio box and keep the default value **1**.
6. Click **Ok** button to apply the settings to the graph. The graph should look like:



Double click on the **Color Scale** to open the **Color Scale Control** dialog and set the dialog as the following graph shows:



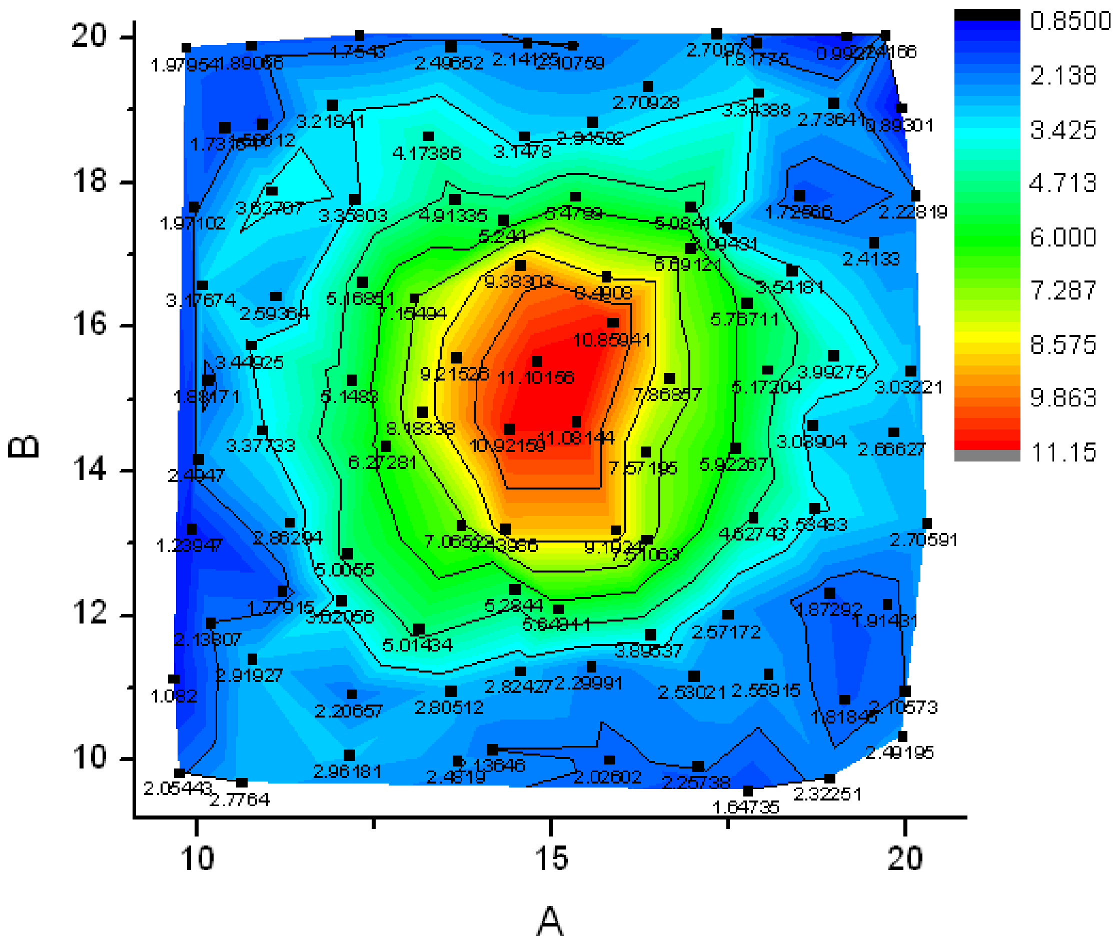
7. Resize and reposition of the color scale. Then modify the X, Y axis tick labels, titles and add the graph title as the example graph shows. The graph should look like



5.5.7 Contour Graph with XY Data Points and Z Labels

Summary

This tutorial will show you how to create a contour from XYZ data and add Z value as label for each XY data point.



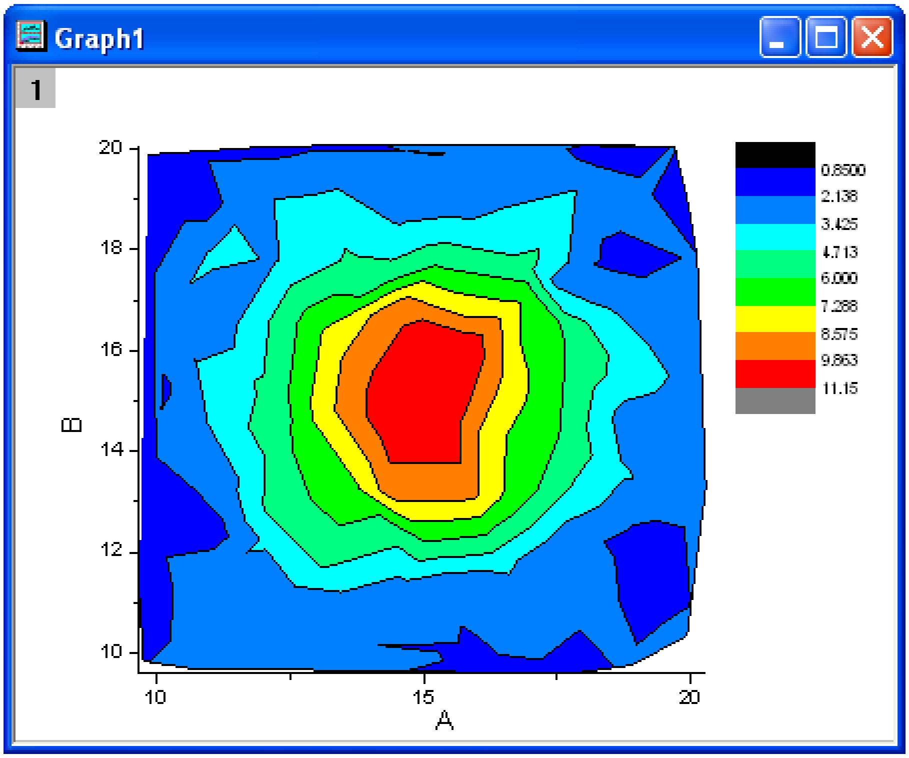
What will you learn

This tutorial will show you how to

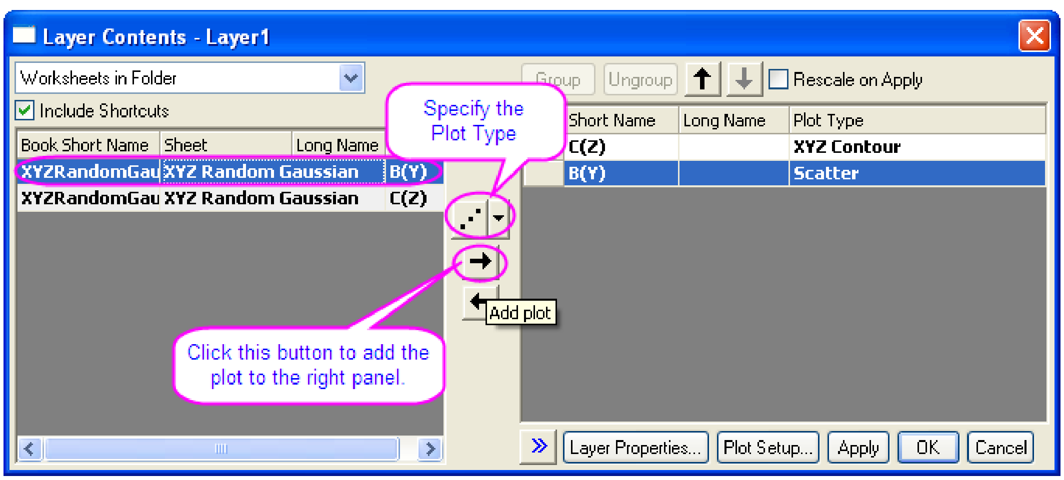
- Create a Contour Plot from XYZ data.
- Customize levels, lines, and color mapping.
- Plot a scatter plot on a contour graph.
- Show the labels for scatter points.

Steps

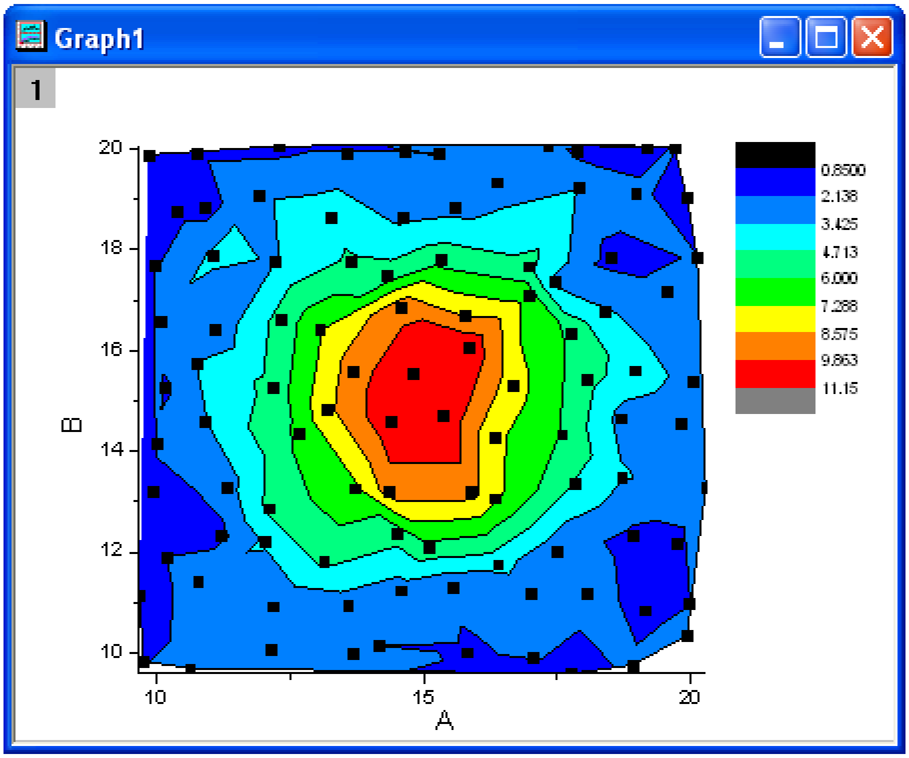
1. Start a new workbook and import the file *Sample/Matrix_Conversion_and_Gridding/XYZ_Random_Gaussian.dat*.
2. Highlight col(C) and set it as *Z*. Then highlight all columns and select **Plot: Contour: Color Fill** from the main menu to plot a contour graph.



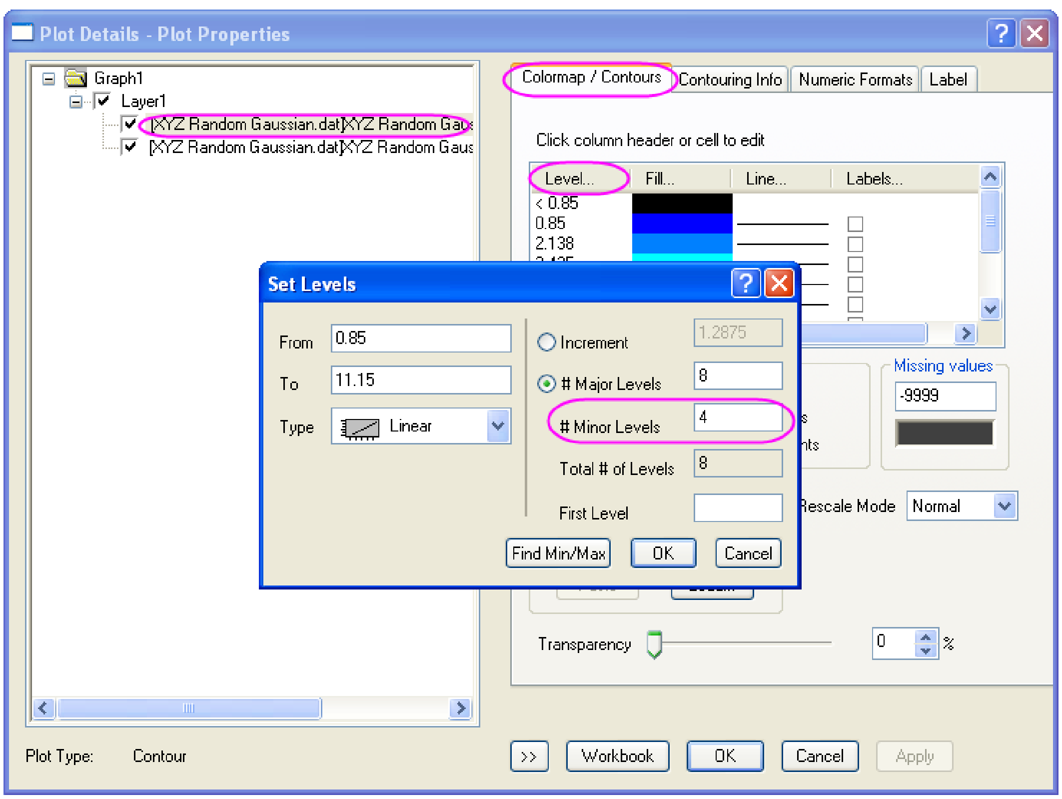
3. Double-click the layer icon to open the **Layer Contents** dialog. In this dialog, select col(B) in the left panel, set the **Plot Type** as *Scatter*, then add it to the right panel.



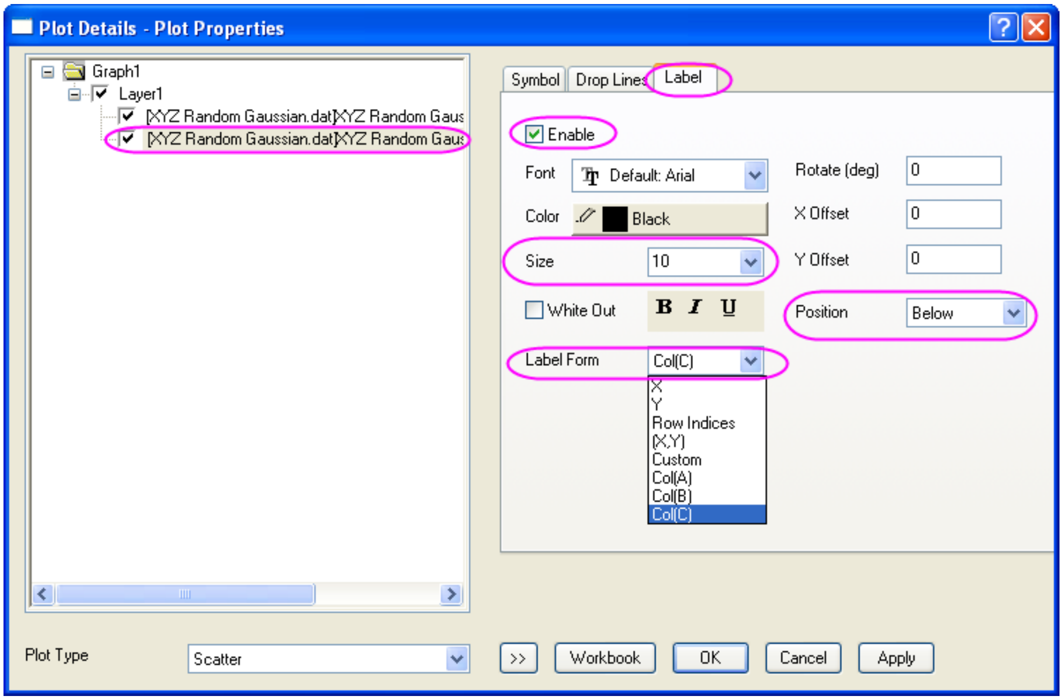
4. Click **OK** button to close the dialog. The scatter plot will be added to the contour graph.



5. Double-click the graph to open the **Plot Details** dialog. In this dialog, go to the **Colormap/Contours** tab, the contour plot in the left panel is selected. Click the **Level...** header to open the **Set Levels** dialog to set the **Minor Levels** as *4*.



6. Click **OK** button to go back the **Plot Details** dialog. Go to the **Label** tab while the scatter plot is selected in the left panel, check the **Enable** check box to activate this tab. Set the **Size** as *10*, **Position** as *Below* and **Label Form** as *Col(C)*.

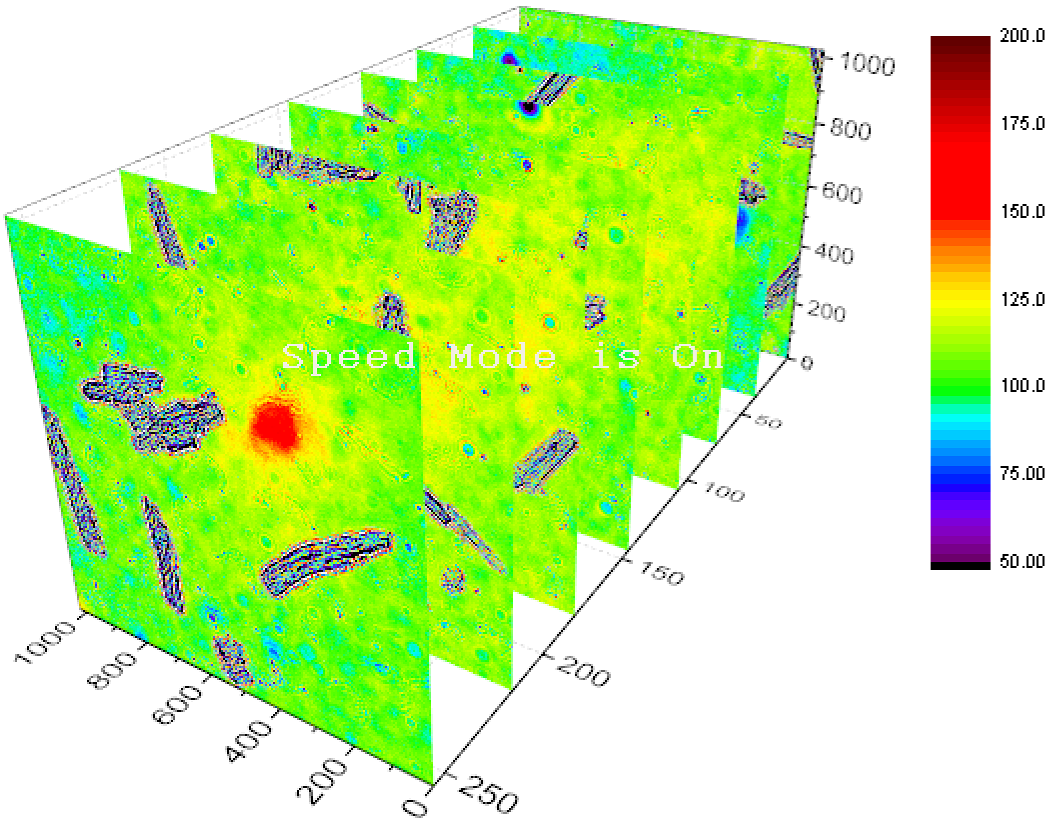


7. Click **OK** button to close this dialog. The Z values will be added as labels to the graph. Click the **Rescale** button  to rescale the graph.

5.5.8 Flattened Colormap Surfaces with Increasing Z Offset

Summary

This tutorial will show you how to plot a stacked flattened colormap surface with Z offset.

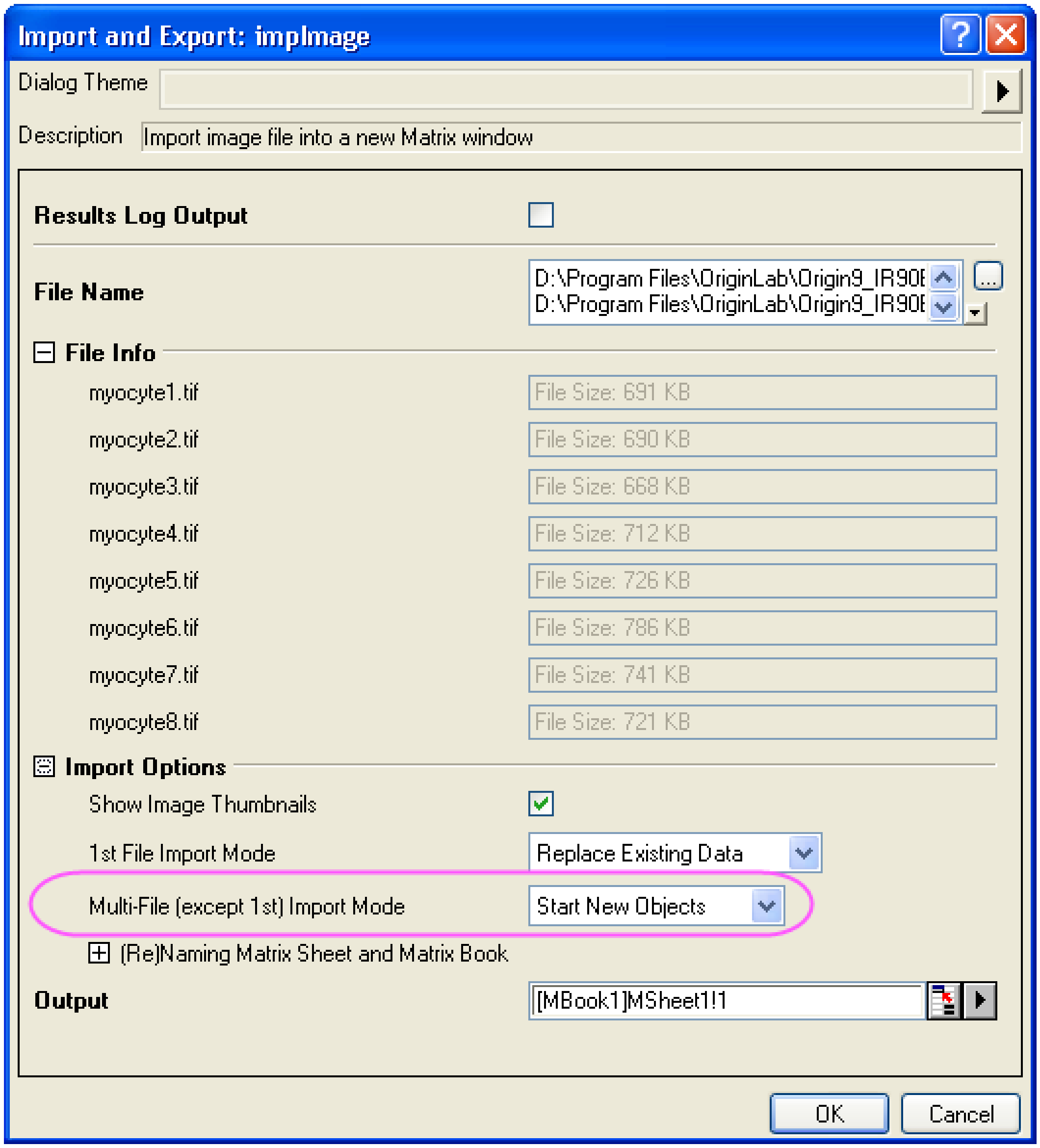


What will you learn

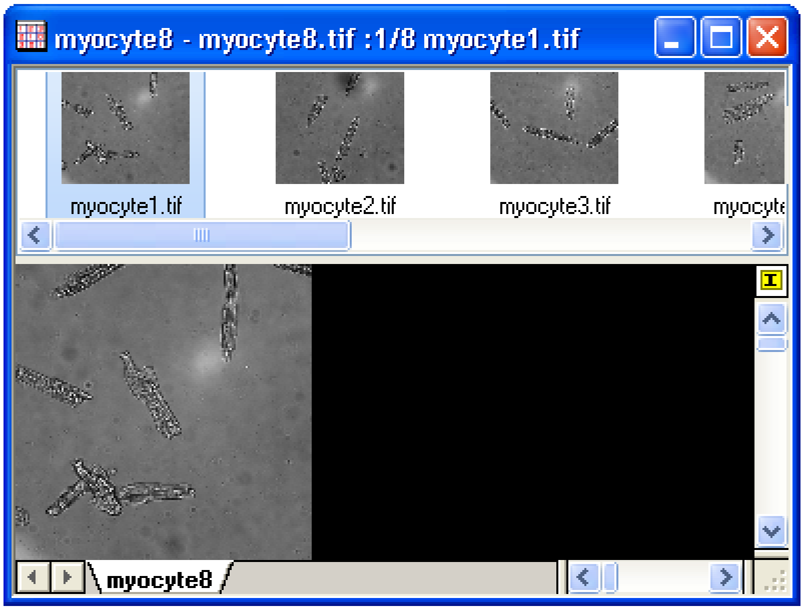
- How to import mulitple matrix into Origin as the matrix object.
- How to plot Multiple Colormap Surfaces in a graph layer.

Steps

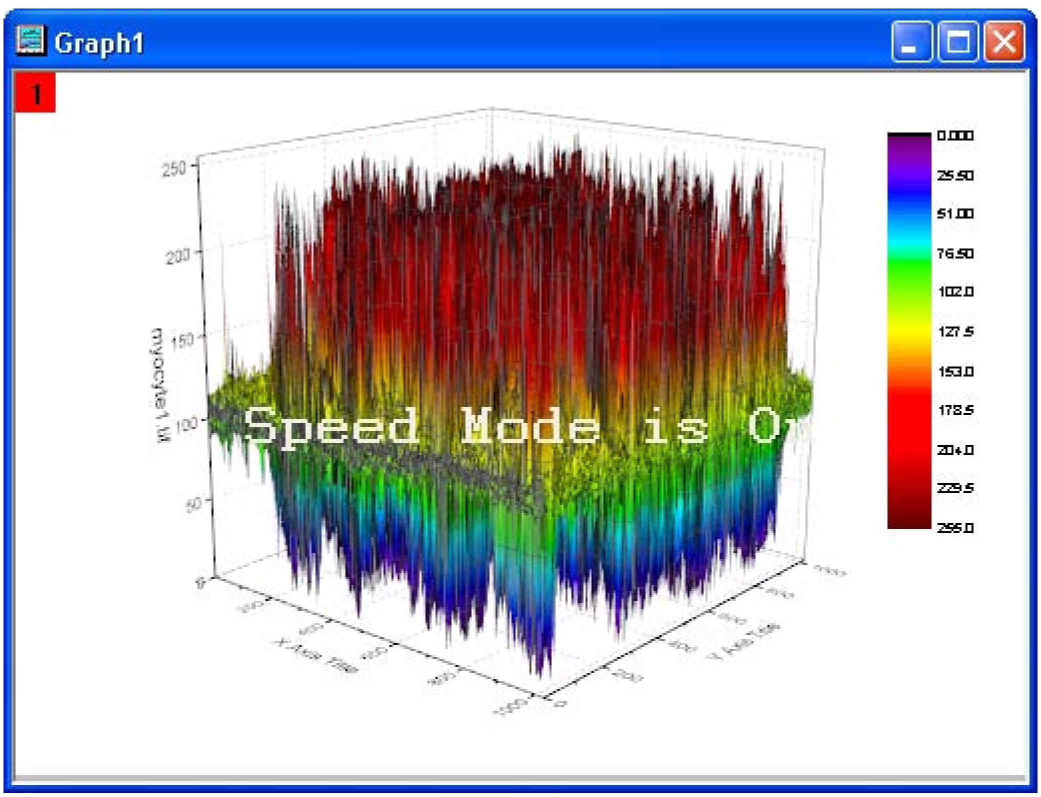
1. Create a new a matrix and select **File: Import : Image** to import **myocyte1~myocyte8** under the *\Sample\Image Processing and Analysis* folder. In the **ImpImage** dialog, keep **Mulit-File (except 1st) Import Mode** as **Start New Objects**.



2. Click the OK button to import the files to matrix as matrix objects.

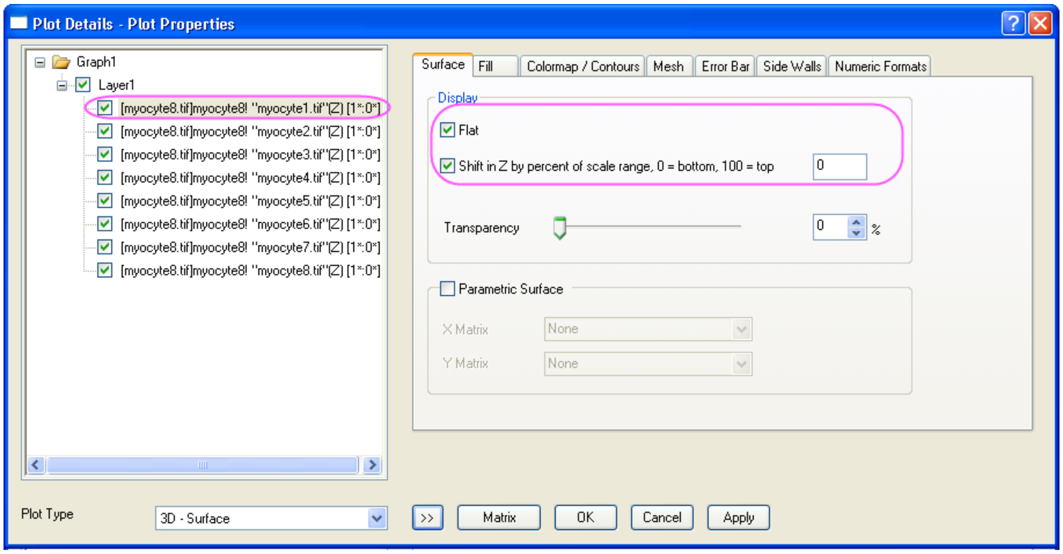


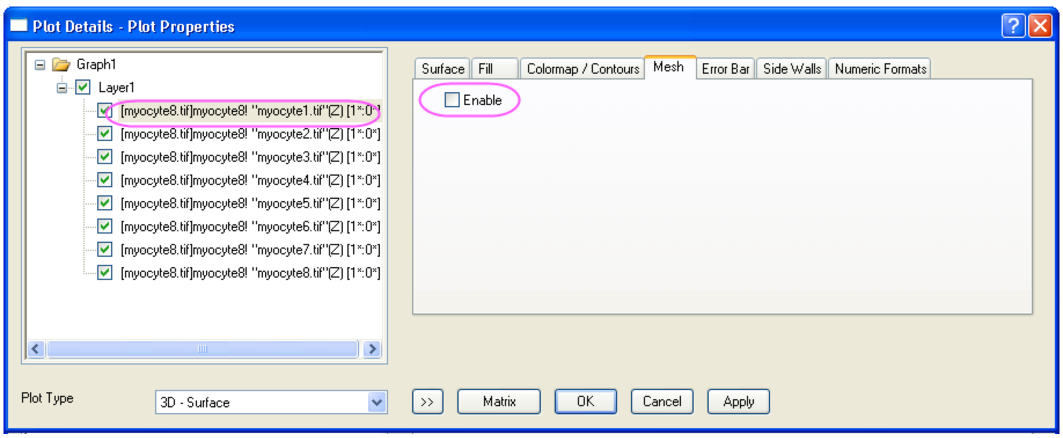
3. Select **Plot: 3D Surface: Multiple Colormap Surfaces** from the main menu to plot a surface graph.



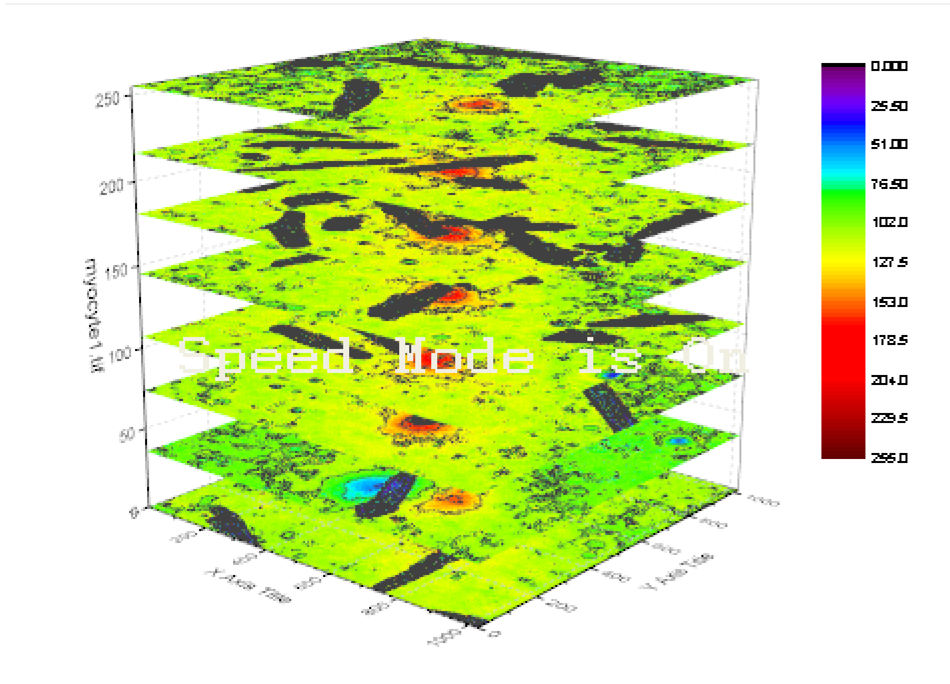
4. Select **Format: Plot properties** from the main menu to open the Plot Details dialog.

5. Select the first plot, check **Flat** and **Shift in Z by percent of scale range**, and set the shift value as 0. And go to the **Mesh** tab, uncheck the **Enable** check box to not show the grid lines.

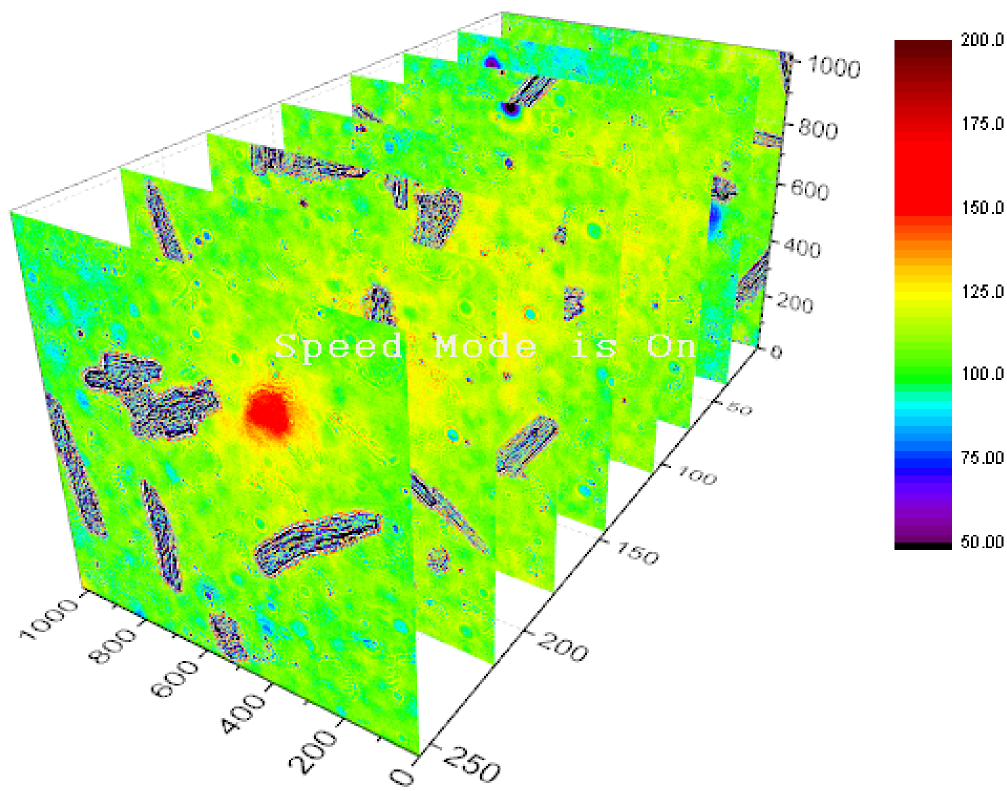




6. Select the 2nd-8th plot in order, check **Flat** and **Shift in Z by percent of scale range**, and set the shift value as 14, 29, 42, 57, 71, 85 and 100. And go to the **Mesh** tab, uncheck the **Enable** check box to not show the grid lines. Click the OK button to close the dialog.



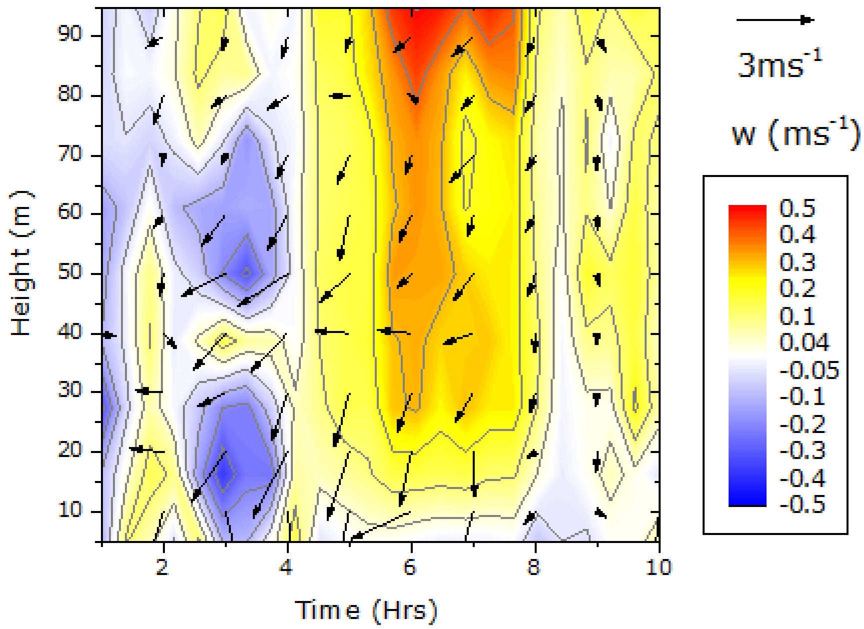
7. Rotate the graph to get a flattened colormap surface as shown below:



5.5.9 Contour Plot with Vector Overlay

Summary

This tutorial will show you how to create a contour plot with vectors overlay.



What will you learn

This tutorial will show you how to:

- Create and customize a contour graph and its color scale
- Create the XYAM vector graph
- Merge the two graphs

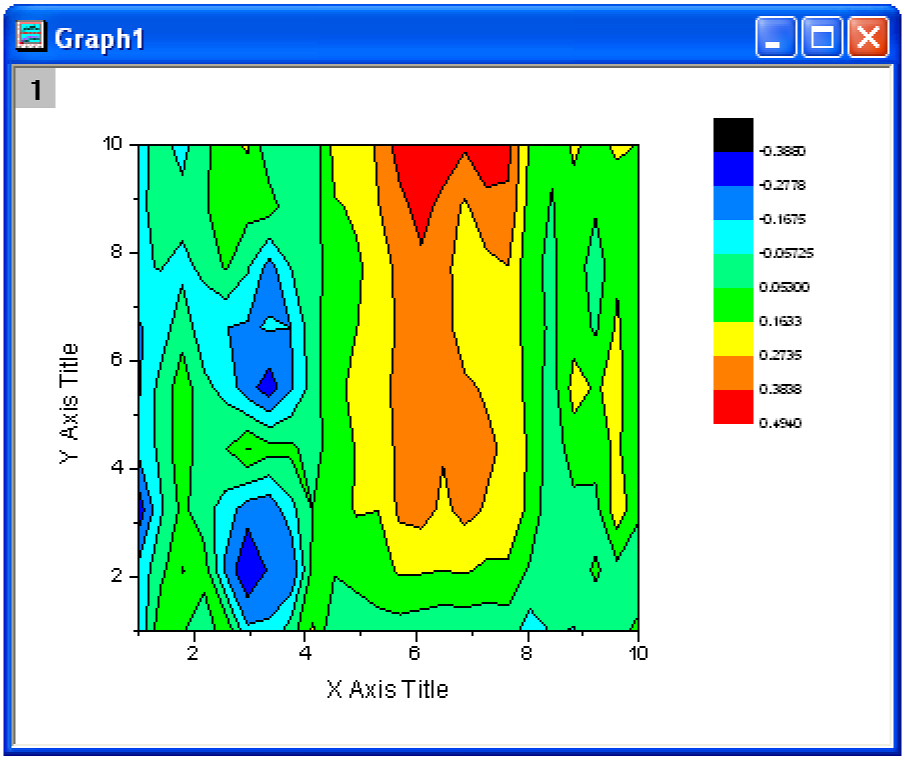
Steps

This tutorial is associated with the 2D and Contour Graphs project: *\Samples\2D and Contour Graphs.opj*.

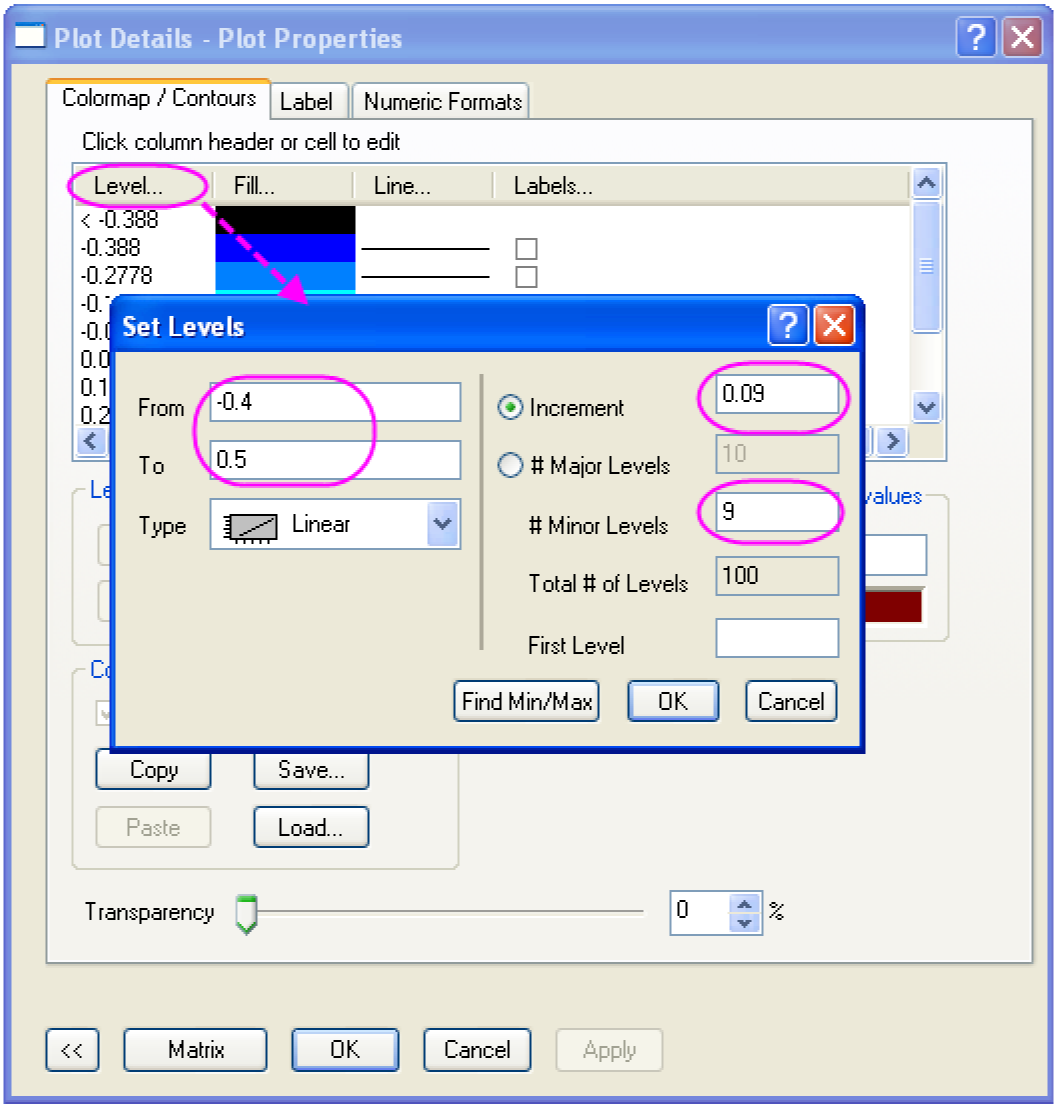
Create the Contour Graph

1. Open the *2D and Contour Graphs: Contour: Contour Plot with Vector Overlay* folder in the **Project Explorer**.

2. Activate the **W147** matrix and select **Plot: Contour: Color Fill** in the menu. The graph should look like:

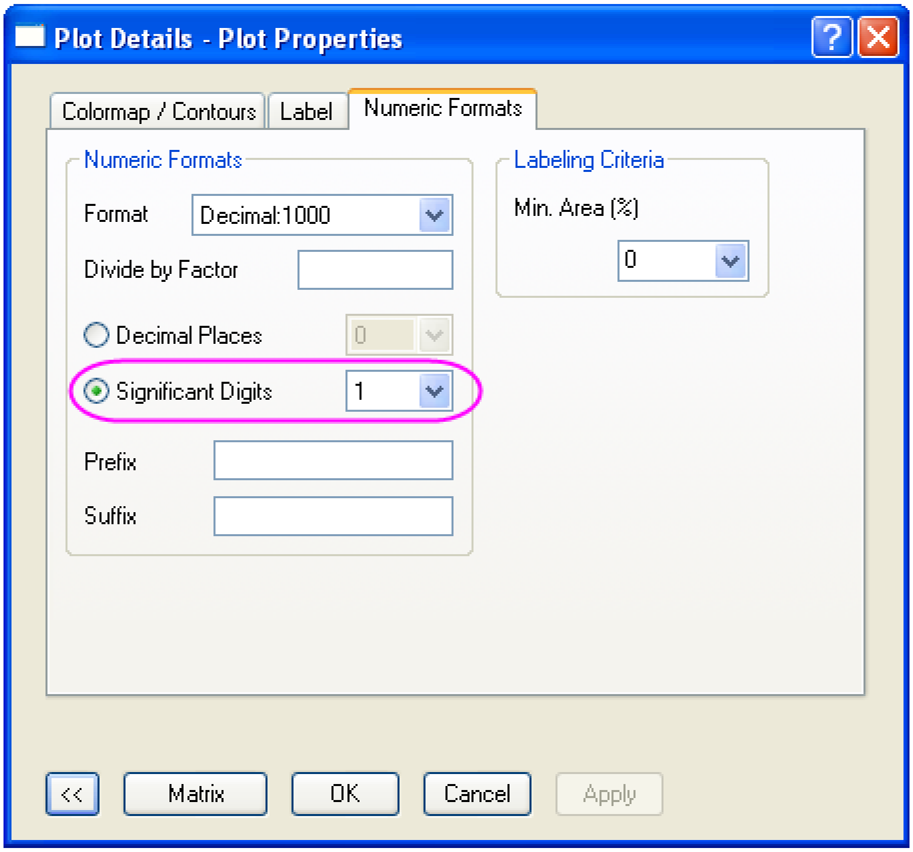


3. Select **Format: Plot Properties** to bring up the **Plot Details** dialog. Go to the **Color Map/Contours** tab. Customize the Contour settings as shown below:
    - Click the **Level** heading and set the dialog as shown in the following screenshot:



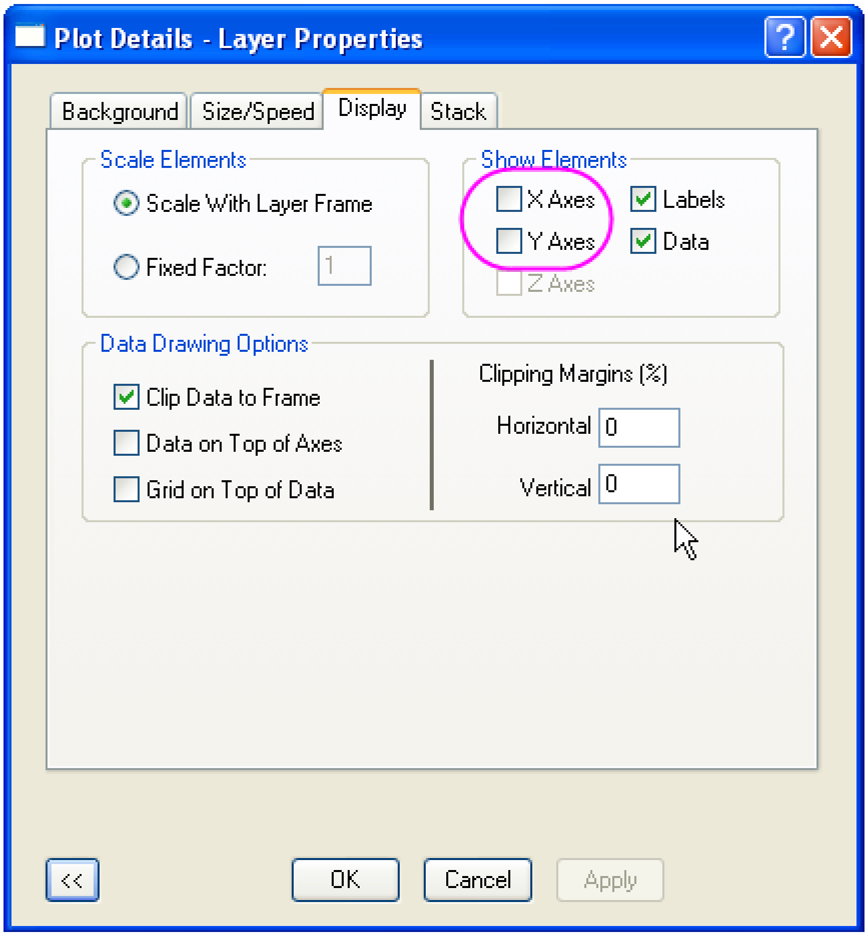
- Click the **Fill** heading to open the **Fill** dialog. Click on **Load Palette**, select **Temperature** from the **Palette** list, and click **OK**.
- Click the **Lines** heading, enable **Apply to All** checkbox and choose **Gray** in the **Color** dropdown list. Click **OK**.

4. Go to **Numeric Formats** tab of the **Plot Details** dialog, select **Decimal Places** radio button and set value to **1**. Click **OK**.

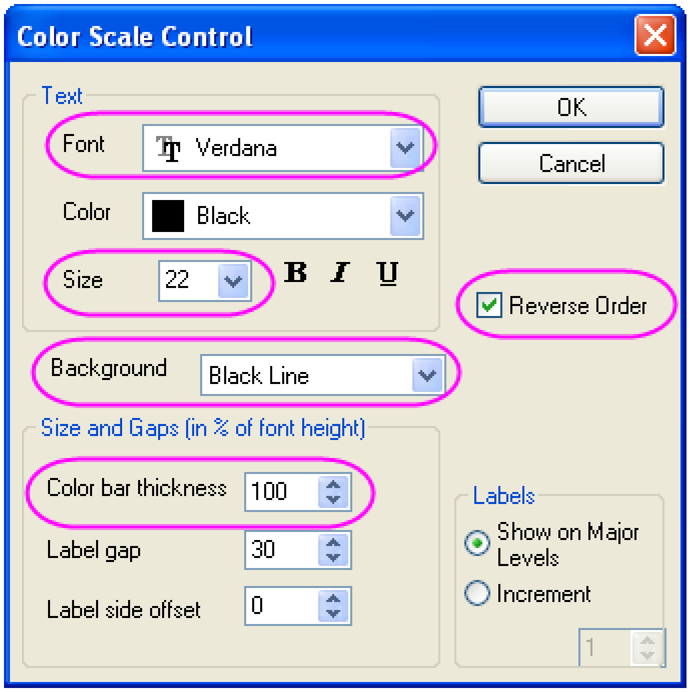


5. To prepare it for merging with the **XYAM Vector** Graph (you will create below), the axes must be hidden. To do that, select **Format: Layer Properties**. Then go to the **Display** tab and disable

the **X Axes** and **Y Axes** checkboxes. Click **OK**.

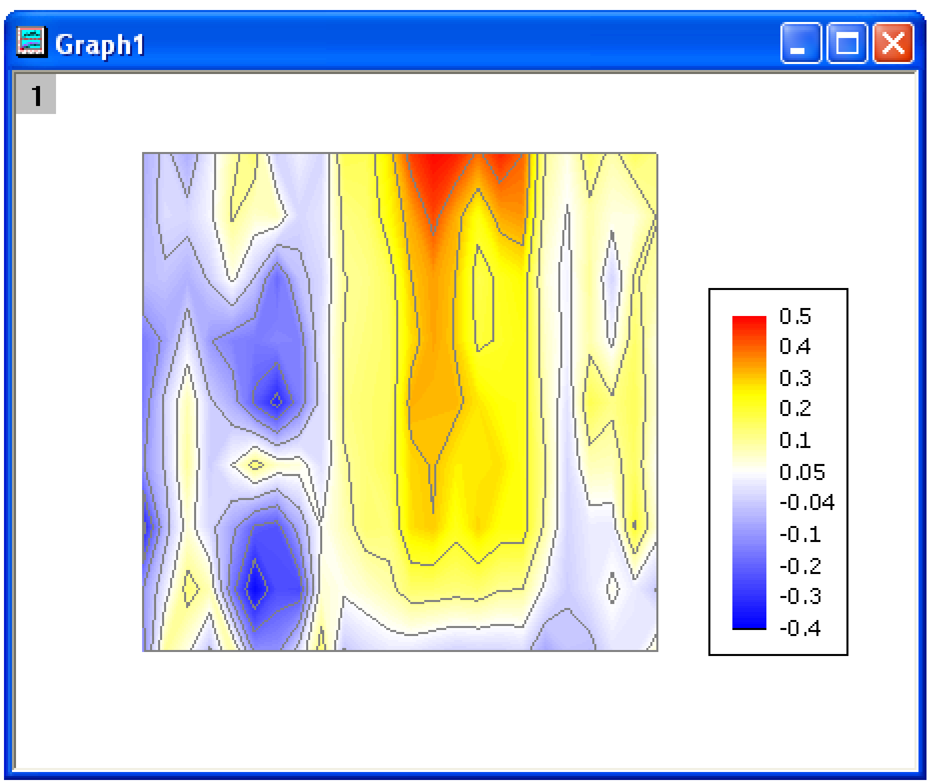


6. Right click on the Y Axis title and choose Delete. Do the same for the X Axis title.
7. To change the color scale, double-click on the color scale in the contour graph. Update its settings as given below:
    - Set the **Font** to **Verdana**
    - Set the **Size** to **22**
    - Select **Black Line** from the **Background** drop-down list
    - Set **Color bar thickness** to **100**
    - Select the **Reverse Order** checkbox



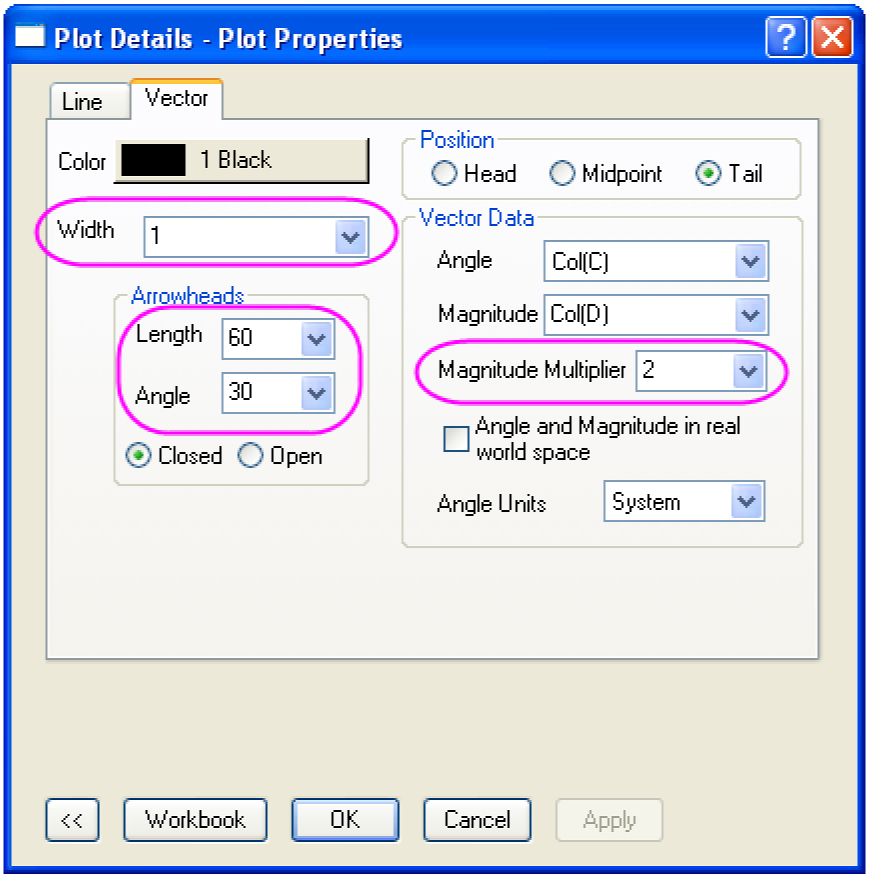
Then click **OK** and position the color scale in the desired location (on the right side of the graph) by clicking and dragging it with your mouse.

The contour plot should now look like:

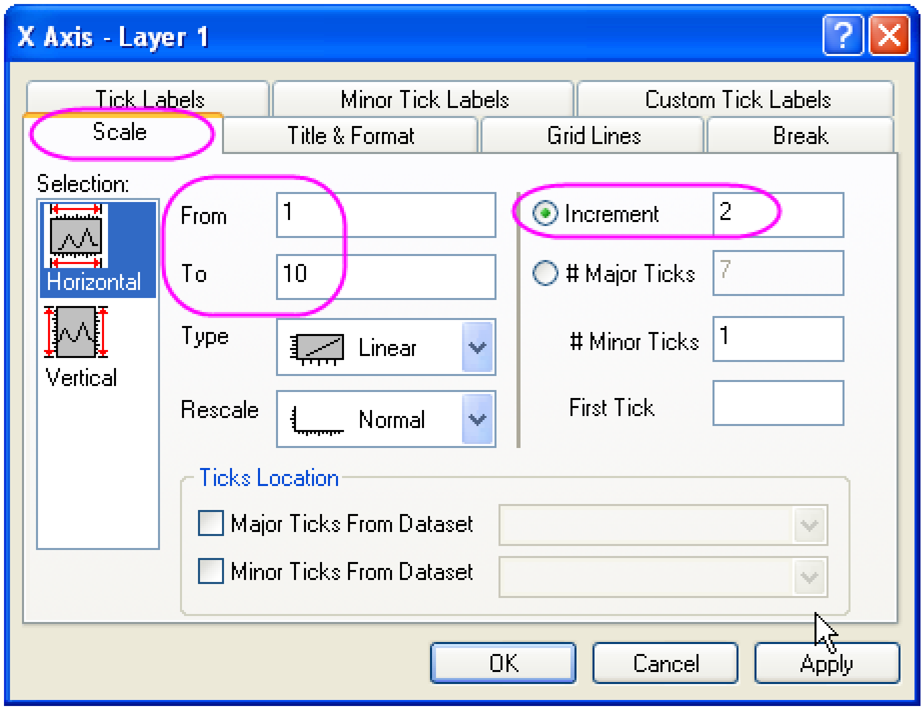


Create the XYAM Vector Graph

1. Activate the **WOR81147** worksheet, highlight the last three columns and select **Plot: Specialized: Vector XYAM** in the menu.
2. Double-click on any vector to bring up the **Plot Details** dialog on the **Vector** tab. Use the settings the following screenshot shows:



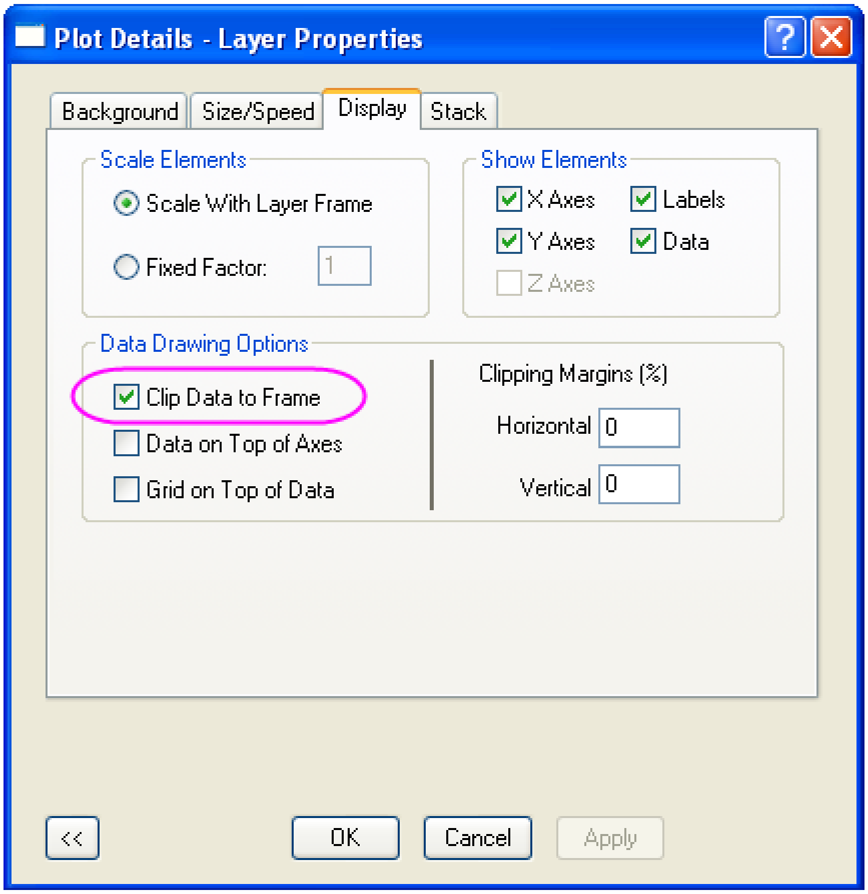
Then click **OK**.
3. To update the axis scales, select **Format: Axes: X Axis...** to open the dialog. Then do the following:
   - In the **Scale** tab, select **Horizontal** icon in the **Selection** list box. Set X **From = 1**, **To = 10** and **Increment = 2**.



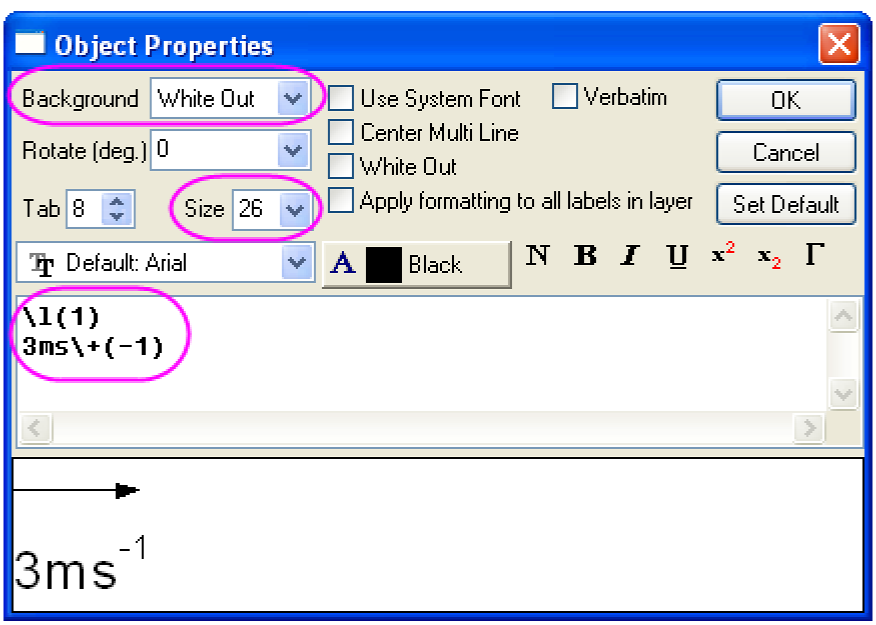
- Select the **Vertical** icon in the **Selection** list box, set the scale of Y as **From = 5**, **To = 95**, and **Increment = 10**.
- Go to the **Title and Format** tab, select the **Top** icon and **Right** icon respectively, and check the **Show Axis & Ticks** checkbox. Click **OK**.

4. At this stage, you may notice that the vectors extend outside the axes (layer frame). To make sure they display within the layer frame only, select **Format: Layer Properties**. Go to the

Display tab and check the **Clip Data to Frame** checkbox. Click **OK**.



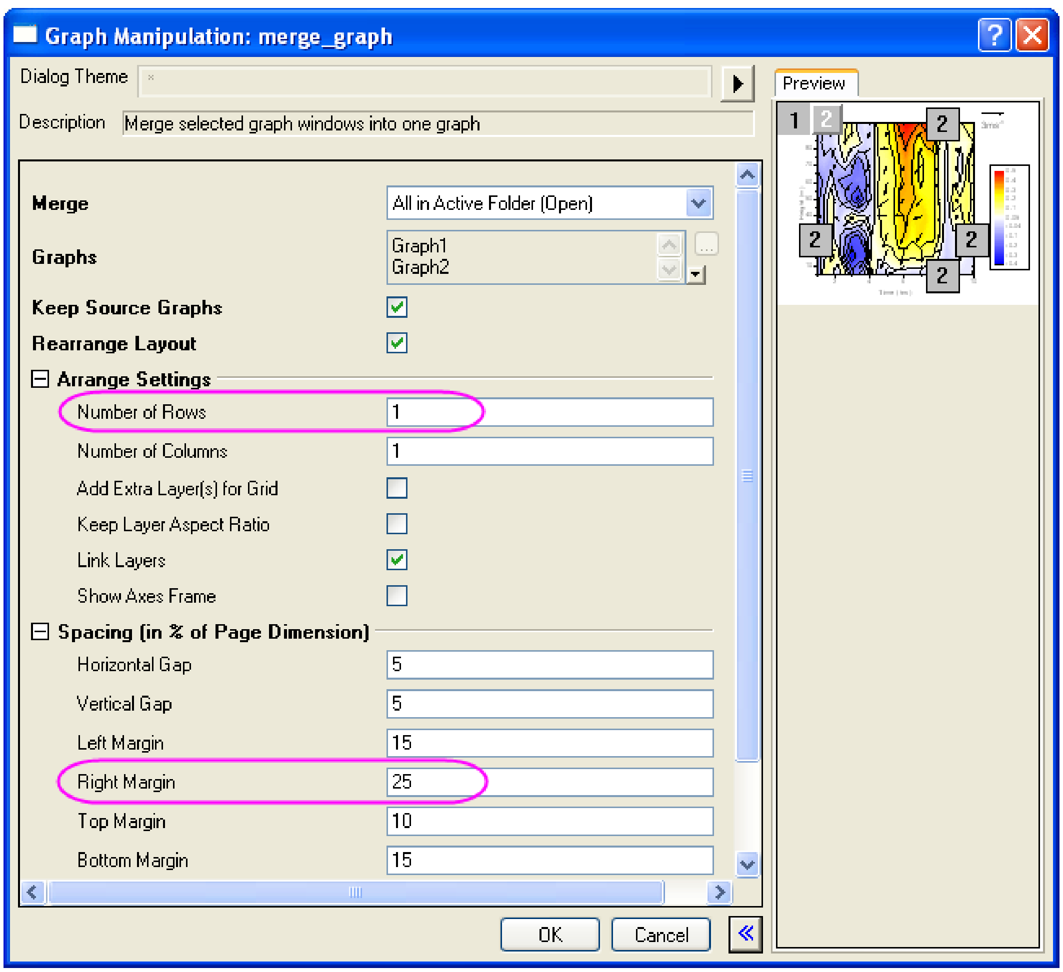
5. Right-click on the legend of the vectors graph, and select **Properties...** in the context menu to open the **Object Properties**. Then set the dialog following the screenshot below:



Merge the Two Graphs

The contour and vector graphs are generated by the steps above, and are ready to be merged into one.

1. **Minimize** or **Hide** all other graphs except for the contour and vector graphs created using the instructions above. With one of the two graphs active, select **Graph: Merge Graph Window...** in the menu.
2. In the **Merge Graph** dialog, specify the settings as the following:
   - Expand the **Arrange Settings** node, and type **1** in the **Number of Rows** text box.
   - Expand the **Spacing (in % of Page Dimension)** node, and enter a value of approximately "25" in the **Right Margin** text box to show the color scale legend from the contour plot. Then click **OK**.



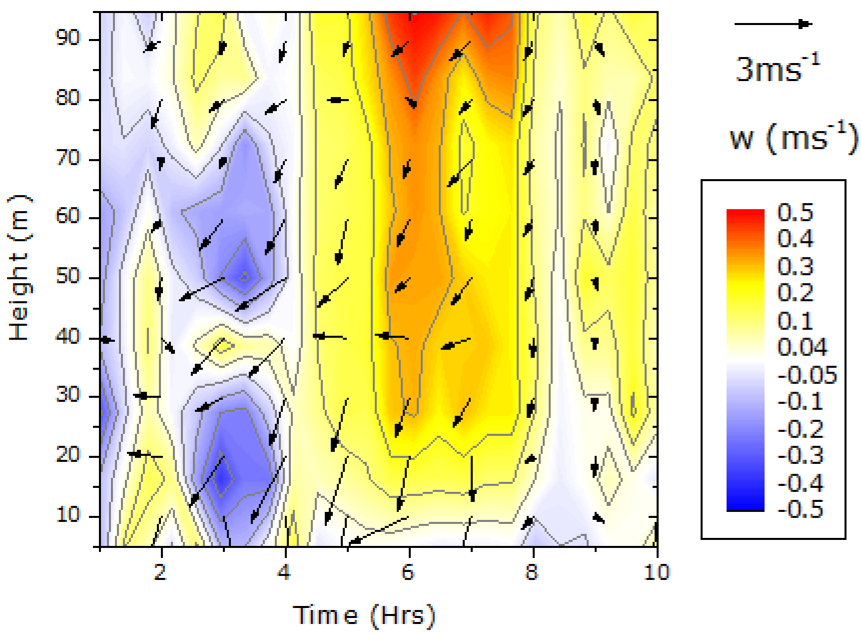
3. Click the **Text Tool** button  to create a text at the top color scale legend, and enter **w (ms-1)**. Highlight **-1** in the text and click on **Superscript** button  in the **Format** toolbar. Then set the font size to **26**.
4. To specify font type of all the objects to **Verdana**, you can:

- Set the objects individually, by clicking on the object and selecting **Verdana** in the **Font** button  in the **Format** toolbar.

Or

- Use **Theme Organizer** to create and apply a font theme to the current graph.

The final graph should look something like this:



5.6 Statistical

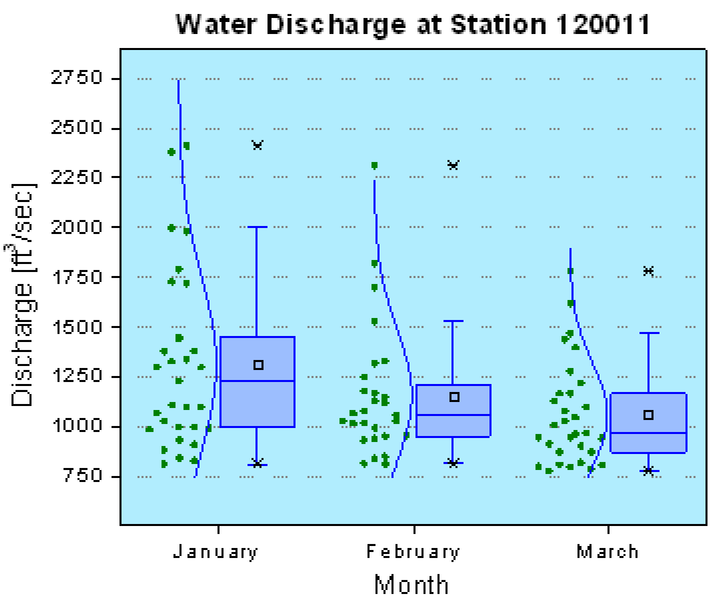
Topics covered in this section:

1. Box Plot
2. Simple Dot Chart
3. Multi-Data Dot Chart
4. Add multiple fitted curves in a Histogram
5. Weibull Probability Plot
6. Q-Q Plot
7. Scatter Matrix

5.6.1 Box Plot

Summary

This tutorial shows how to create the following box plot in the Statistical and Specialized Graphs project (\Samples\Statistical and Specialized Graphs.opj).



What you will learn

- Create a data overlapped box plot
- Customize box plot settings
- Change layer background

Steps

This tutorial is associated with the Statistical and Specialized Graphs project: *\Samples\Statistical and Specialized Graphs.opj*.

1. Open the Statistical and Specialized Graphs project and browse to folder *Statistical and Specialized Graphs: Statistical: Box Chart*.
2. Activate worksheet **Box Plot Data**, and select columns January(Y), February(Y), and March(Y) and plot them to the Box Chart from **Plot: Statistics: Box Chart**.
3. Double-click on one of the box charts to bring up **Plot Details** dialog. Select **None** from **Border Color** drop-down list and click **Apply** button, so no data plot properties are set to automatically increment. Then, update the properties for the group as follows (if a property is not mentioned, leave it at its default):

   Box Tab:

Type:	Box[Right]+Data[Left]

4. **Data** Tab:

Distribution Curve Type:	Normal
Automatic Binning:	Disabled/Unchecked
Bin Size:	100
Begin:	750
End:	3600
Bin Height	100

5. **Line** Tab:

Style:	Solid
Width:	2
Color:	Blue

6. **Symbol** Tab:

Size:	5
Symbol Color:	Olive
Shape:	Circle
Interior:	Solid

7. **Pattern** Tab:

Border Color:	Blue

Border Style:	Solid
Border Width:	2
Fill Color:	Custom (R:156, G:190, B:254)
Pattern:	None

8. **Percentile** Tab:

Size:	5
Edge Color:	Black
Fill Color:	Black

9. In the **Plot Details** dialog, select the layer in the left panel to activate the tabs of layer level. Go to **Background** tab, set the color as a the color (R:177, G:237, B:254). Click **OK** to accept the setting and close the dialog.
10. Double-click on the Y Axis and select the Scale tab to set axis scale as follow:

From:	510
To:	2900
Increment:	250
#Minor Ticks:	0

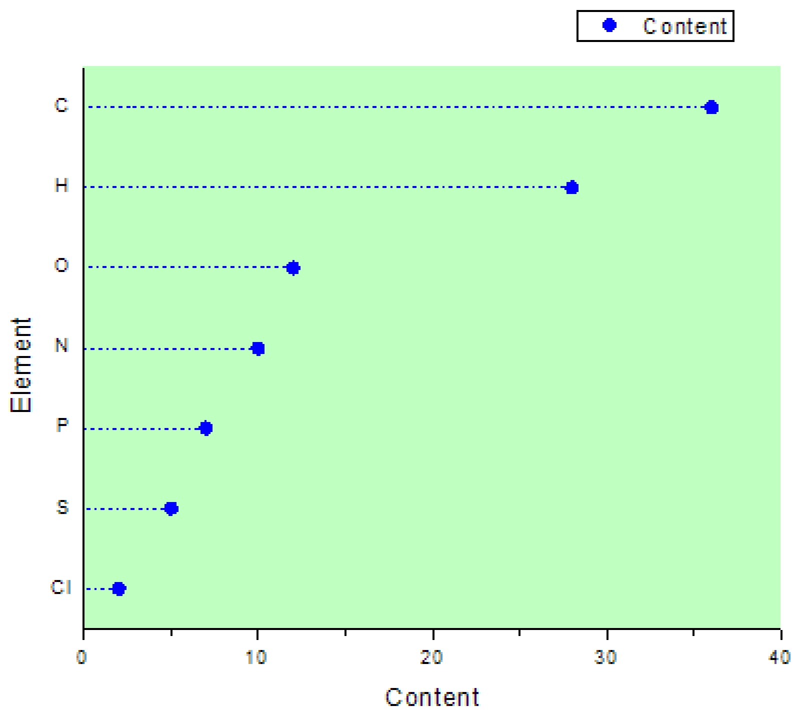
11. Activate the **Grid Lines** tab. Make sure the **Horizontal** icon is selected in the **Selection** list box so you can edit the grid lines for the Y axis. Enable the **Major Grids** check box and set the **Line Color** = *LT Gray* and **Line Type** = *Dot*.
12. Enable the **Opposite Line** check box on the **Grid Lines** tab for both the *Vertical* and *Horizontal* selections in the **Selection** List box.
13. Delete the legend and update the X and Y axis titles if desired.
14. Right-click the layer and select **Add/Modify Layer Title**, and add title as you want.

5.6.2 Simple Dot Chart

Contents

Summary

Dot chart is a statistical chart which consist of data points plotted on a simple scale. It is often used as a substitute for the pie chart, as it allows for quantities to be compared easily. This tutorial will teach you how to create a simple dot chart plot.



Minimum Origin Version Required: Origin 8.0 SR6

What you will learn

This tutorial will show you how to:

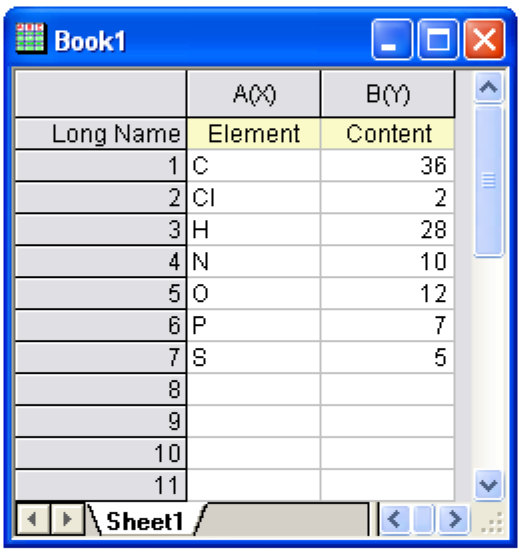
- Create a scatter graph
- Change the X-Y Axis
- Use Plot Setup dialog to customize your graph

Steps

Let us start with the following data which represents various elements in a compound:

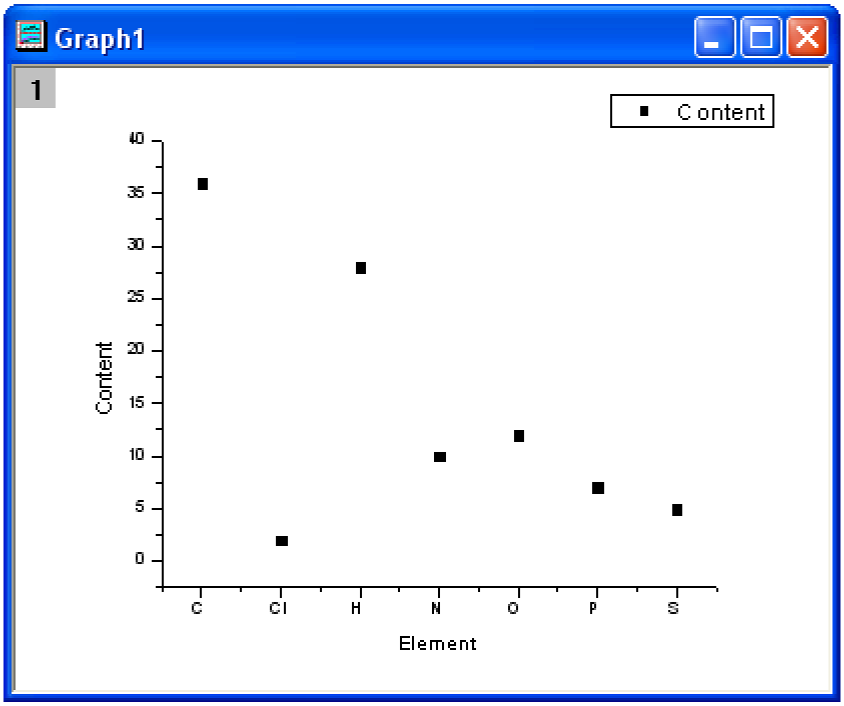
Element	Content
C	36
Cl	2
H	28
N	10
O	12
P	7
S	5

1. Create a new workbook, and input the data.

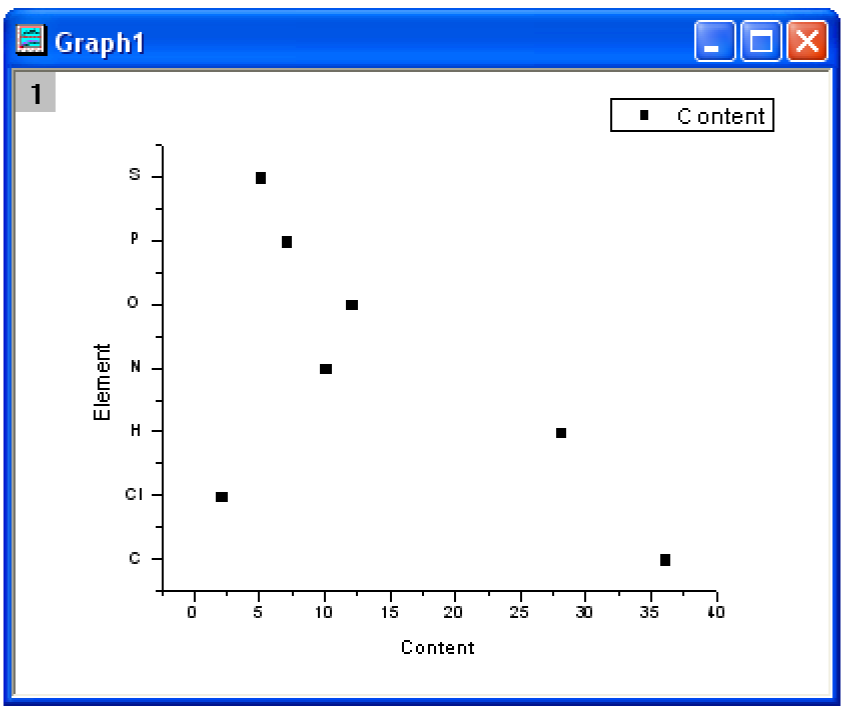


	A(X)	B(Y)
Long Name	Element	Content
1	C	36
2	Cl	2
3	H	28
4	N	10
5	O	12
6	P	7
7	S	5
8		
9		
10		
11		

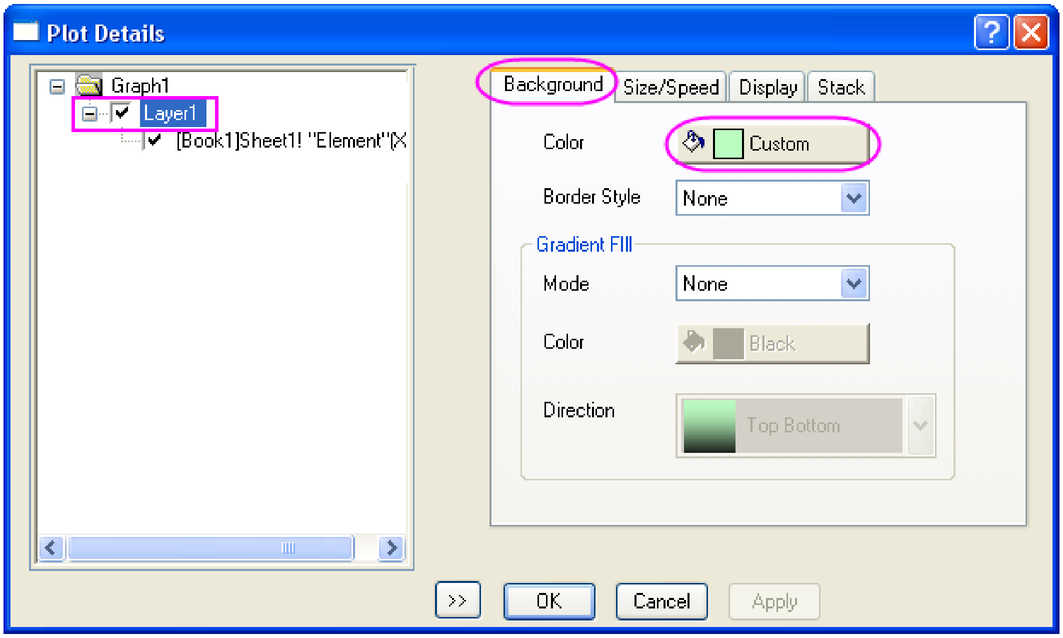
2. Highlight col(A) and col(B), and then select the **Plot: Symbol: Scatter** menu item from the Origin menu to create a scatter plot.

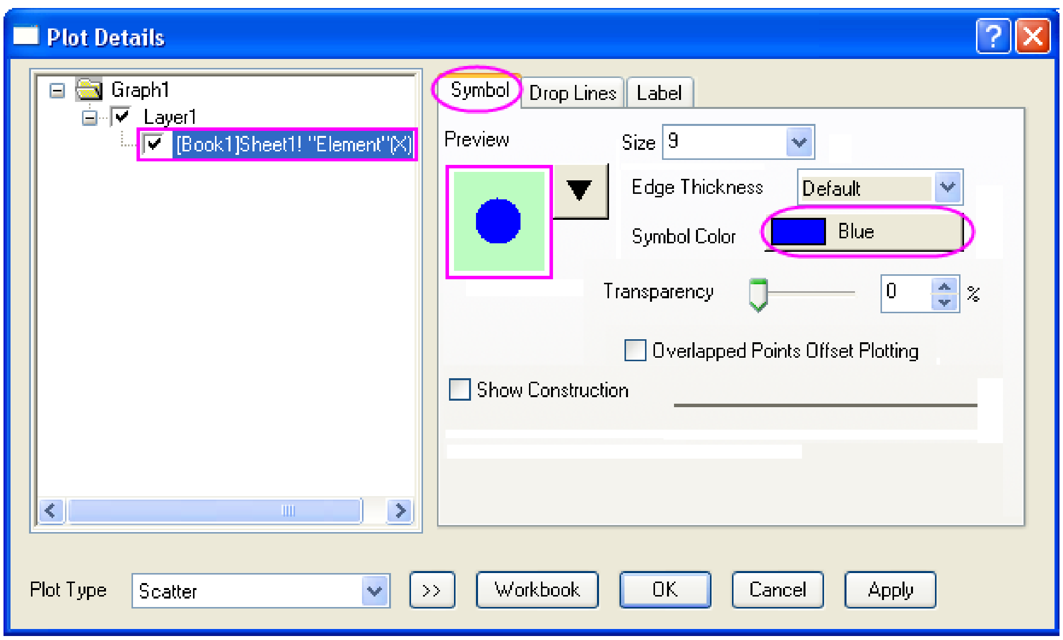


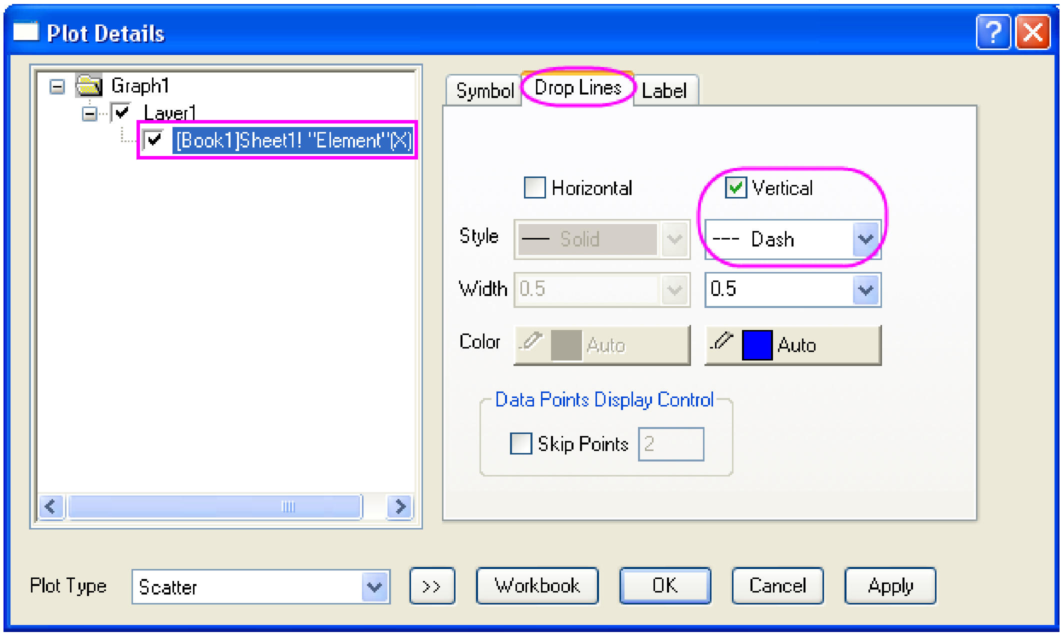
3. Select **Graph: Exchange X-Y Axis** from the menu.



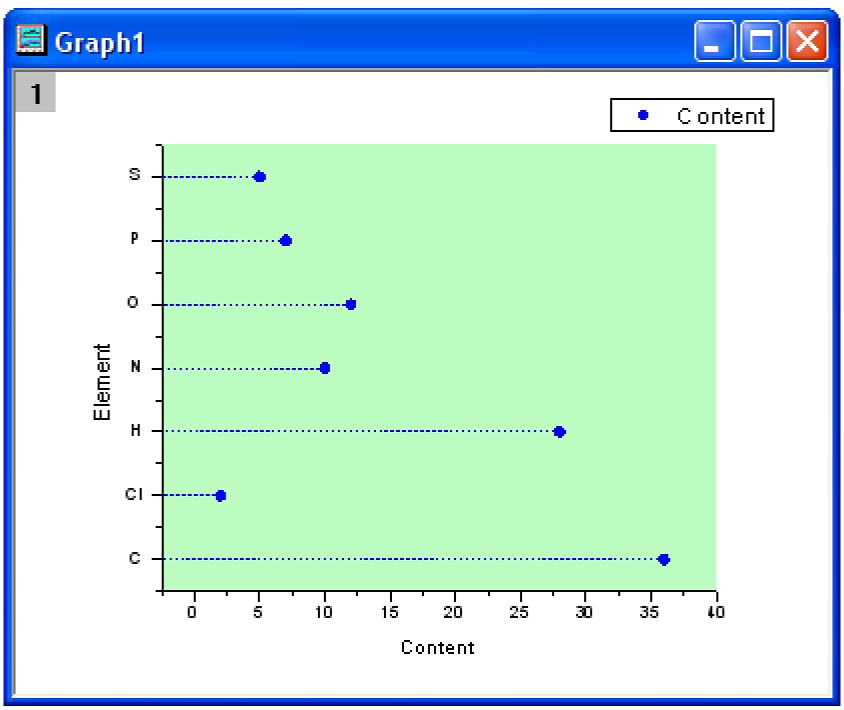
4. Double-click on the graph to bring up the **Plot Details** dialog, change the symbols and the symbol color as in the following image:


Plot Details
Graph1
Layer1
[Book1]Sheet1! "Element"(X
Background
Size/Speed
Display
Stack
Color
Custom
Border Style
None
Gradient Fill
Mode
None
Color
Black
Direction
Top Bottom
>>
OK
Cancel
Apply

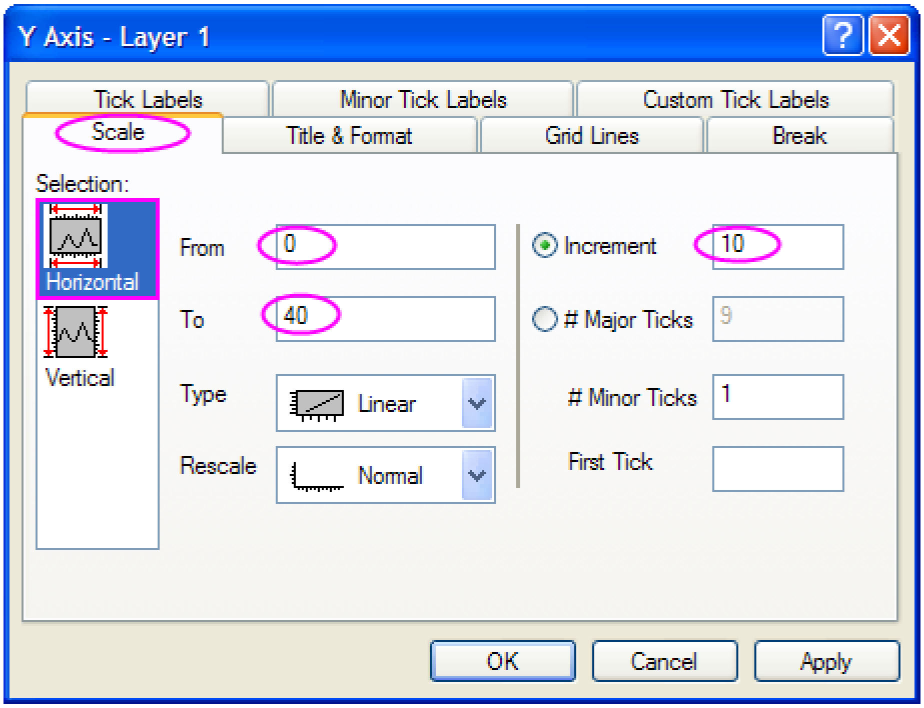

Plot Details
Graph1
Layer1
[Book1]Sheet1! "Element"(X)
Symbol
Drop Lines
Label
Preview
Size
9
Edge Thickness
Default
Symbol Color
Blue
Transparency
0
%
Overlapped Points Offset Plotting
Show Construction
Plot Type
Scatter
>>
Workbook
OK
Cancel
Apply



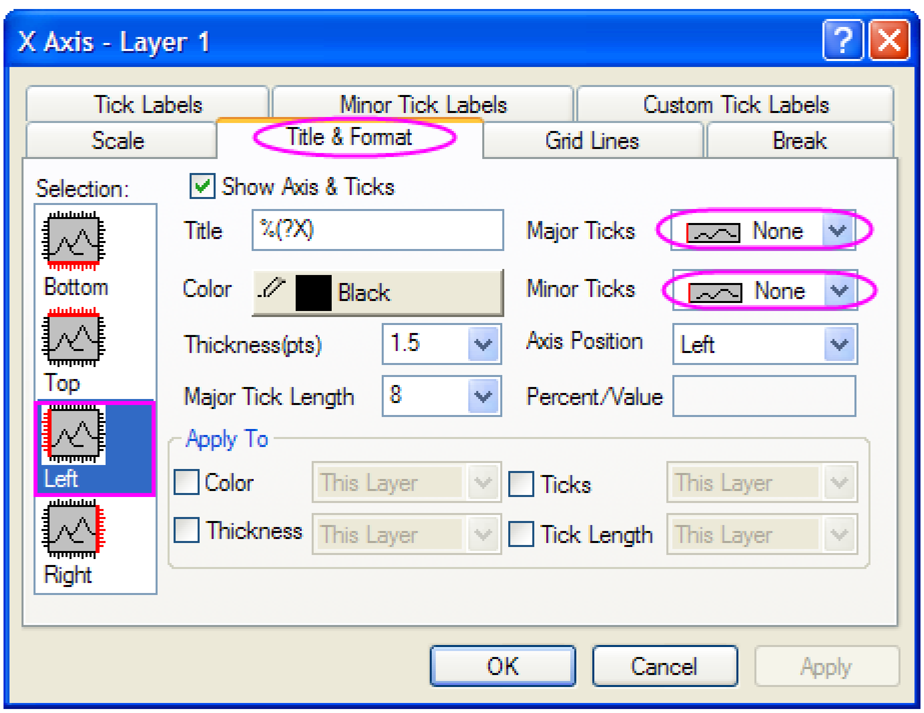
5. Click the **OK** button to close the dialog. Your graph should look like the image below:



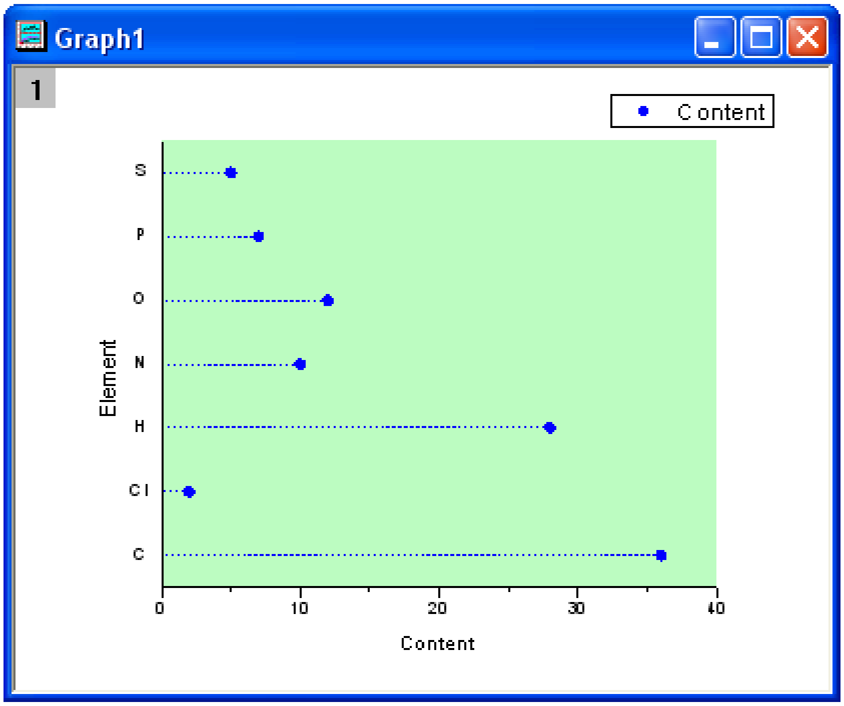
6. Let us reset the X and Y Axes. Double-click the x axis to open the **Axis Properties** dialog. In the **Scale** tab, set **From** as **0** and **To** as **40**. Set the **Increment** as **10**.



7. In the **Title and Format**, make sure **left** is selected in the **Selection** list, and then set **Major Ticks** and **Minor Ticks** as **None**. Finally, click **OK** button.



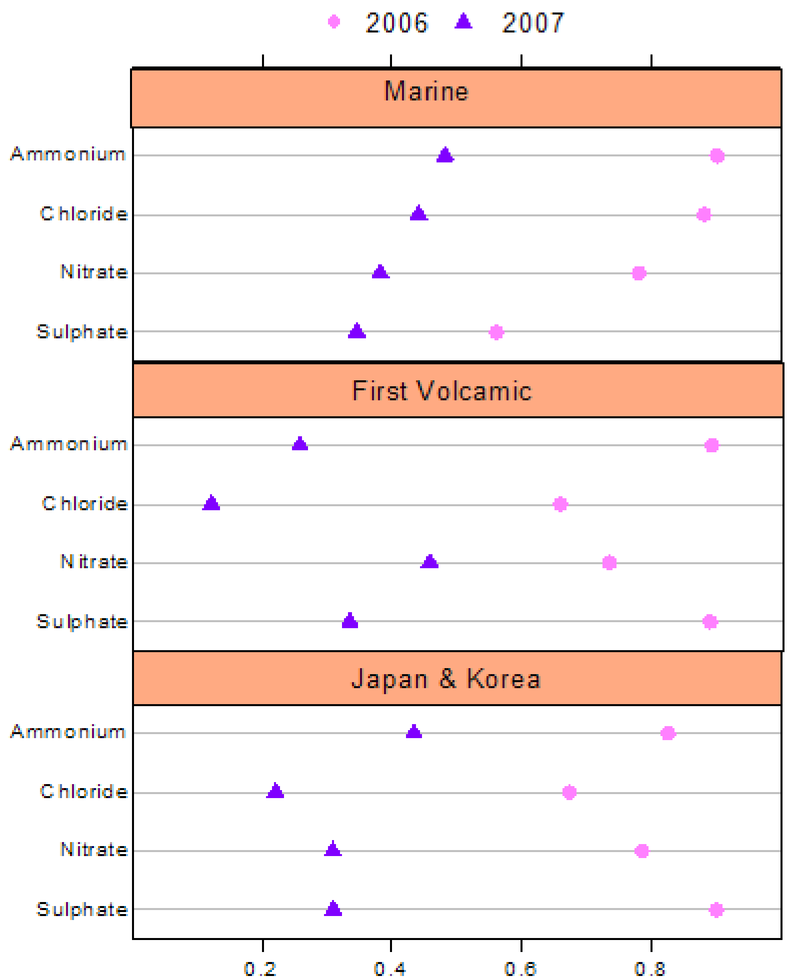
Now the Dot chart is plotted successfully and should look like below:



5.6.3 Multi-Data Dot Chart

Summary

A **Dot Chart** is a statistical chart which consists of data points plotted on a simple scale. It is often used as a substitute for the pie chart because it can make the comparing of quantities easy. This tutorial will teach you how to create the Multi-Data Dot Chart.



Minimum Origin Version Required: Origin 8.0 SR6

What you will learn

This tutorial will show you how to:

- Create a scatter graph
- Change the X-Y Axis
- Use the Plot Setup dialog to customize your graph
- Use Layer Management
- Customize the axis
- Add objects on the graph

Steps

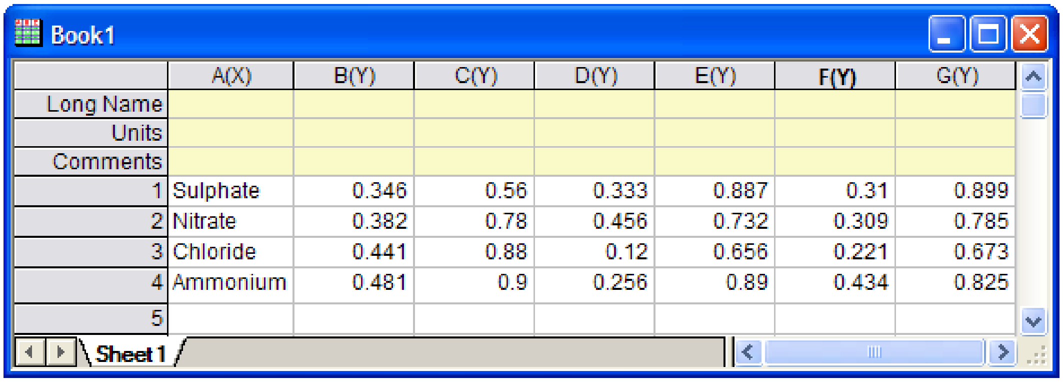
Let us learn how to create a multi-data dot chart. Here is some data about the element content of several areas in different time. We can use it to create a dot chart.

Sulphate	0.346	0.560	0.333	0.887	0.310	0.899
Nitrate	0.382	0.780	0.456	0.732	0.456	0.732

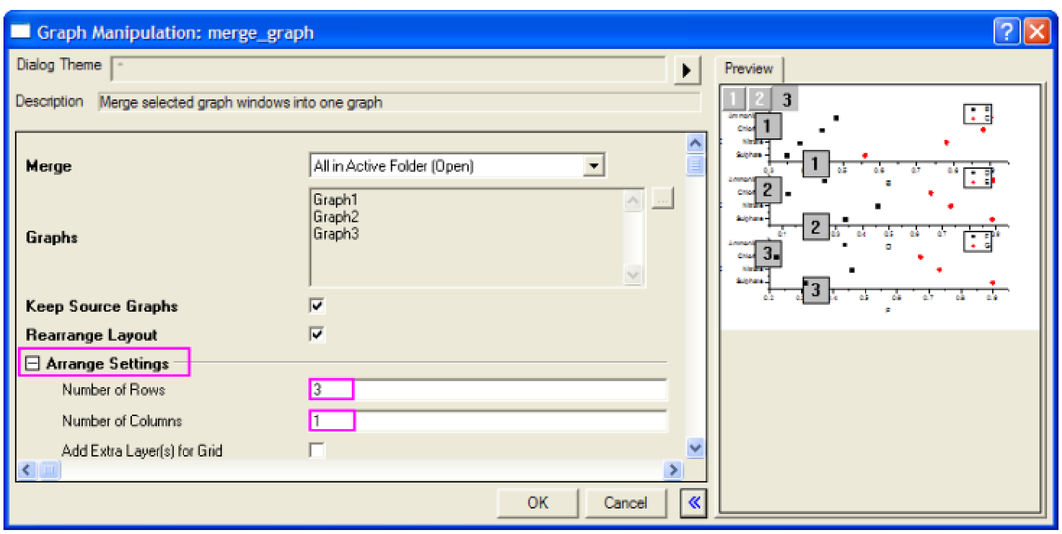
Chloride	0.441	0.880	0.120	0.656	0.221	0.673
Ammonium	0.481	0.900	0.256	0.890	0.434	0.825

Now, let us begin.

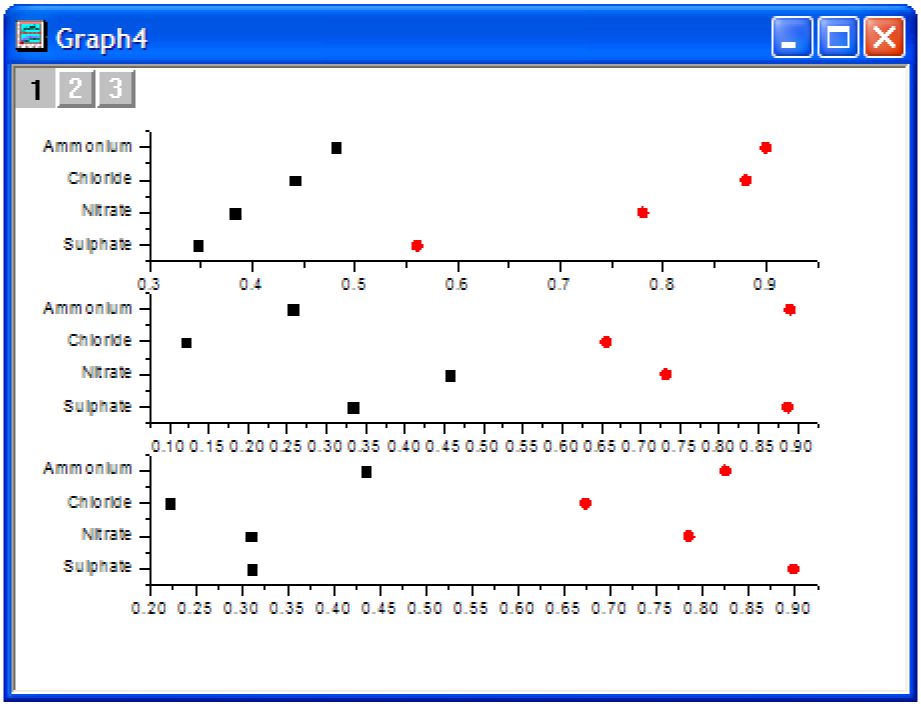
1. Create a new workbook and input the data.



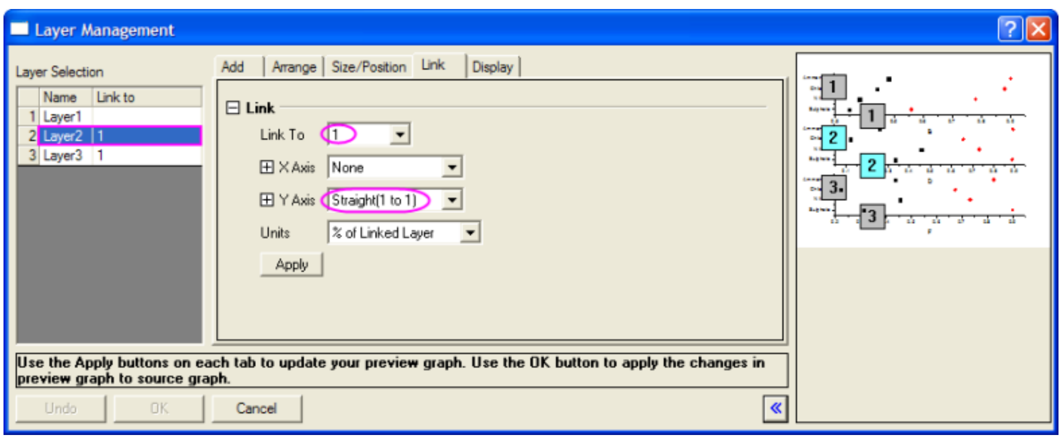
2. Highlight col(B) and col(C), select **Plot: Symbol: Scatter** in the main menu to draw a graph, then select **Graph: Exchange X-Y Axis**.
3. Repeat step 2 to create one graph with col(D) and col(E), and another graph with col(F) and col(G).
4. Merge these three graphs. Select **Graph: Merge Graph Windows: Open dialog**. Expand **Arrange Settings**, set **the Number of Rows** as **3** and **Number of Columns** as **1**. Click the **OK** button. Now you get a new graph which contains three layers.



5. You can delete the legend and the XY axis labels in the graphs to clean up the graph.

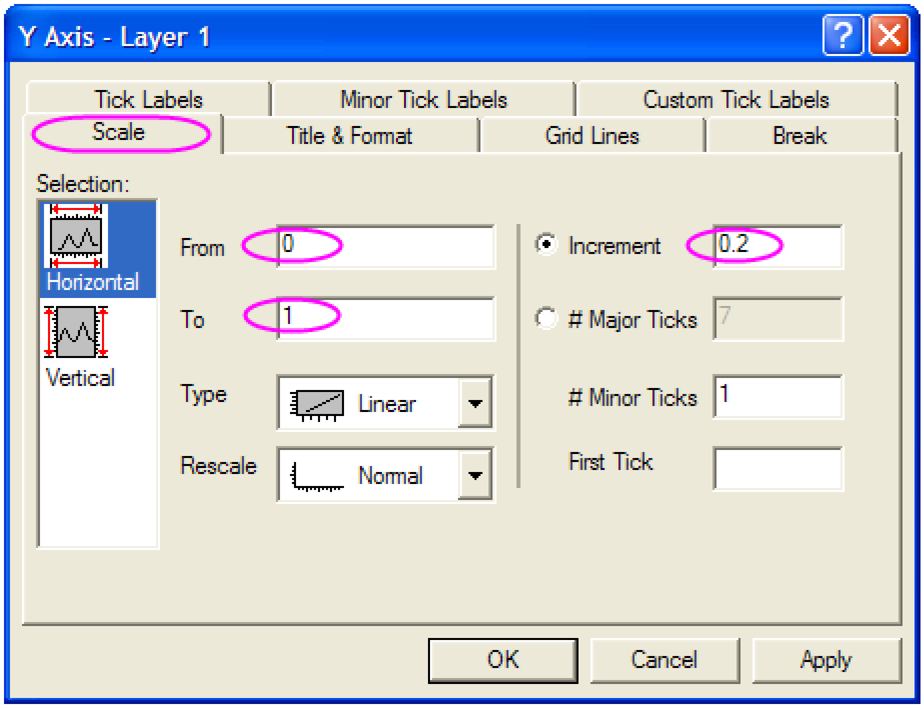


6. Select **Graph: Layer Management** from the main menu. Select Layer 2 on the left. Then on the right panel, make sure that the **Link** tab is active. In this tab, set **Link to** as **1** and **Y Axis** as **Straight (1 to 1)**. Click the **Apply** button.

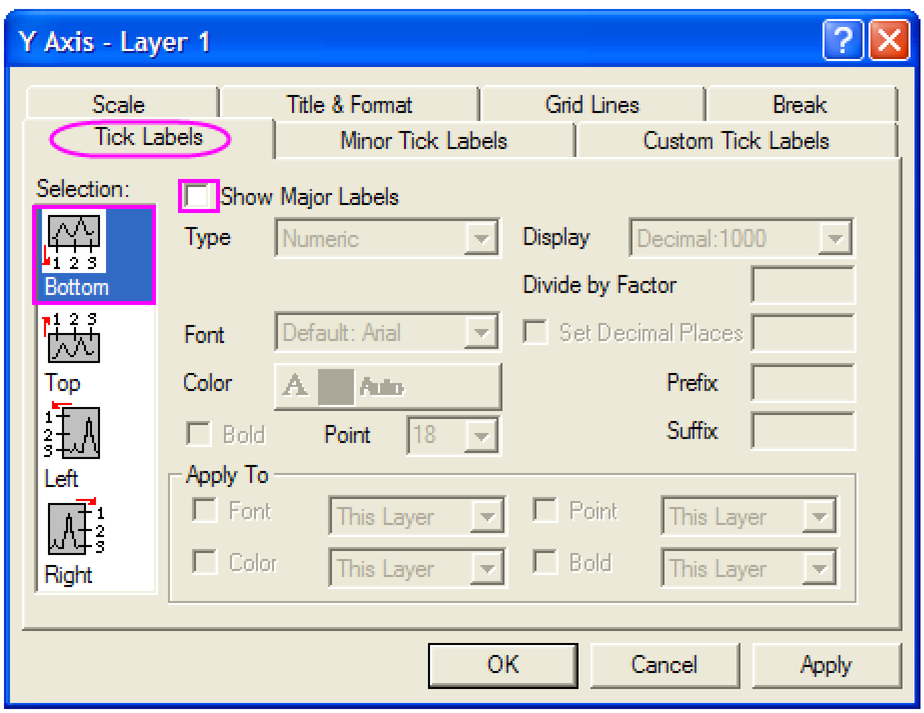


7. Select Layer 3 on the left. Also in the **Link** tab, set **Link to** as **1** and **Y Axis** as **Straight (1 to 1)**. Then click the **OK** button.

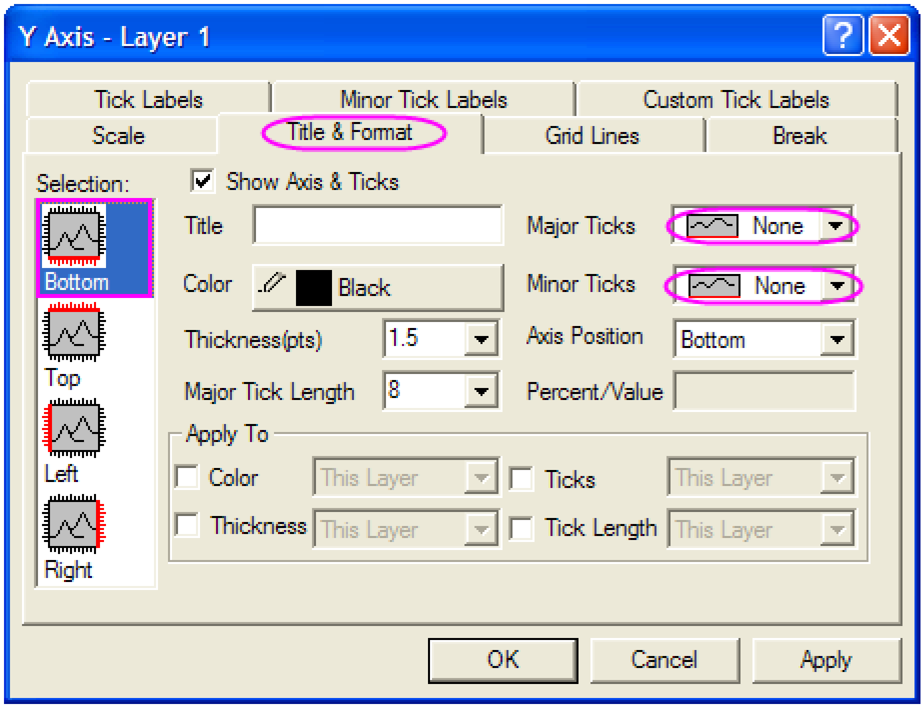
8. Reset X and Y Axes. Make sure **Layer 1** is active, double-click the X axis of Layer 1. Set the scale of X **From 0 To 1**, and the **Increment** as **0.2**.



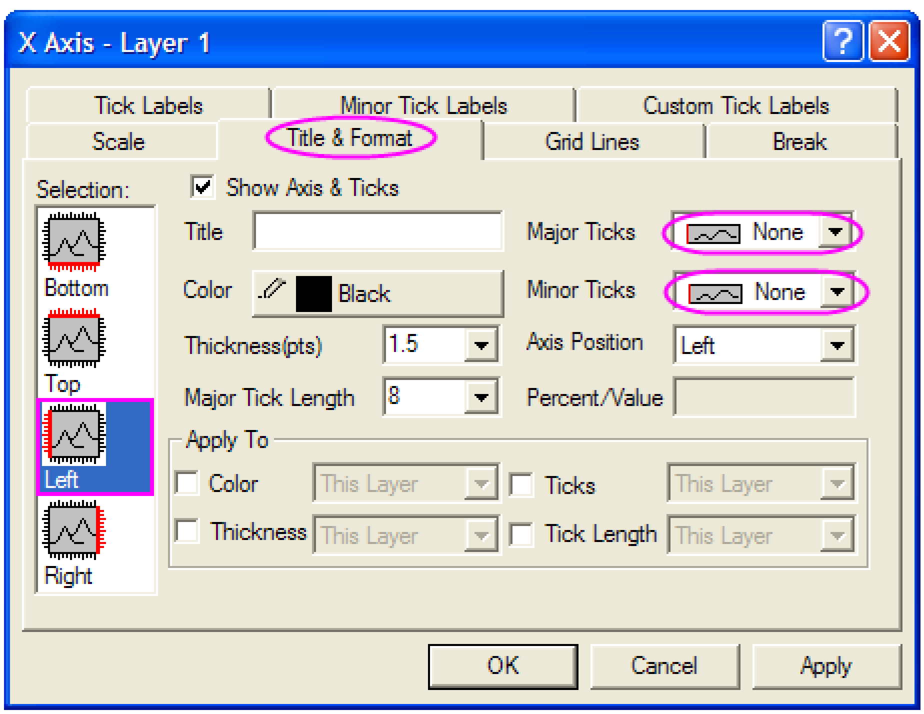
9. In the **Tick Labels** tab, uncheck the **Show Major Label** box.



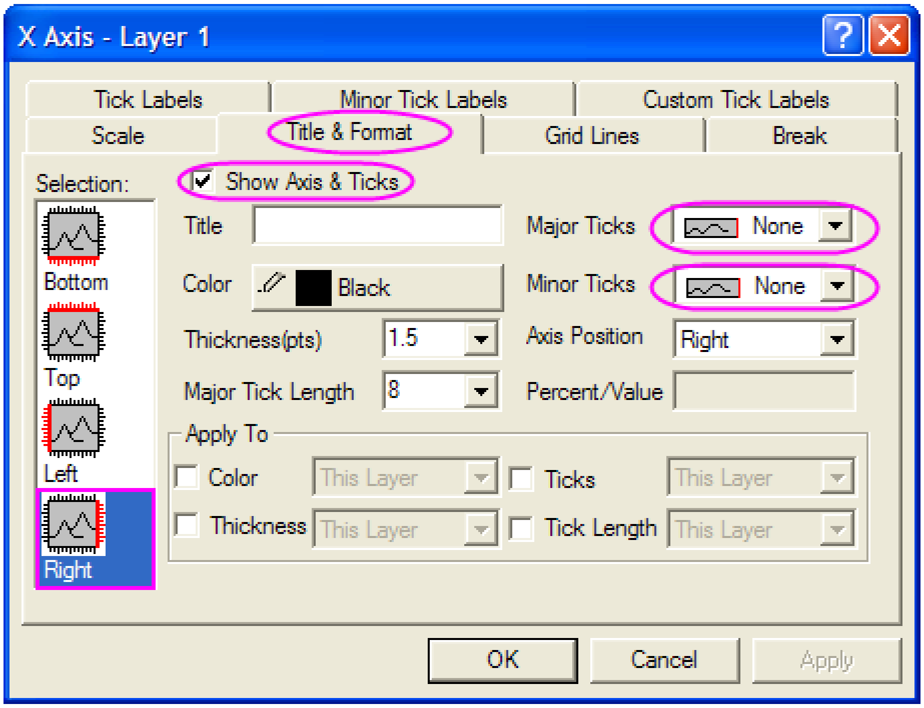
10. In the **Title and Format** tab, set **Major Ticks** and **Minor Ticks** as **None**.



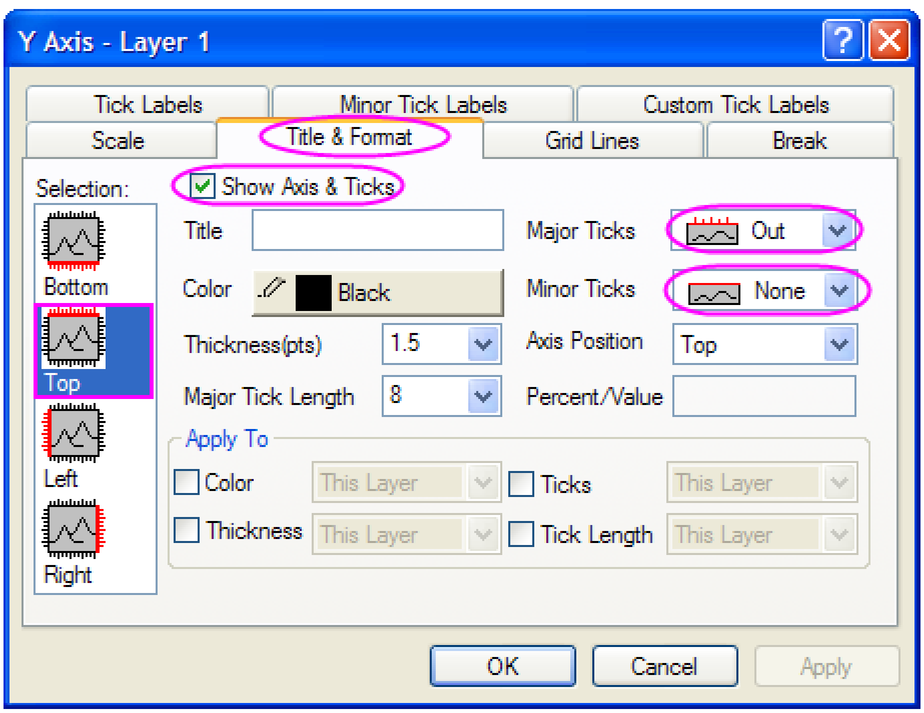
11. Click **Left** in the **Selection** list, select the **Show Axis & Tick** check box and set **Major Ticks** and **Minor Ticks** as **None**.



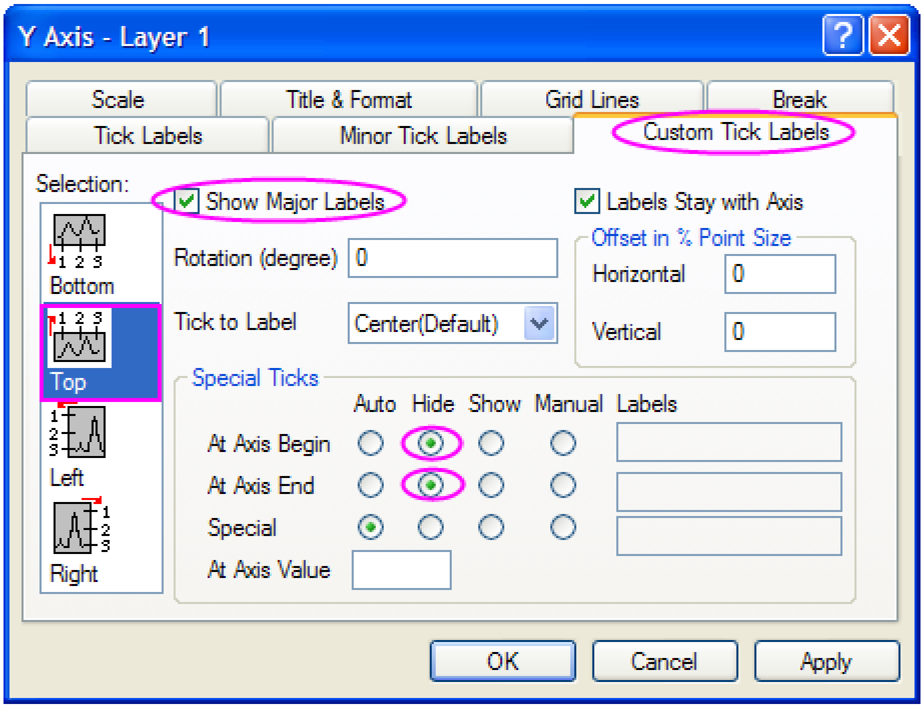
12. Click **Right** in the **Selection** list, select the **Show Axis & Tick** check box and set **Major Ticks** and **Minor Ticks** as **None**.



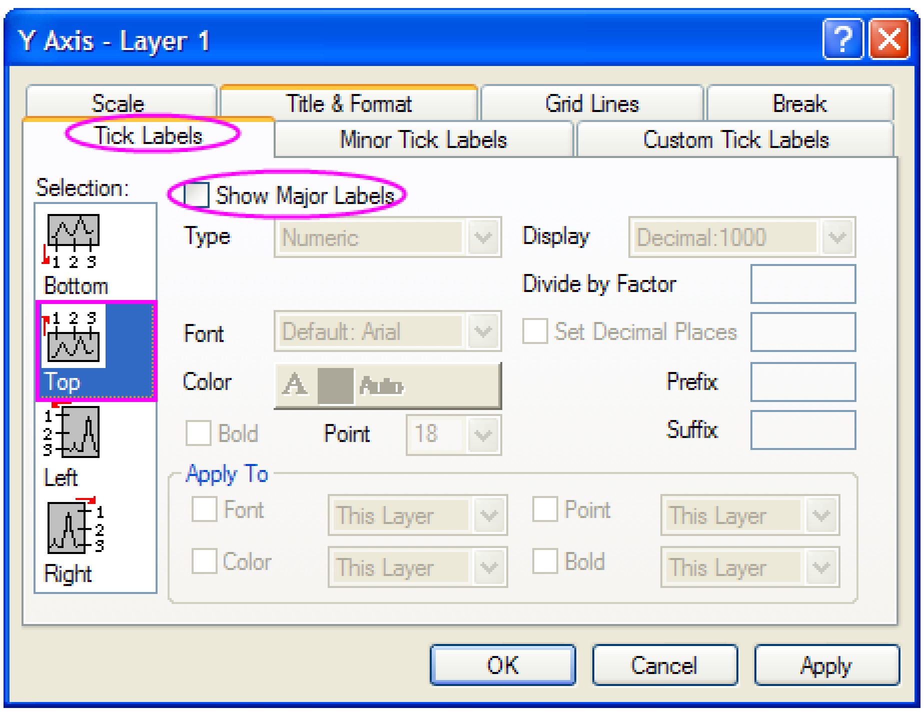
13. Select **Top** in the **Selection** list, check the **Show Axis & Tick** box. set **Major** as **Out** and **Minor** as **None**.



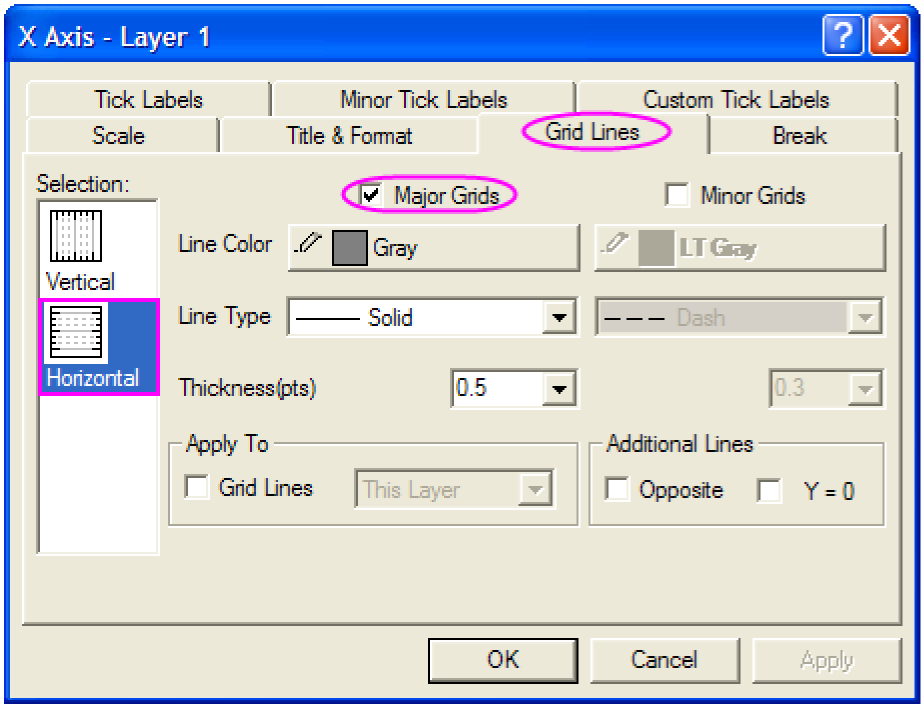
14. Select the **Custom Tick Labels** tab, highlight **Top** in the Selection, choose the **Hide** radio button both with **At Axis Begin** and **At Axis End**.



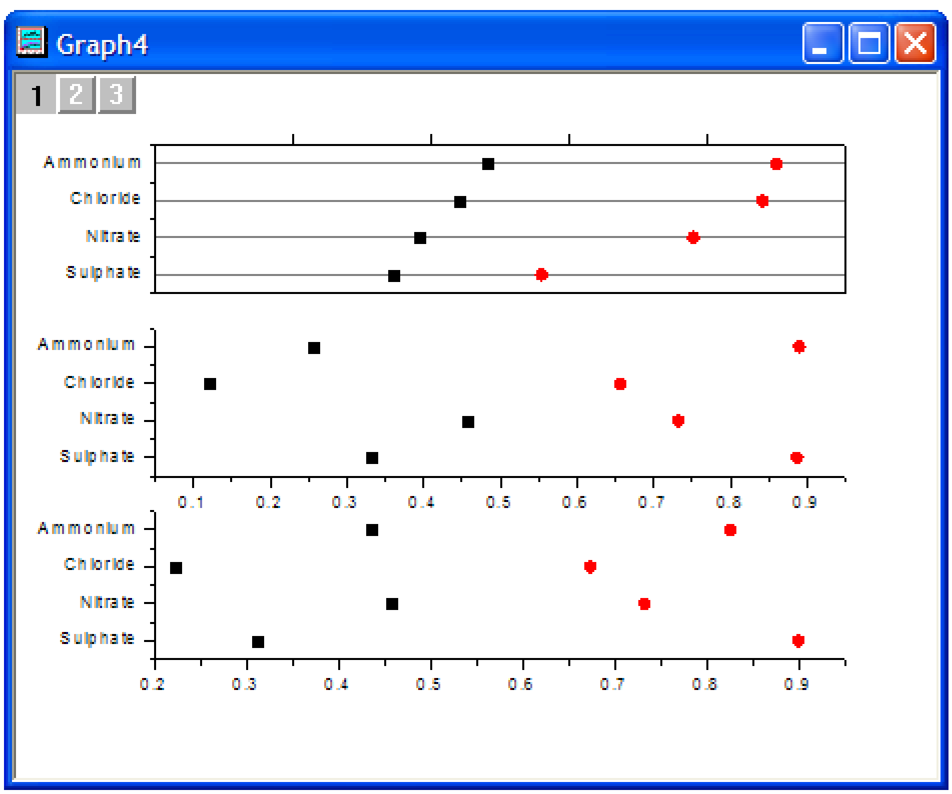
15. In the **Tick Labels** tab, uncheck the **Show Major Label** box.



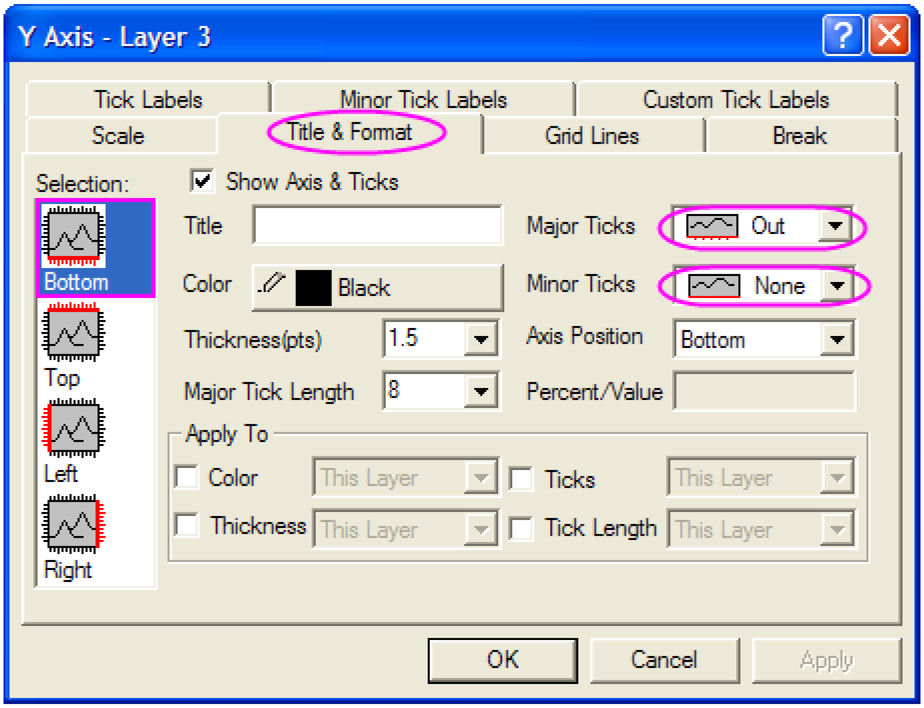
16. In the **Grid Lines** tab, select **Horizontal** in the **Selection** list and then select the **Major Grid** checkbox. Also, choose the color and the style for the grid lines. Click the **OK** button.



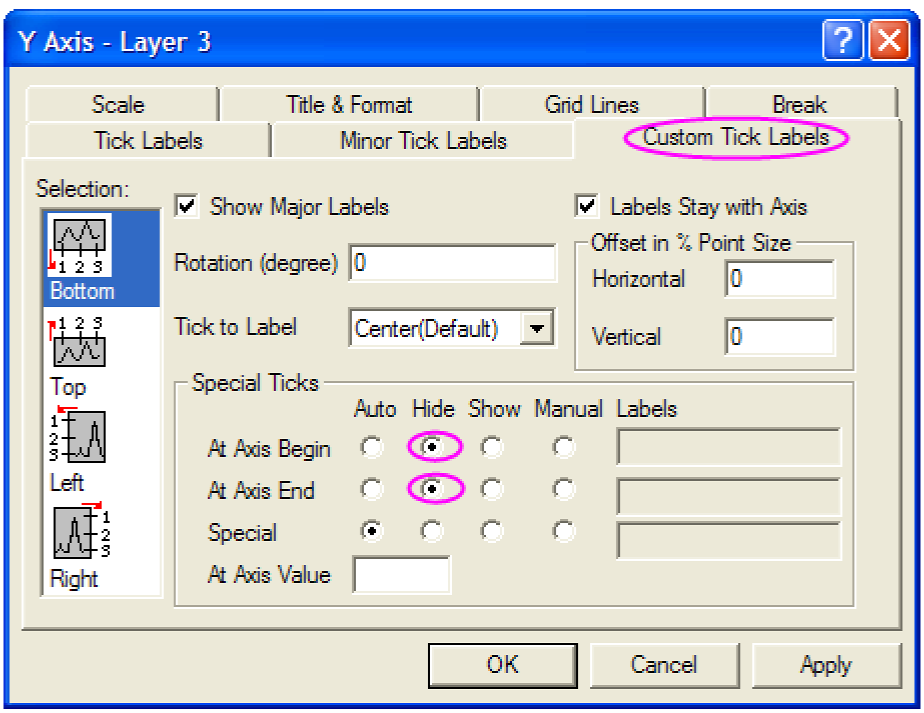
17. Activate Layer 2, repeat the steps from 9 to 12 and step 16.



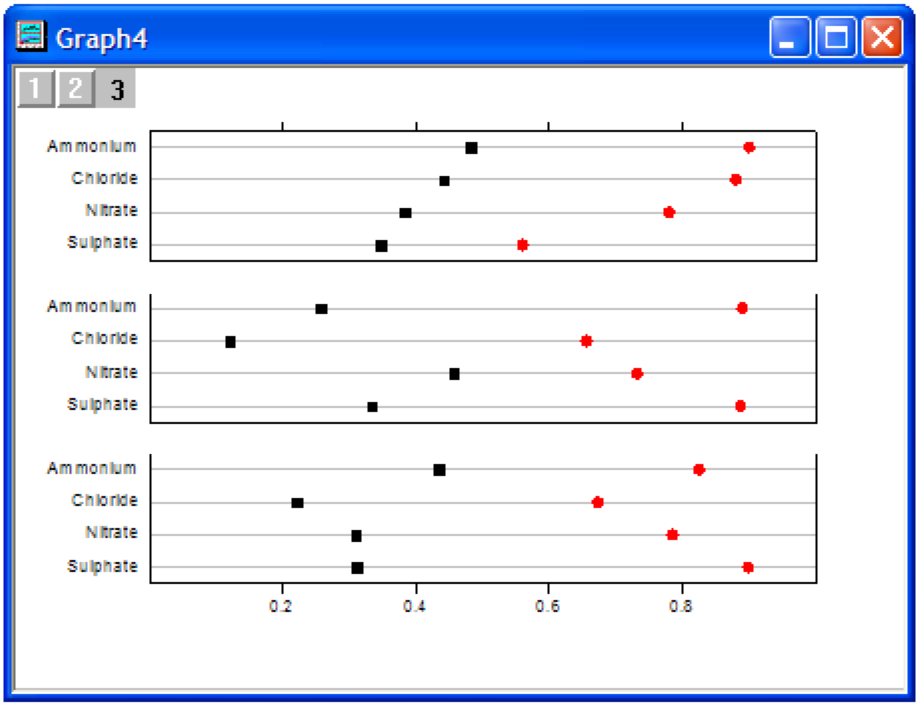
18. Activate Layer 3, also repeat steps from 9 to 12 and step 16 except step 10. In the **Title and Format** tab, set **Major Ticks** as **out** and **Minor Ticks** as **None**.



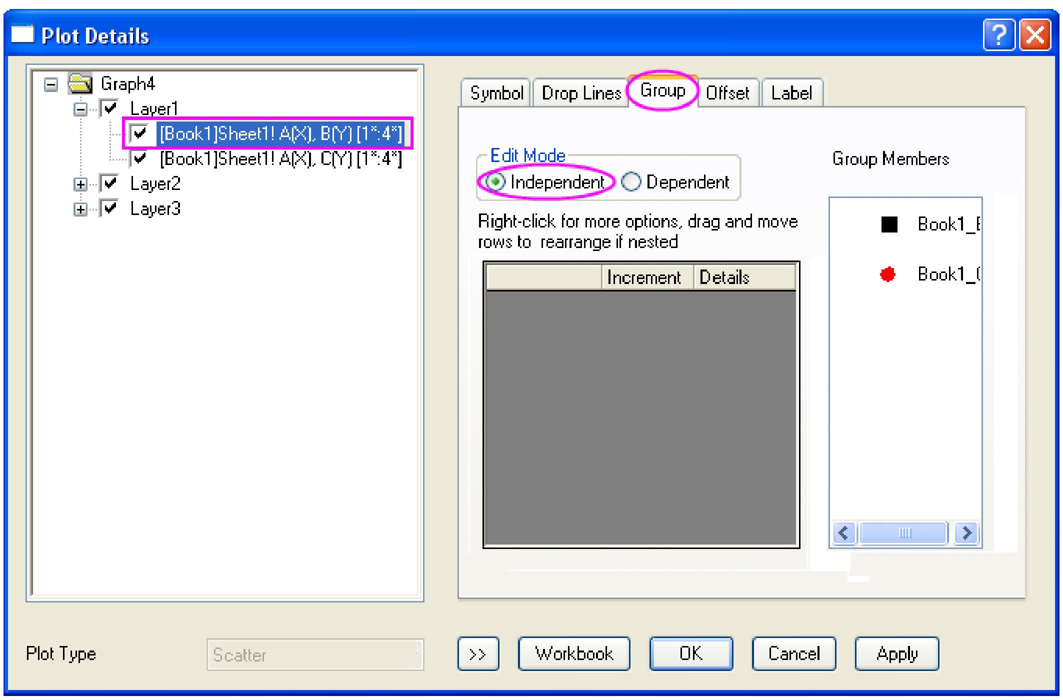
19. Select the **Custom Tick Labels** tab, highlight **Bottom** in the **Selection**, choose the **Hide** radio button both with **At Axis Begin** and **At Axis End**.

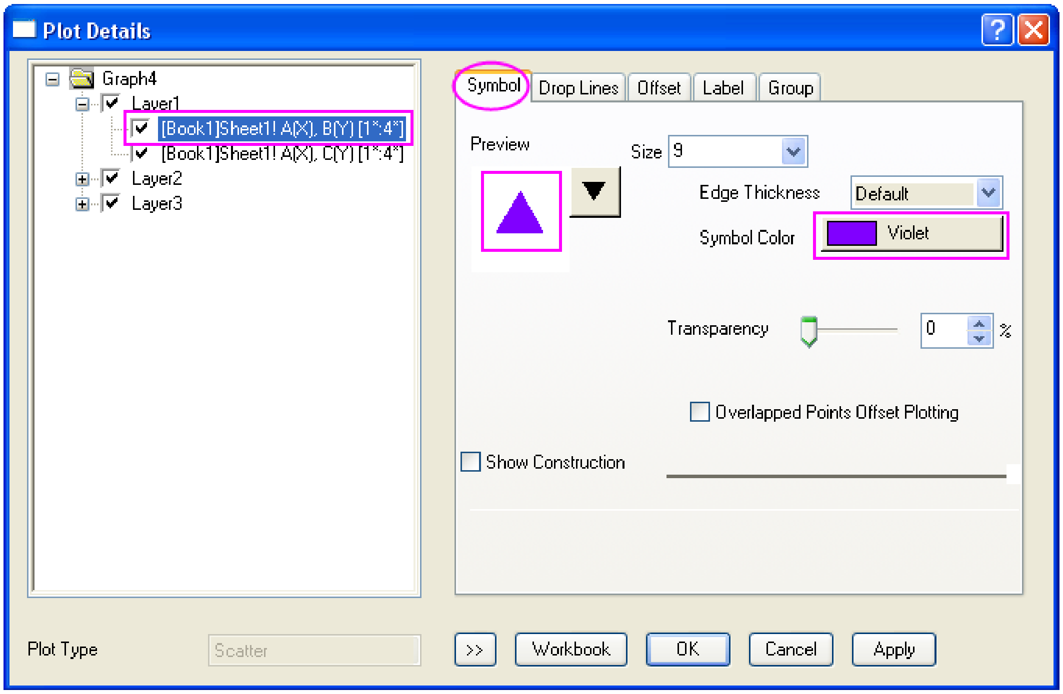


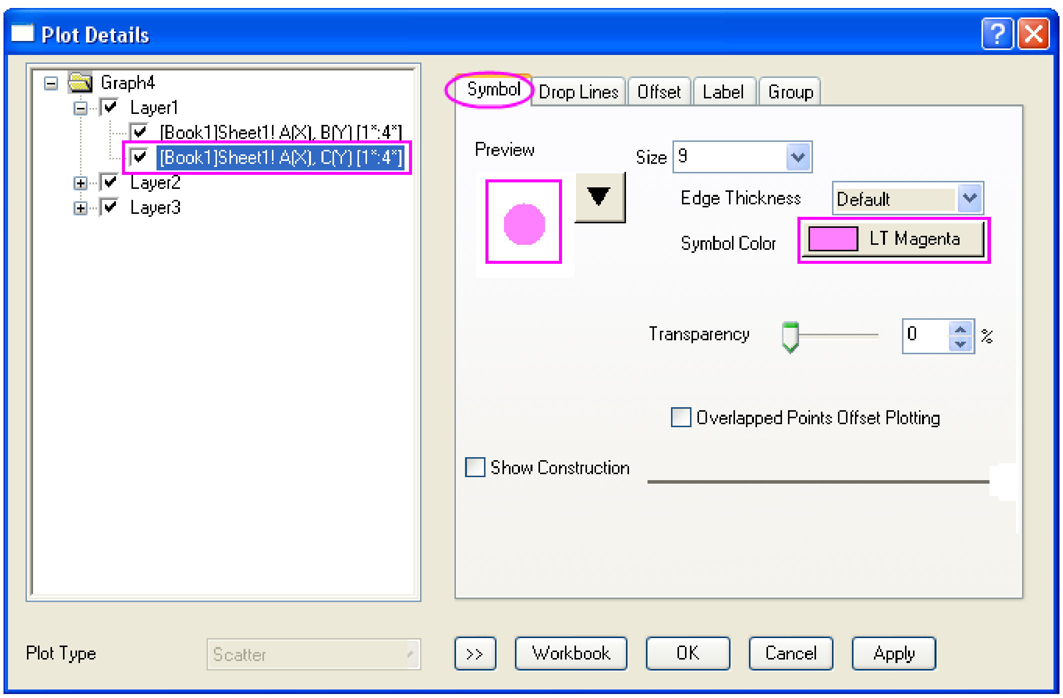
20. Click **OK**. Now you will see the graph below.



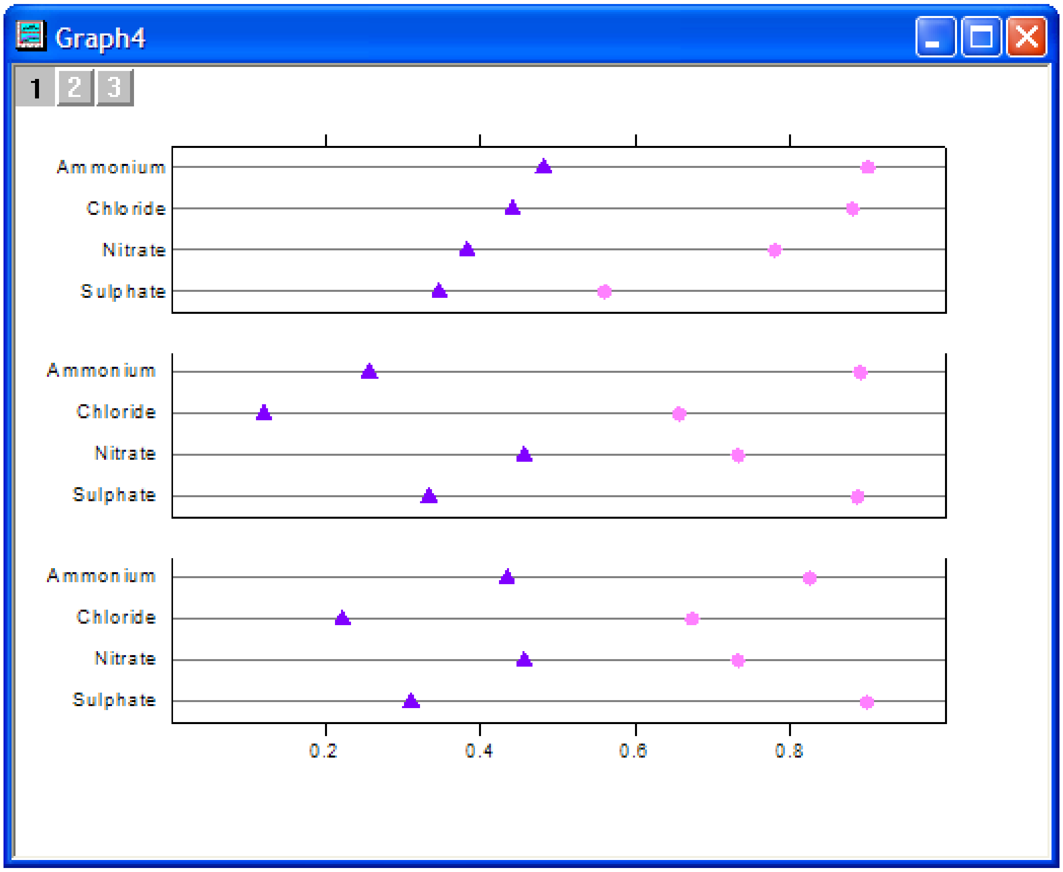
21. Double-click the graph to bring up the **Plot Details** dialog. Change the options as the following screenshot.

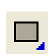


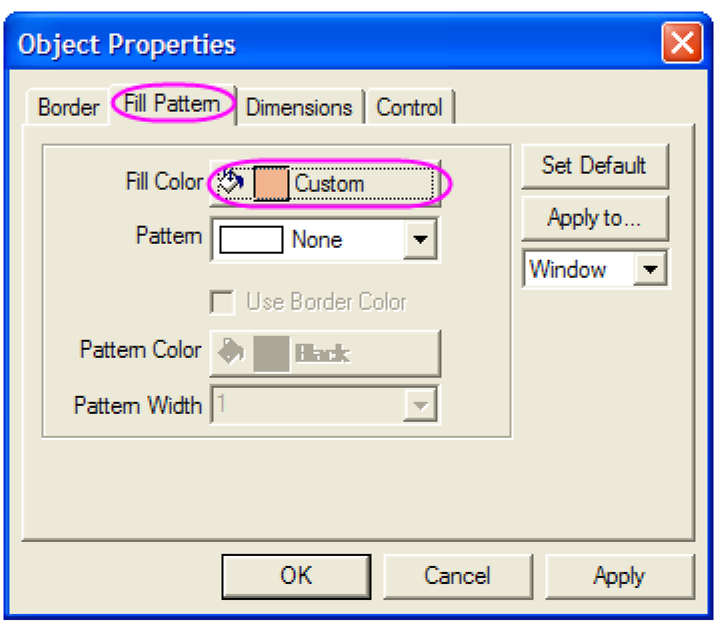




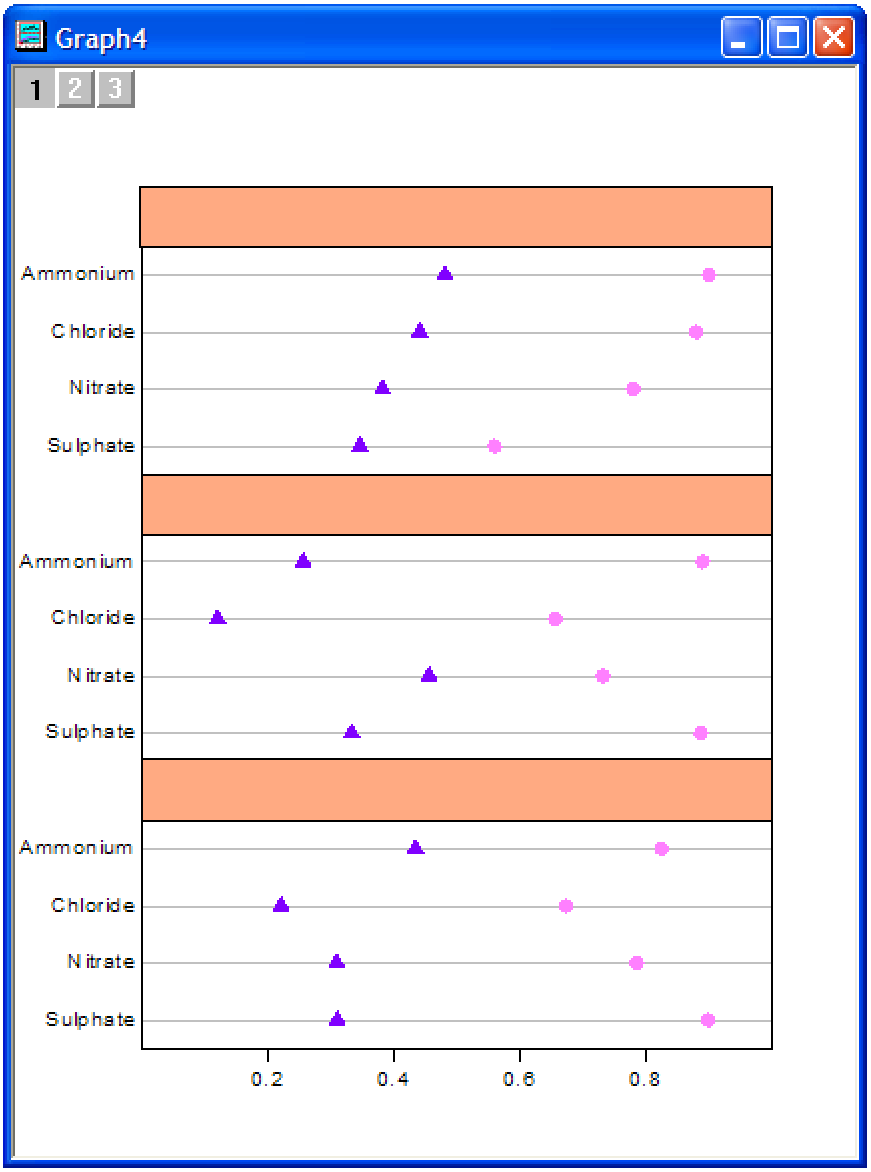
22. Repeat the steps of 21 for the Layer 2 and Layer 3. Click the **OK** button and you will see this graph.



23. Select  in the **Tools** toolbar to draw three rectangles on the graphs. For each rectangle:
    1. Double-click on the rectangle to bring up the **Object Properties** dialog.
    2. In the **Fill Pattern** tab, set the desired **Fill Color**.

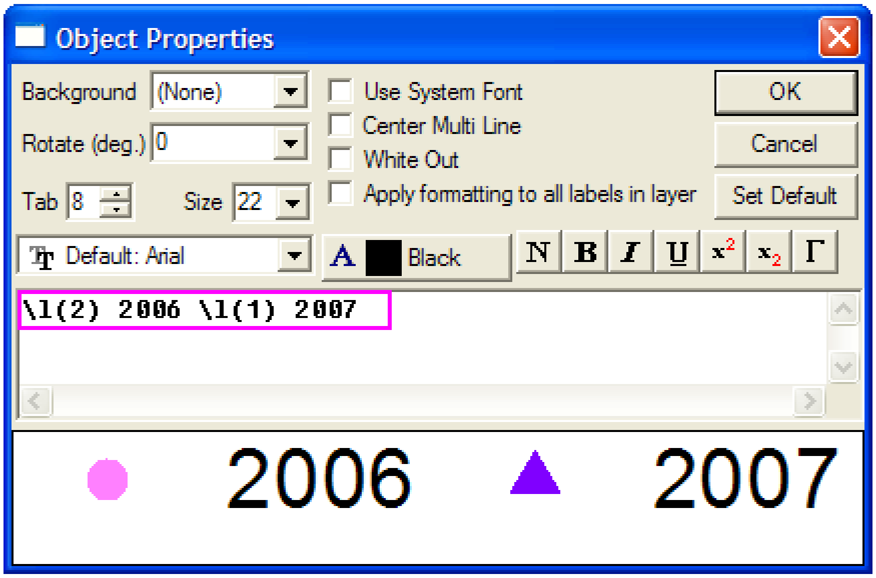


3. In the **Dimensions** tab, adjust the size and position of the rectangle if so desired.



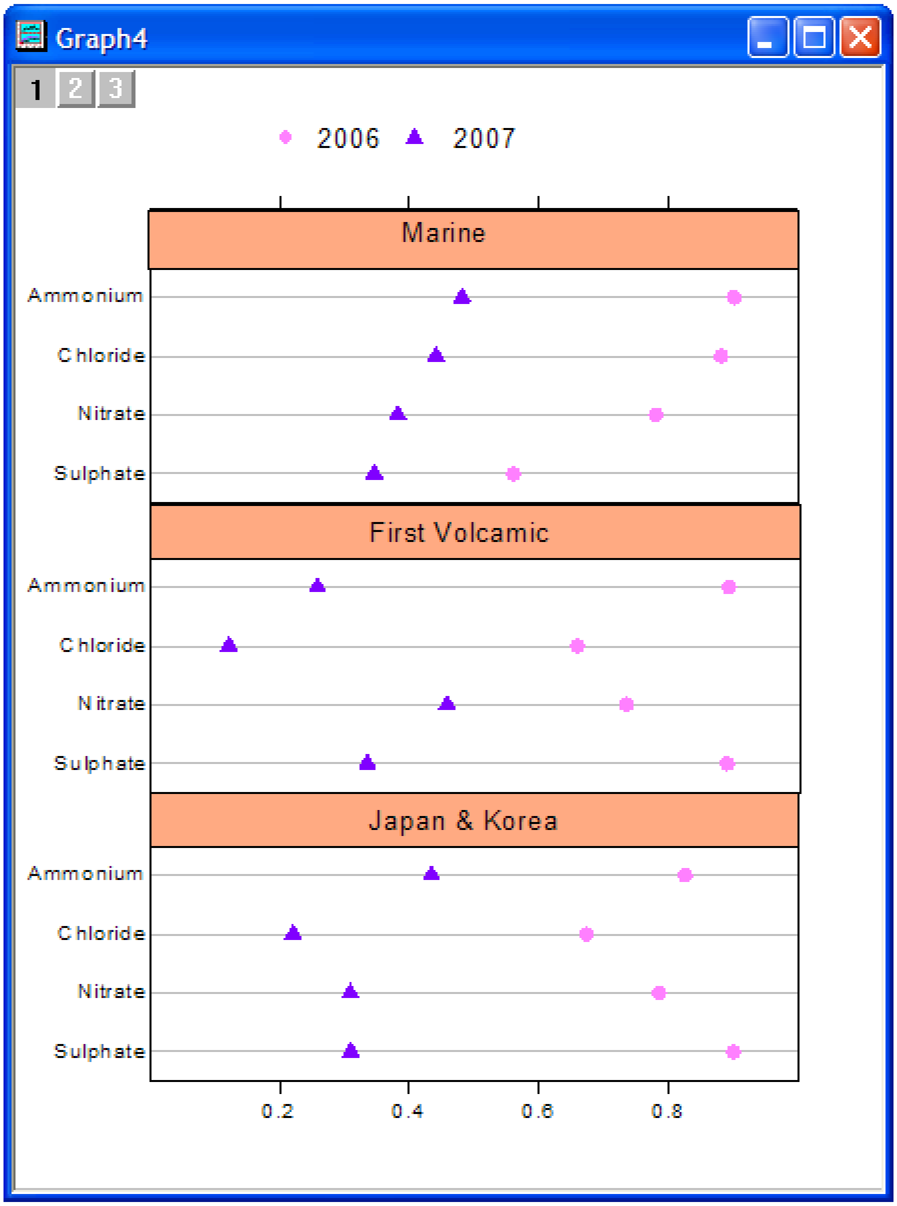
24. Select the Text Tool  in the Tools toolbar and click inside the rectangles to add the text that you want.

25. Select **Graph: New Legend** from the main menu. Move the legend to a suitable place, then right-click on it and select **Properties**. Change the settings as below:



26. Activate the top x axis of Layer 1 to move it to a suitable place.

Now the multi-data dot chart is finished. You can see the graph below.

1. 

5.6.4 Add multiple fitted curves in a Histogram

Summary

After you plot a Histogram, Origin allows you to overlay a distribution curve on the binned data by selecting **Normal**, **Lognormal**, **Poisson**, **Exponential**, **Laplace**, or **Lorentz** from the **Type** drop-down list in the Data tab of the Plot Details dialog. If you want to add multiple distribution curves to the Histogram, the procedure involves a few more steps.

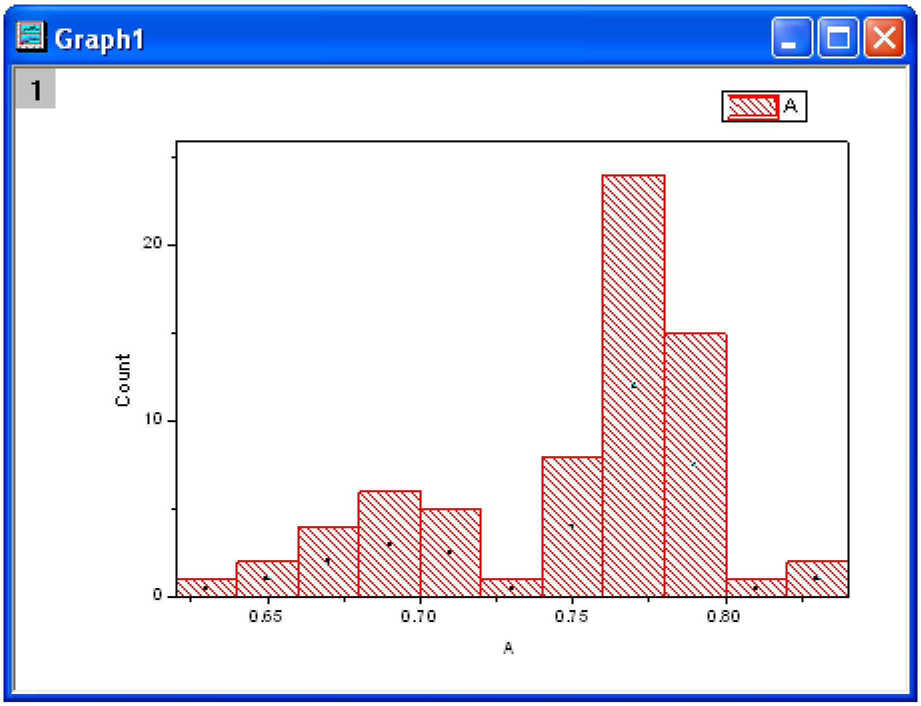
What you will learn

- Plotting the Histogram

- Using Frequency Count to do statistics
- Using Peak Analyzer to find peaks and do fitting
- Adding new layers

Steps

Copy and paste the sample data into Origin and set the column as Y (Highlight the column and choose **Column: Set As Y** from the Origin menu). Plot this data as a Histogram by clicking **Plot: Statistics: Histogram** from the menu.



Frequency Count

1. Highlight the sample data, then open the **Frequency Count** dialog by selecting **Statistics: Descriptive Statistcs: Frequency Count**.
2. Click **OK** to finish. A new result sheet will be generated.

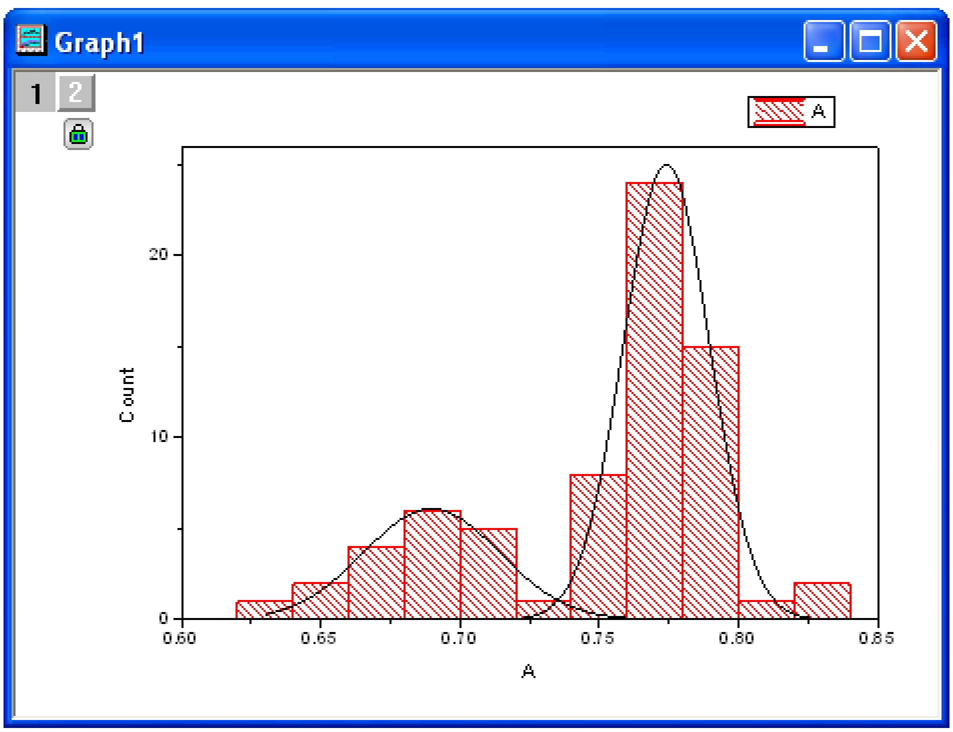
Fit peaks

1. Select Col(Counts) and then open the **Peak Analyzer** dialog from **Analysis: Peak and Baseline: Peak Analyzer**.
2. In the start page, select **Fit Peaks** as **Goal**, then click **Next**.
3. In the **Baseline Mode** page, set the baseline as **Y=0** when the **Custom** radio box is checked.
4. Click the **Next** button twice to go to the **Find Peaks** page. Click the **Find** button under the **Enable Auto Find** check box to find two peaks.
5. Click the **Next** button again to open the **Fit Peaks** page. Click **Fit Control** at the bottom of this page to open the **Peak Fit Parameters** dialog.

6. In this dialog, the default fitting function is **Gaussian**, which is the right function for normalizing the data. Close the **Peak Fit Parameters** dialog and go back to the **Peak Analyzer** dialog. Click **Finish** to complete the fitting.

Add the fitted curves

1. Activate the Histogram graph and add a layer by selecting **Graph: New Layer(Axes): Right-Y** from the main menu.
2. Right-click the **Layer2** icon and select **Plot Setup** from the short menu to open the **Plot Setup** dialog.
3. Select the sheet **FitPeakCurve1** from the top panel, then set col(A3) as X and col(A4) as Y, and add them into the Layer **RightY** in the bottom panel.
4. Do the same things for col(A5) and col(A6). After that, both fitted peaks have been added into the Layer **RightY**.
5. Click OK. Two fitted curves had been added to the Histogram.
6. Double-click the graph to open the **Plot Details** dialog. Select **RightY** from the left panel, then open the **Link Axes Scales** tab in the right panel and select **Straight(1 to 1)** for both **X Axis Link** and **Y axis Link**. Click OK to close the dialog.
7. The fitted curves are added into the Histogram with the proper scale. The following is the result graph, with the right Y-axis removed.



Sample Data

0.631
0.642

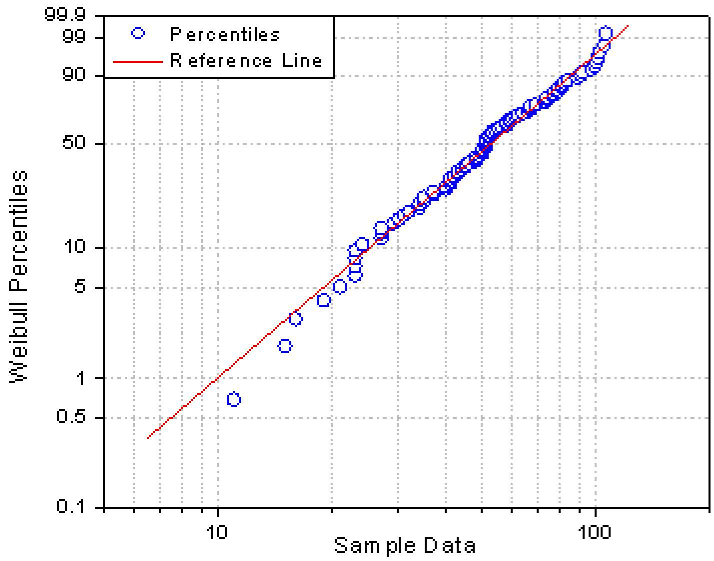
0.652
0.662
0.669
0.676
0.677
0.69
0.691
0.696
0.697
0.699
0.699
0.7
0.7
0.708
0.712
0.718
0.731
0.744
0.749
0.751
0.752
0.753
0.758
0.758
0.759
0.761
0.761
0.763
0.763
0.763
0.765
0.767
0.768
0.768
0.769
0.769
0.77
0.771
0.771
0.772
0.774
0.775
0.775

0.776
0.776
0.776
0.777
0.778
0.779
0.78
0.78
0.781
0.784
0.784
0.785
0.785
0.789
0.789
0.791
0.794
0.795
0.796
0.798
0.798
0.803
0.82
0.831

5.6.5 Weibull Probability Plot

Summary

The probability plot shows a graph with observed cumulative percentage on X axis and expected cumulative percentage on Y axis. The Weibull probability plot is used to test whether or not a dataset follows Weibull distribution. And its X scale type and Y scale type are Log10 and Double Log Reciprocal respectively. If all the scatter points are close to the reference line, we can say that the dataset follows the Weibull distribution.



Minimum Origin Version Required: Origin 8.1 SR0

What you will learn

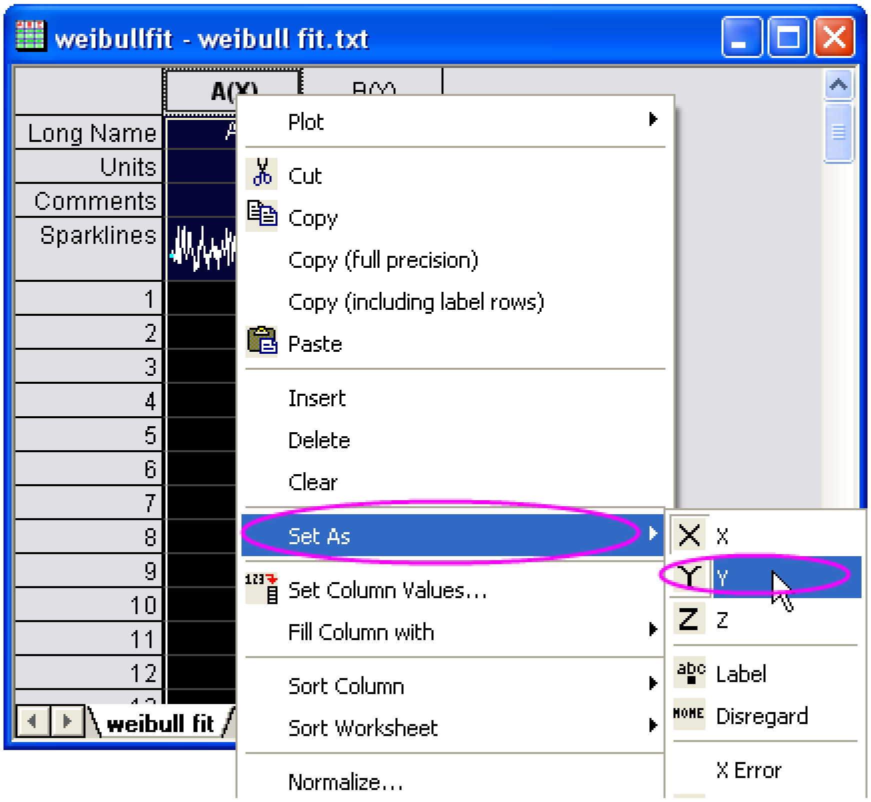
- How to create a Weibull probability plot
- How to change the shape of the symbol
- How to change the width of the line

Steps

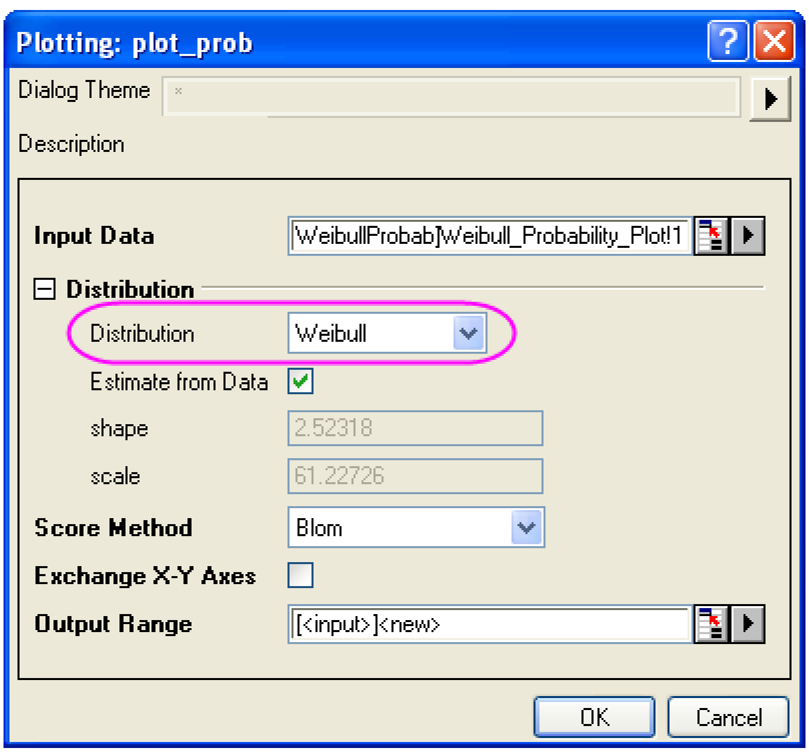
This tutorial is associated with the Statistical and Specialized Graphs project: *\Samples\Statistical and Specialized Graphs.opj*. (If you don't have the OPJ, you can also download the data from here)

1. Open the project, browse to the folder *Statistical and Specialized Graphs: Statistical: Probability, QQ Plot* in the **Project Explorer**. Active the worksheet *Probability Plot Data*. Highlight column A

and right-click to select **Set As: Y** from the context menu.



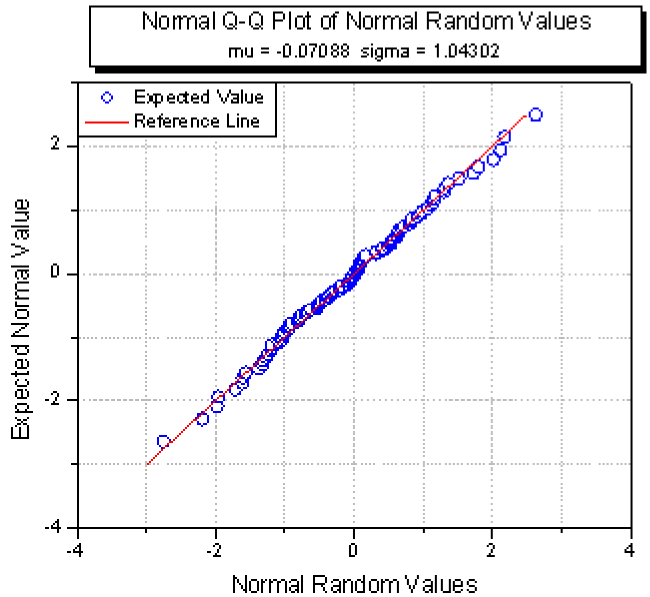
2. Keep this column highlighted, select **Plot: Statistics: Probability Plot** from the Origin main menu to open the **plot_prob** dialog. In this dialog, change **Distribution** to **Weibull** and then click the **OK** button to create a Weibull probability plot with column A.



5.6.6 Q-Q Plot

Summary

Q-Q plot is a graphic method to test whether or not a dataset follows a given distribution. In a Q-Q plot, observed values on X axis and expected values on Y axis are shown. If all the scatter points are close to the reference line, we can conclude that the dataset follows the given distribution.



Minimum Origin Version Required: Origin 8.1 SR0

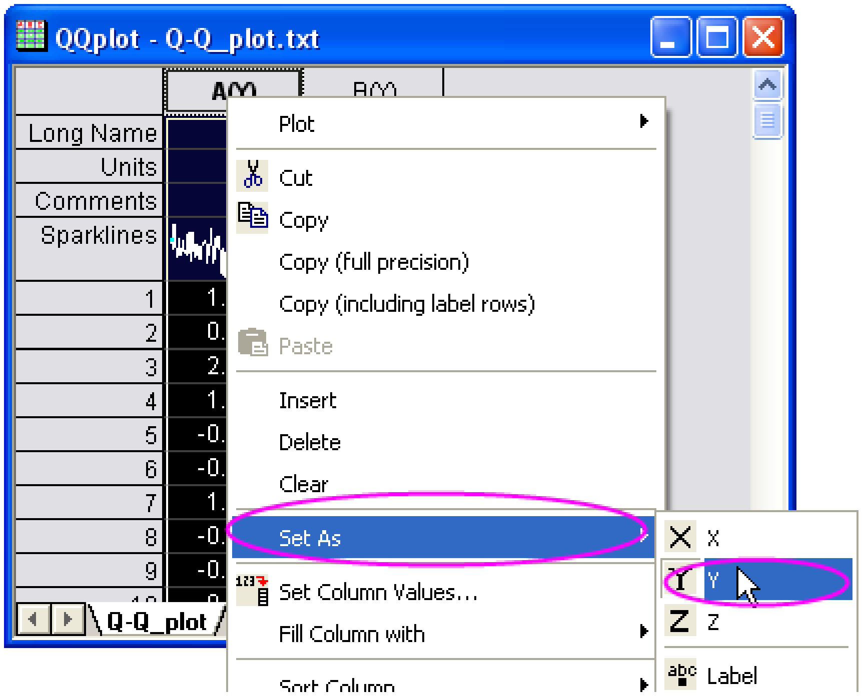
What you will learn

- How to create a normal Q-Q plot
- How to change the color and shape of the symbol
- How to change the color of the line

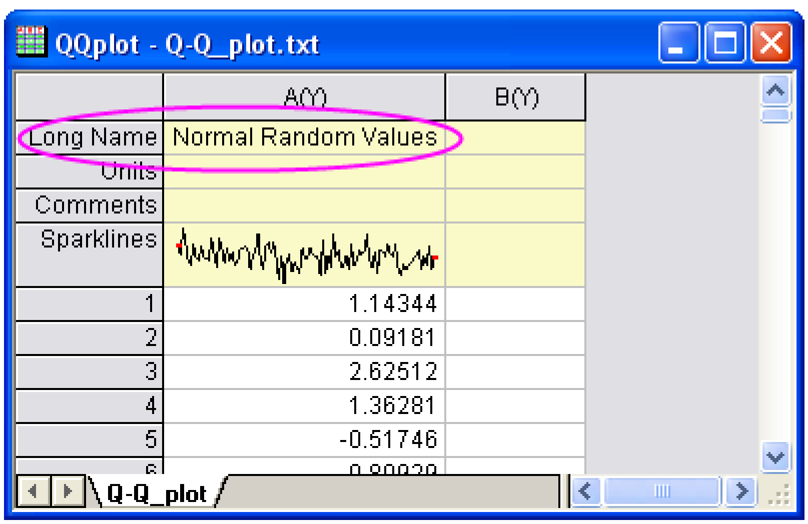
Steps

This tutorial is associated with the Statistical and Specialized Graphs project: *\Samples\Statistical and Specialized Graphs.opj*. (If you don't have the OPJ, you can also download the data from here)

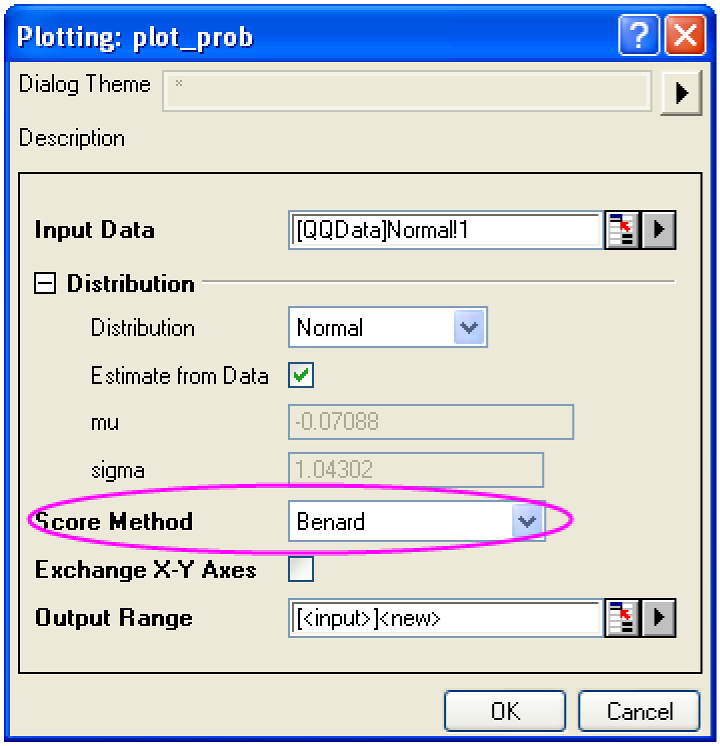
1. Open the Statistical and Specialized Graphs project, open the folder *Statistical and Specialized Graphs: Statistical: Probability, QQ Plot* in the **Project Explorer**. Activate worksheet *Q-Q Plot Data*, highlight column A and right-click to select **Set As: Y** from the context menu.



2. Add a long name to this column, say **Normal Random Values**.



3. Highlight this column and select **Plot: Statistics: Q-Q Plot** from the Origin main menu to open the **plot_prob** dialog. In this dialog, change **Score Method** to **Benard**.

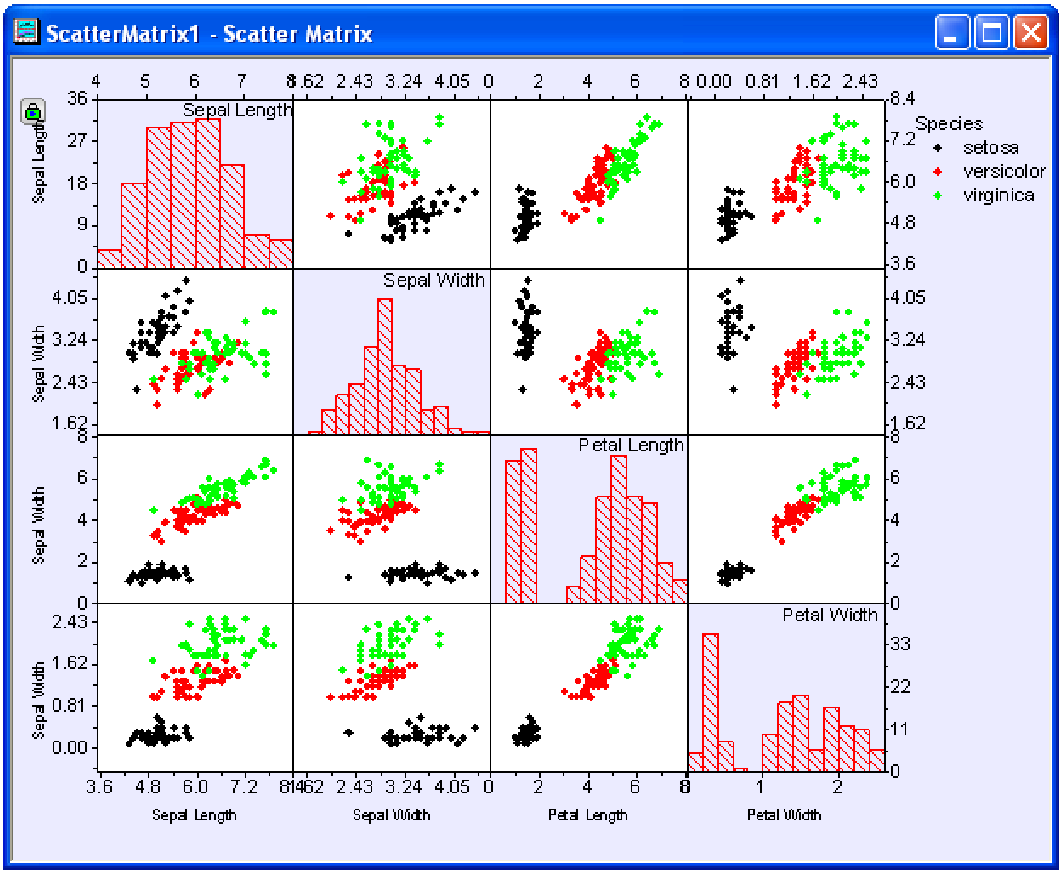


4. Click the **OK** button to create a normal Q-Q plot with column A.

5.6.7 Scatter Matrix

Summary

A scatter matrix is consists of several pair-wise scatter plots of variables presented in a matrix format. It can be used to determine whether the variables are correlated and whether the correlation is positive or negative. This tutorial will show you how to create a Scatter Matrix plot.



Minimum Origin Version Required: Origin 9.0 SR0

What you will learn

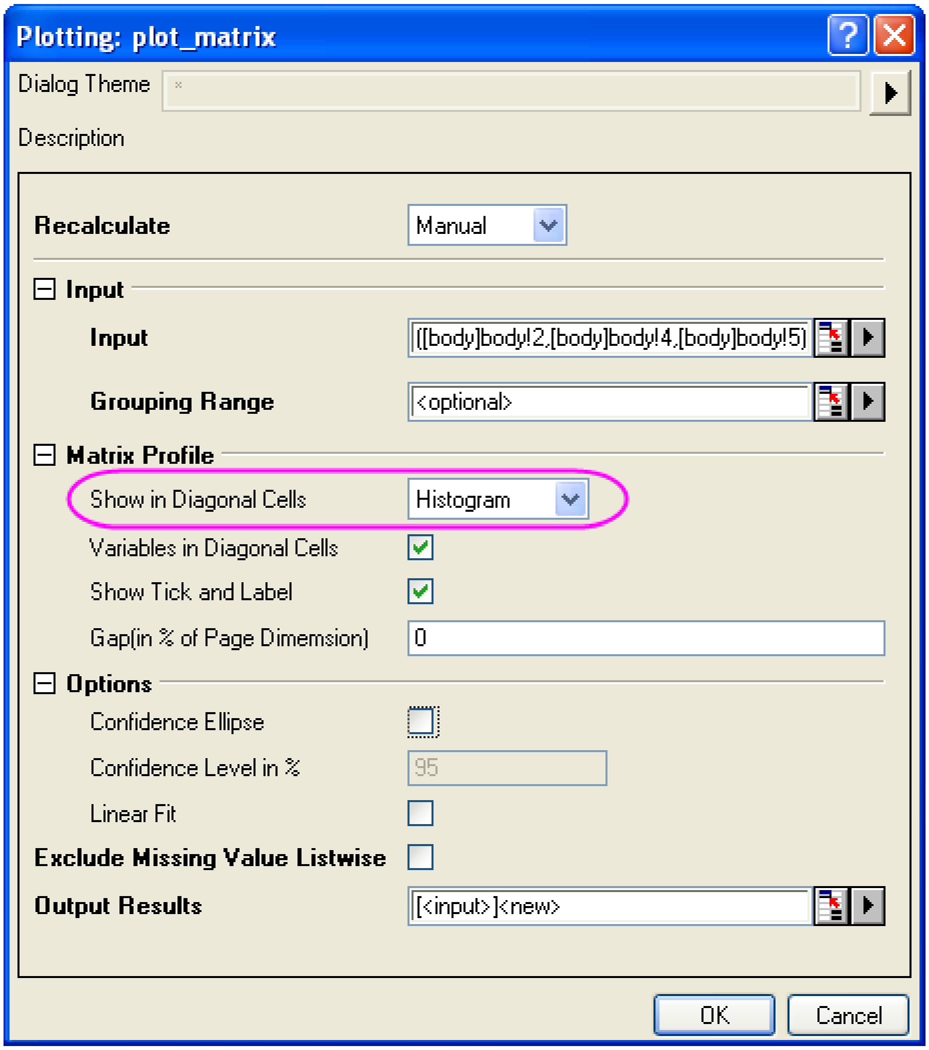
- How to create a Scatter Matrix plot with histogram
- How to customize Scatter Matrix plot
- How to set grouping range for showing color index

Steps

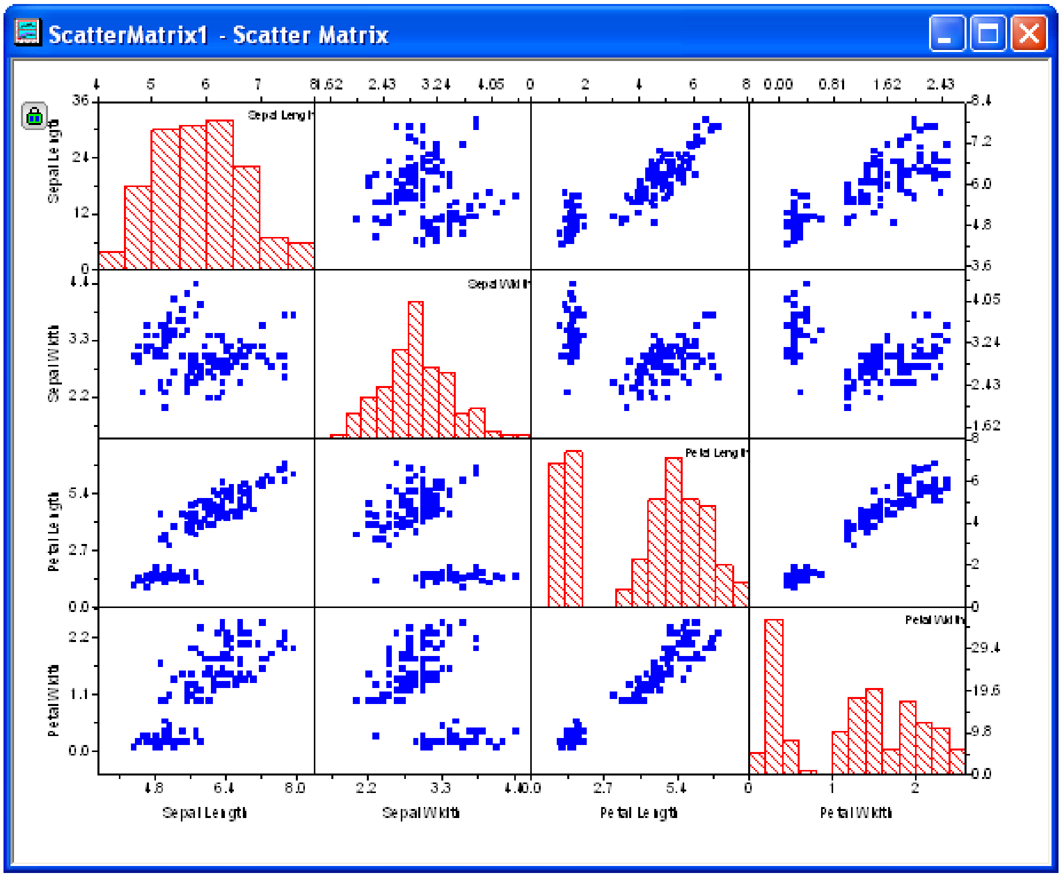
Creating Scatter Matrix plot

1. Start with an empty worksheet, select **File: Import: Single ASCII...** to open the Import Single ASCII dialog, browse to the *\Samples\Statistics* subfolder of the Origin program folder, and import the file *Fisher's Iris Data.dat*.
2. Highlight columns (A)~(D), and then select **Plot: Statistics: Scatter Matrix** from the Origin menu.

3. In the dialog, select **Histogram** in the **Show in Diagonal Cells** drop-down list.



4. Click **OK** to close the dialog. The graph should look like the following:

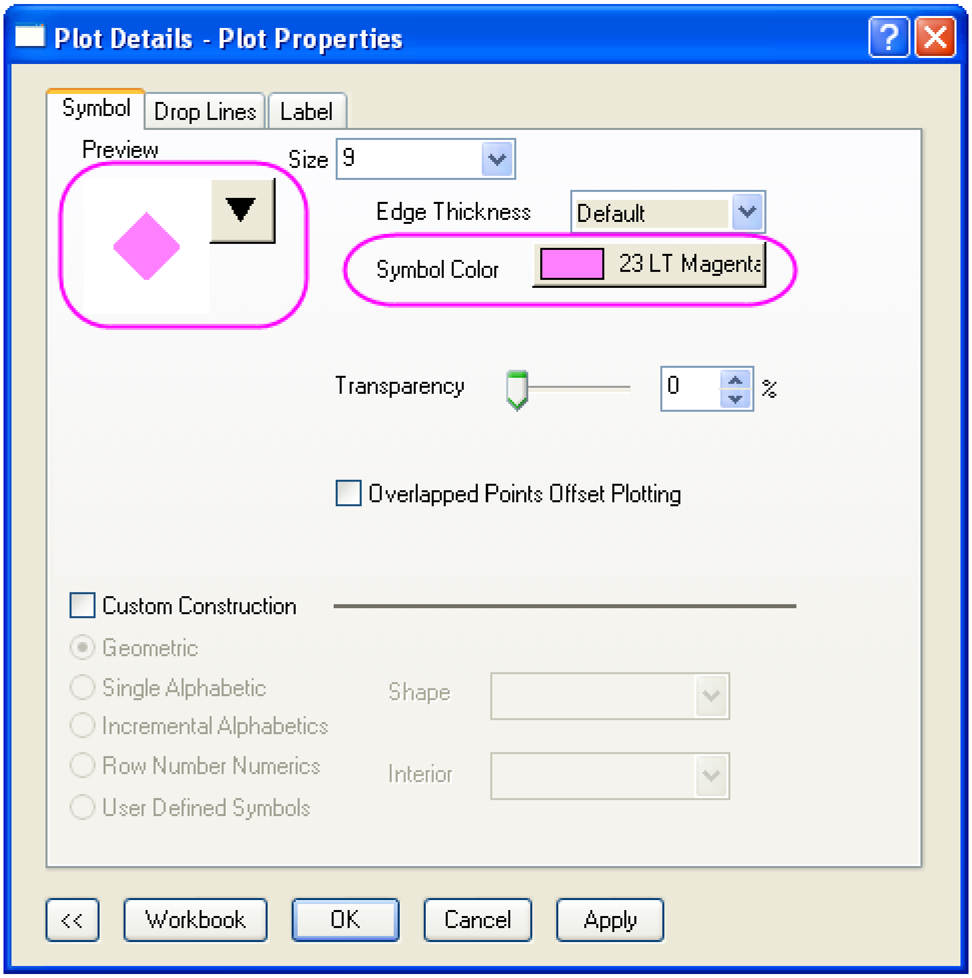


And the **PlotData1** sheet for the scatter matrix plot is generated in the same workbook.

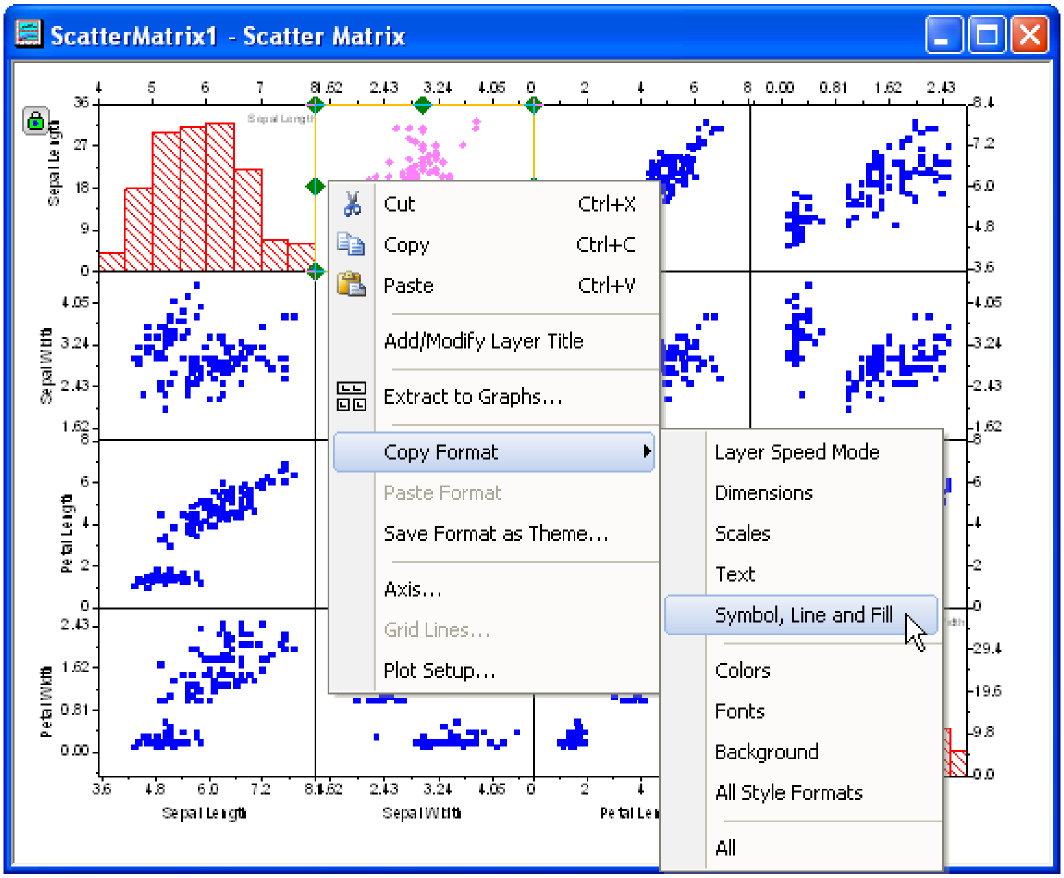
Customizing Scatter Matrix plot

There are multiple layers in the Scatter Matrix graph. This section will show you how to customize the background color, the type and color of a data plot and the tick label of axis of the scatter matrix.

1. Double click on a layer except in the diagonal cells to open the **Plot Details** dialog. Specify the type and color of the symbol as shown in the following image, and click **OK**.



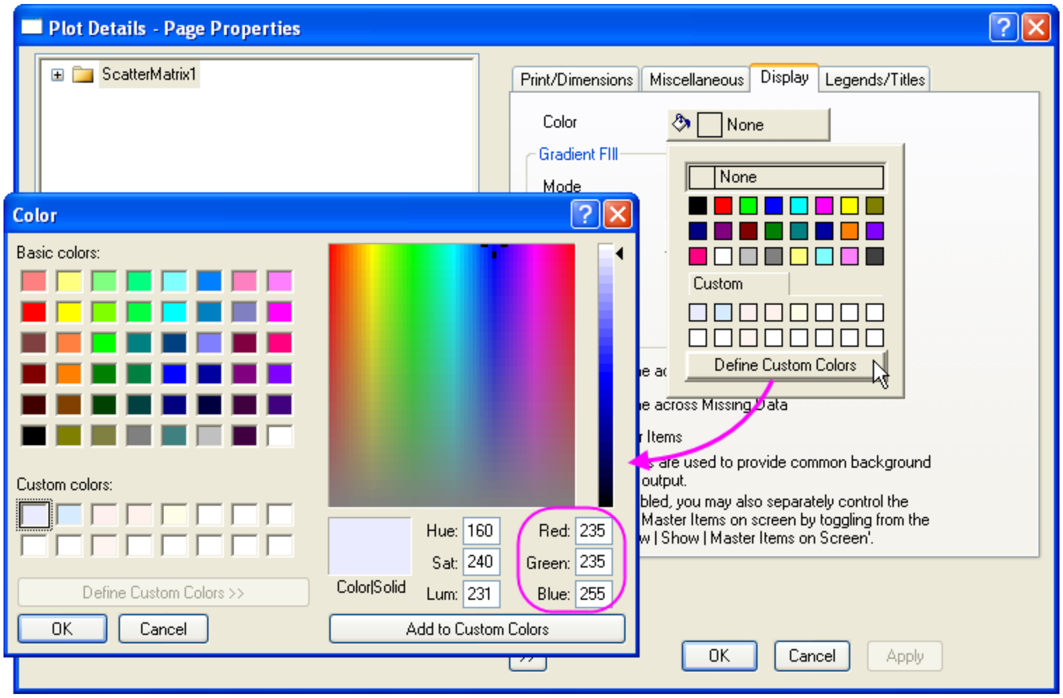
2. Click to select the layer to be updated, right-click on it and select **Copy format: Symbol, Line and Fill**.



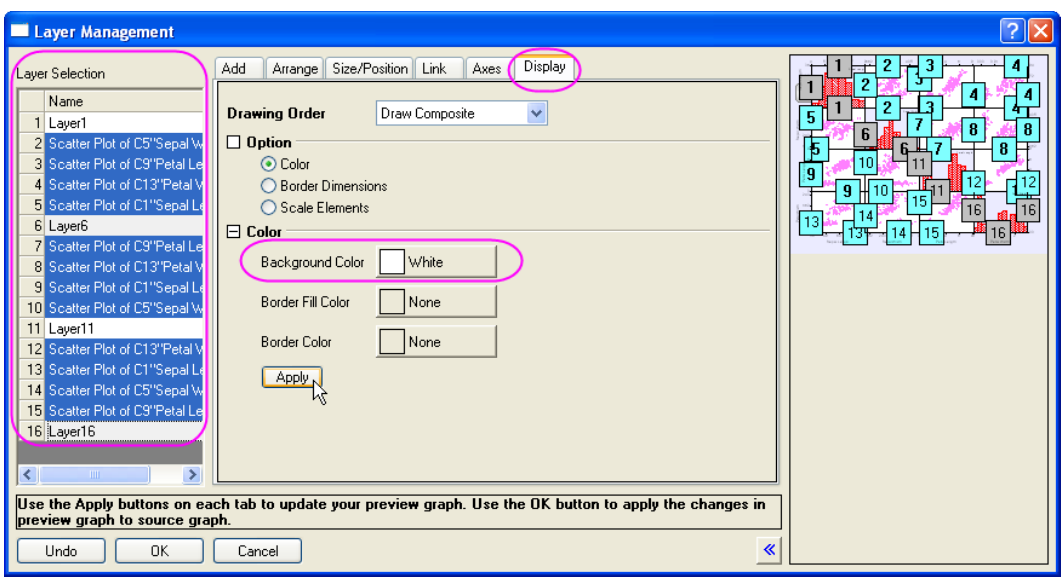
Then click on the white space of the graph, right-click, and select **Paste Format**. You can use the same method to copy the symbol color to other layers.

3. Right-click on that layer again, and select **Copy Format: Colors**. Then click on the white space of the graph, right-click, and select **Paste Format**.
4. Select **Format: Page Properties** to open the **Plot Details** dialog. Go to the **Display** tab, click on the **Define Custom Colors** button for the Color option.

5. In the **Color** dialog, specify the color as Red=235, Green=235, Blue=255. Then add as custom color and click **OK**.



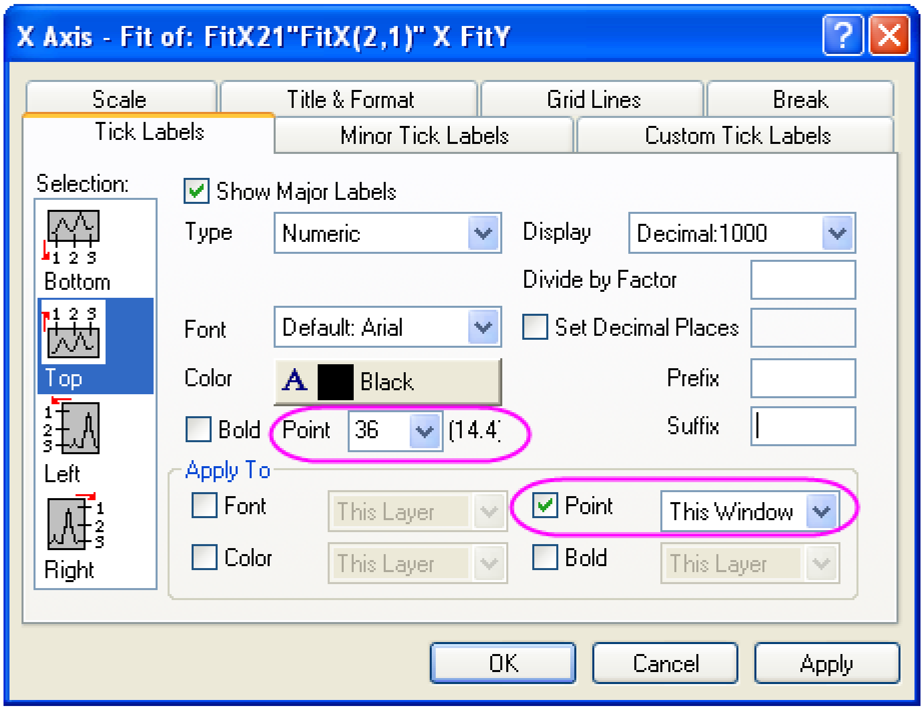
6. To specify background color of the layers with scatter plots, select the **Graph: Layer Management** to open the dialog, and go to the **Display** tab. Highlight all the layers listed in **Layer Selection** except the diagonal cells, and then specify the **Background Color** as **White**. Click on the **Apply** button to preview the change in the right panel of the dialog.



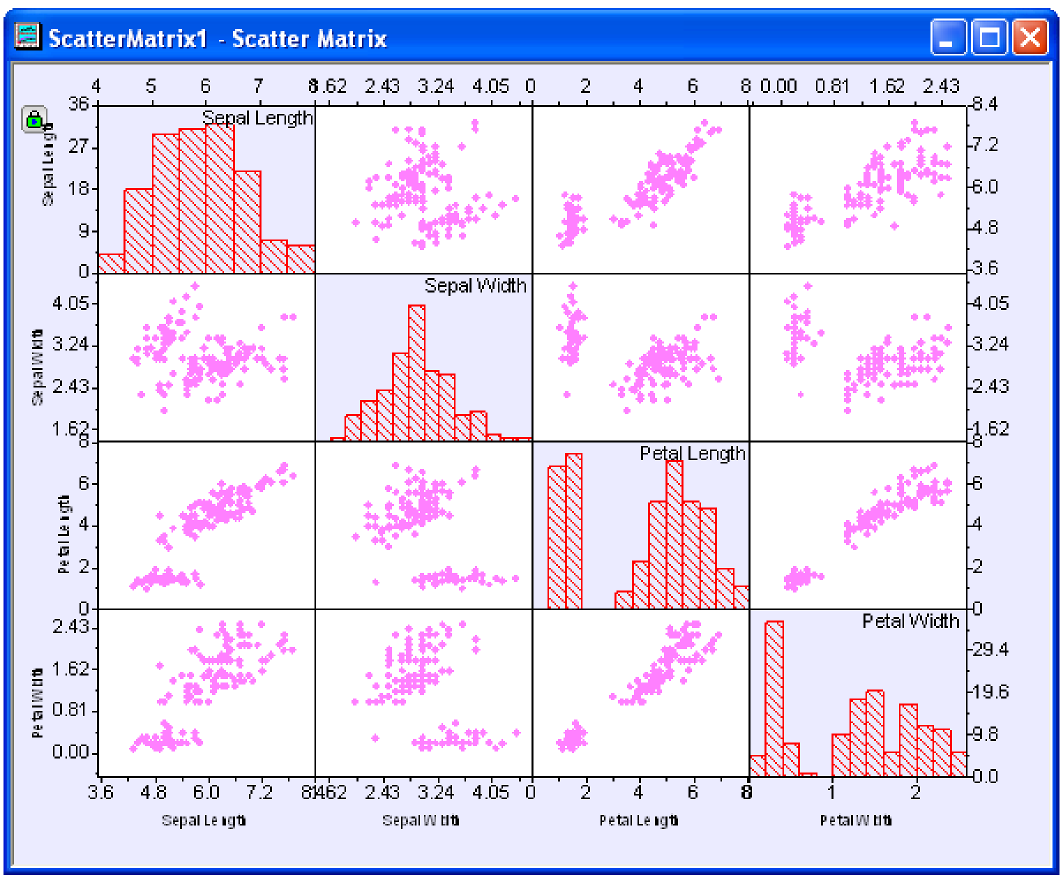
Then click **OK** in the dialog.

7. Double click on a tick label in the graph to open the **Axis** dialog. In the **Tick Labels** tab, specify **Point** to **36**. Then select the **Point** checkbox in the **Apply To** section, and choose **This Window**

in the drop-down list.

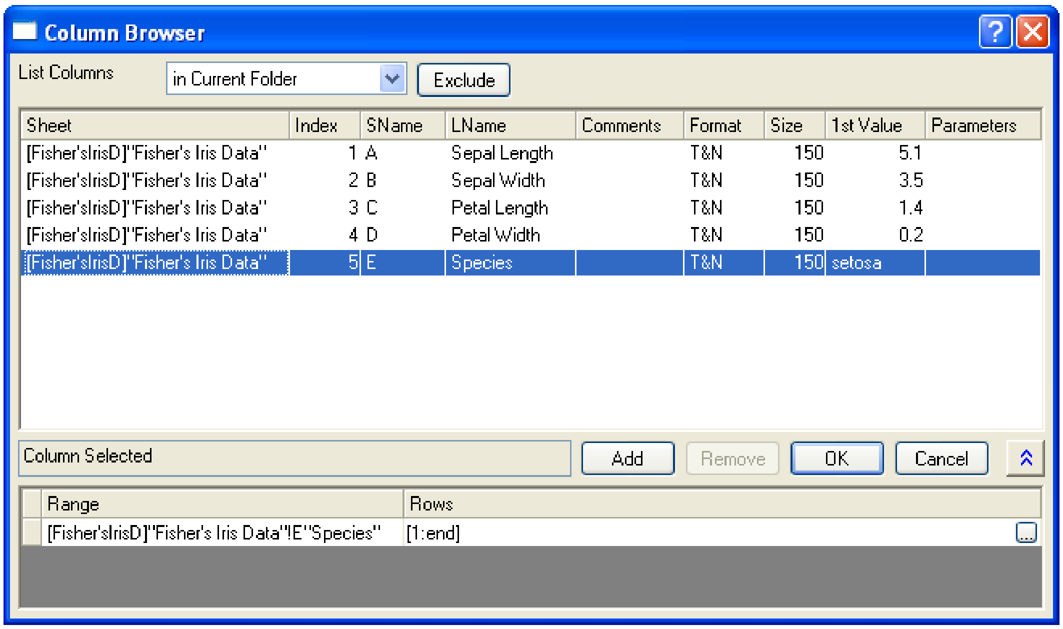


8. Highlight the variables in the diagonal cells, and use the **Size** button  in the **Format** toolbar to set the size to **36**. Then the graph will look like the following:



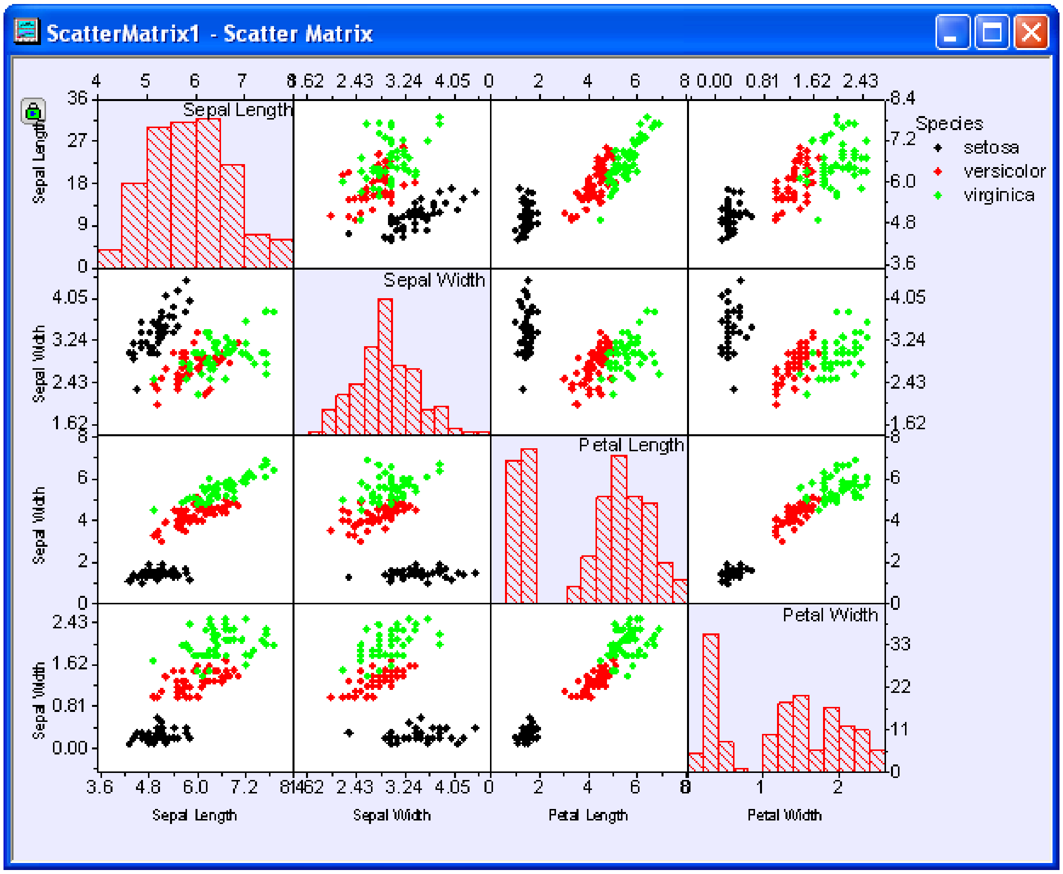
Add Grouping Range

1. To add a **Grouping Range**, click on the green lock icon on the upper-left corner. And then select **Change Parameters** to bring back the **Plotting: plot_matrix** dialog.
2. Click the triangle button next to the **Grouping Range** option. Click on the **Select Columns** to open the **Column Browser** dialog, and then choose column E (Species) as the group range. Click **OK**.



3. Click the **OK** button in the **Plotting: plot_matrix** dialog.

Your final graph should look like this:



5.7 Polar

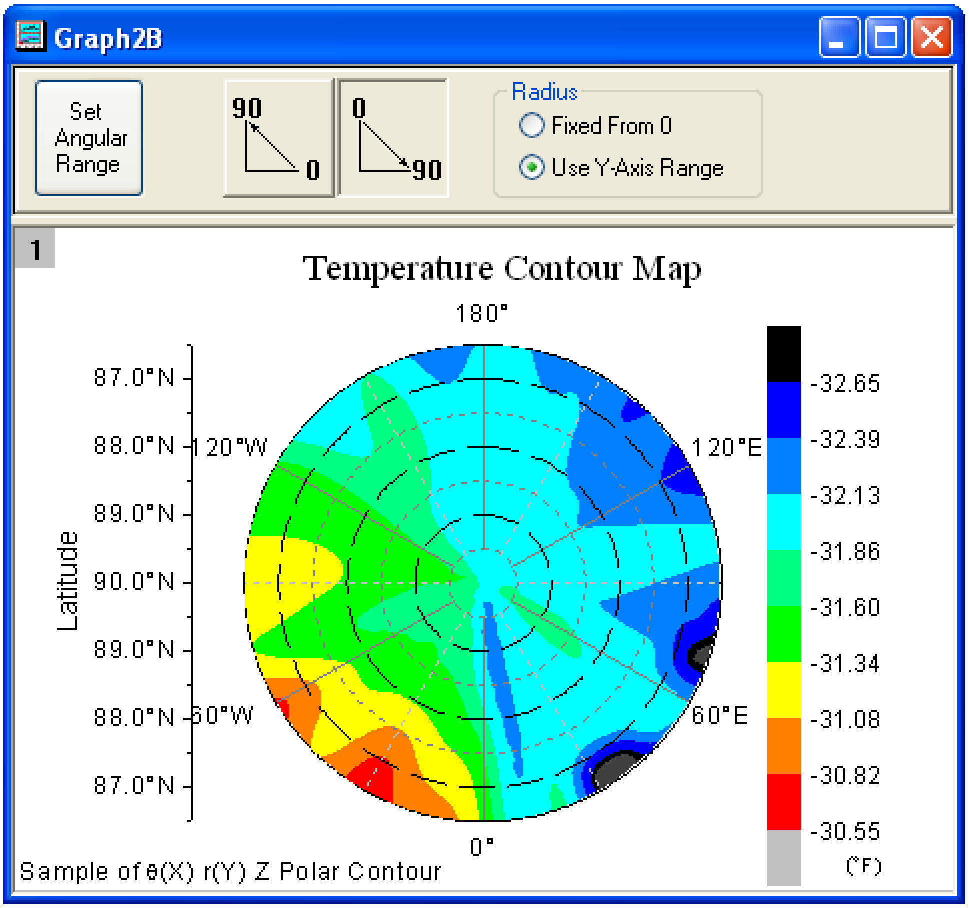
Topics covered in this section:

1. Polar Contour
2. Polar Plot with Error Bar

5.7.1 Polar Contour

Summary

This tutorial will show you how to create a Polar Contour graph.



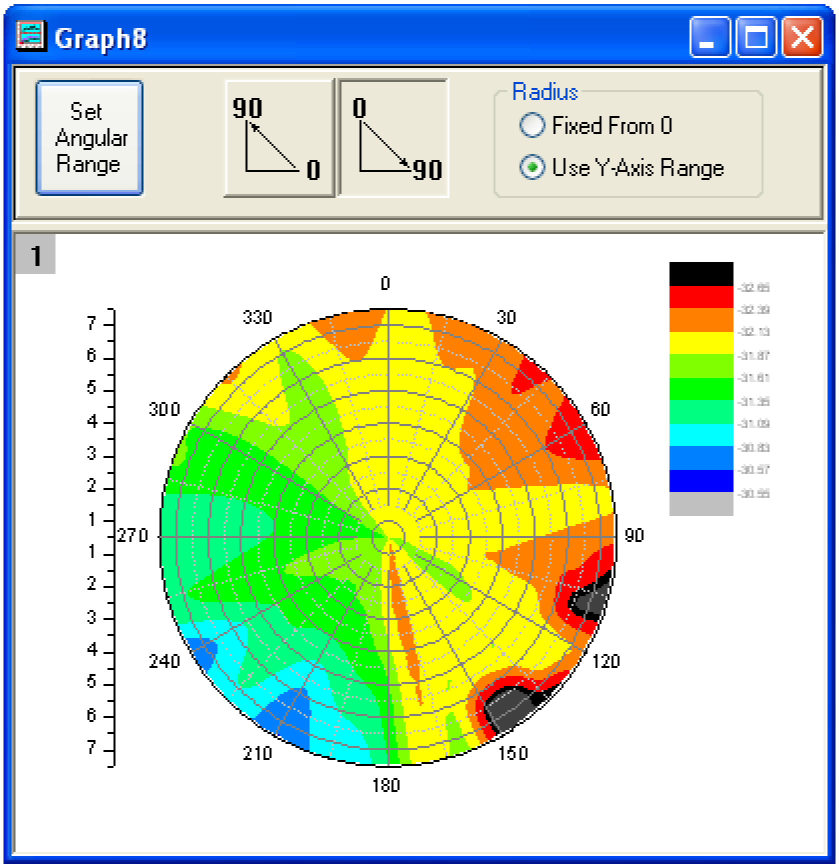
What will you learn

- Create a Polar Contour graph
- Customize the graph by using the Plot Details dialog

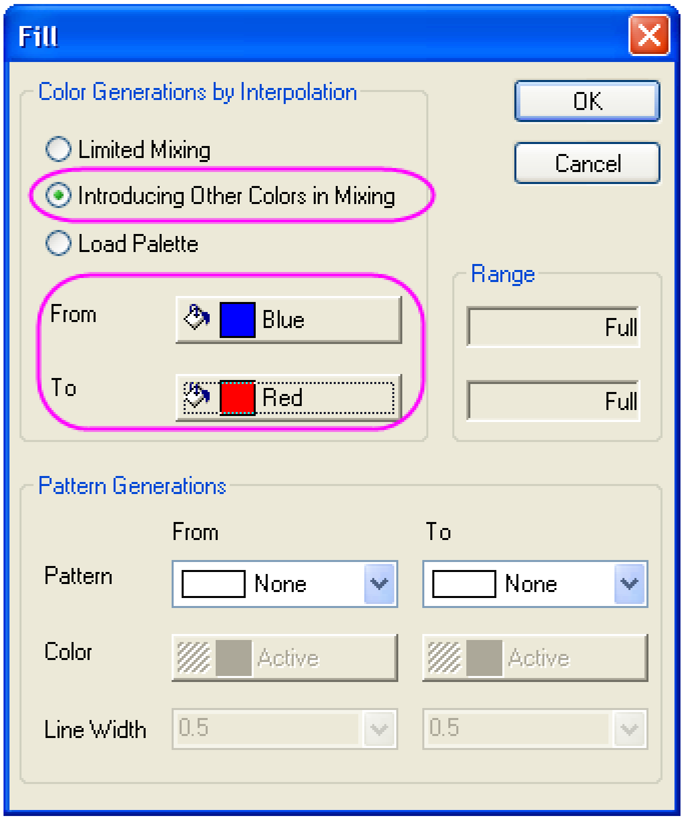
Steps

This tutorial is associated with the 2D and Contour Graphs project: *\Samples\2D and Contour Graphs.opj*.

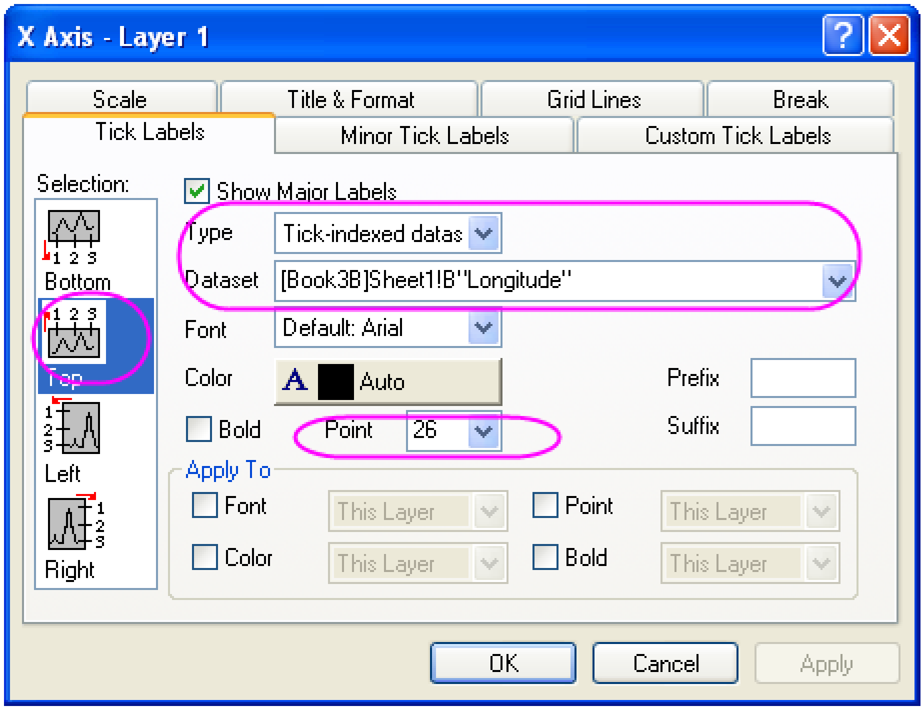
1. Open the *2D and Contour Graphs: Contour: Polar Contour* folder in the **Project Explorer**. Activate **Book2B** with the temperature and location data and select column **C**. Select **Plot: Contour: Polar Contour Theta(X) r(Y)** from the menu to create a polar contour graph. The graph should look like:



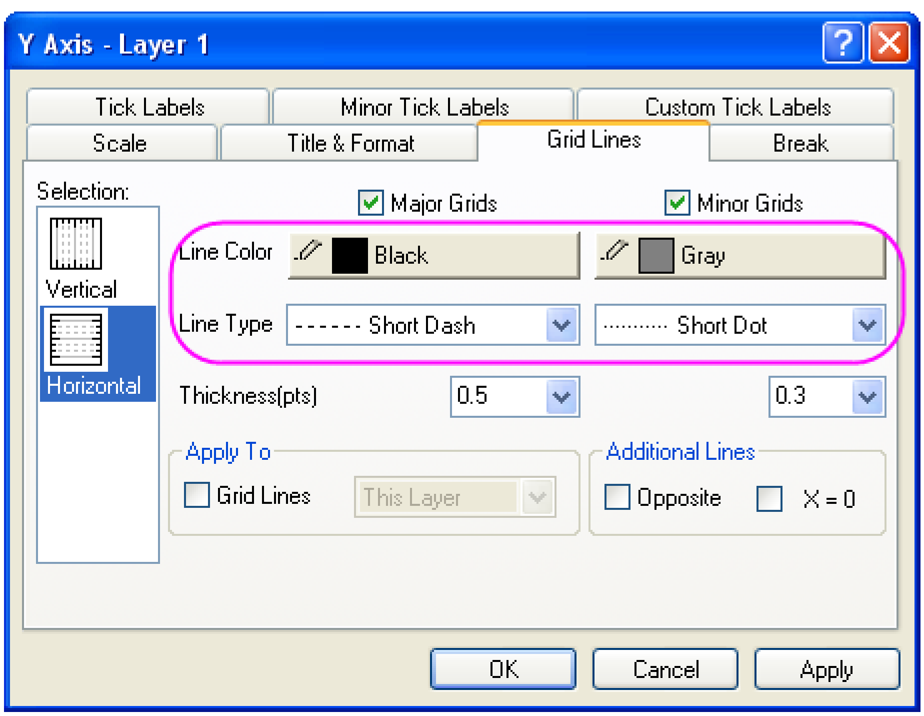
2. Then we wil customize the contour graph by using the **Plot Details** dialog. Double-click on the contour plot to bring up Plot Details dialog, select the **Color Map/Contours** tab and click the **Fill** heading, and then set the dialog as the follwing graph shows.



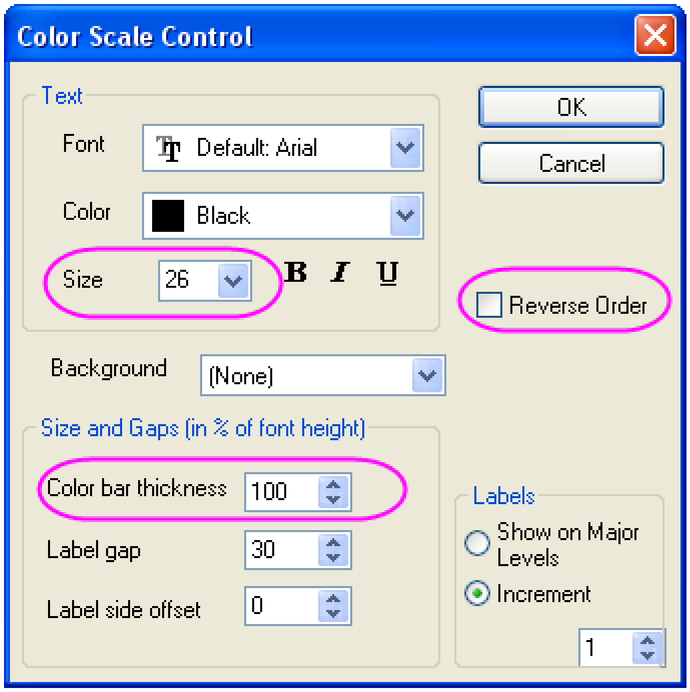
3. Click the **Contouring Info** Tab, select the **Layer Boundary** radio button.
4. Click OK button to close the **Plot Details** dialog.
5. Then we will customize the axes of the graph. Double-click on any tick label to open the **Axis** dialog.Then do the following things
    - Select the **Scale** tab, choose **Horizontal** in the **Selection** list, set **Increment** to 60.
    - Select **Vertical** in the **Selection** list, set **From** to **0**, set **To** to **7** and set **Increment** to **2**.
    - Select **Tick Labels** tab and select **Top** in the **Selection** list. Then select **Tick-Indexed Dataset** with the **Type** drop-down list. Choose **[Book3B]Sheet1!B** for **Dataset**. Change **Point** to **26**.



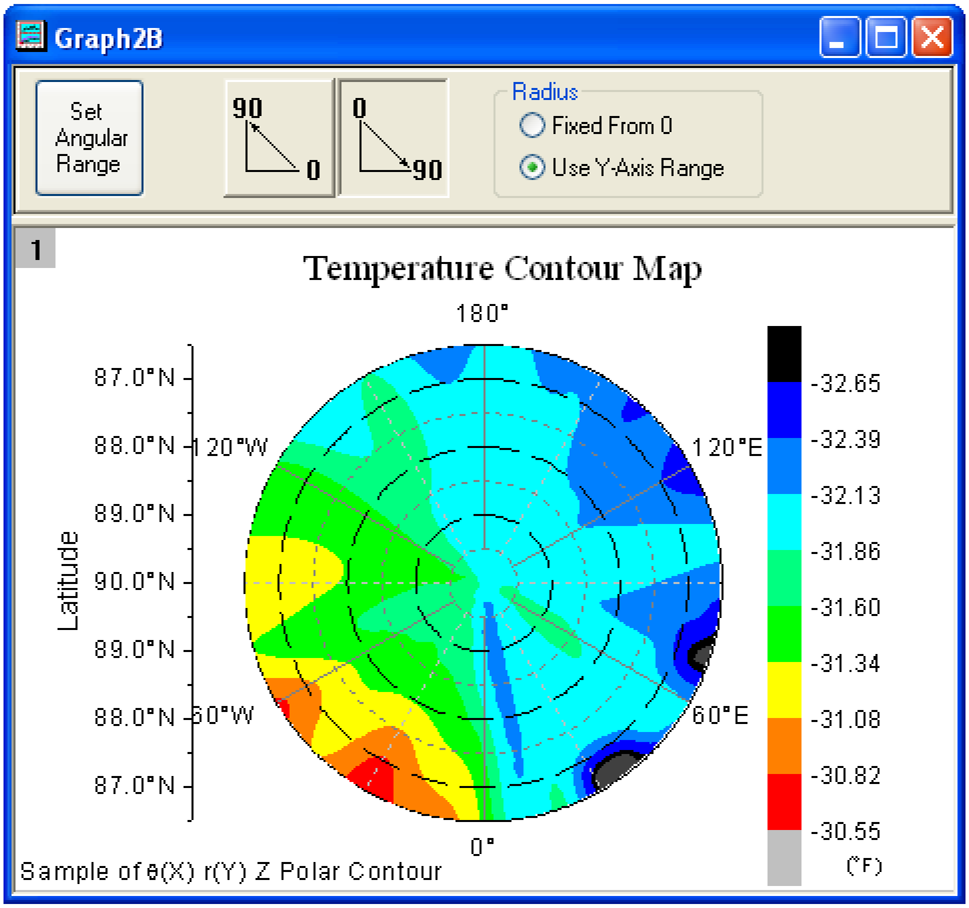
- o Select **Left** in the **Selection** list. Then select **Tick-Indexed Dataset** with the **Type** drop-down list. Choose **[Book3B]Sheet1!A** for **Dataset**. Change **Point** to **26**.
- o Select **Right** in the **Selection** list. Then select **Tick-Indexed Dataset** with the **Type** drop-down list. Choose **[Book3B]Sheet1!A** for **Dataset**. Change **Point** to **26**.
- o Go to the **Grid Lines** tab and make sure **Horizontal** has been selected in the **Selection** list. Then set the dialog as the following image shows. Click **OK** to close **Axis** dialog.



6. Double click on the Color Scale to open the **Color Scale Control** dialog and set the dialog as the following image shows.



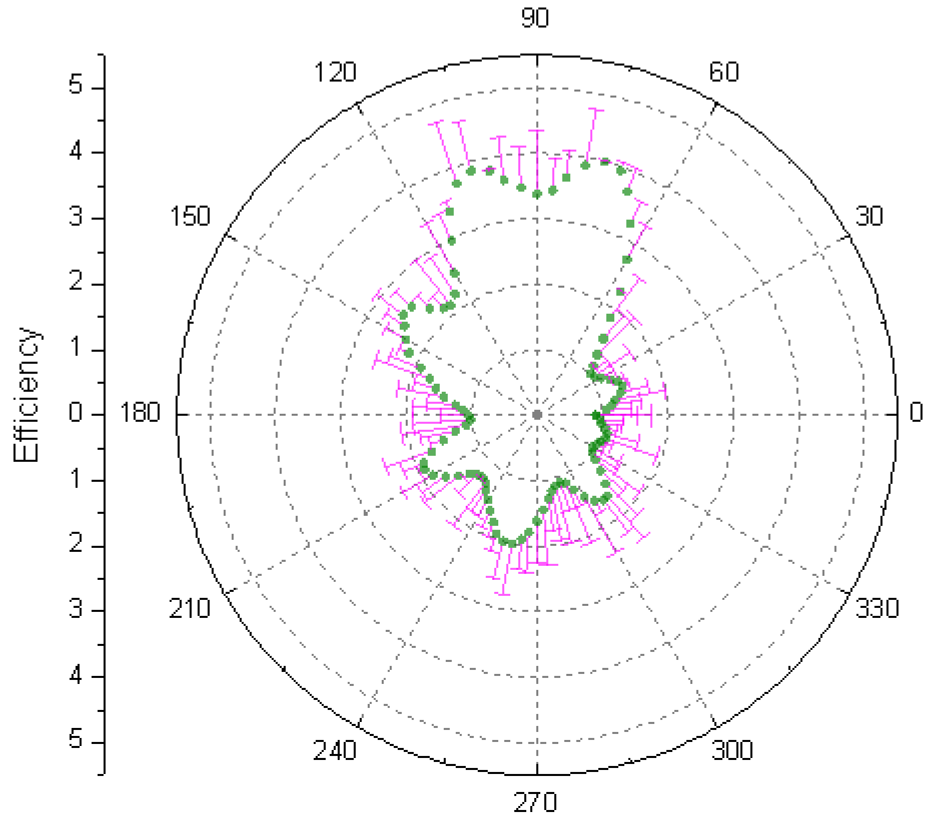
7. The graph should look like



5.7.2 Polar Plot with Error Bar

Summary

This tutorial will show you how to create a Polar graph with error bar.

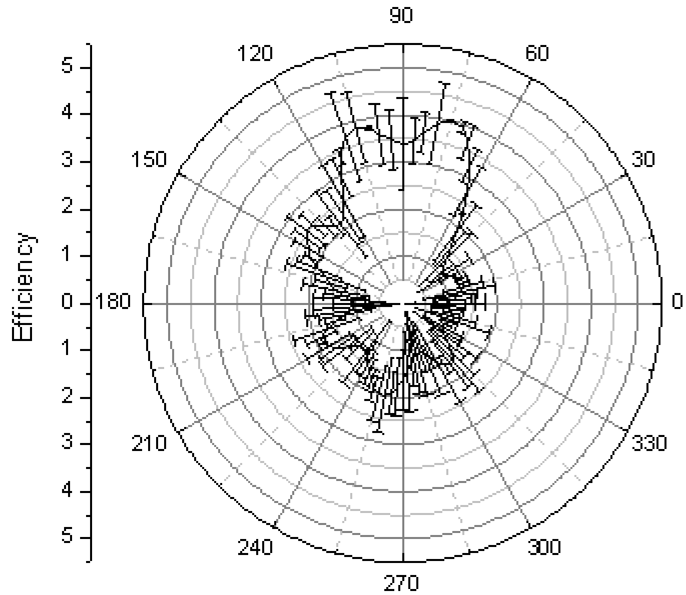


What will you learn

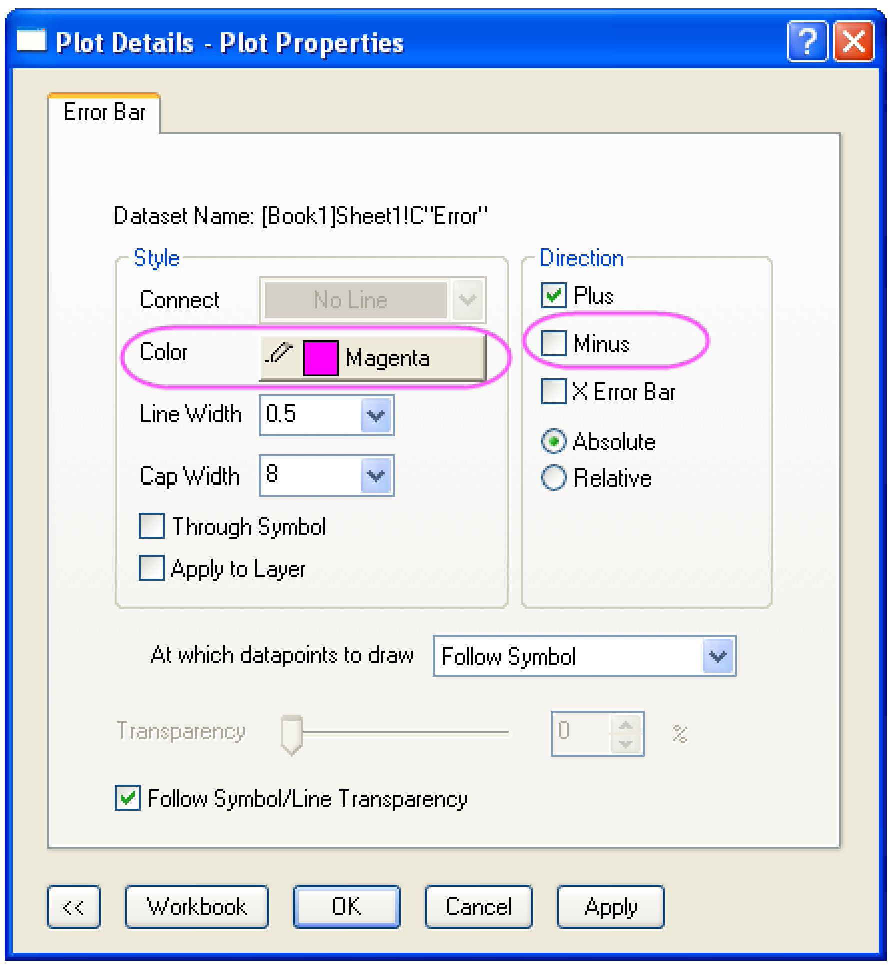
- Create a Polar graph with error bar
- Customize the graph using the Plot Details and Axis dialogs

Steps

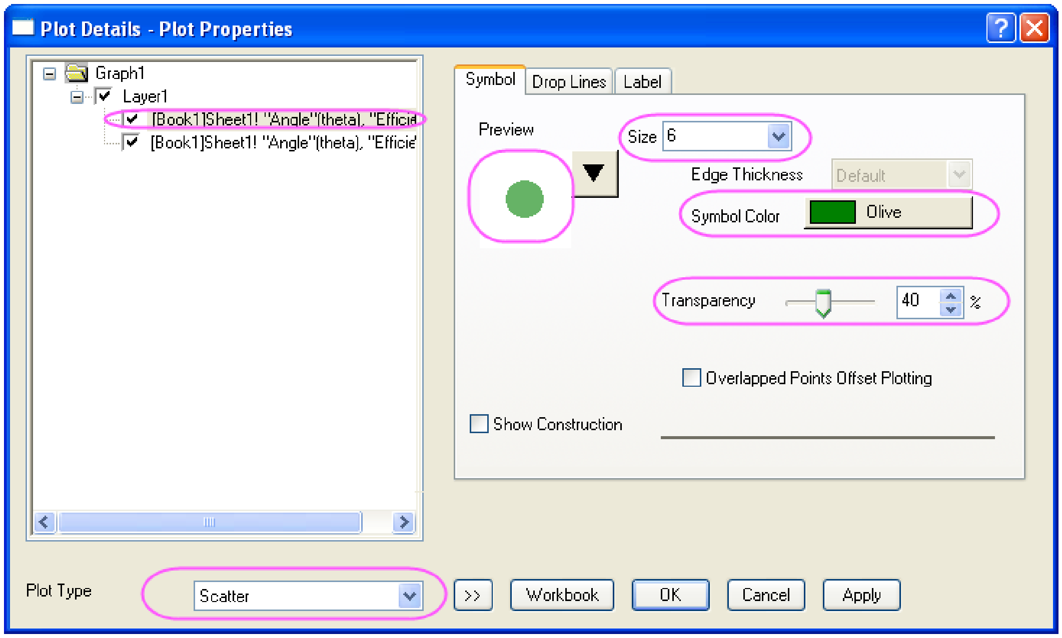
1. Import the sample data.
2. Highlight column C and set as Y error.
3. To plot a Polar graph with error bar: Highlight all columns. In the main menu, click Plot, point to Specialized, and then click Polar theta(X) r(Y). Alternatively, you can simply click the Polar theta(X) r(Y) button on the 2D Graphs toolbar.



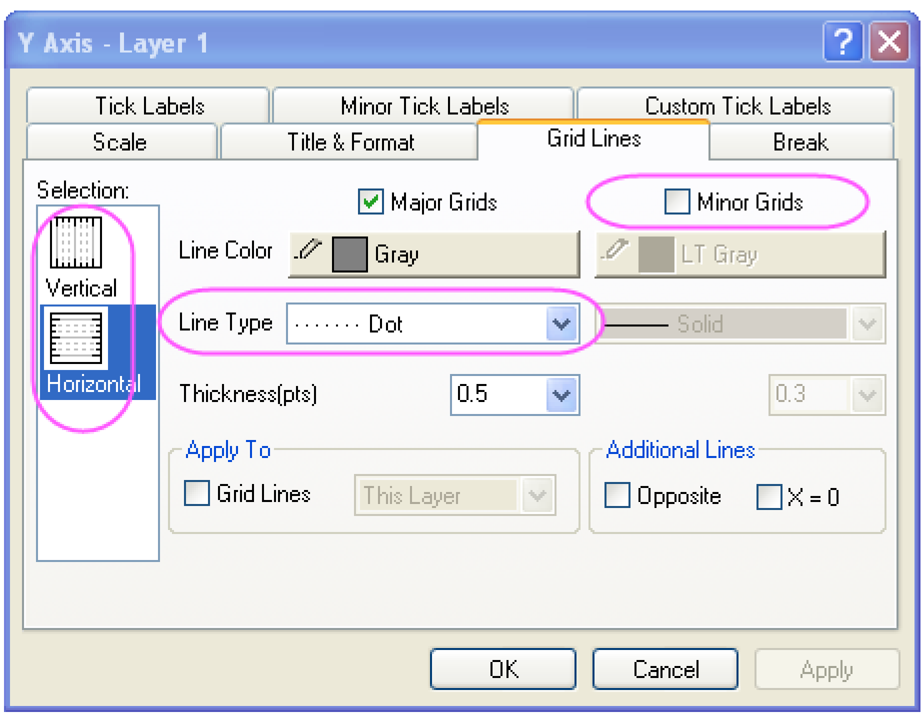
4. Double-click the plot to open the Plot Details dialog. In the left menu, select the Error plot. Change the color of the error bar to Magenta and clear the Minus check box (as below).



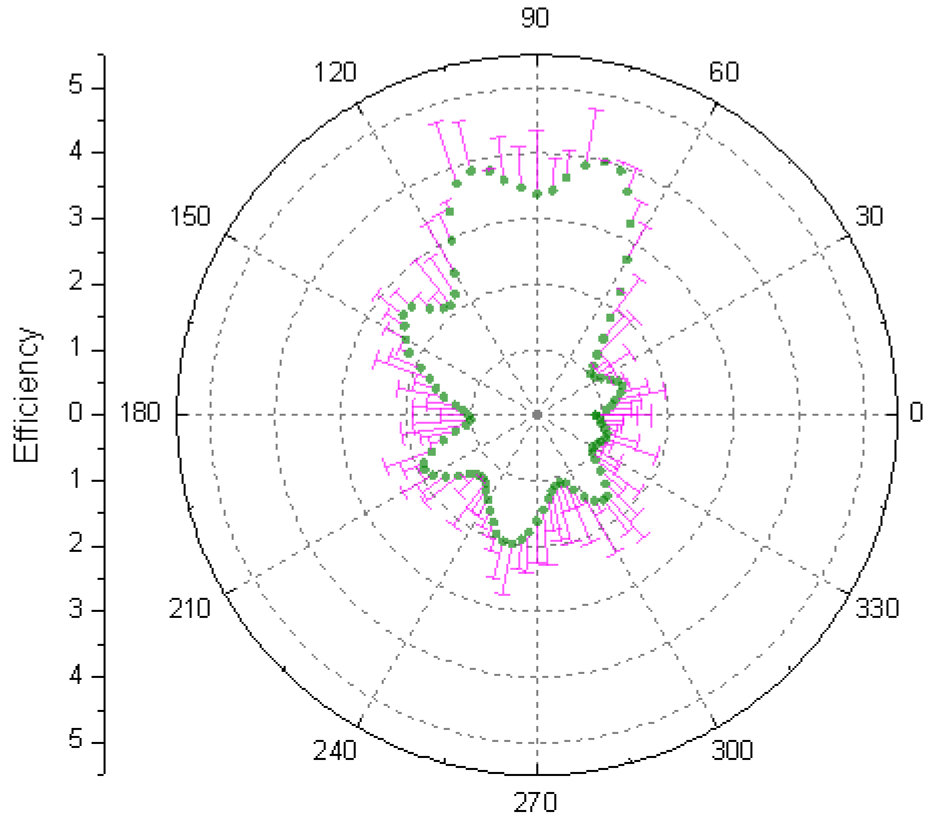
5. In the left panel, select the line plot and change its plot type to Scatter. Then, on the Symbol tab, change the symbol style, color, and transparency as below. Click OK to close the dialog.



6. On the left side of the graph, double-click the vertical axis to open the Axis dialog. On the Grid Lines tab, clear the Minor Grids check box and change the Major Grids Line Type to Dot. Select Horizontal in the Selection box at left, and repeat these steps for the Horizontal axis.



7. Click OK to close the Axis dialog. Your final graph should look like this:



5.8 3D

Topics covered in this section:

1. 3D Plotting
2. 3D Pie Chart
3. 3D Scatter Plot with Line Projections of Core Drilling Locations
4. 3D Surface Plot with Skipping Gridlines and Ignoring Missing Values
5. Creating Intersecting Surface Plots from Worksheets
6. Intersecting Color Surfaces
7. Surface with Symbols and Droplines
8. Colormap from Second Matrix
9. Color Map Surface Graph
10. Parametric Surface with Colormap from Data
11. Stacked 3D Surface Plots

5.8.1 Basic 3D Plotting

Summary

In Origin, most 3D plots -- including 3D surface, wire frame/wire surface, 3D bar plot and 2D contour -- are created from an Origin matrix. In most cases, the raw data is XYZ data and you should convert it to a matrix first, using one of Origin's built-in gridding routines.

Minimum Origin Version Required: Origin 8.0 SR6

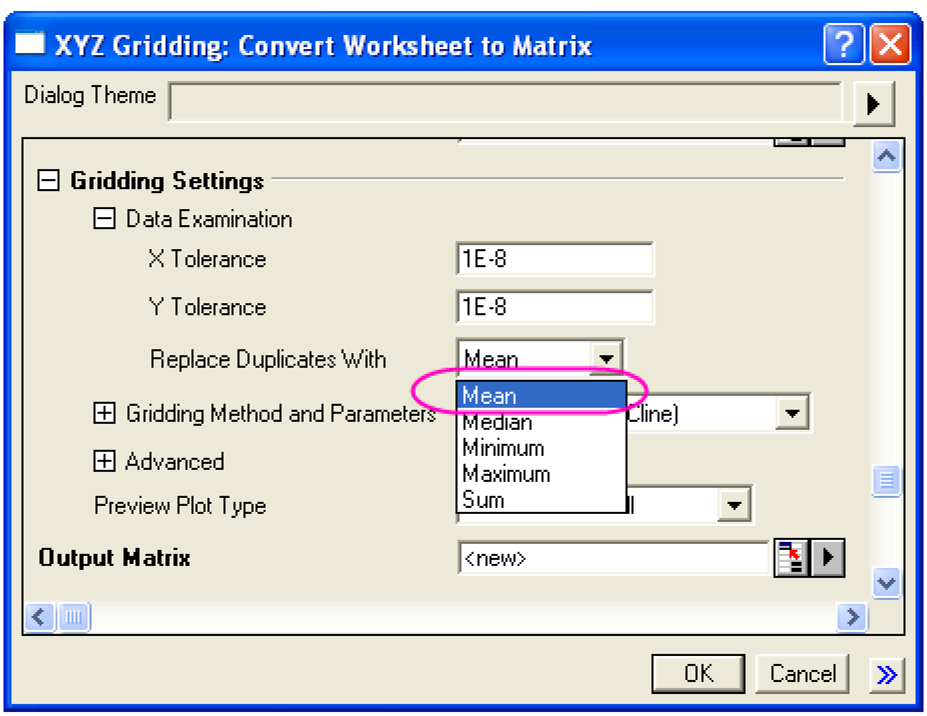
What you will learn

This tutorial will show you how to:

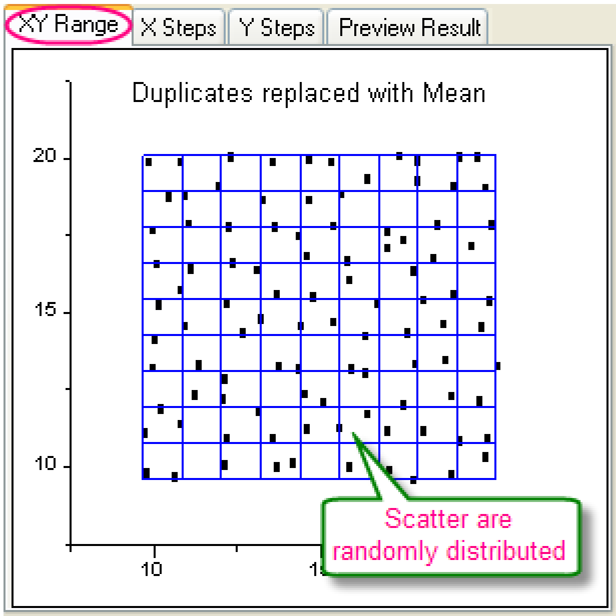
- Create a 3D graph in Origin
- Convert Worksheet data to a Matrix
- Use the layer contents dialog to add/remove a dataset
- Use the Plot Details dialog to modify graph

Steps

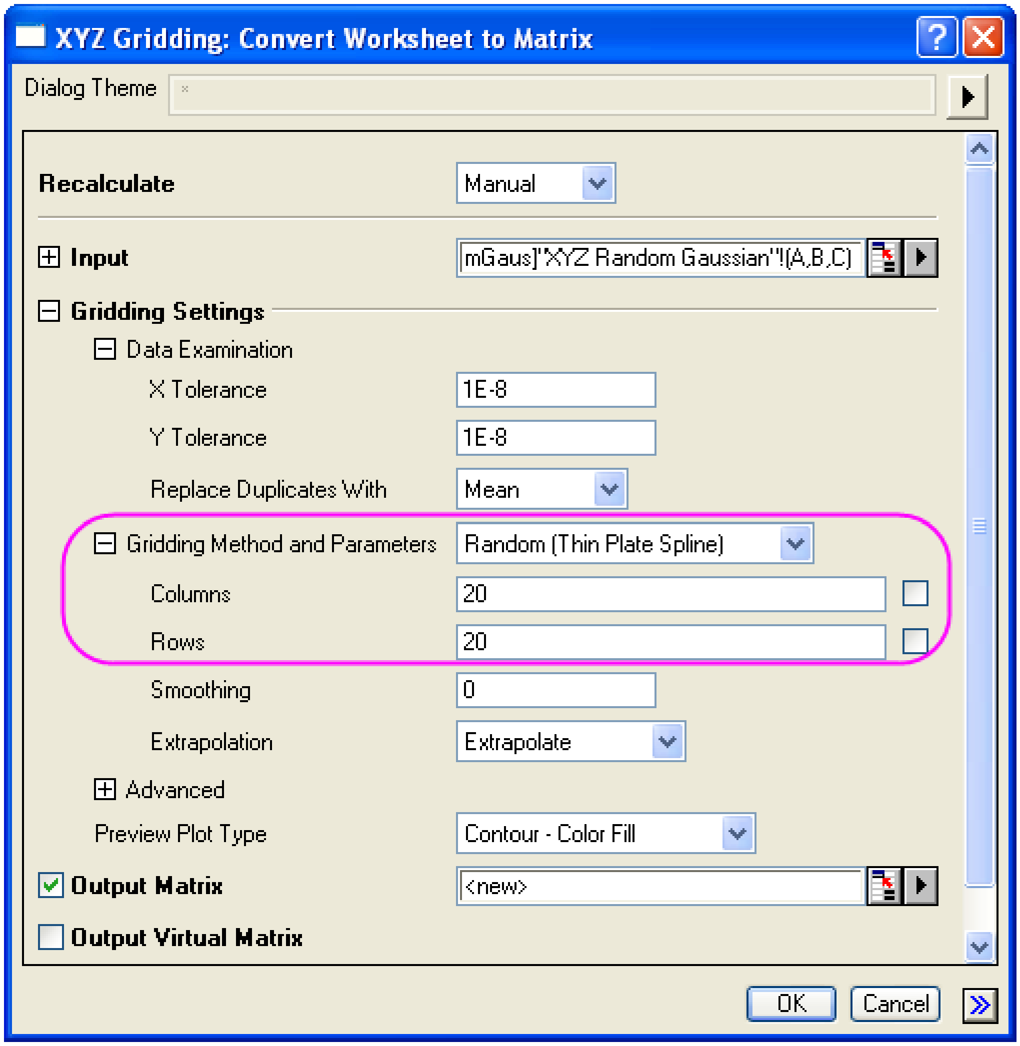
1. Import the file *\Samples\Matrix Conversion and Gridding\XYZ Random Gaussian.dat*.
2. Highlight the 3rd column, right-click, and select **Set As: Z** from the context fly-out menu.
3. To convert the worksheet XYZ data into a matrix, highlight the whole worksheet, select **Worksheet: Convert to Matrix: XYZ Gridding** to bring up the **XYZ Gridding** dialog. Confirm **Replace Duplicates with** item is set to **Mean** as shown below:



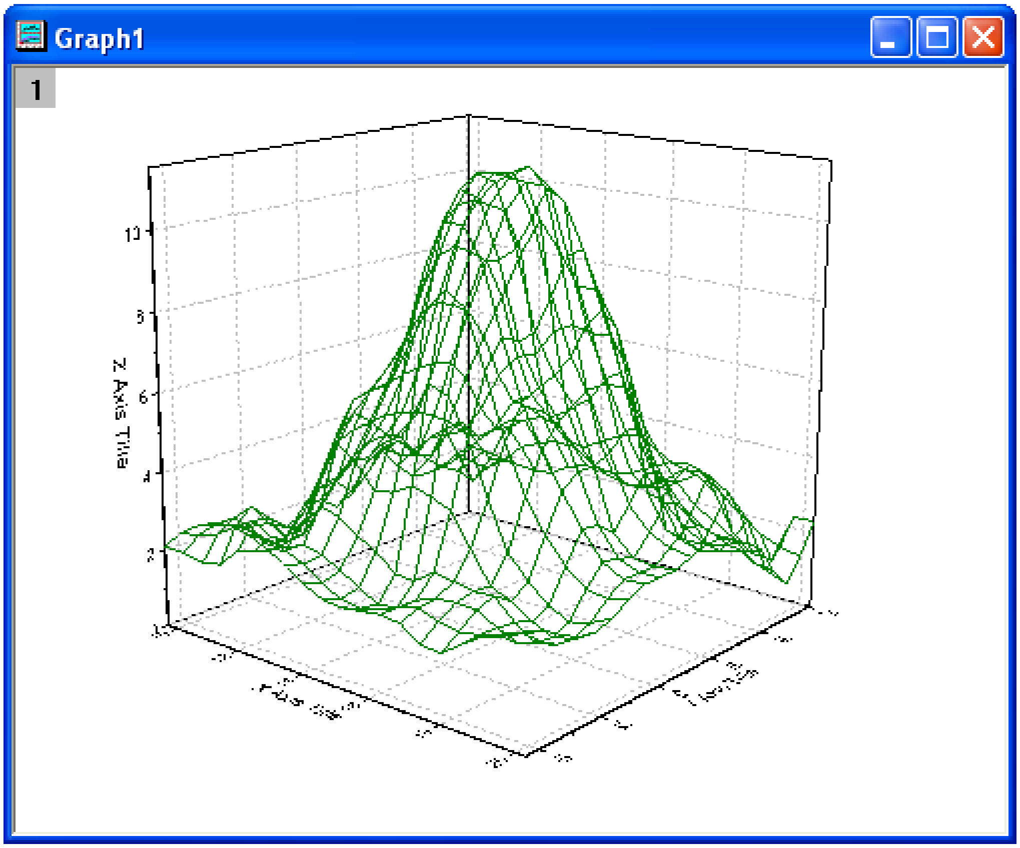
4. And you can see the right preview panel as below. Since the XY data are randomly distributed, a random gridding method should be used.



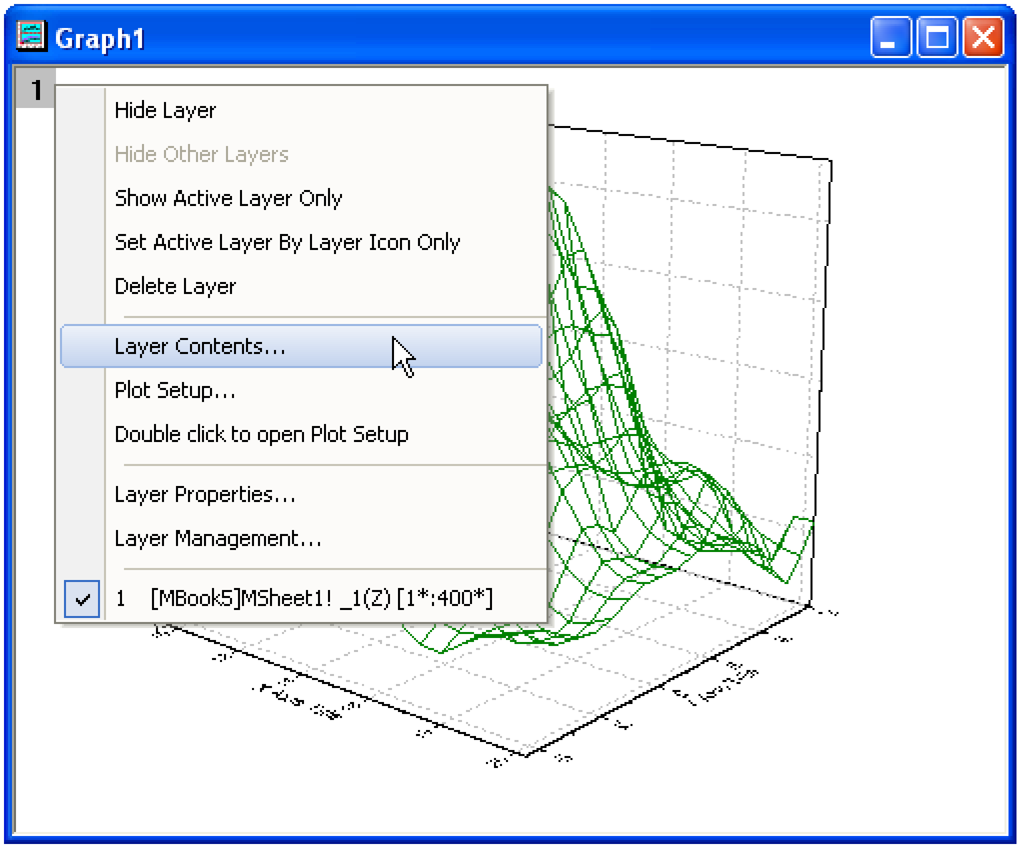
5. Use the following settings and click **OK** to convert the XYZ columns of data into a matrix of data. The Thin Plate Spline gridding method will generate a smooth surface.



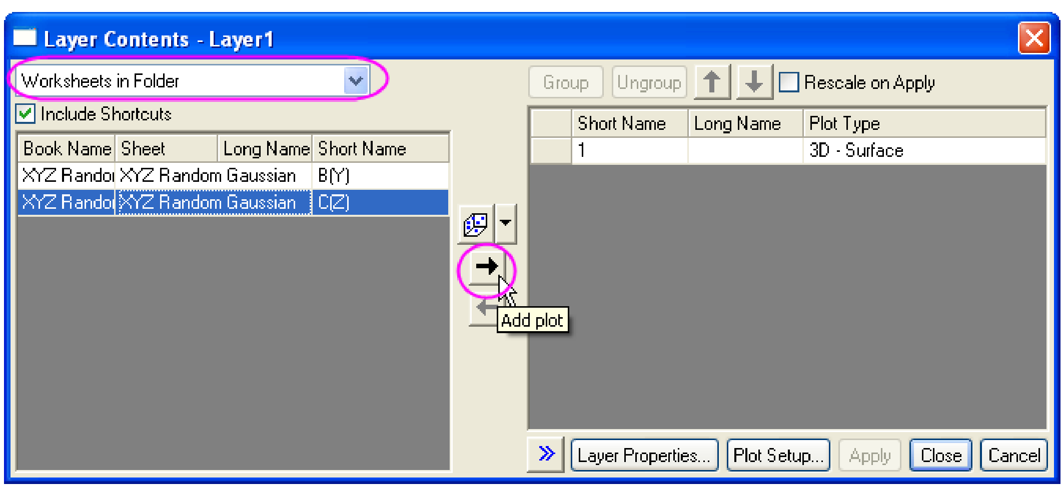
6. Set the newly generated matrix as the active window and select **Plot: 3D Surface: Wire Frame** from the menu to plot a 3D mesh:



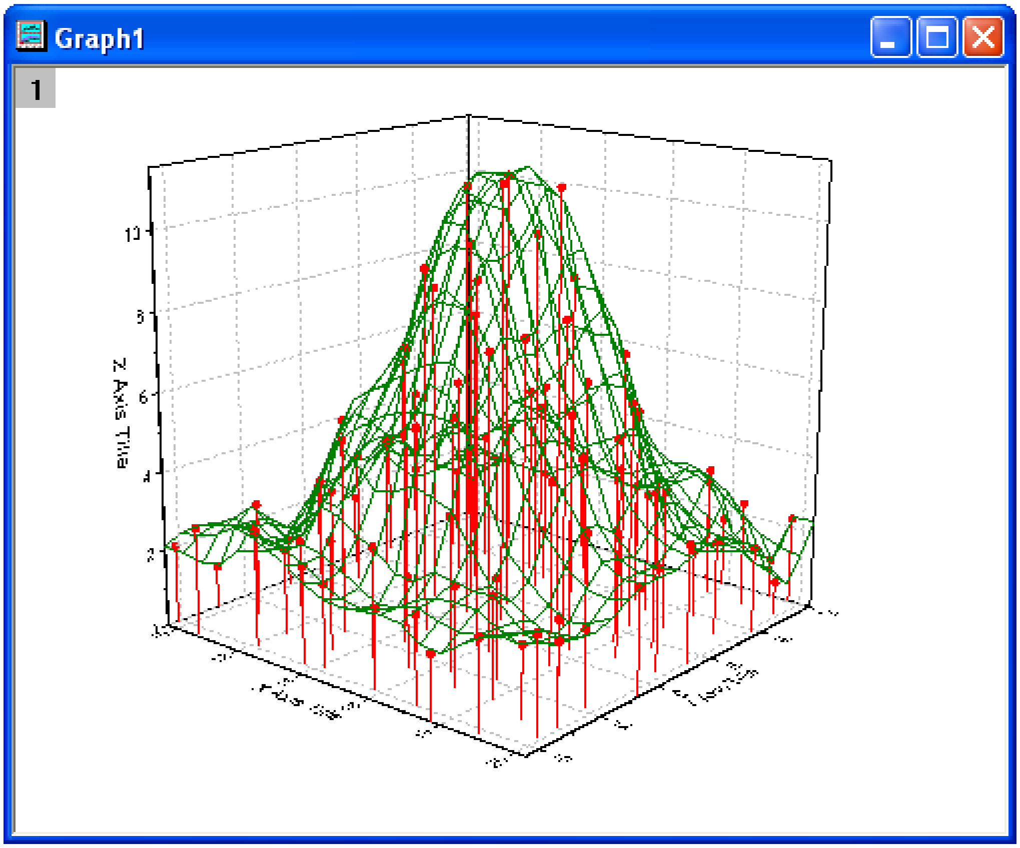
7. In order to plot the original data points on the graph, you can use the **Layer Contents**. Right-click on the layer icon (the small grey box in the upper left corner of the graph window) and select **Layer Contents**.



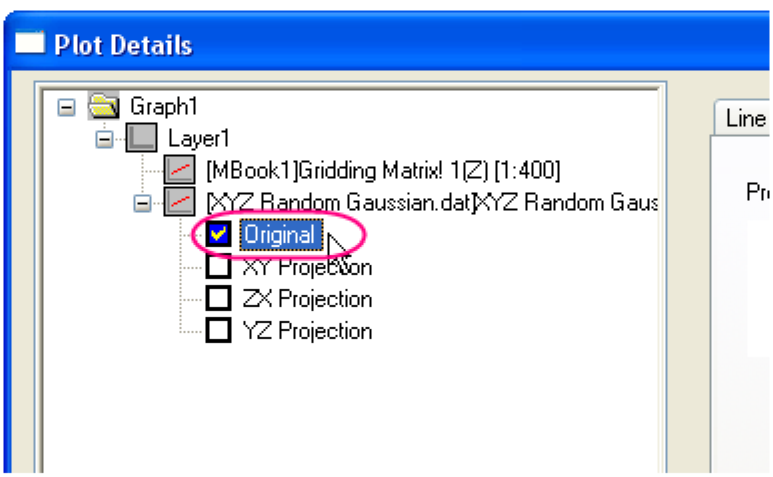
In the Layer Contents dialog, select **Worksheets in Folder** from the available data box (in the upper left corner of the dialog). Click on the **Plot Type** button , and select **3D Scatter / Trajectory / Vector** in the list. Then select the worksheet Z column and click the **->** button to add it to the right panel list.



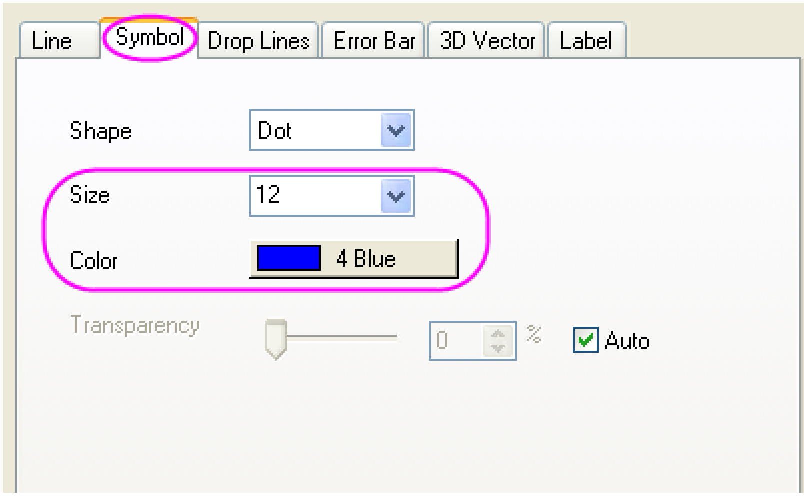
After you click the **Apply** button, the source data will be added to the layer.



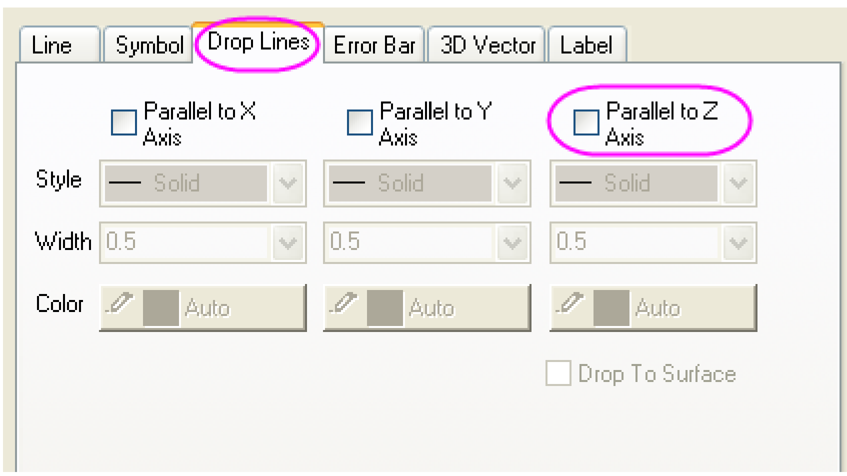
8. You can now use the **Plot Details** dialog to modify the appearance. Double-click on the graph to bring up the **Plot Details** dialog. On the left panel, select the 3D scatter data



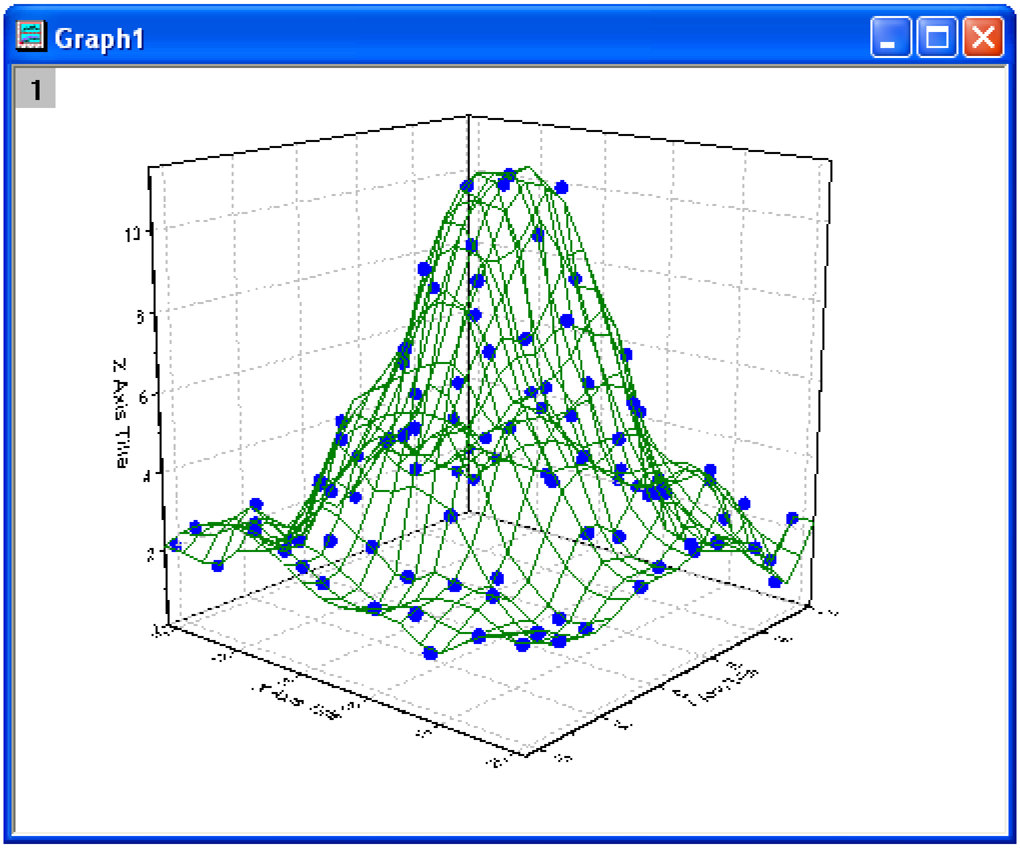
and then go to the **Symbol** tab located on the right panel and adjust the size and color of the data.



Remove the drop lines on the **Drop Lines** panel:



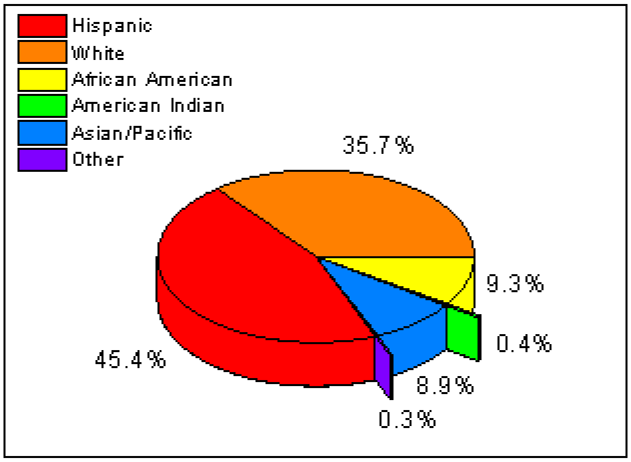
9. When done, click **OK** to accept the modifications:



5.8.2 3D Pie Chart

Summary

Origin's 3D Pie Chart gives you complete control over the look of the plot. Set the thickness of the pie slice, displacement, view angle, and size and rotation of the chart. You can even choose to explode the view of one or more pie slices.



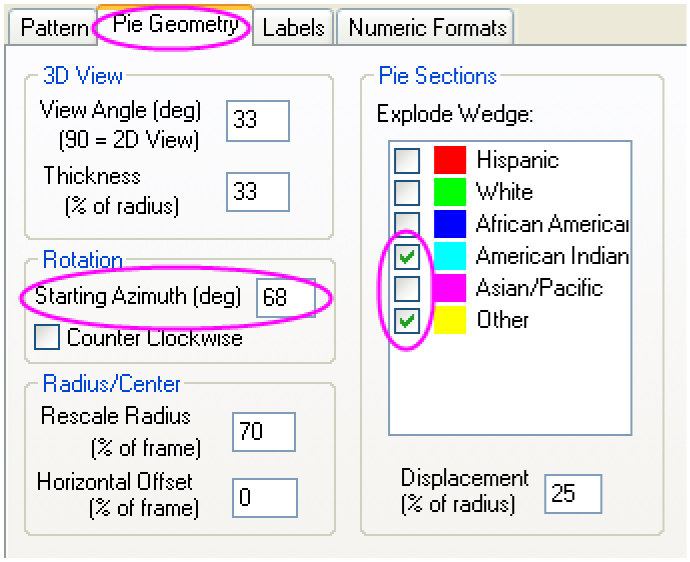
What you will learn

- Create 3D pie chart and change the view angle
- Explode the view of pie slices
- Customize pie slices individually

Steps

This tutorial is associated with the folder *Statistical and Specialized Graphs: Pie Chart* of the Statistical and Specialized Graphs project: *\Samples\Statistical and Specialized Graphs.opj*.

1. Import the data *\Samples\Graphing\3D Pie Chart.dat* into worksheet, and rename the column long name as *Demographic* and *Percent*.
2. Highlight the 2nd column and create a 3D Pie Chart from **Plot: Column/Bar/Pie: 3D Color Pie Chart**.
3. Double-click the pie chart to bring up the **Plot Details** dialog. In the **Pattern** tab, set the **Fill color** to **Increment** and red as starting color. Next, select the **Pie Geometry** tab and change the **Rotation** to 68 degrees. Then check the checkbox for *American Indian* and *Other* in the **Explode Wedge** group as shown below and click **OK**.

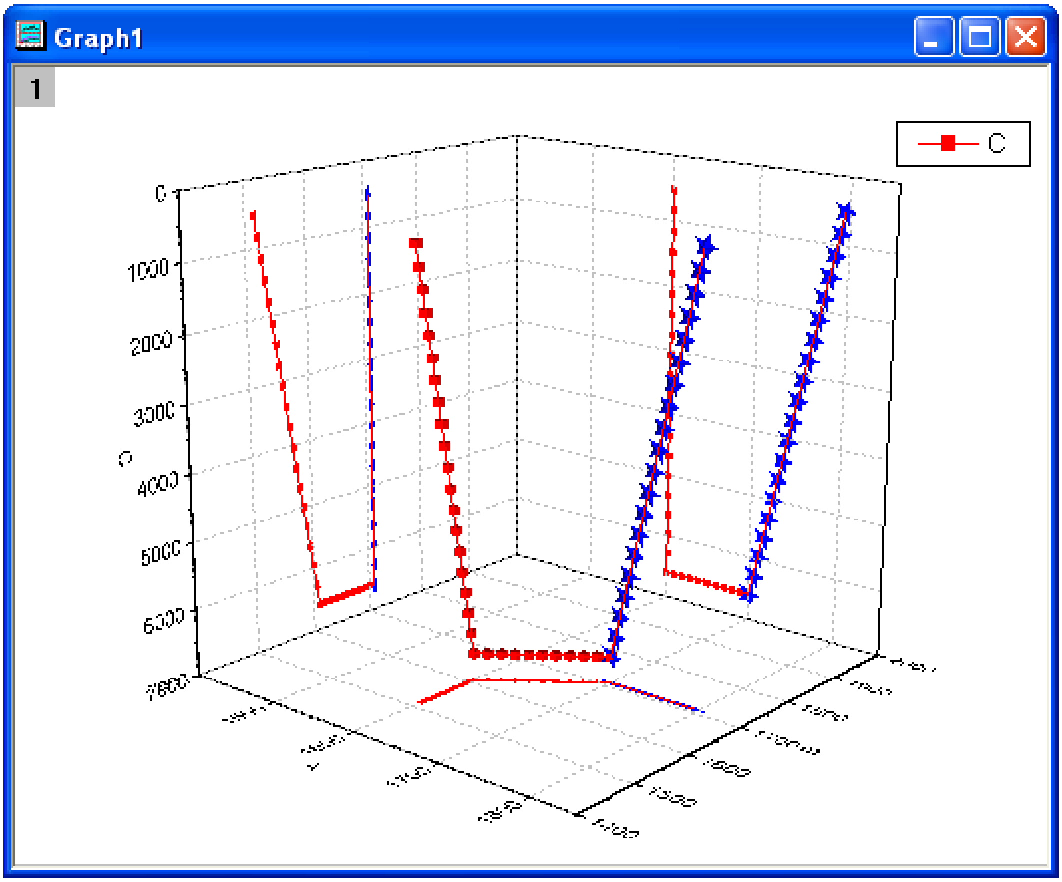


4. Then hold the **Ctrl** key and double click each pie slice to open the **Plot Details** dialog and change the **Fill Color** in the **Pattern** tab.

5.8.3 3D Scatter Plot with Line Projections of Core Drilling Locations

<u>Summary</u>

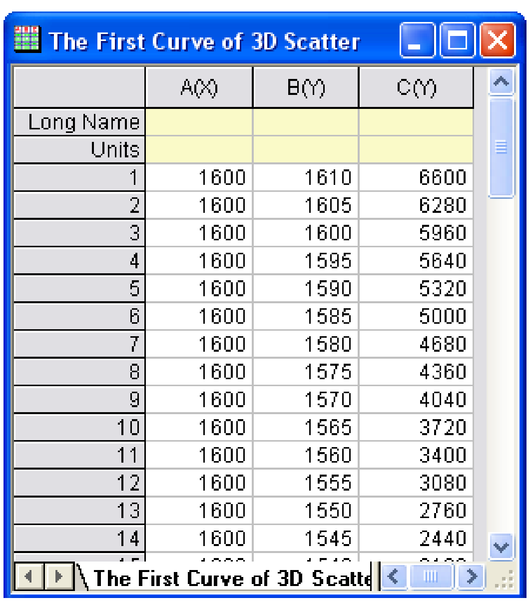
This tutorial will show you how to create a 3D scatter plot and how to show the projections of the plots.



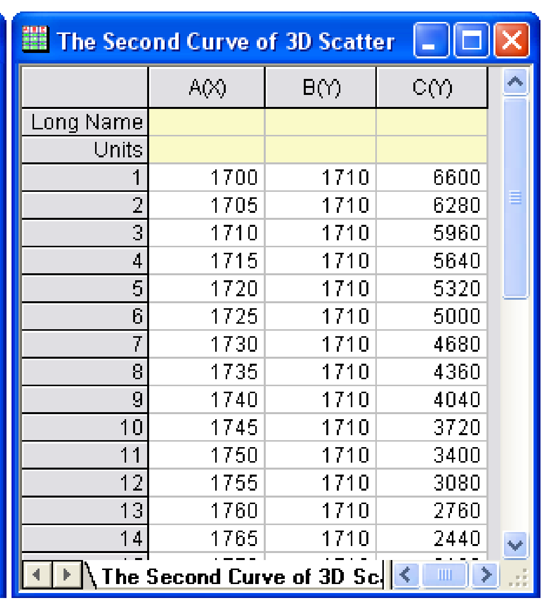
Minimum Origin Version Required: Origin 8.5.1

Steps

1. Create two new workbooks. Import the data into the workbooks

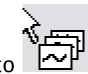

The First Curve of 3D Scatter

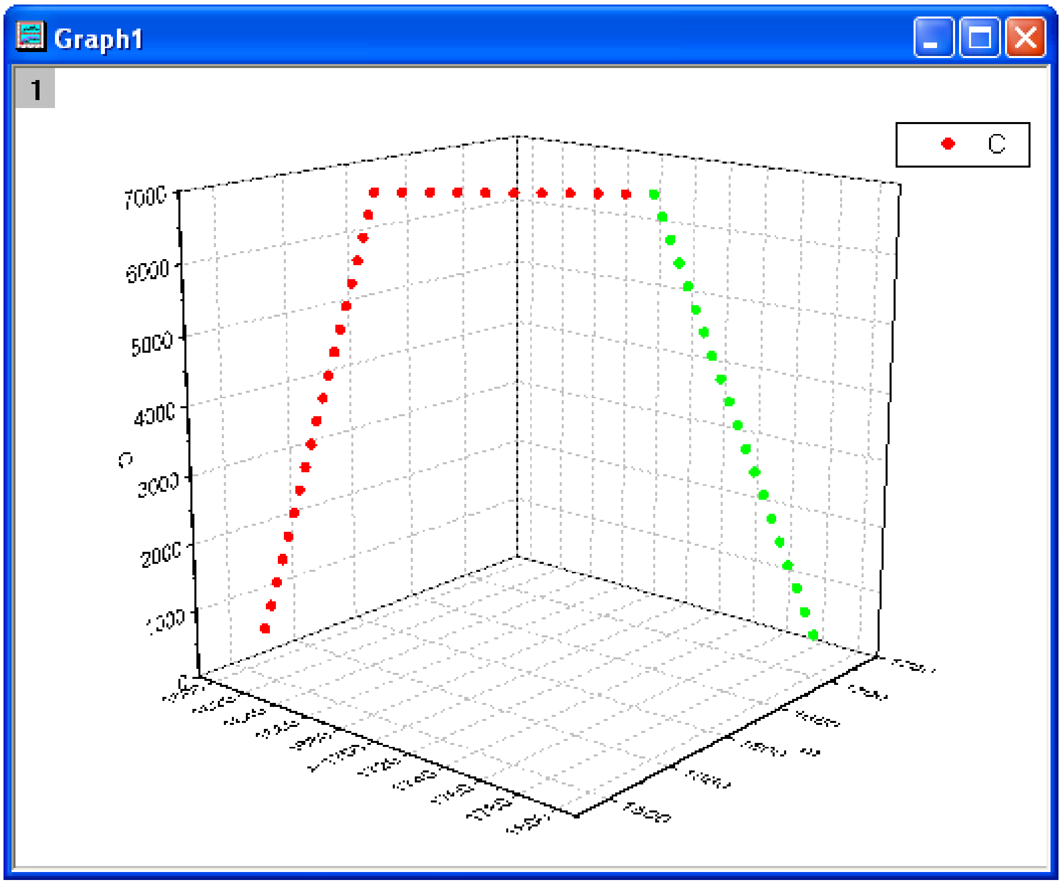
	A(X)	B(Y)	C(Y)
Long Name			
Units			
1	1600	1610	6600
2	1600	1605	6280
3	1600	1600	5960
4	1600	1595	5640
5	1600	1590	5320
6	1600	1585	5000
7	1600	1580	4680
8	1600	1575	4360
9	1600	1570	4040
10	1600	1565	3720
11	1600	1560	3400
12	1600	1555	3080
13	1600	1550	2760
14	1600	1545	2440


The Second Curve of 3D Scatter

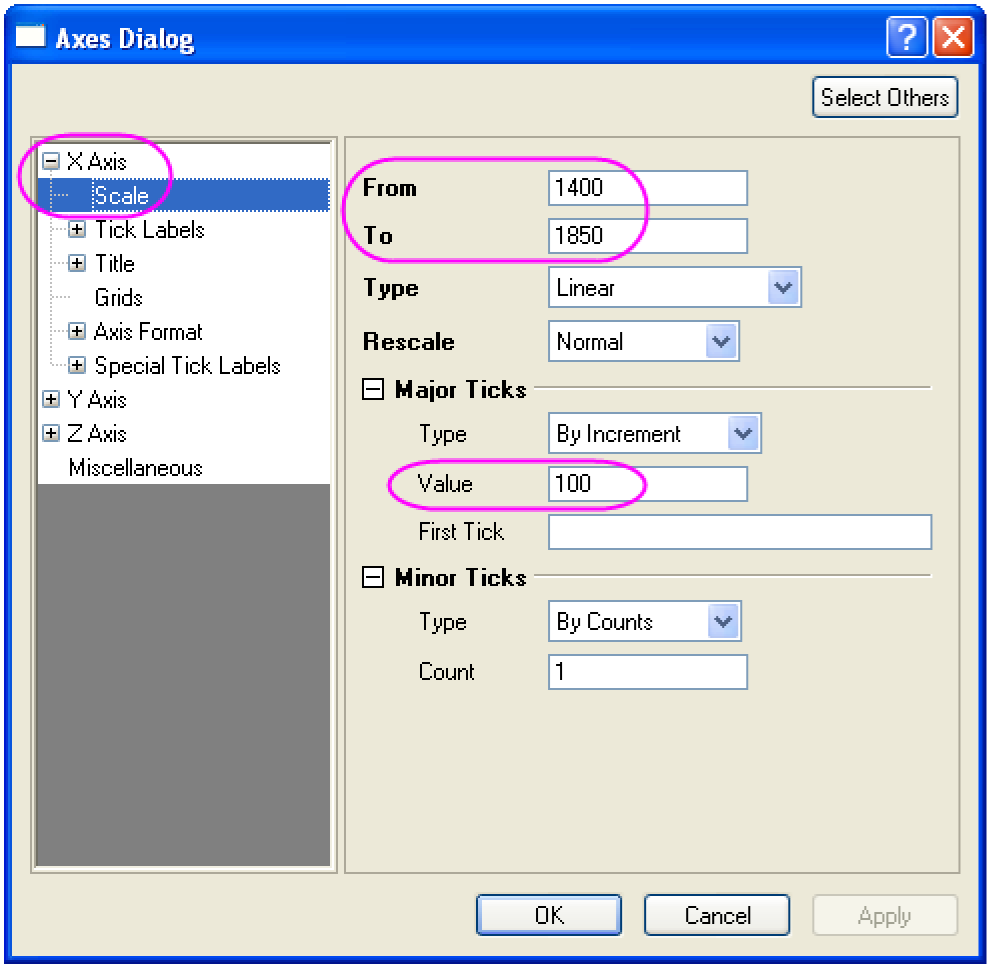
	A(X)	B(Y)	C(Y)
Long Name			
Units			
1	1700	1710	6600
2	1705	1710	6280
3	1710	1710	5960
4	1715	1710	5640
5	1720	1710	5320
6	1725	1710	5000
7	1730	1710	4680
8	1735	1710	4360
9	1740	1710	4040
10	1745	1710	3720
11	1750	1710	3400
12	1755	1710	3080
13	1760	1710	2760
14	1765	1710	2440

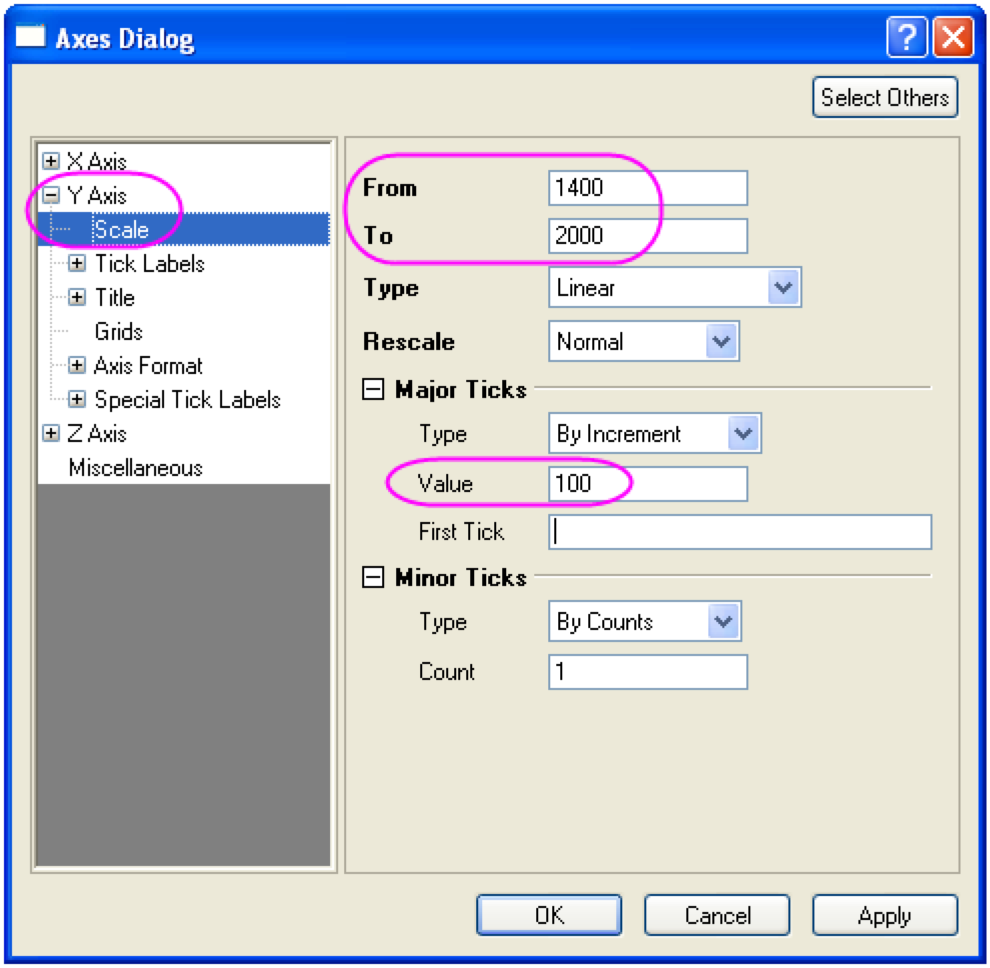
2. Next, select the **The First Curve of 3D Scatter** worksheet. Highlight column **C**. Right-click on it to open the context menu and select **Set As: Z**. Generate a plot by first selecting all data in the sheet followed by choosing **Plot: 3D Symbol/Bar: 3D Scatter** from the main menu.

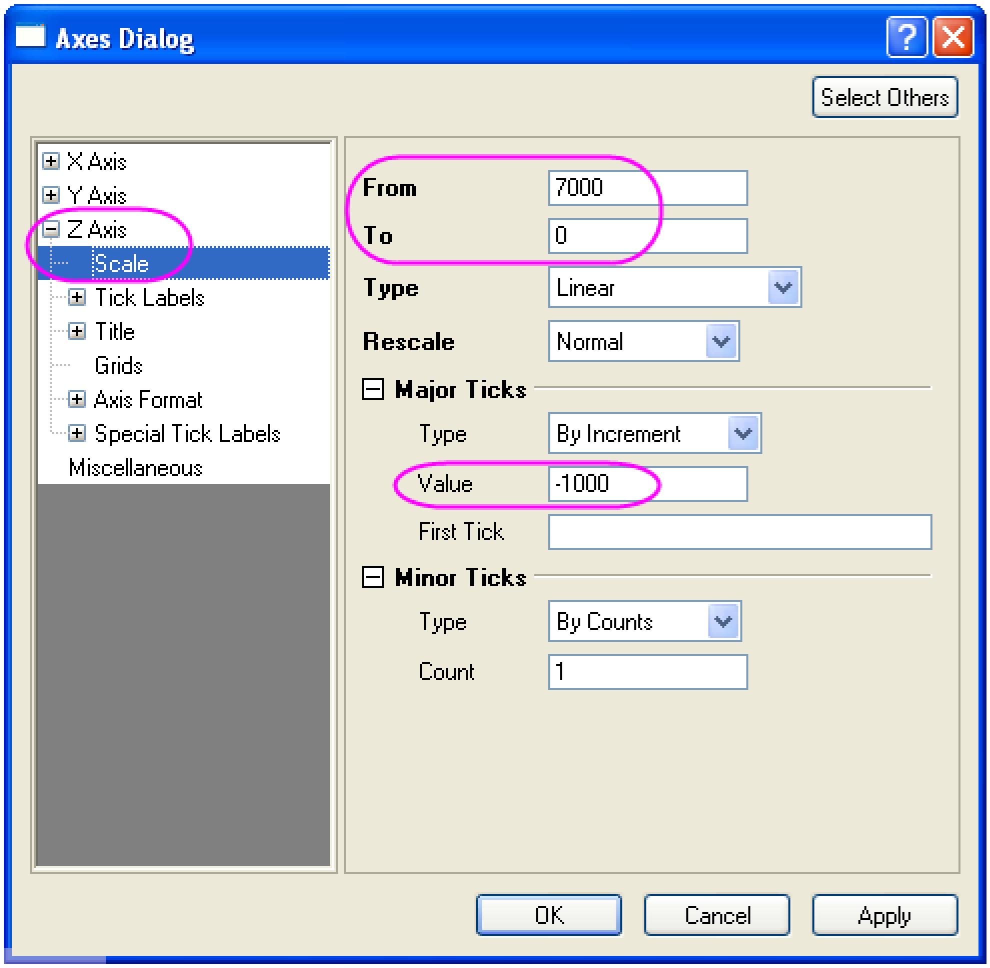
3. Activate **The Second Curve of 3D Scatter** worksheet. Highlight column **C**. Right-click on it and select **Set As: Z** from the context menu. Then move the mouse cursor to the right edge of the selection area until the shape of the cursor changes to . Then hold down the left mouse button and drag the highlighted data into the newly created graph window.
The resulting graph should resemble



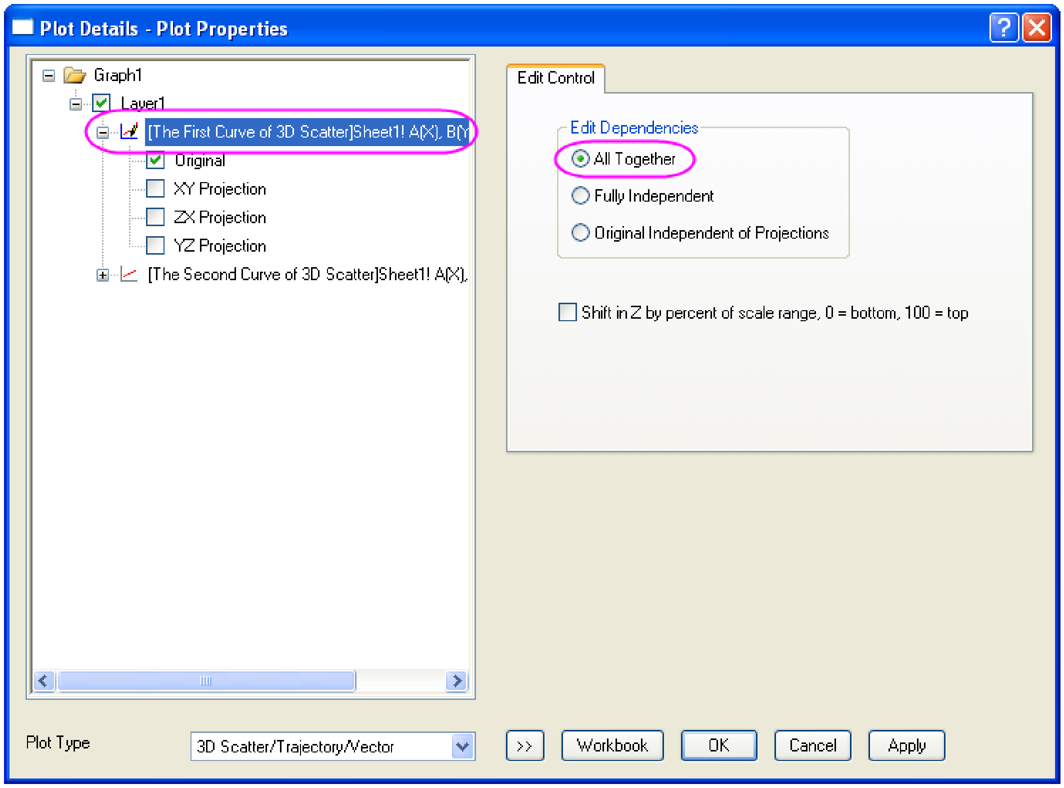
4. Double-click on either the X-, Y- or Z-coordinate axes to open the **Axis Dialog**. For each axis, set the **Scale** section options according to those shown below.


Axes Dialog
Select Others
X Axis
Scale
Tick Labels
Title
Grids
Axis Format
Special Tick Labels
Y Axis
Z Axis
Miscellaneous
From
1400
To
1850
Type
Linear
Rescale
Normal
Major Ticks
Type
By Increment
Value
100
First Tick
Minor Ticks
Type
By Counts
Count
1
OK
Cancel
Apply

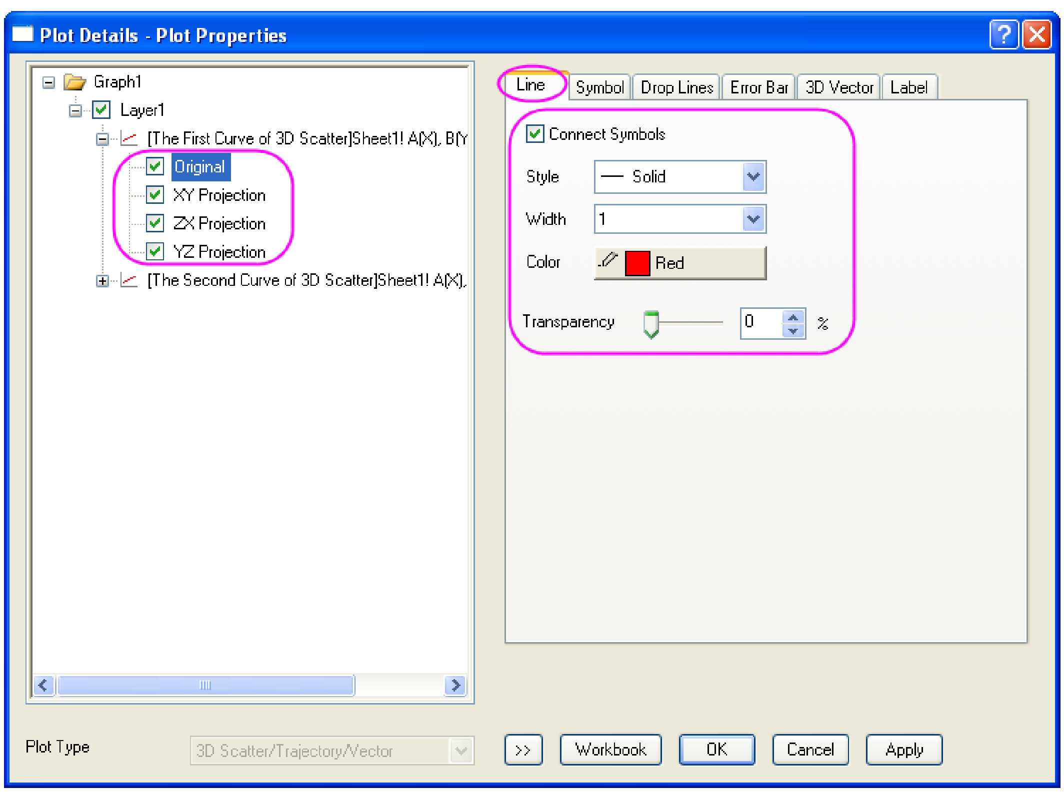

Axes Dialog
Select Others
X Axis
Y Axis
Scale
Tick Labels
Title
Grids
Axis Format
Special Tick Labels
Z Axis
Miscellaneous
From
1400
To
2000
Type
Linear
Rescale
Normal
Major Ticks
Type
By Increment
Value
100
First Tick
Minor Ticks
Type
By Counts
Count
1
OK
Cancel
Apply



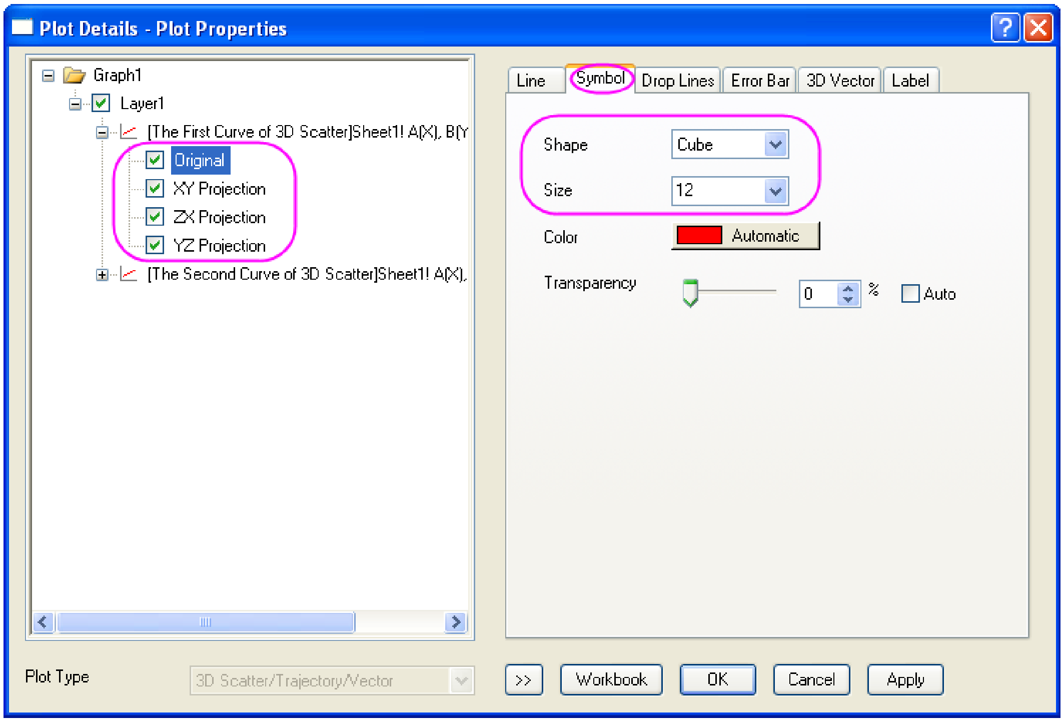
5. Select **Format: Layer Properties** from the main menu to open the **Plot Details** dialog. Expand the **Layer1** node. Choose the **first** plot and select **All Together** from **Edit Dependencies**.



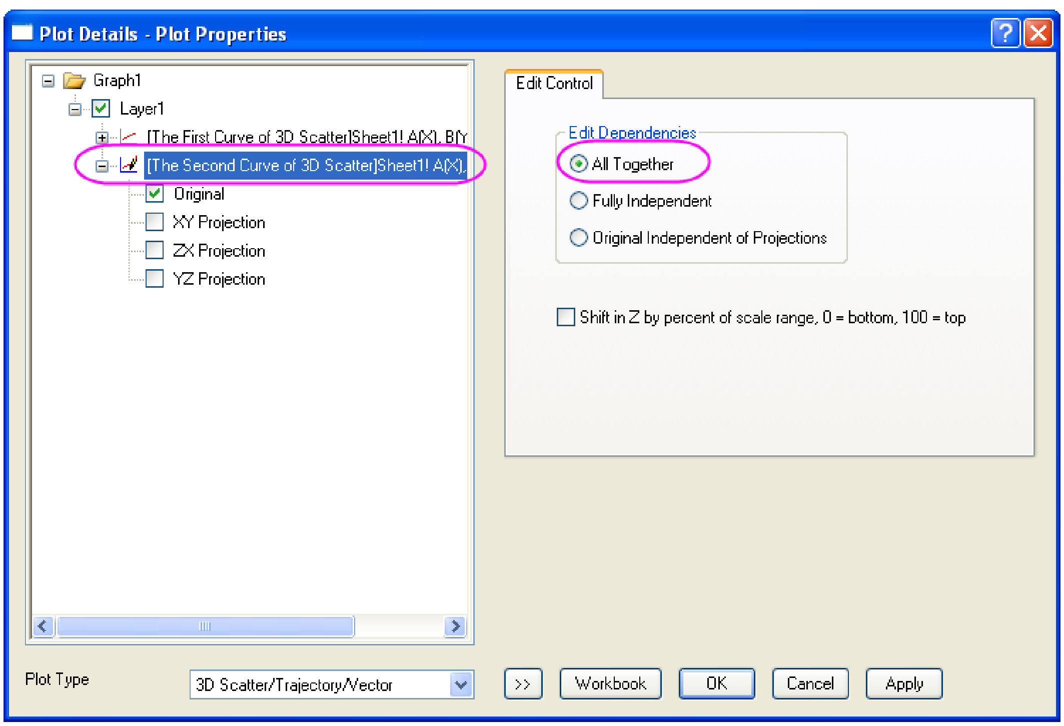
Expand the first plot node and select the **Original** plot. Then set the dialog options to those shown in the screenshot below.

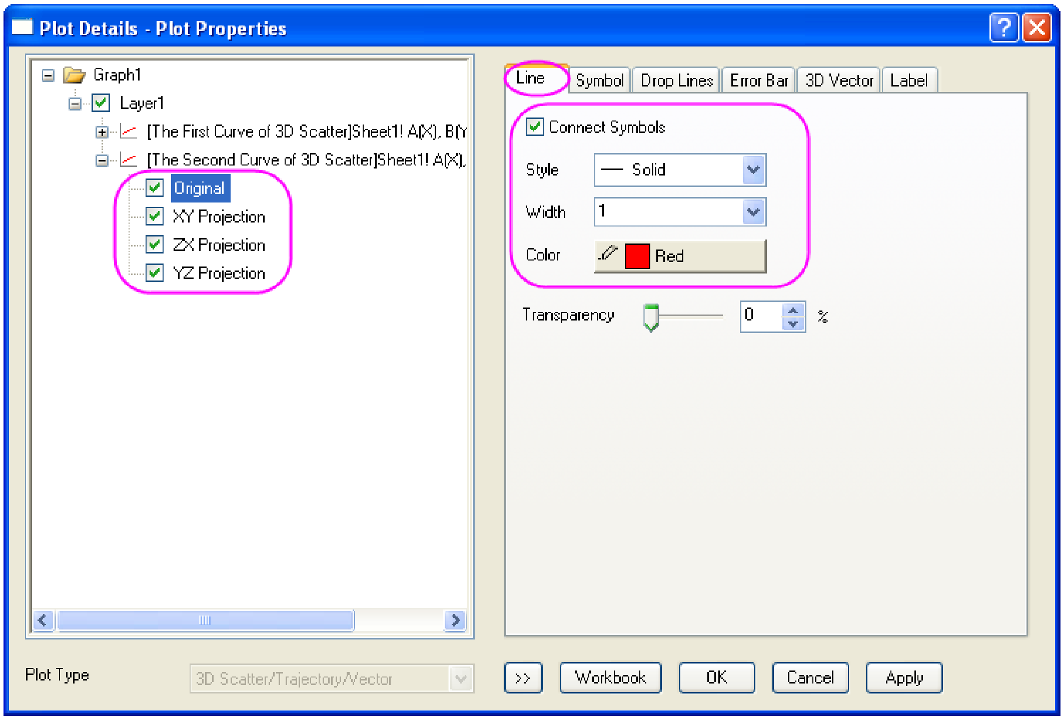


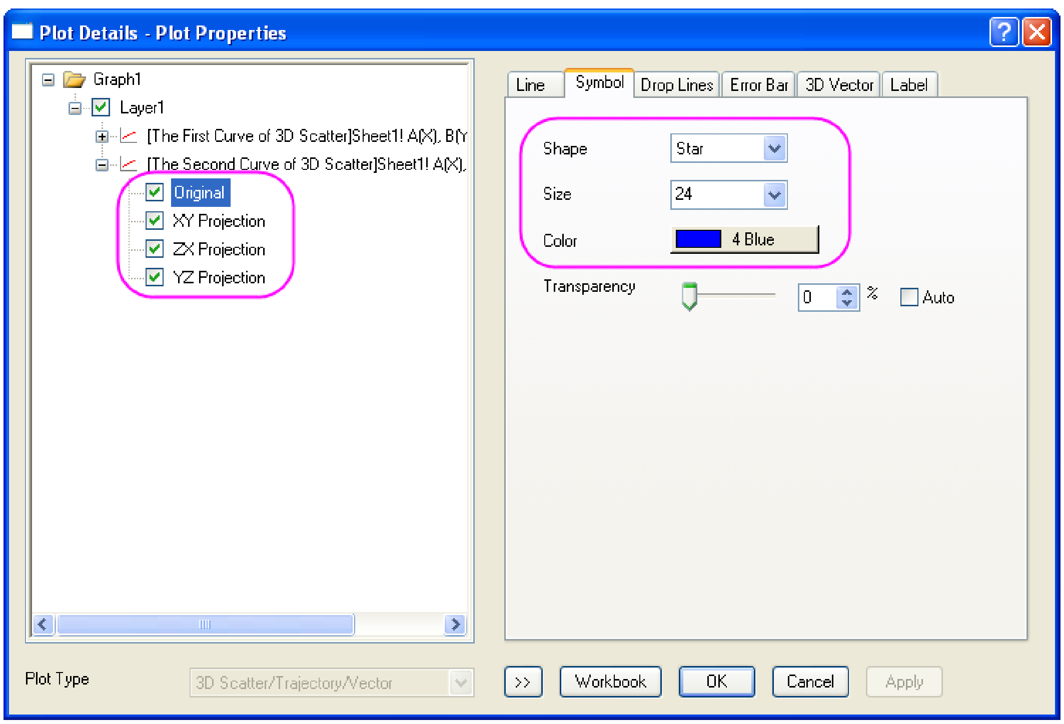
6. Select the **Symbol** tab and set the dialog options according to the screenshot below.



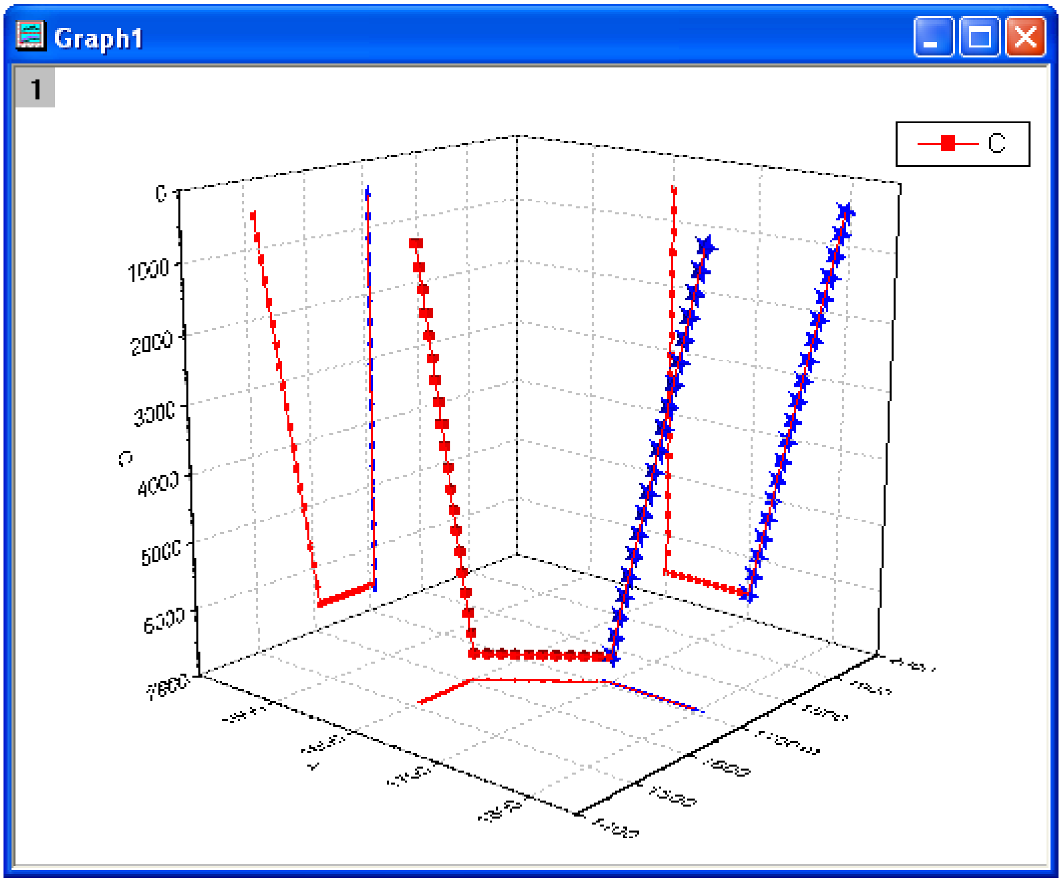
7. In a similar fashion, select the second plot and set the dialog options to match those in the screenshots below.



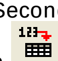




8. The final graph should resemble



Sample Data

Download the **The_First_Curve_of_3D_Scatter.txt** file and **The_Second_Curve_of_3D_Scatter** file from http://www.originlab.com/ftp/graph_gallery/data/The_First_Curve_of_3D_Scatter.txt and http://www.originlab.com/ftp/graph_gallery/data/The_Second_Curve_of_3D_Scatter.txt. Create a new worksheet. Click the **Import Single ASCII button**  and select the **The First Curve of 3D Scatter.txt** to import it into Origin. Then select **File: New: Workbook** from the main menu to create another empty workbook and import the **The Second Curve of 3D Scatter.txt** with the **Import Single ASCII button**.

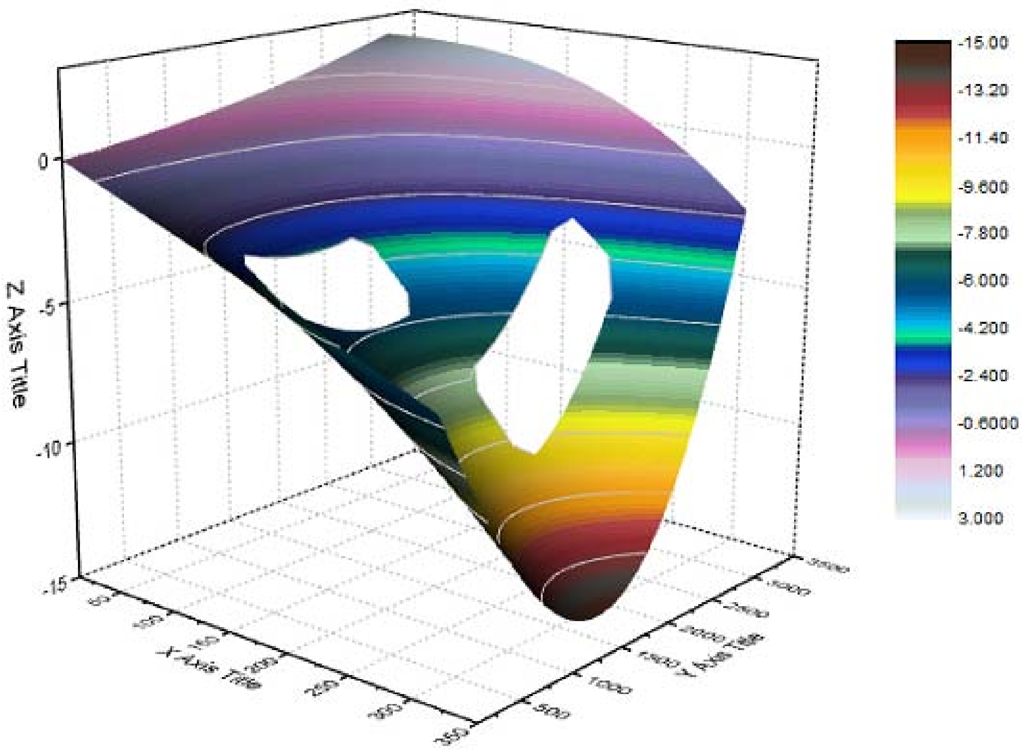
The following table contains part of the sample data.

A(X)	B(Y)	C(Z)
1600	1610	6600
1600	1605	6280
1600	1600	5960
1600	1595	5640
1600	1590	5320
1600	1585	5000
1600	1580	4680
1600	1575	4360
1600	1570	4040
1600	1565	3720

5.8.4 Surface with Missing Values

Summary

In this tutorial, a 3D color map surface graph will be created from a matrix with missing values. Also, gridlines can be skipped by customization.



Minimum Origin Version Required: Origin 8.5 SR1

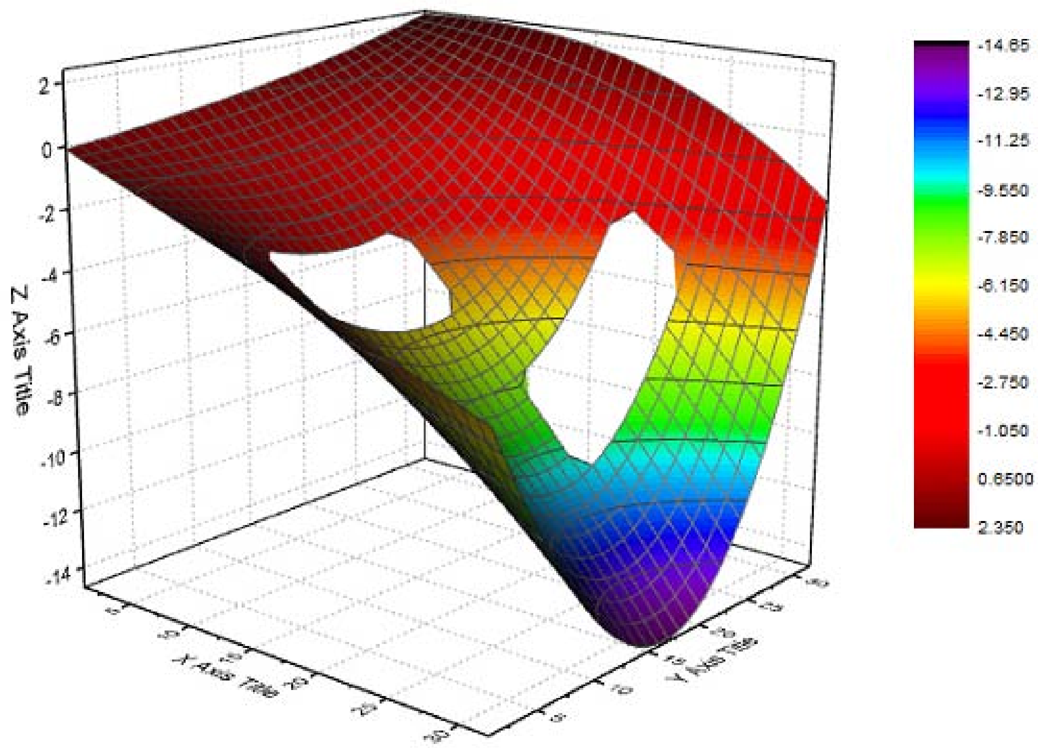
What you will learn

- Create a 3D surface plot from a matrix by ignoring missing values
- Skip gridlines of a 3D surface plot
- Set contour lines
- Set contour levels and fill colors
- Re-scale axes ticks and labels

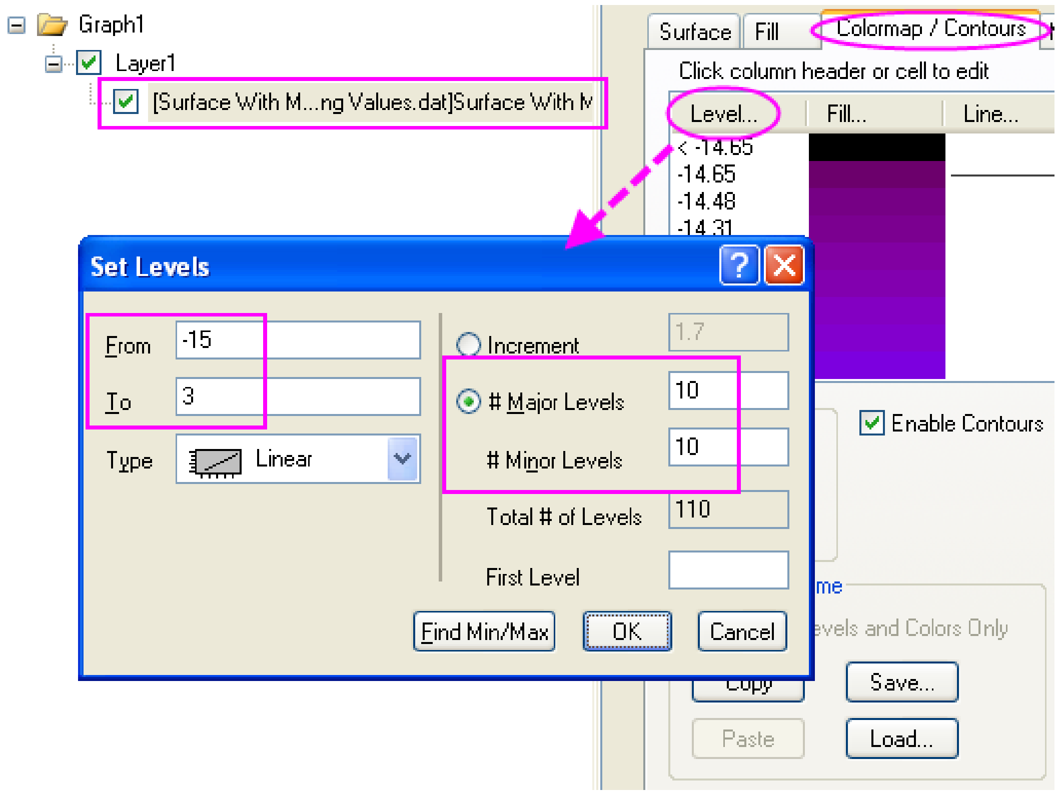
Steps

1. Start with a new matrix and import the file *\Samples\Graphing\Surface With Missing Values.dat*. Activate the matrix and select **Plot: 3D Surface: Color Map Surface** from the Origin main menu to create a graph. From the plotting dialog set the **Data Format** to **X**

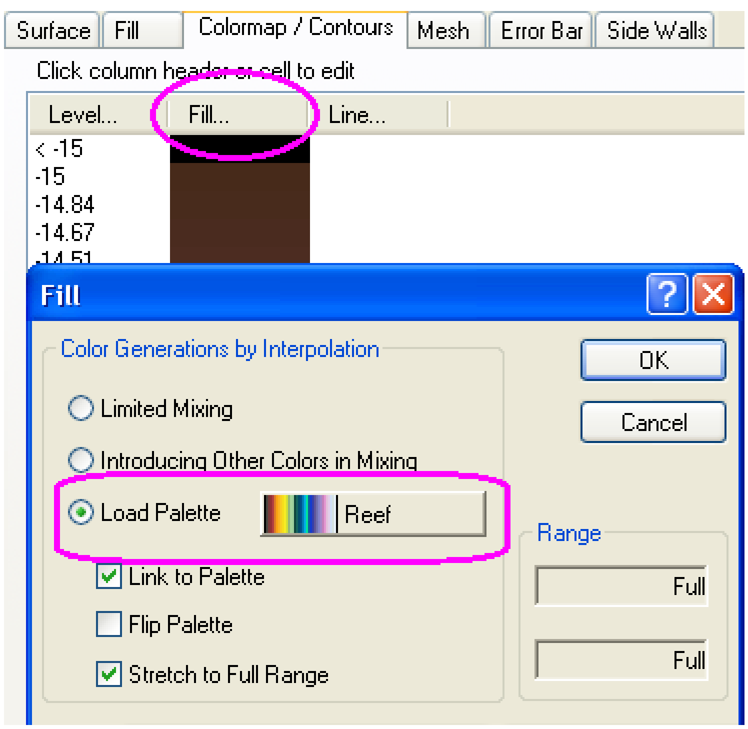
across columns, **X Values in** to **None**, and **Y Values in** to **1st column in selection**. The graph should look like



2. Double click on the graph to open the **Plot Details** dialog. Expand the tree node on the left panel to make sure that the box under **Layer1** is checked. Activate the **Colormap / Contours** tab on the right panel. Click the **Level** heading to open the **Set Levels** dialog and set the dialog as the following sreenshots shows:

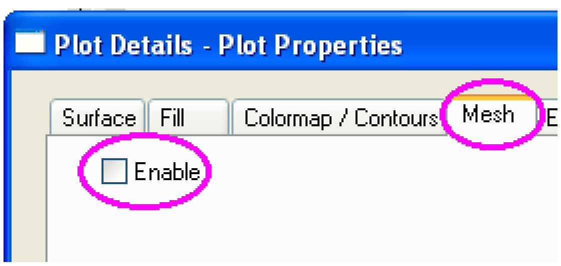


3. Click on the **Fill** heading to open the **Fill** dialog. In this dialog, choose the **Load Palette** radio button and then click the **Select Palette** button to choose the **Reef** palette. And make sure the **Link to Palette** check box has been enabled. Click the **OK** button to return to the **Plot Details** dialog.

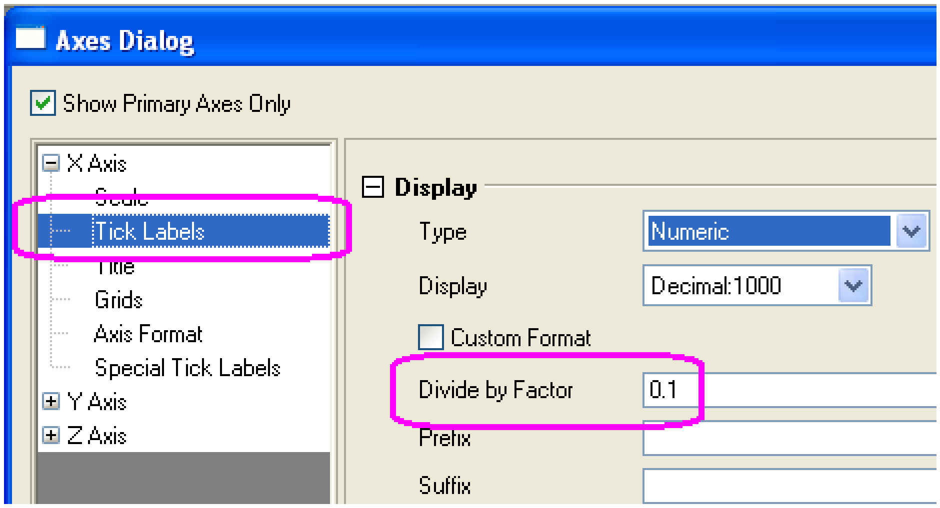


4. Click on the **Line** heading to open the **Contour Lines** dialog. Make sure **Apply to All** is checked, and set **Color** to **LT Gray**. Click the **OK** button to return to the **Plot Details** dialog.

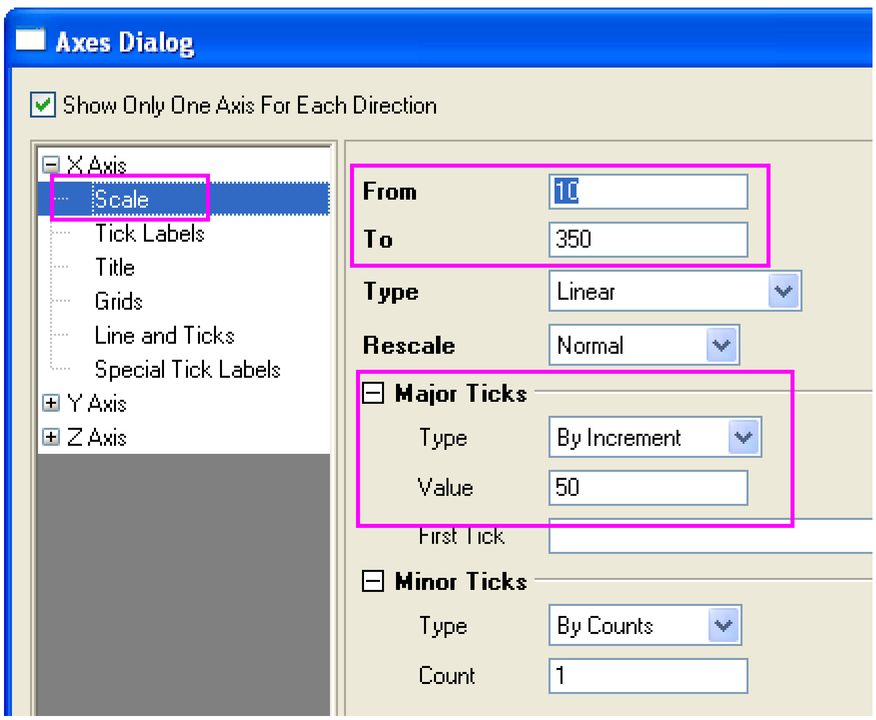
5. Activate the **Mesh** tab on the right panel. Uncheck the box before **Enable** as the following screenshot shows. Click **OK**.



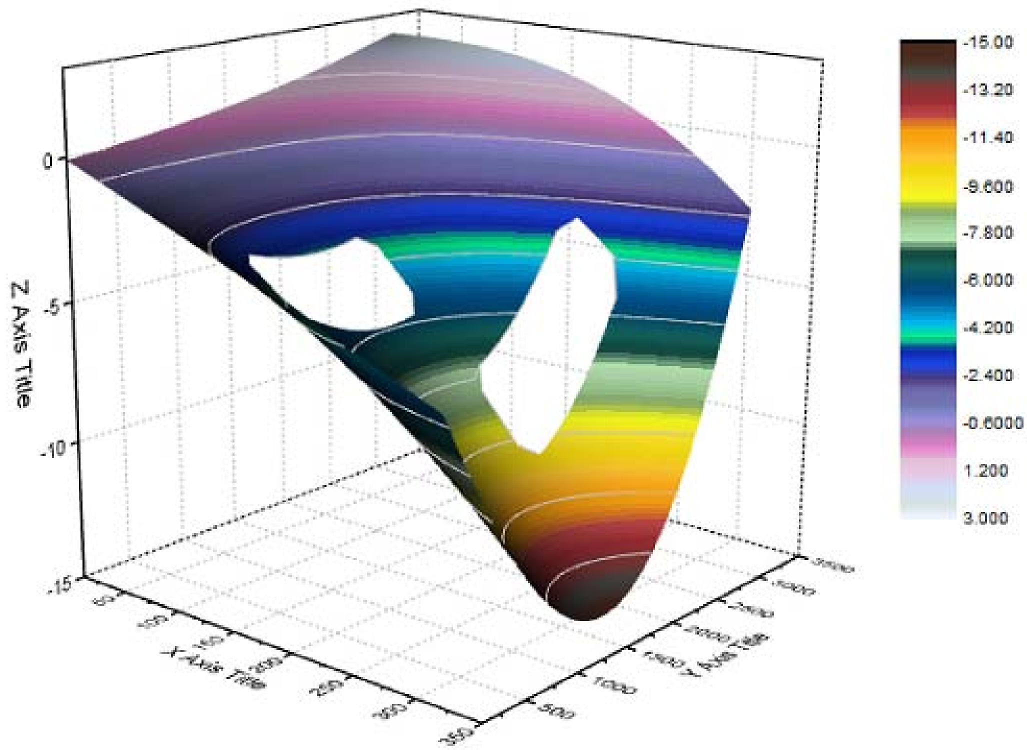
6. Double-click the X axis to open the **Axes Dialog**. In this dialog, select **Tick Labels** under the **X Axis** branch, set the **Divide by Factor** equal to **0.1** on the right panel, then click the Apply button.



7. Go to the **Scale** option on the left panel, set **From** and **To** equal to **10** and **350**, respectively. Under the **Major Ticks** option, set Type to **By Increment** and set Value to **50**.



8. In a similar fashion set options for the **Y Axis**. First, click on the **Y Axis** node to expand the list of options. Select **Tick Labels** and set the **Divide by Factor** to 0.01. Next, select the **Scale** option and set **From** and **To** equal to 100 and 3500, respectively. Then set Value to 500 under the Major Ticks option.
9. Finally, select the Z axis and choose the **Scale** option to set the **From** and **To** options equal to **-15** and **3**, respectively. Next, click the OK button. The final graph should resemble



5.8.5 Creating Intersecting Surface Plots from Worksheets

Summary

Many plot types, such as 3D Surface, 3D Bars, and Contour, can be created from data contained in a matrix object or from data arranged in a block of cells in a worksheet. The latter arrangement is referred to as a **Virtual Matrix**. Whereas a typical matrix object only supports a linear mapping of X and Y coordinates, a virtual matrix supports nonlinear mapping as well. In this tutorial, we show you how to create intersecting color map surface plots from virtual matrix data.

What you will learn

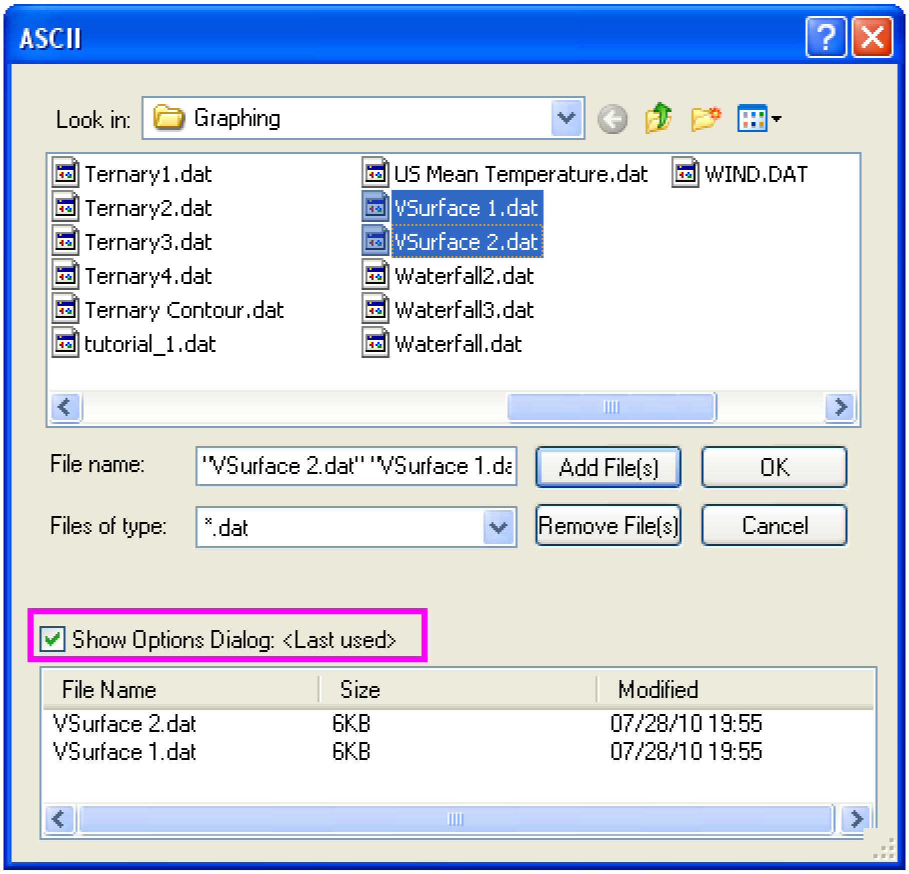
This tutorial will show you how to:

- Create a Color Map Surface plot using virtual matrix data from a worksheet
- Add one surface graph to another to create intersecting surface plots
- Set transparency

Create Surface Plots from Virtual Matrix Data

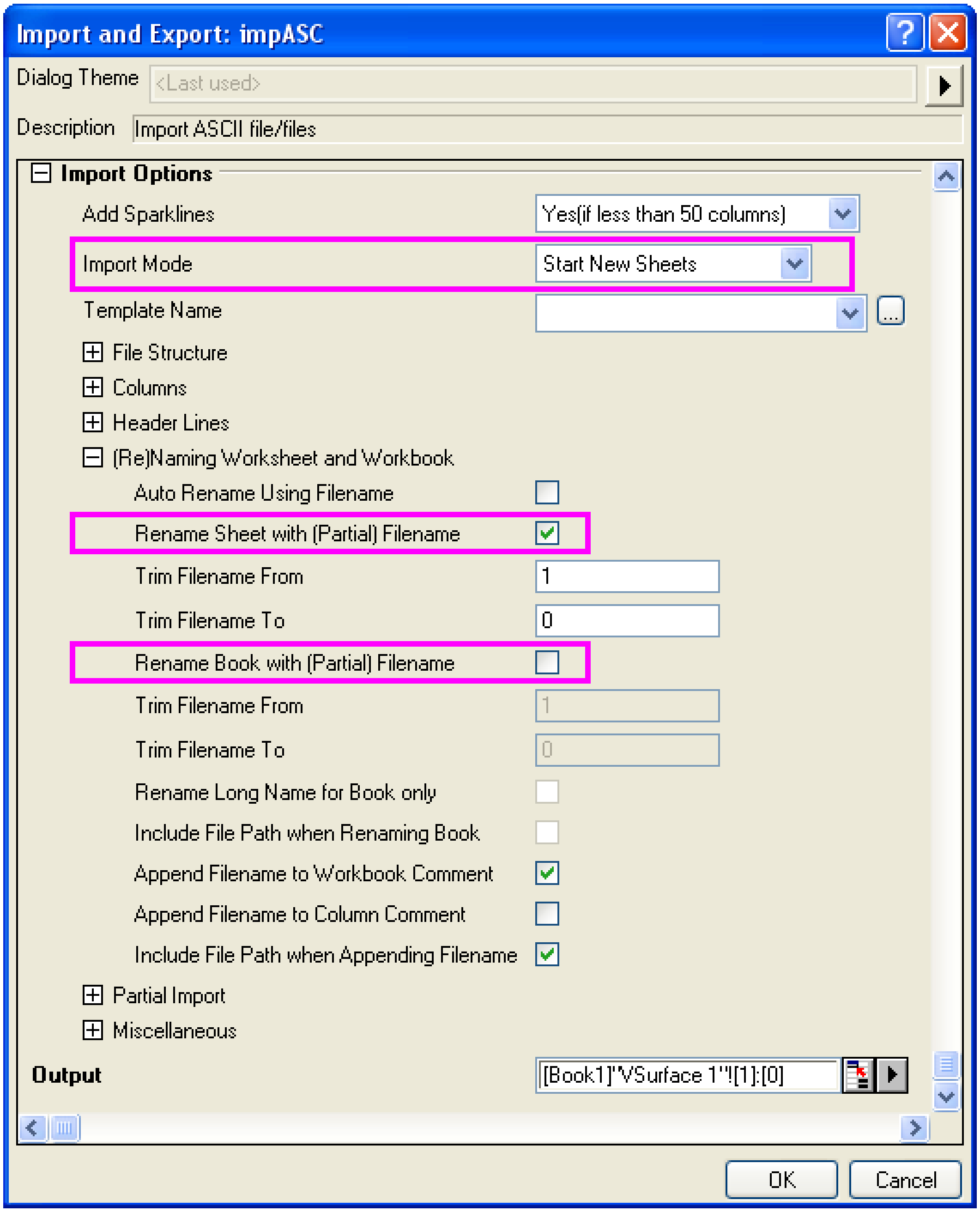
In this section, we will show you how to create surface plots from data in a worksheet.

1. Start a new project. Click the **Import Multiple ASCII** button  on the **Standard** toolbar. In the dialog that opens, navigate to the *\Samples\Graphing* folder and select *VSurface 1.dat* and *VSurface 2.dat*. Select the **Show Options Dialog** check box.

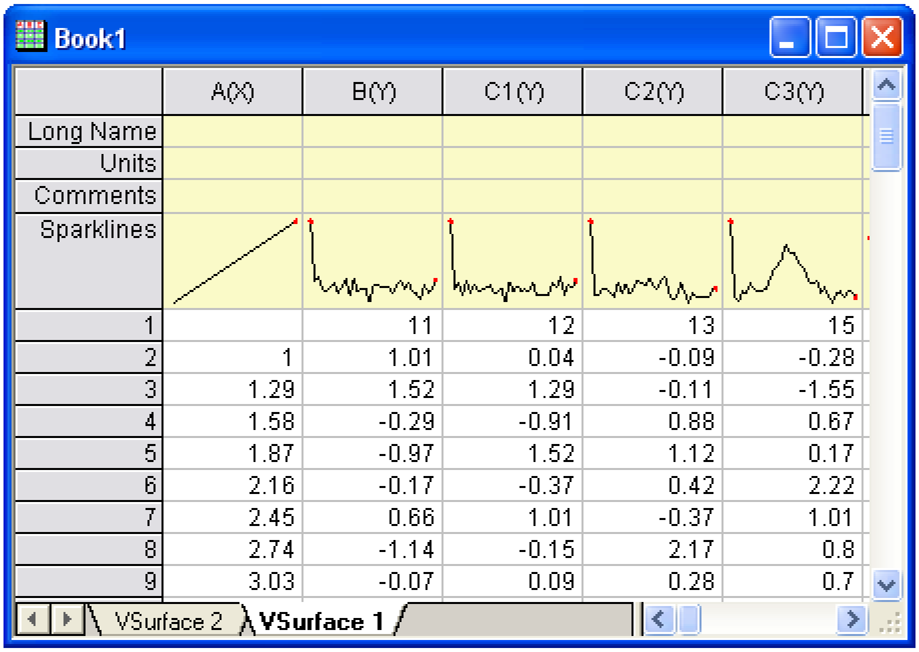


Press the **OK** button.

2. In the **impASC** dialog, select **Start New Sheets** from the **Import Mode** drop-down list, to import data files to different worksheets of the same workbook. Expand the **(Re)Naming Worksheet and Workbook** branch of **Import Options**. Check **Rename Sheet with (Partial) Filename** and uncheck **Rename Book with (Partial) Filename**, to rename the worksheet with the file name.

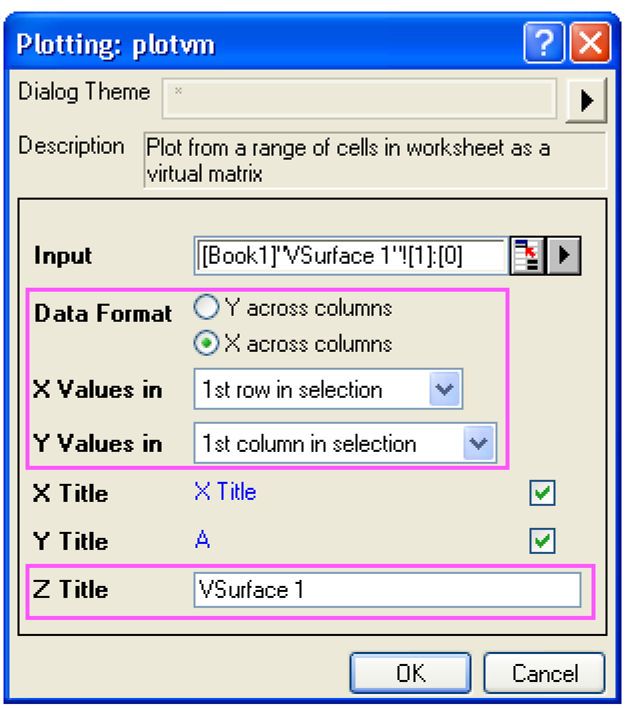


Click **OK** to import these two data files.

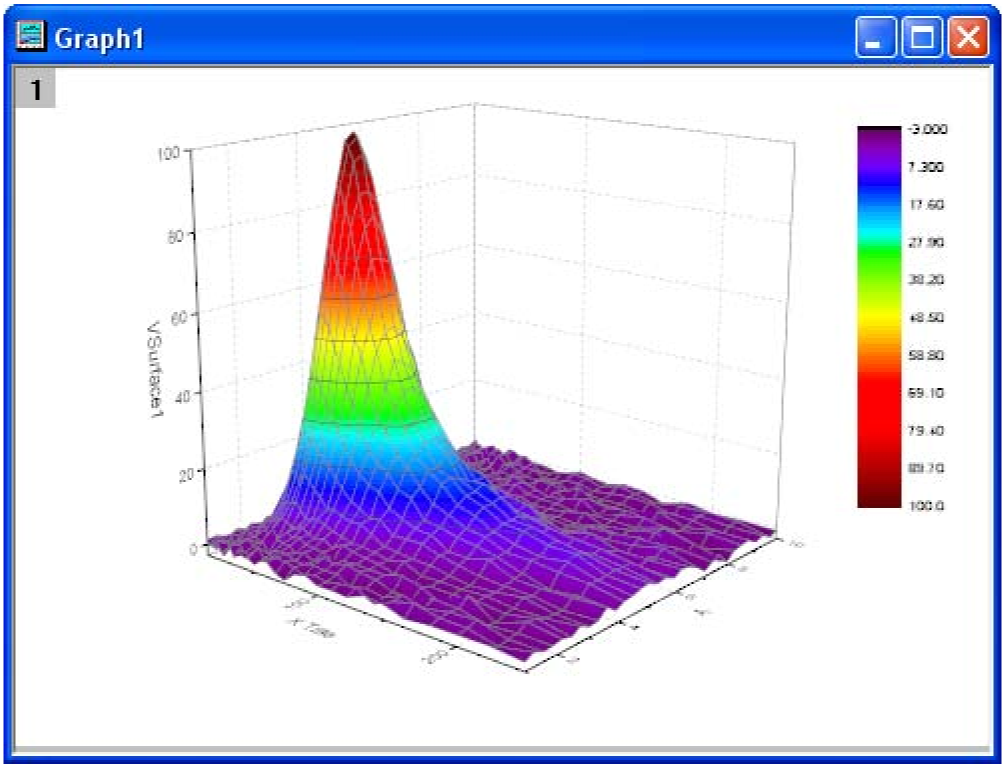


The data has X coordinate values in the top row and Y coordinate values in the first column. Note that the X coordinates have nonlinear spacing.

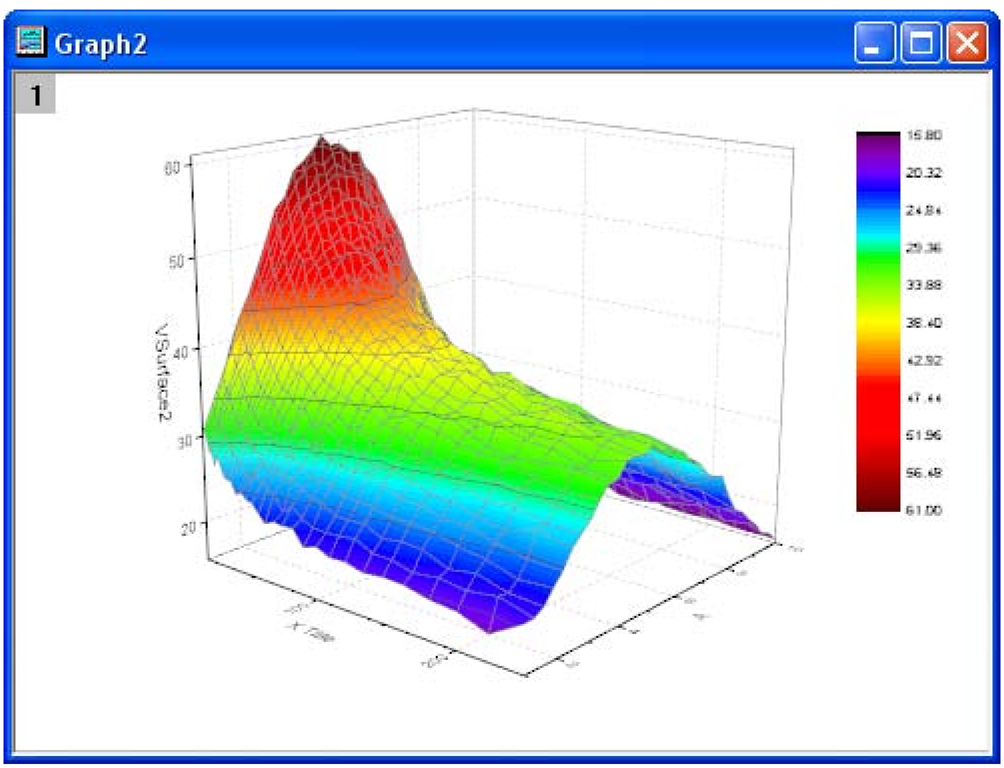
3. With the "VSurface 1" sheet active, click on top left corner of the workbook to select the entire sheet. Then select the menu item **Plot: 3D Surface: Color Map Surface**. This will open the **plotvm** dialog. This dialog opens any time a 3D or Contour plot menu item is invoked with a group of worksheet cells (a virtual matrix) selected. The dialog allows you to define where the X and the Y coordinate values are located. In the dialog, select **X across columns** from the **Data Format** drop-down list. Set the **X Values in** drop-down list to **1st row in selection** and the **Y Values in** drop-down list to **1st column in selection**:



Click **OK** to plot a Color Map Surface plot.



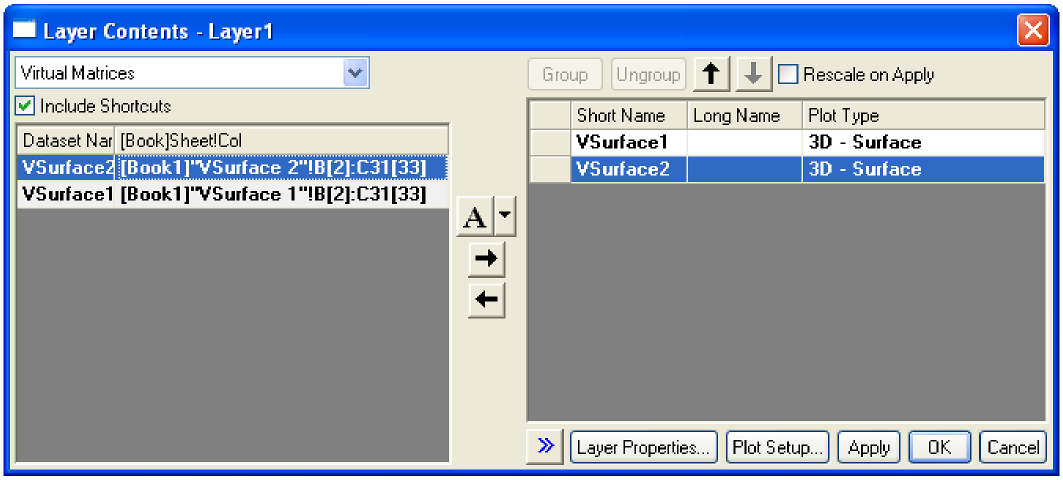
4. Repeat the last step to plot another Color Map Surface plot using data from the VSurface 2 worksheet. This time use **VSurface 2** as Z title in the **plotvm** dialog.



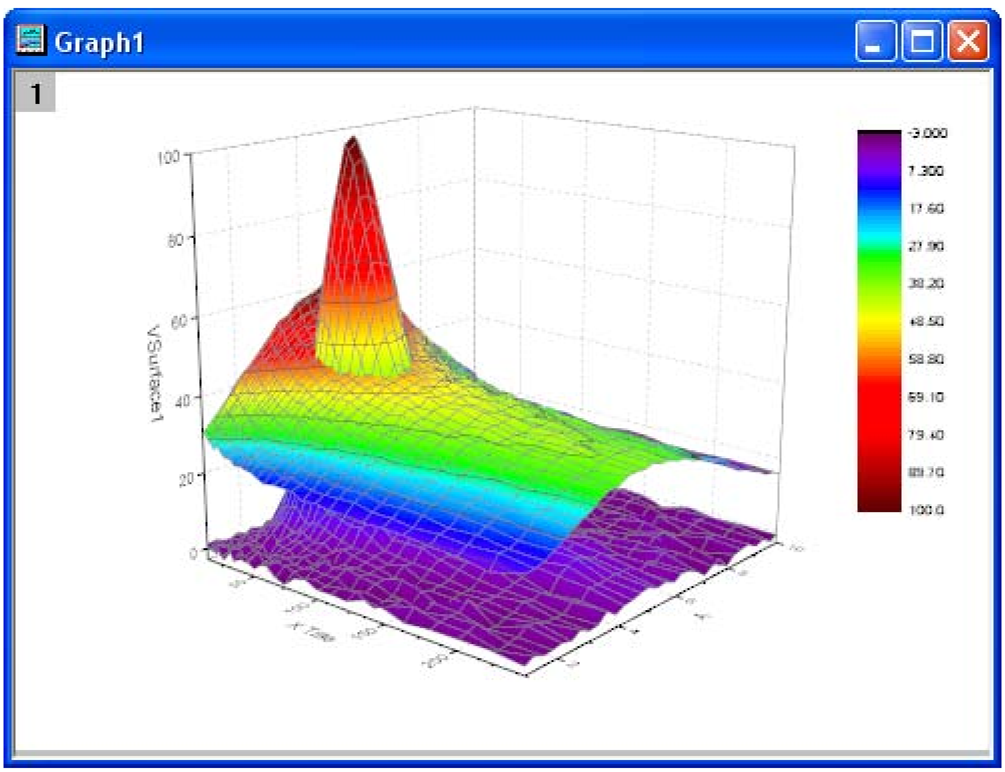
Add another Surface Plot to Layer

In this section, we will show you how to add a surface plot to a graph layer already containing another surface plot.

5. Activate Graph1. Right-click on the Layer 1 icon on the top left corner of the graph and select **Layer Contents** from the context menu. In the **Layer** dialog that opens, select vsurface2 in the Available Data box and add it to the Layer Contents box.

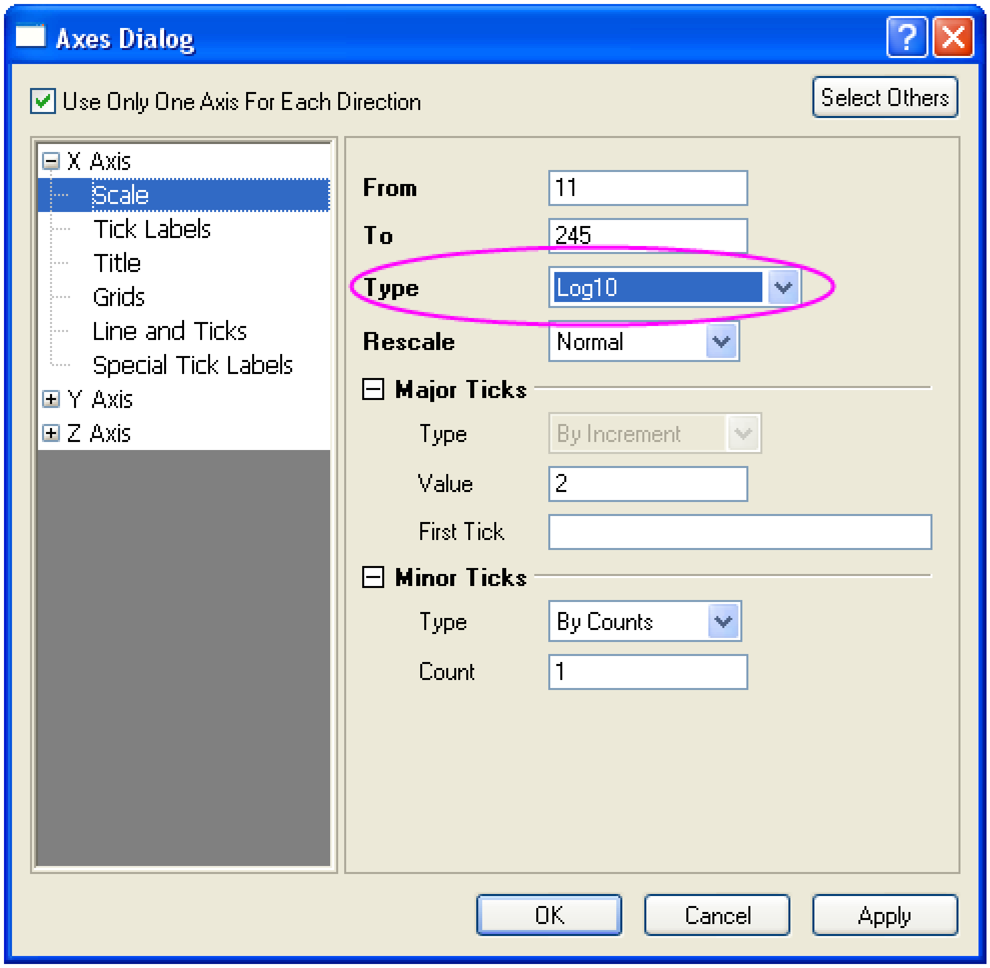


Click **OK** to add the second surface.



Note that Origin displays the two surfaces with proper intersection.

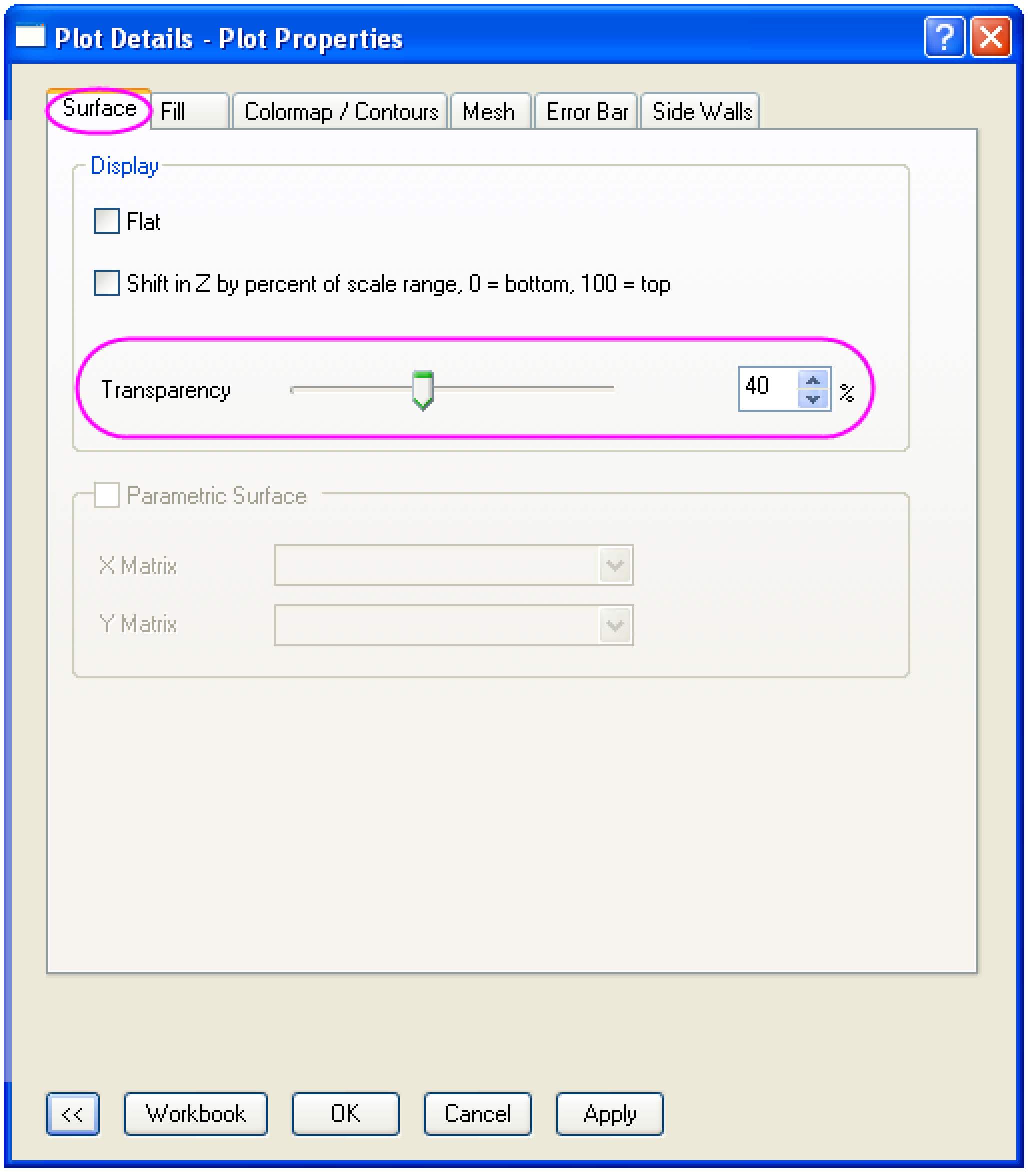
6. Double-click on X axis to open the **Axis Dialog**. On the **Scale** section, select **Log10** from the **Type** drop-down list.



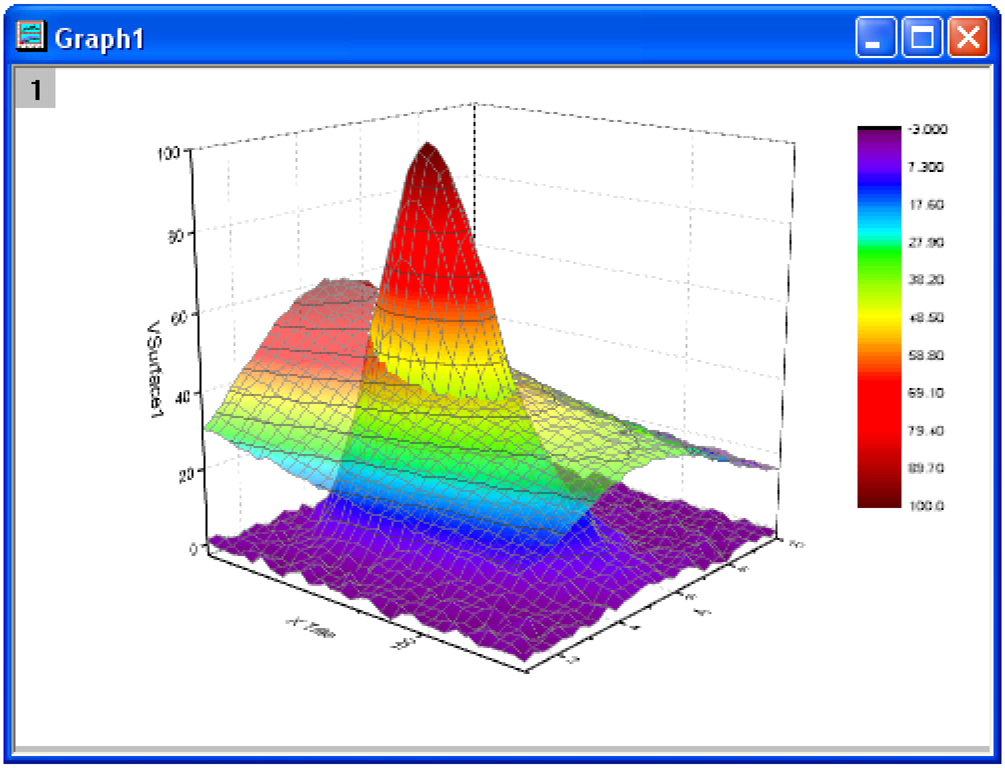
Set Surface Transparency

We will now set the transparency of the second surface so that parts of the first surface are visible in this intersection plot.

7. Switch to the **Surface** tab and move the **Transparency** slider to 50%.



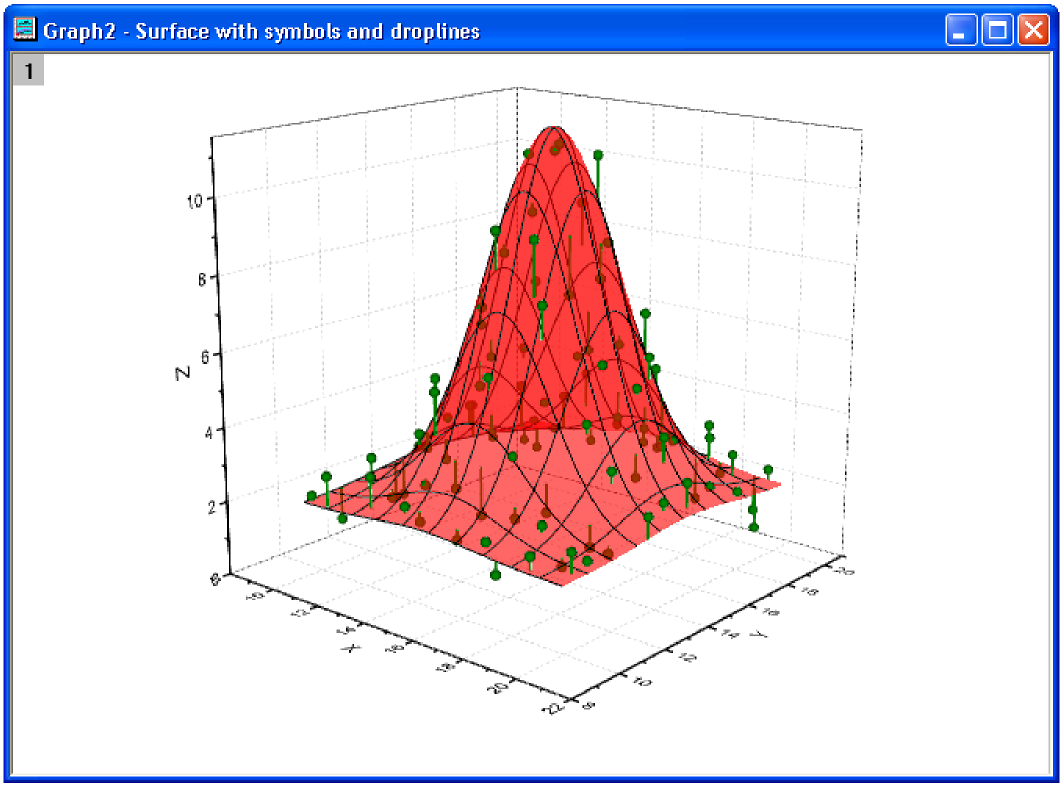
The final graph should look like the image below:

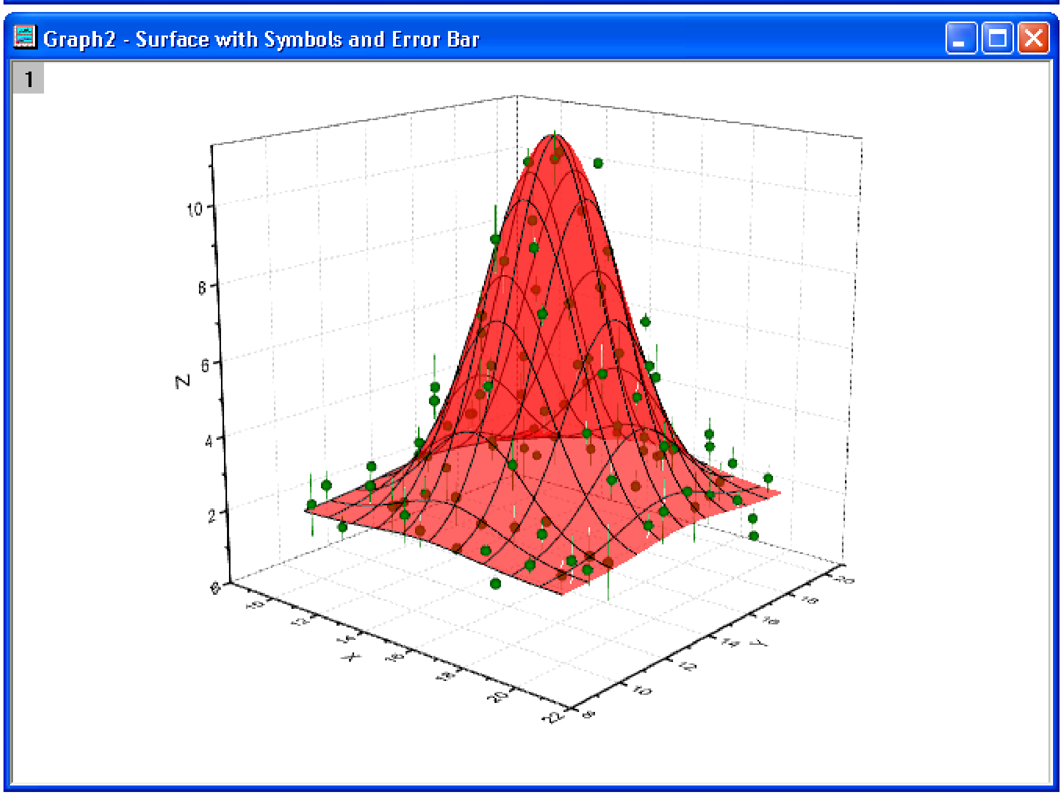


5.8.6 Surface with Symbols and Droplines

Summary

Origin supports transparency in most graph types. This feature permits the visibility of parts of a graph that would otherwise be obscured by overlapping plots. In this tutorial, we will add a fitted surface to a scatter plot, set transparency to make data points "behind" the surface visible, and add drop lines and error bars for data points.





Minimum Origin Version Required: Origin 8.5 SR0

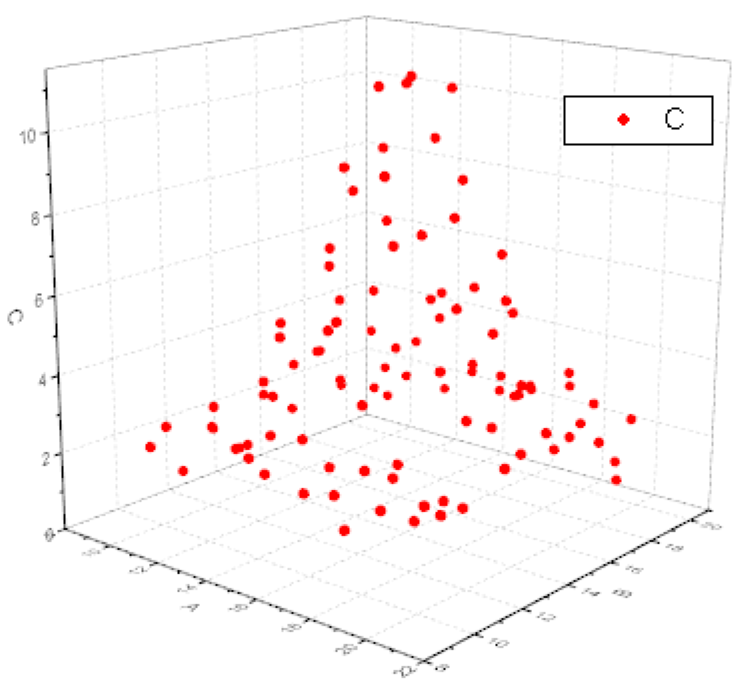
What you will learn

- Create a 3D scatter plot from worksheet data and add a color fill surface to it.
- Draw drop lines or error bars for data points.
- Set surface transparency.

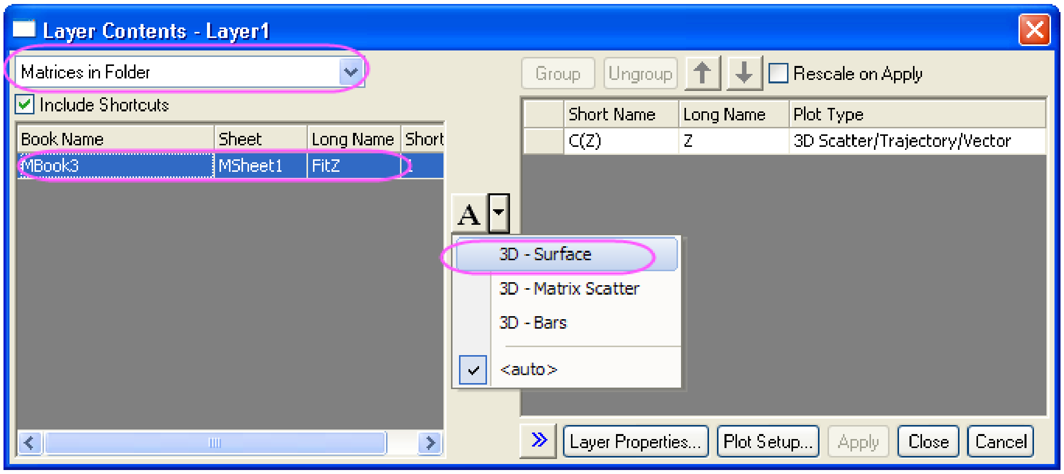
Steps

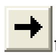
Create a surface graph with symbols and droplines

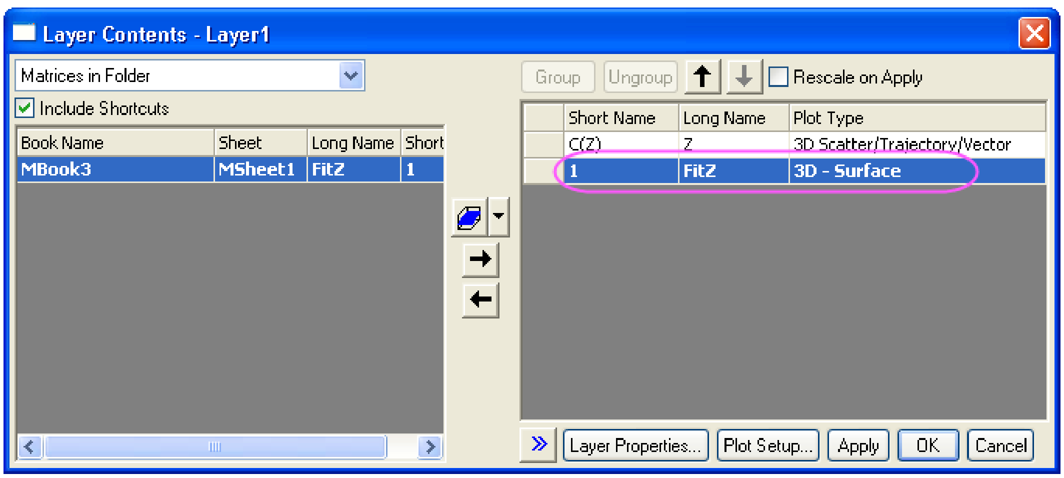
1. Open the **3D openGL Graphs** folder by clicking **File: Open Sample Projects: 3D OpenGL Graphs** from the main menu.
2. Under this folder, go to **3D OpenGL Graphs: Graph with Transparency: Surface with symbols and droplines** folder.
3. Active the worksheet *XYZRandomGauA*, highlight the XYZ to plot a 3D scatter by selecting **Plot: 3D Symbol/Bar/Vector: 3D Scatter** from the main menu.



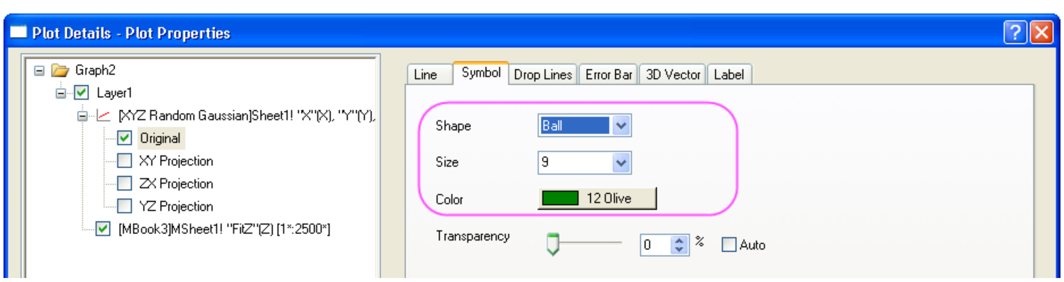
4. Now we will add a 3D Color Fill Surface plot to this 3D scatter graph. In the top left corner of the graph window, double-click the layer **1** icon to open the **Layer Contents** dialog.
5. In the **Layer Contents** dialog box, select **Matrices in Folder** from the dropdown menu in the upper left. In the left panel, select **MBook 3**, Click the **triangle button** next to **A** and select **3D ? Surface**.



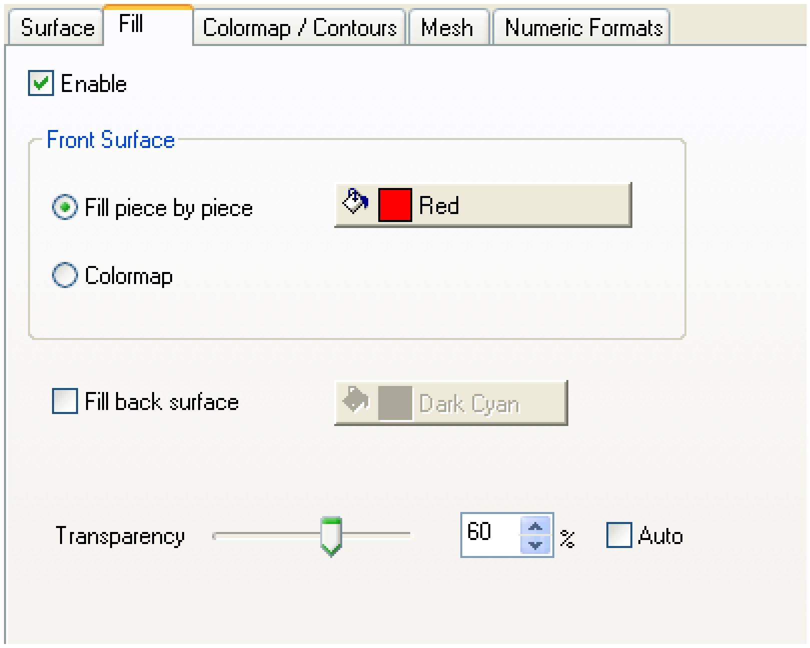
Click  to add it to the right panel.

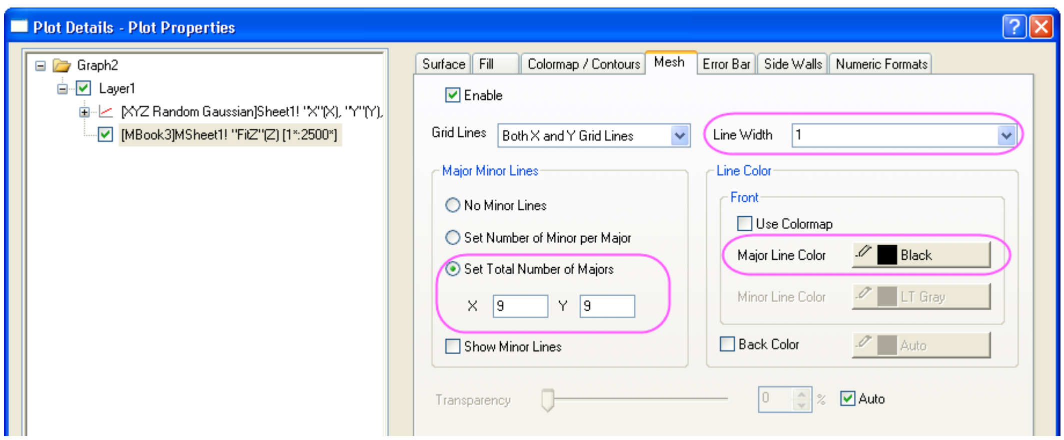


6. Click **OK** button to apply the change and close the **Layer Contents** dialog. Double-click the graph to open the **Plot Details** dialog box. In the left panel of this dialog, expand all items and select **Original** (or choose **Plot Properties** from the **Format** menu). In the right panel, select the **Symbol** tab. Set **Shape** as *ball* and set **Size** as *9* and **Color** as *Olive*.

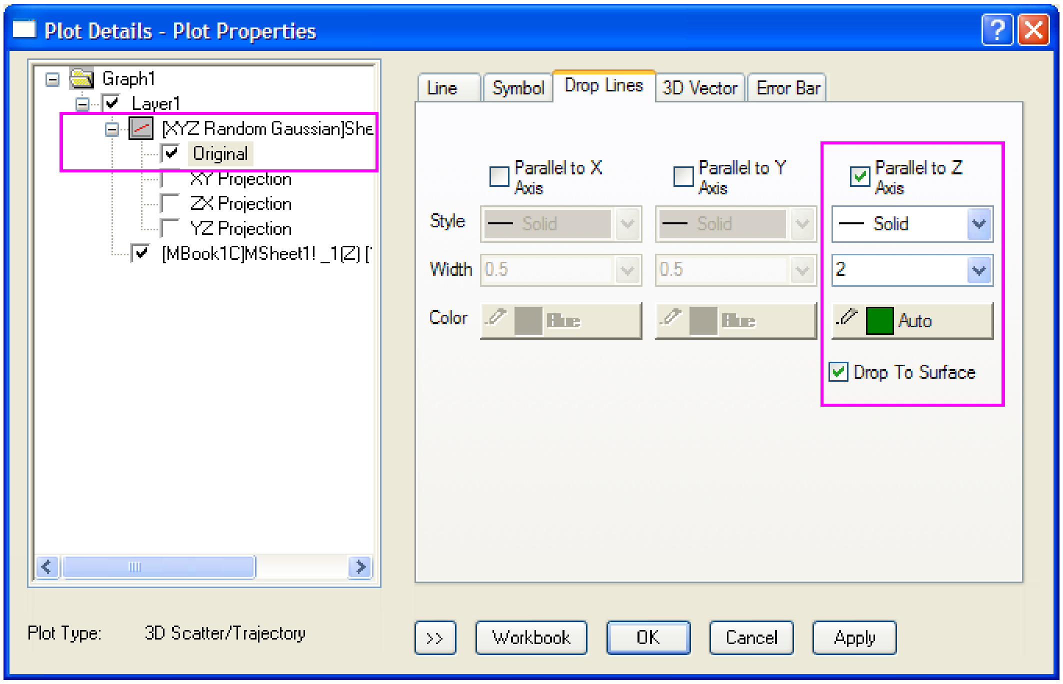


7. In the left panel, select the surface plot, and then go to the **Fill** tab in the right panel, change the **Fill piece by piece** as **Red**. Set **Transparency** to **60**. Go to the **Mesh** tab, set the **Line Width** as **1** and **Set Total Number of Majors** as *X=9* and *Y=9*.

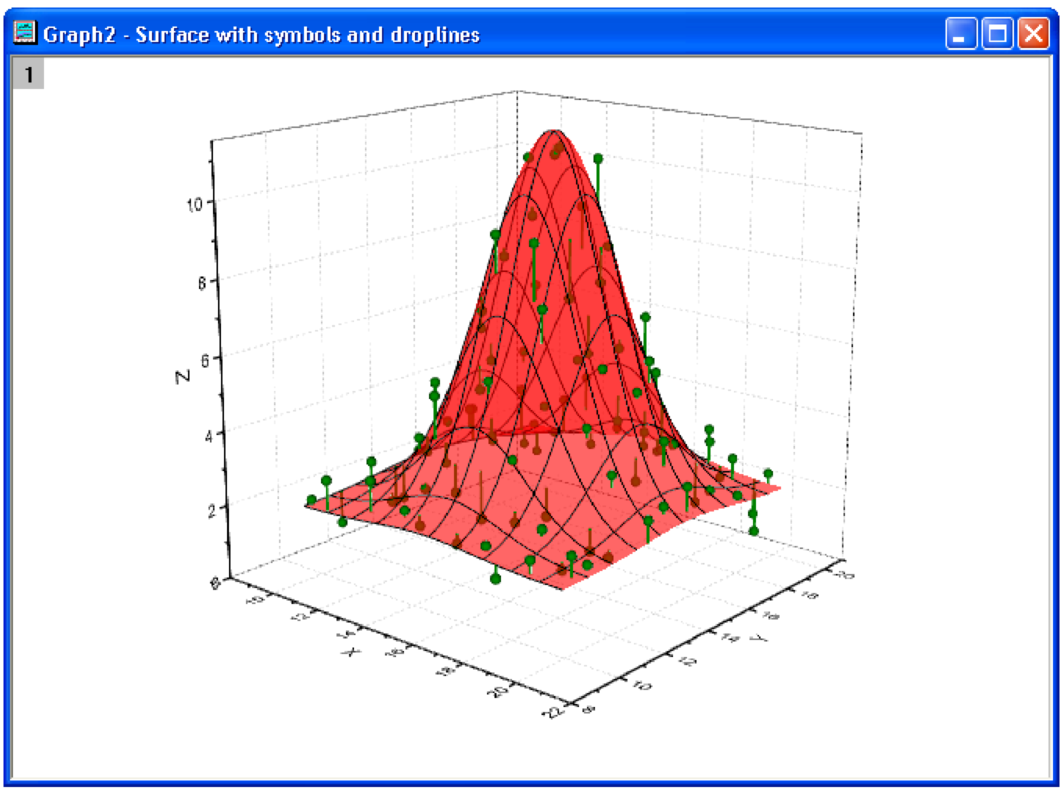




8. Now we will draw the drop lines. In the left panel, select the scatter plot (Original). In the right panel, select the **Drop Lines** tab. Select the **Parallel to Z Axis** check box. Select the **Drop to Surface** check box. Change the drop line **Width** to **2**, and the color to **Auto**.



9. Click **OK** to close the dialog. Your final graph should look like this:

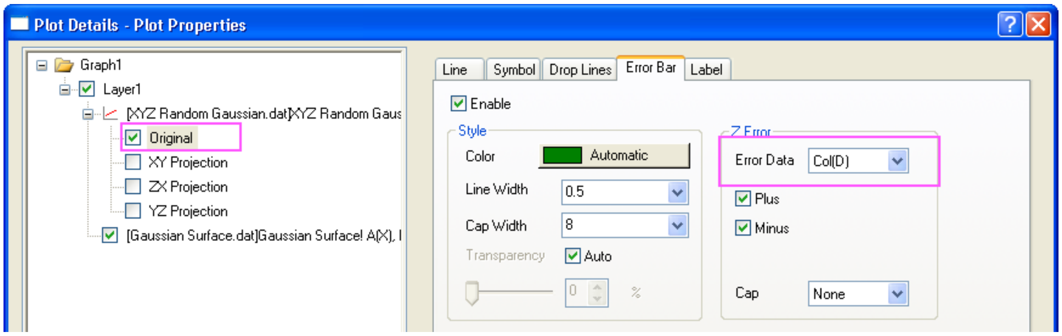


Create a surface with symbols and error bar

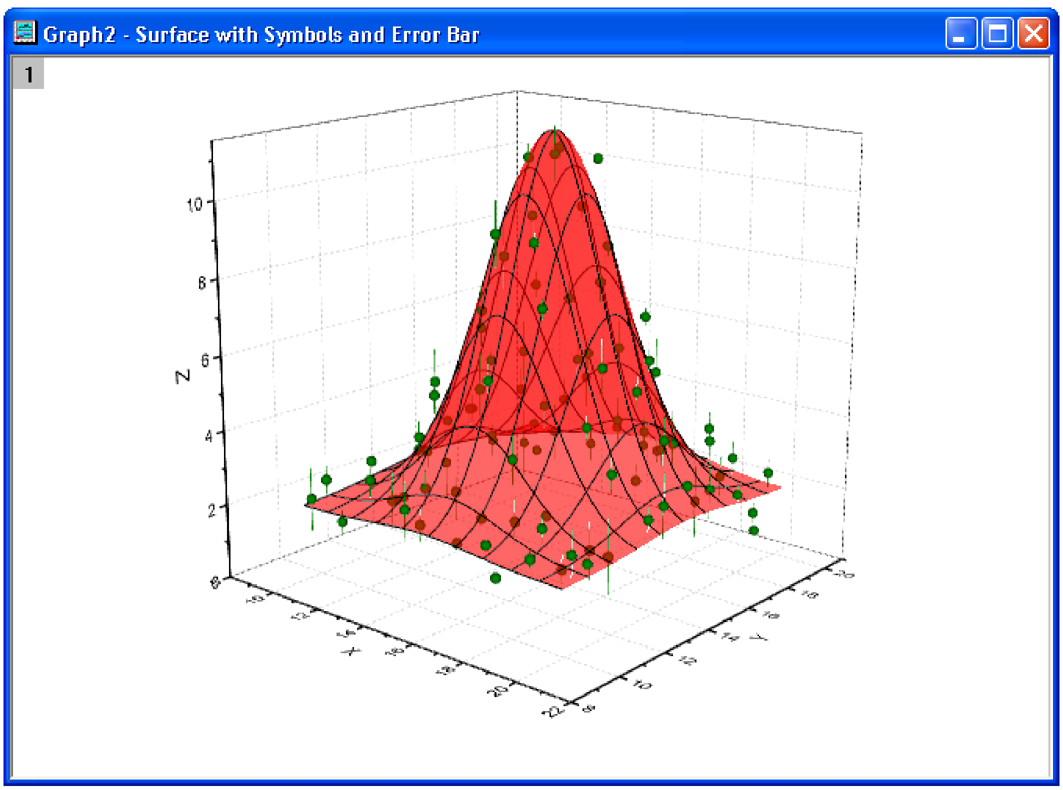
1. Follow steps 1 through 7 above to create a 3D surface graph with a scatter plot.

2. Add a new column in the worksheet "XYZRandomGauA". Select the column and right-click. In the **Fill Column with** context menu, click **Uniform Random Numbers.**

3. Open the **Plot Details** dialog. Select the plot (Original) in the left panel. On the **Error Bar tab**, select the **enable** check box, and select **Col(D)** from the **Error Data** dropdown menu.



4. Click **OK** to close the dialog. Your final graph should look like this:



5.8.7 Intersecting Color Surfaces

Summary

Origin supports multiple intersecting surfaces.

Minimum Origin Version Required: Origin 8.5 SR0 - however, this tutorial uses a project file only available starting with 8.5.1

What you will learn

This tutorial will show you:

- How to create intersecting color surfaces from different matrix objects.
- How to customize the colormap surface plot.
- How to construct additional color scale for multiple colormap surfaces.

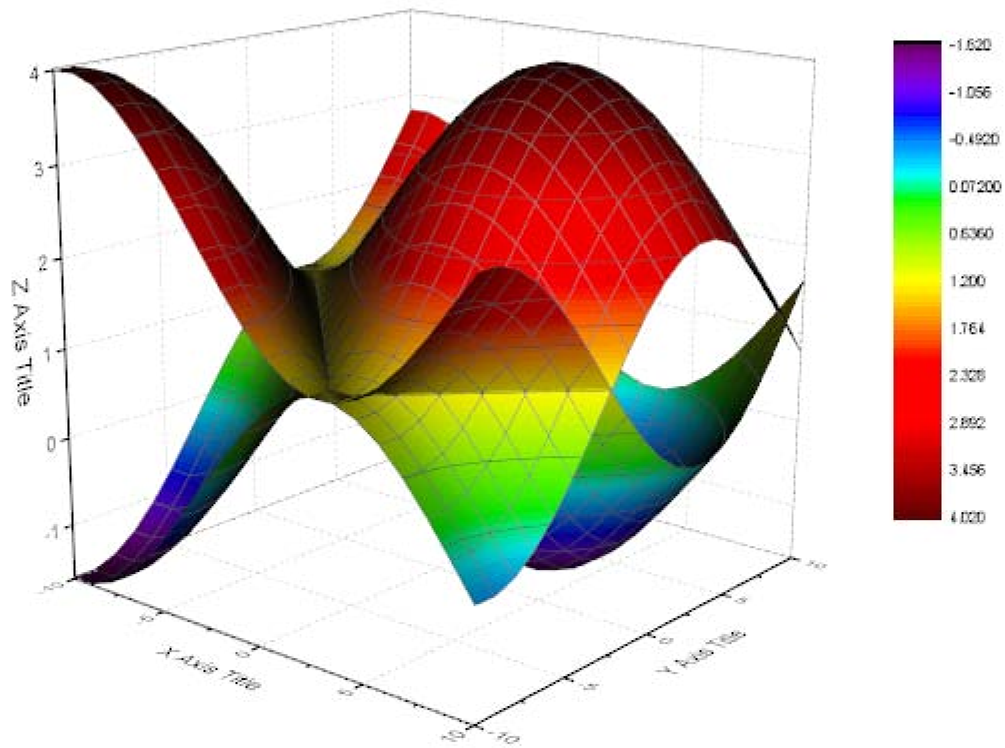
Steps

1. Choose "File:Open" and browse to open the matrix book **Intersecting_Color_Surfaces.ogm** in the **Samples\Graphing** folder. Note that the matrix displays two thumbnail images above the data, one for each matrix object in the matrix.

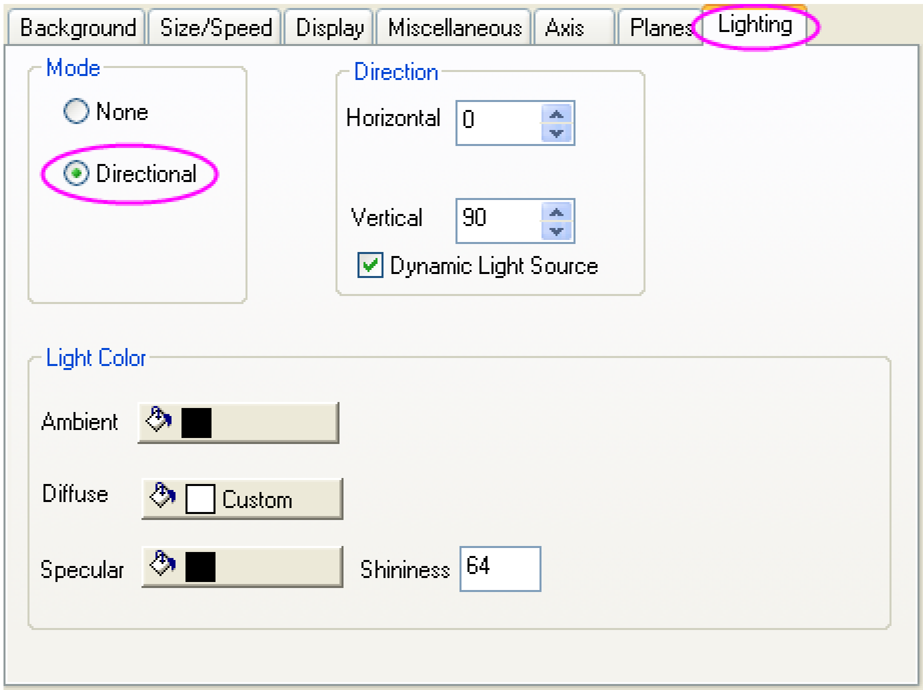
To plot intersecting surfaces, each matrix object must have identical dimensions and XY Mapping. This requirement is automatically fulfilled when the two objects are contained in the same matrix.

See Hierarchy of Origin Objects for additional information.

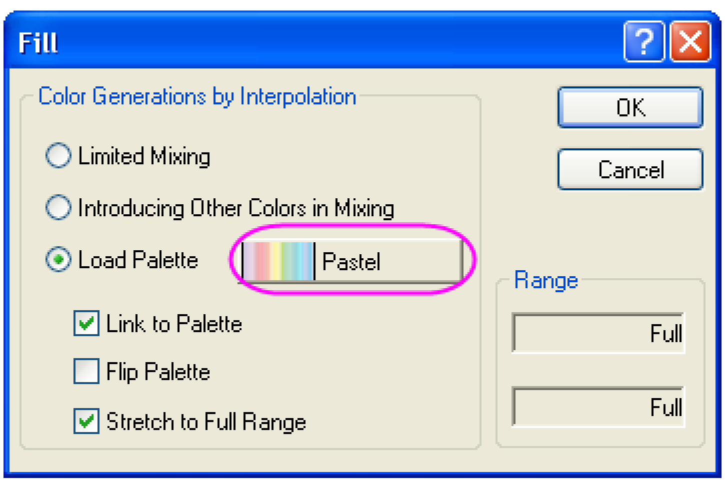
2. Select either thumbnail image. From the main menu, select **Plot**, then point to **3D Surface**, and then click **Multiple Colormap Surfaces**. This will generate an intersecting surface plot of every matrix object in the matrix sheet.



3. Open the **Layer Properties** dialog by double-clicking on the graph layer, or by selecting **Format:Layer Properties...** from the main menu. Go to the **Lighting** tab, and change the **Mode** to **Directional** to turn on the lighting effect.



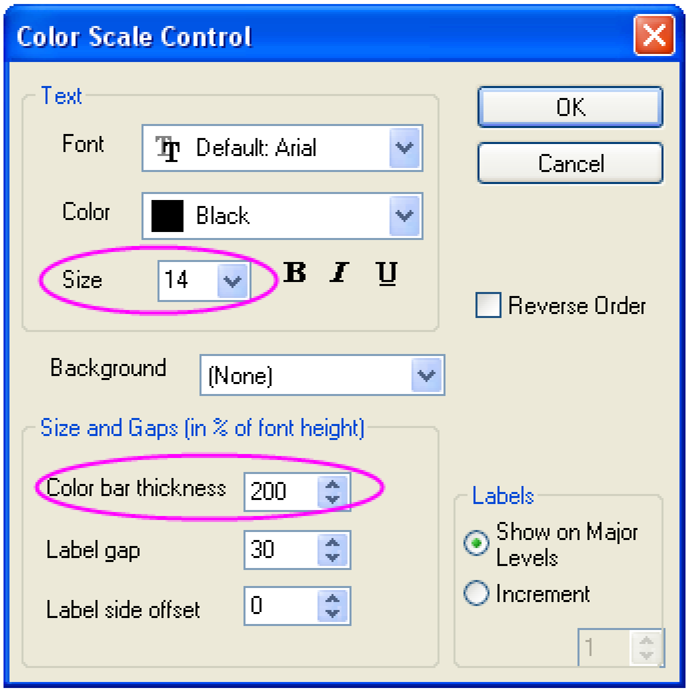
4. Expand the **Layer 1** branch in the left panel, select **[MBook36]MSheet1!_1(Z)[1*:400*]** to open the **Plot Properties** dialog. In the right panel, go to the **Colormap/Contour** tab, uncheck the **Enable Contours** check box to turn off contour lines
5. Click the **Fill...** column header to open the **Fill** dialog box. Click the default **Rainbow** palette to open the palette menu. On the palette menu, select **Pastel**. Click **OK** to close the **Fill** dialog box.

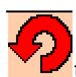


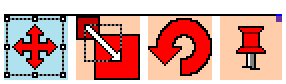
6. Select **[MBook36]MSheet1!_2(Z)[1*:400*]** in the left panel. Uncheck the **Enable Contours** check box. And click **Apply** to see the modifications so far.
7. Keep **[MBook36]MSheet1!_2(Z)[1*:400*]** selected, and go to the **Mesh** tab and clear the check box before **Enable** to turn off the mesh lines. Do the same for **[MBook36]MSheet1!_1(Z)[1*:400*]**.

   Click **OK** to close the **Plot Details** dialog box and update the graph.

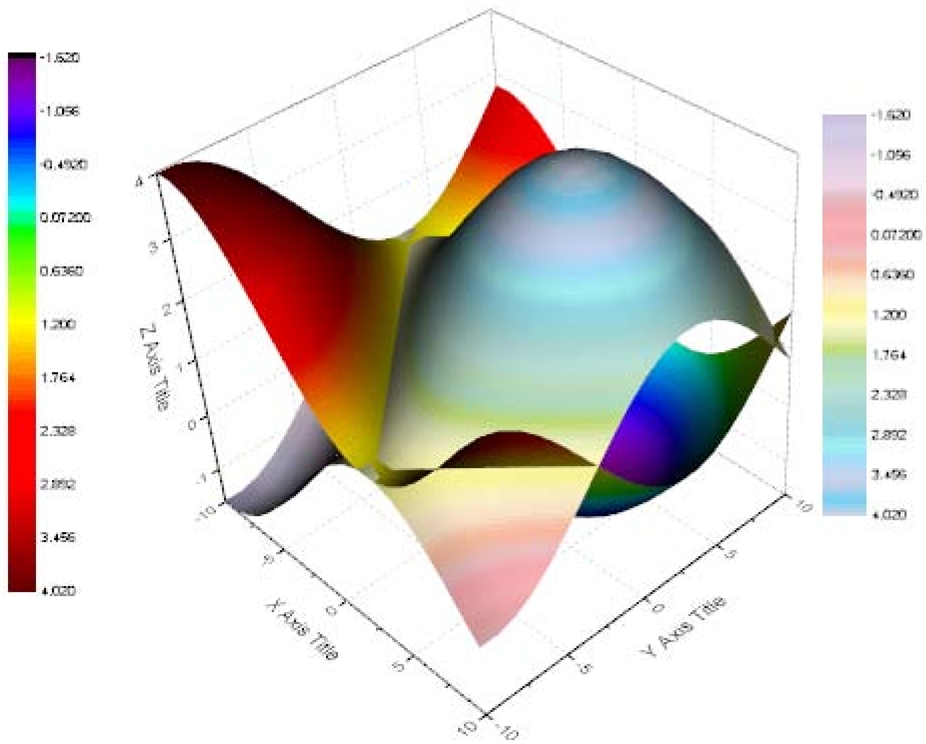
8. Now we will construct the an additional color scale for the second surface, change that plot to be active by click **Data:2 [MBook36]MSheet1!_1(Z)[1*:400*]**, then click **Graph:New Color Scale** to create color scale for this plot.
9. Double click on the newly added color scale object to open the **Color Scale Control**, change the settings according to the screenshot below, then drag and put the color scale object to a desired place on the layer:



10. You can use the 3D rotation button  to rotate the graph and gain an overhead perspective. This button is a part of the 3D annotation toolbar which can be brought out by clicking the graph layer once



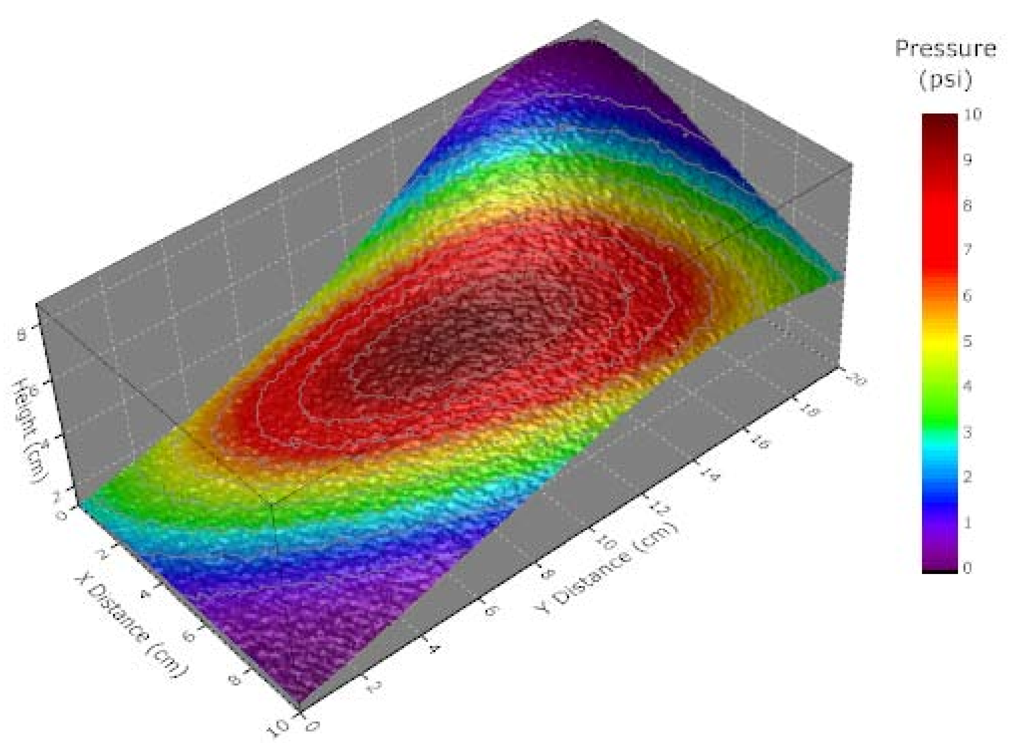
Your final graph should look something like this:



5.8.8 Colormap from Second Matrix

Summary

Origin can represent four-dimensional data by color-mapping a surface plot using a second matrix.



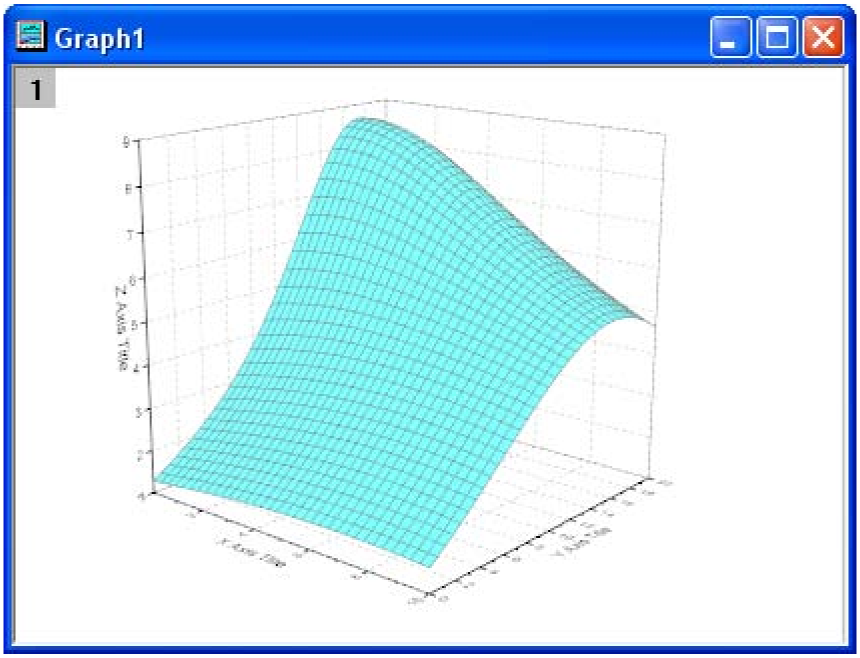
Minimum Origin Version Required: Origin 8.5 SR0

What you will learn

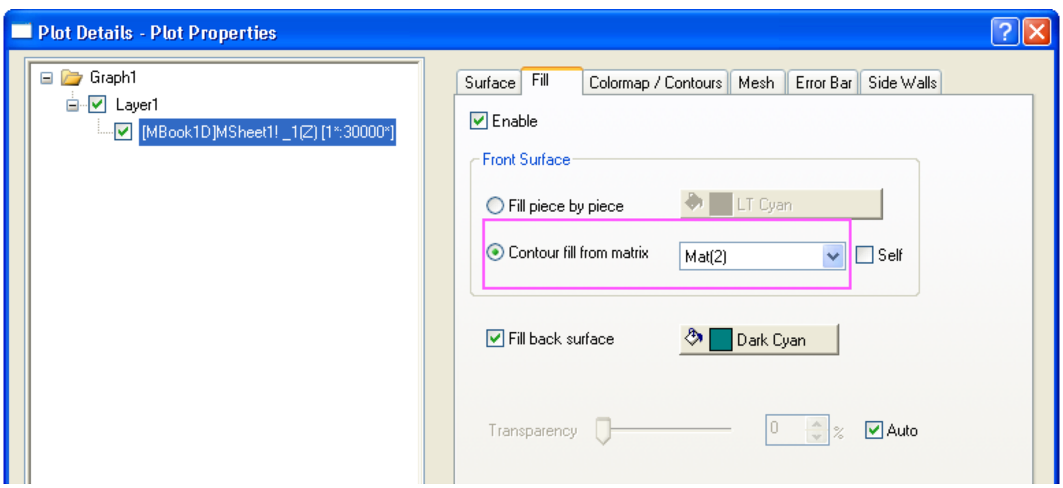
- Create a Color Fill Surface from a matrix.
- Color map a surface plot using a second matrix.
- Customize color map levels and palette.
- Control lighting on the graph (From Origin 9 SR0)

Steps

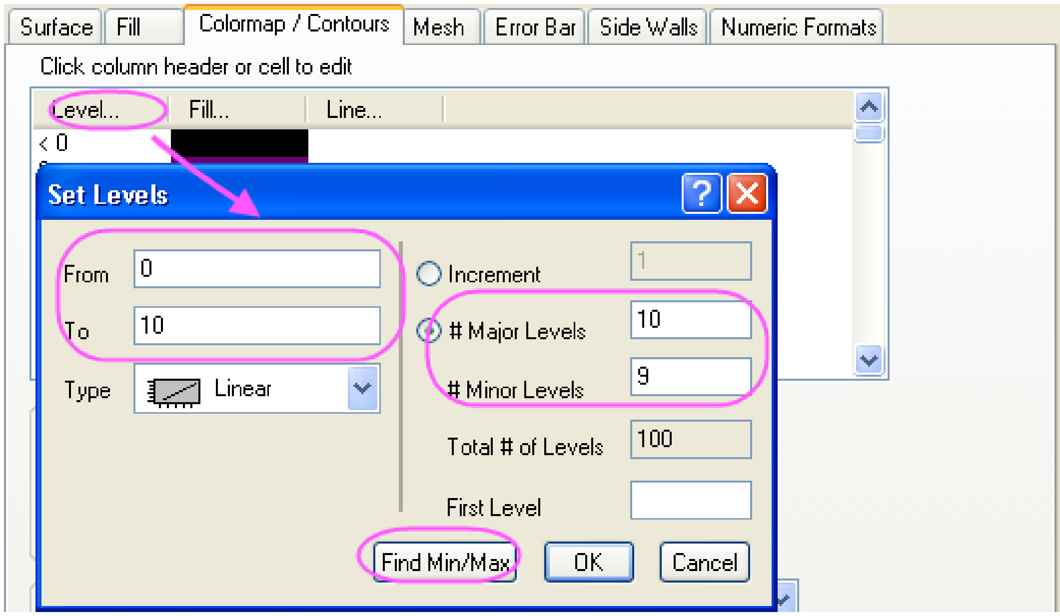
1. Click **File : Open** to open *Colormap_from_Second_Matrix.ogm* under the folder **Sample\Graphing**. You should see two image thumbnails above the matrix data, just under the title bar. (If you do not see image thumbnails, right-click on the matrix title bar and select **Show Image Thumbnails**.) Select image thumbnail **1**.
2. On the main menu, click **Plot**, point to **3D Surface**, then click **Color Fill Surface** to generate a surface plot.



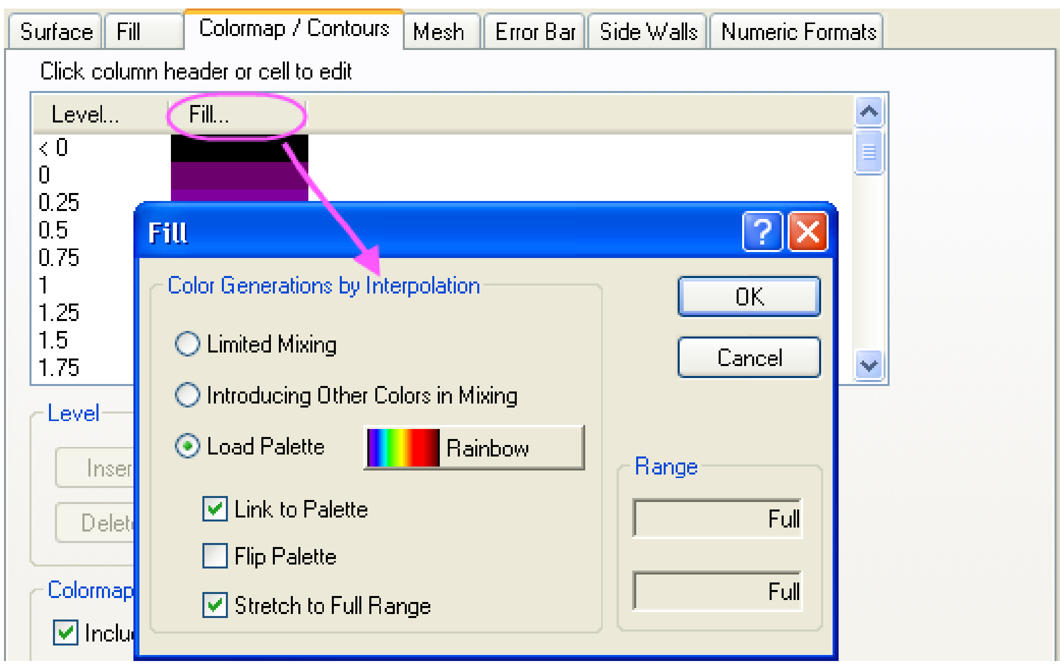
3. Double click on the plot to open the **Plot Details** dialog. In the left panel, select **(MBook1D)MSheet1**. In the right panel, select the **Fill** tab. Choose **Coutour Fill from Matrix**. and select **Mat(2)** for front surface.



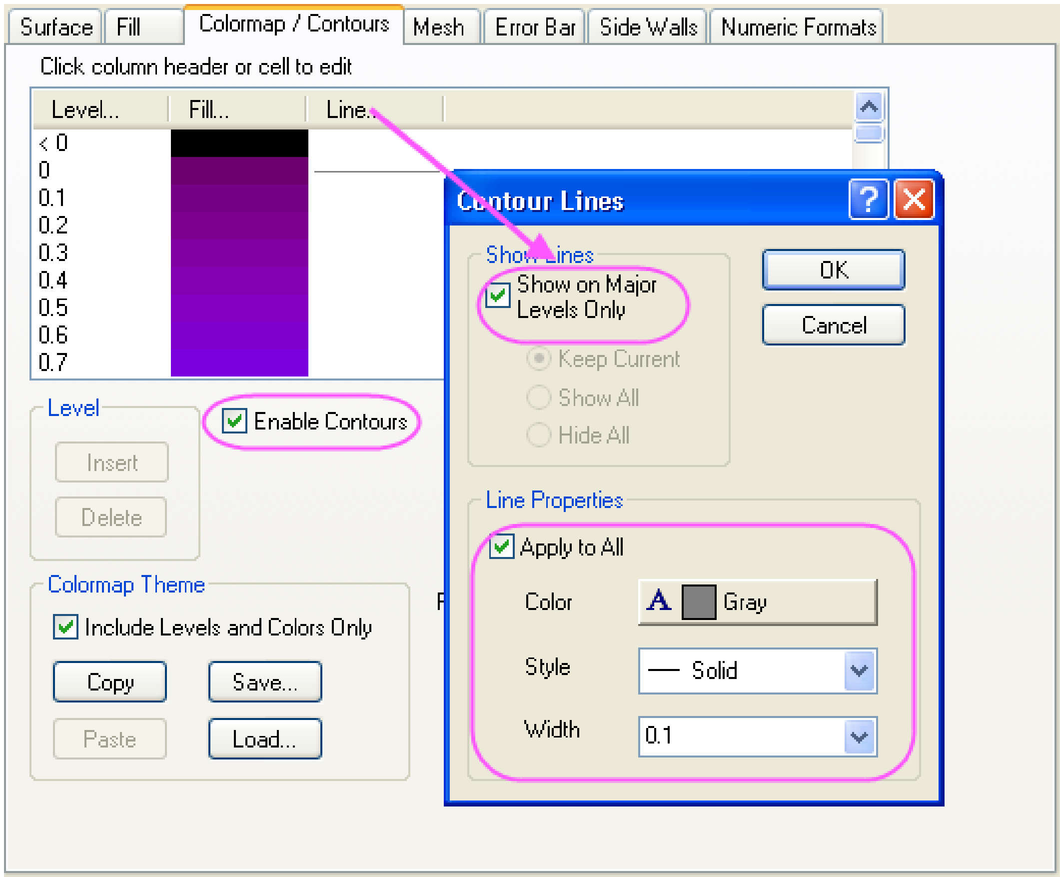
4. Go to **Colormap/Contours** tab, click the **Level...** title to open the Set Levels dialog. In this dialog, click **Find Min/Max** button and set the **#Major Levels** and **#Minor Levels** as *10* and *9*.Click OK button to close the dialog.



5. Click the **Fill...** title to open the **Fill** dialog. select **Load Palette** and then click **Select Palette** button to select **Rainbow** palette.Click OK button to close this dialog.

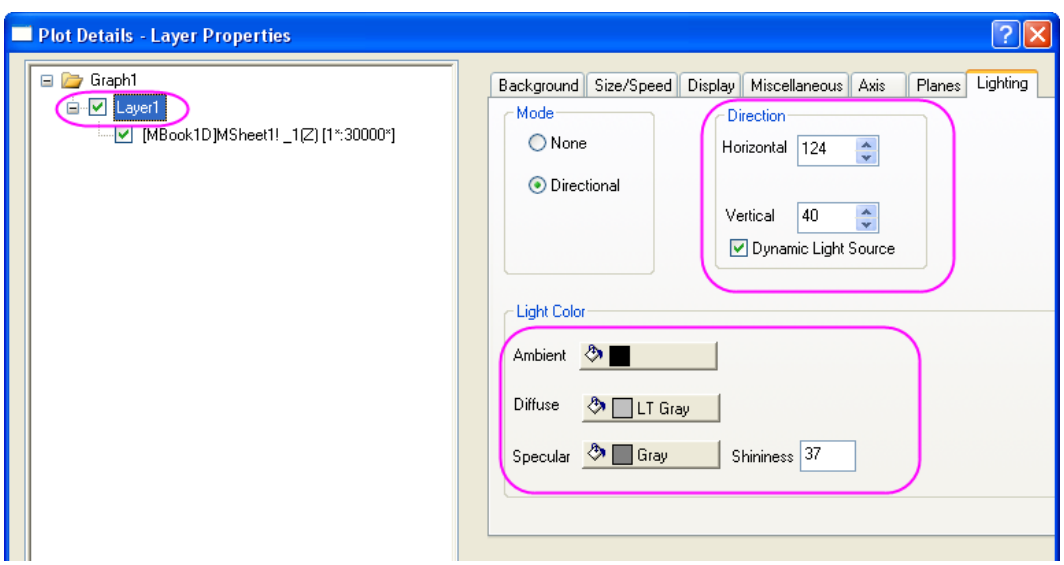


6. Check the **Enable Contours** check box. And then click **Line...** to open the **Contour Lines** dialog. In this dialog, check the **Show on Major Levels Only** check box and set the **Line Properties** as below. Click OK button to close the dialog.

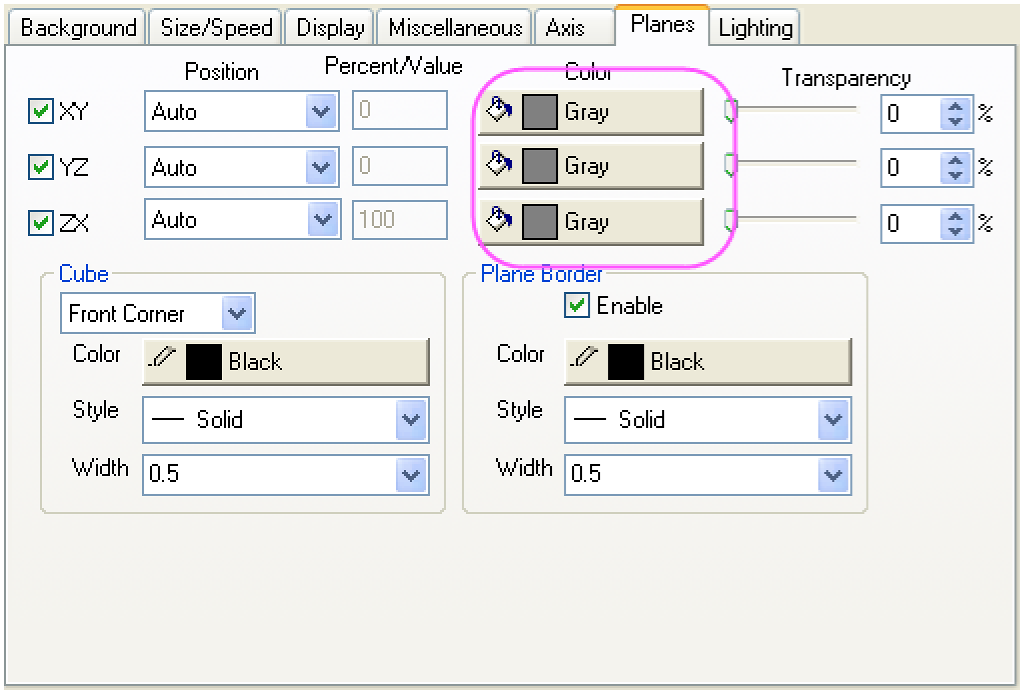


7. Go to the **Mesh** tab,uncheck the **Enable** box to disable the mesh lines.

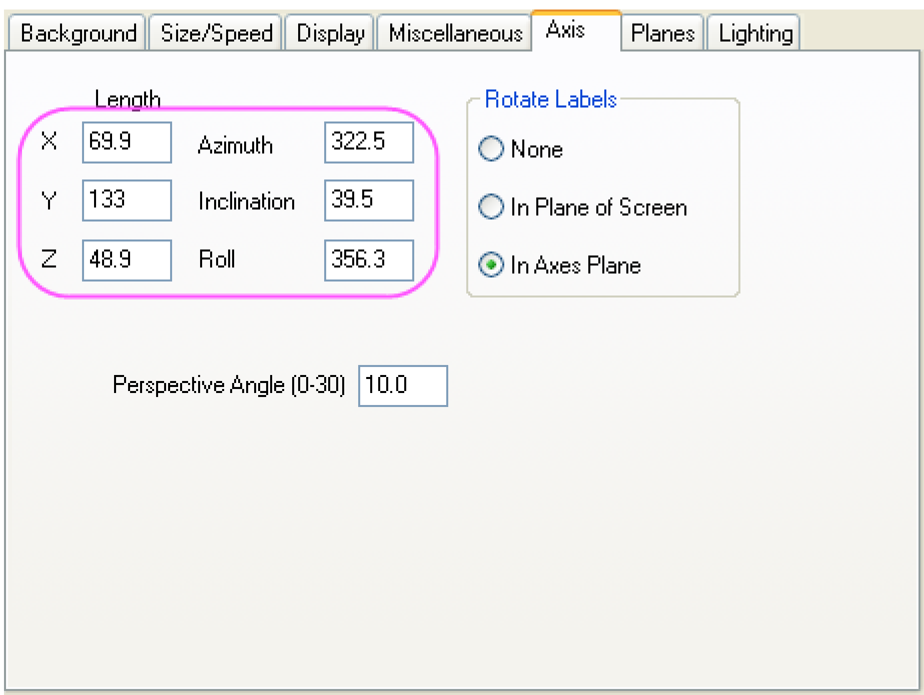
8. Select **Layer1** in the left panel, go to **Lighting** tab in the right panel. Select **Directional** under **Mode**. Set **Horizontal** and **Vertical** as *124* and *40* , and change the color of **Diffuse** as *LT Gray* and **Specular** as **Gray**. Then set the **Shininess** as *37*.



9. Go to the **Planes** tab, set the color as **Gray** and select **Front Corner** for the Cube drop-down menu.

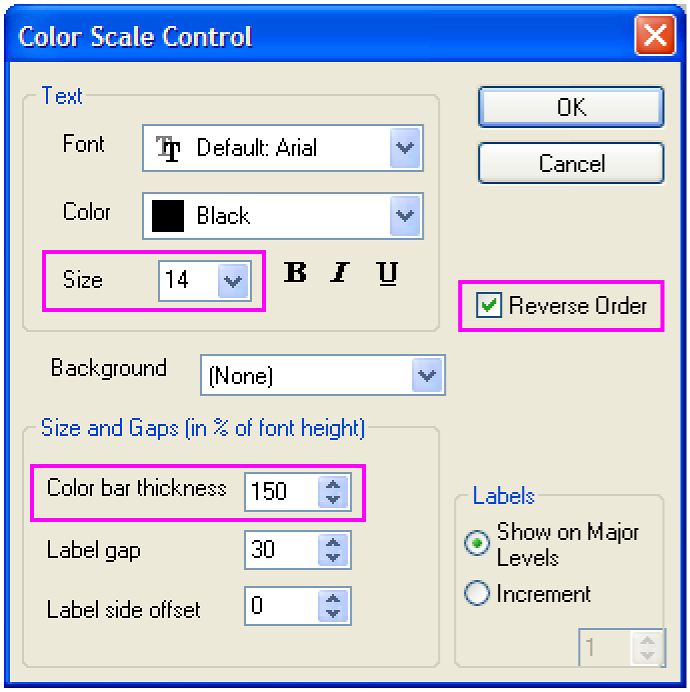


10. Go to Axis tab and do settings as below. Click **OK** button to close this dialog.



11. Keep the graph window active. Select **Graph: New Color Scale** to add a new color scale in the graph window.

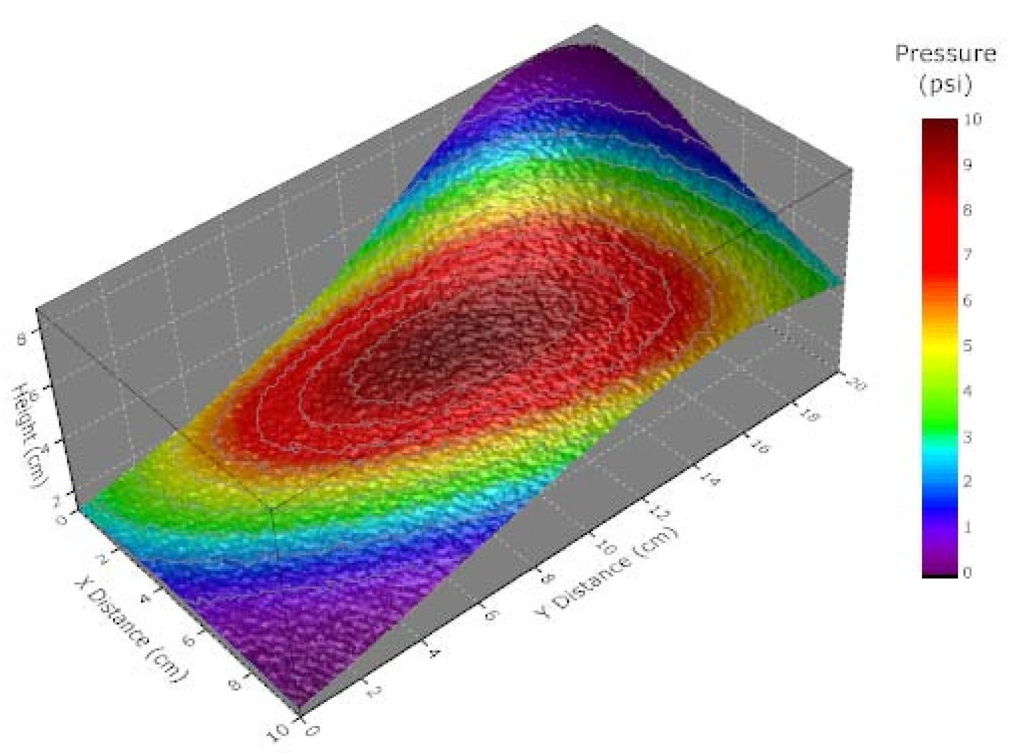
12. Double-click anywhere on the color scale to open the **Color Scale Control** dialog box. Change text **Size** to **14**, and **Color bar thickness** to **150**. Select the **Reverse Order** check box.



Click **OK** to close the **Color Scale Control** dialog box.

13. Double-click the X, Y and Z axis title and enter "X distance", "Y distance" and "Height". Add a text object "Pressure(psi)" above the color scale.

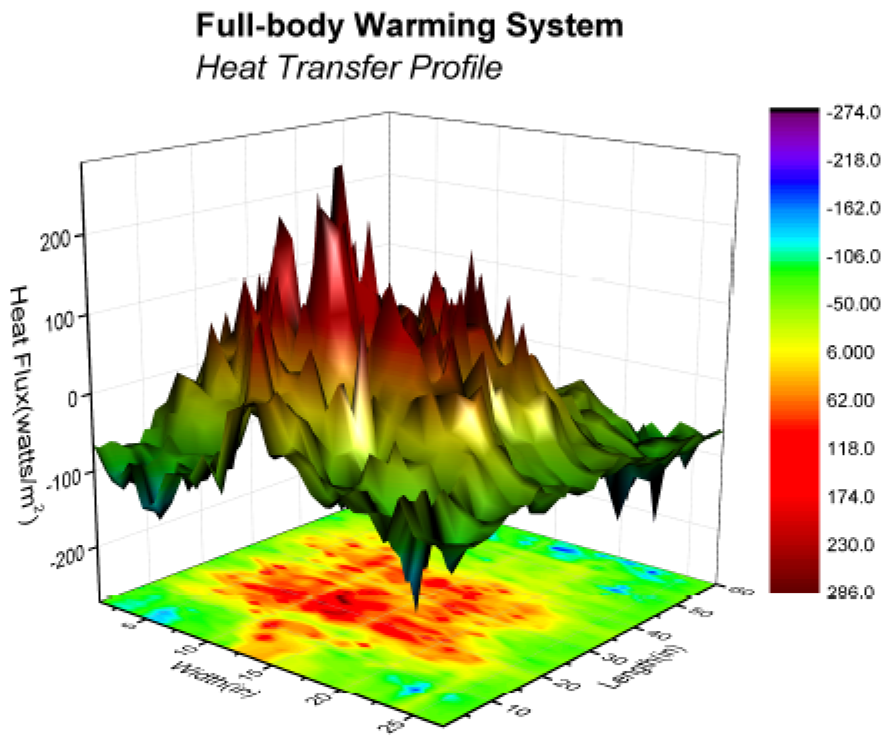
14. Your final graph should look like this:



5.8.9 Color Map Surface Graph

Summary

This tutorial will show you how to create a 3D color map surface.



What will you learn

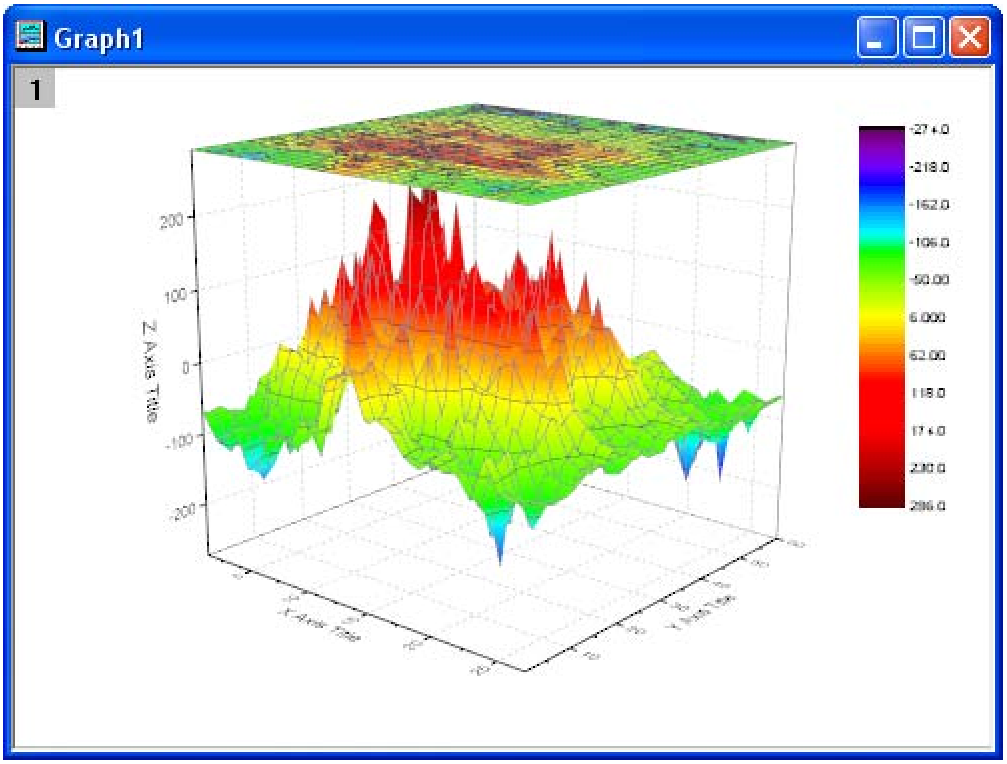
This tutorial will show you how to

- Create a 3D color map surface graph
- Customize the 3D surface graph

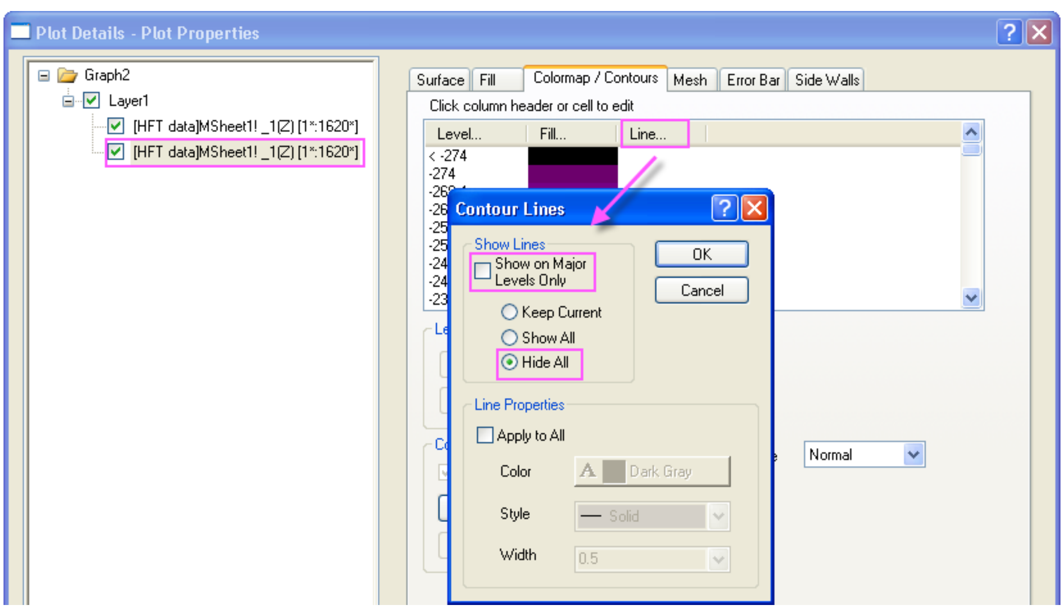
Steps

This tutorial is associated with this Graph Galley page.

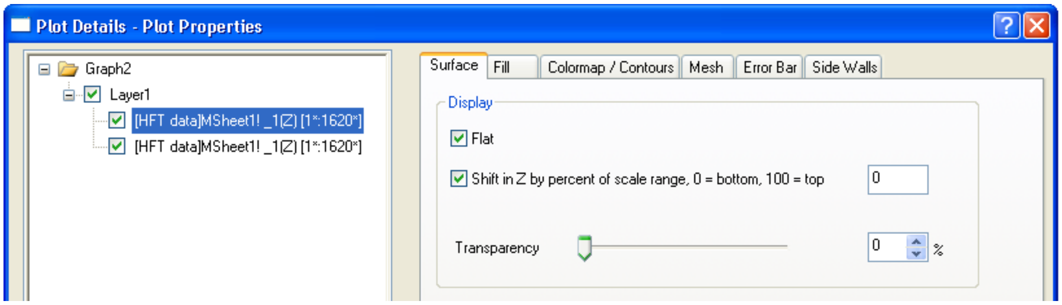
1. Download the project by clicking the Download Project link at the top of this page.
2. Open the project in the zipped file. Highlight **HFT data** matrix and select **Plot: 3D Surface: Colormap Surface with Projection** to create a 3D graph.



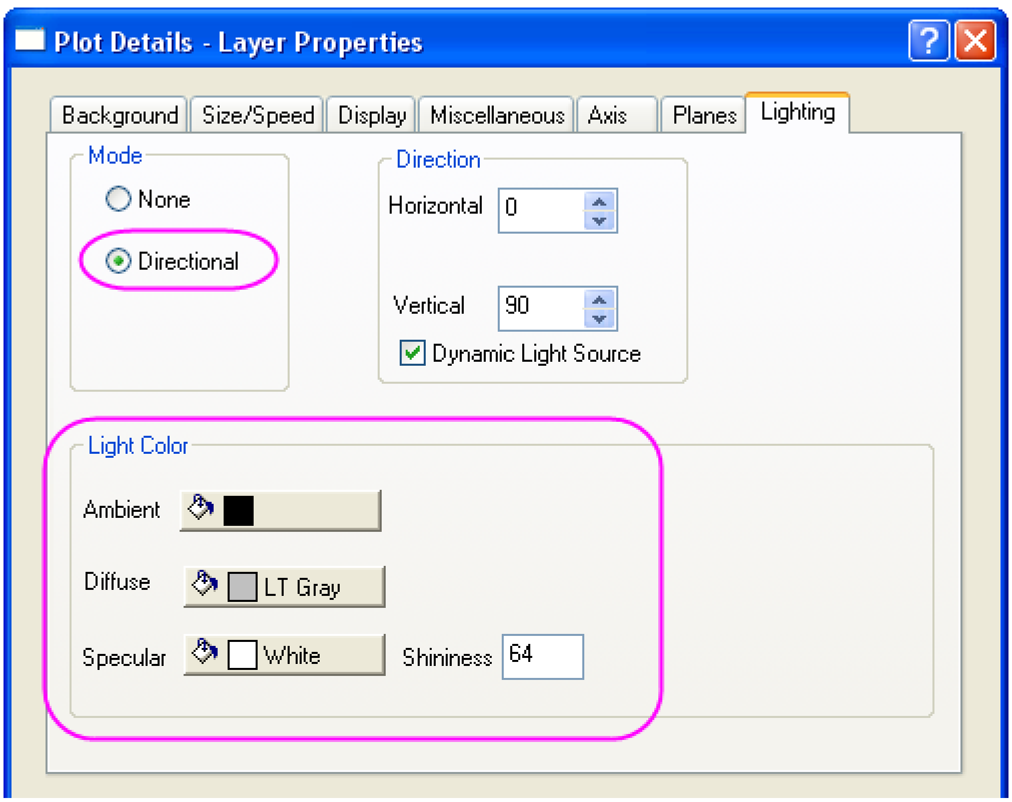
3. Choose **Format: Plot Properties** to go to the plot level of the **Plot Details** dialog.
4. Select the surface plot, go to the Mesh tab, uncheck the **Enable** box to turn off the mesh lines. Go to **Colormap/Contours** tab, click Line title to open the **Contour Lines** dialog. In this dialog, uncheck **Show on Major Levels Only** and select **Hide All** to hide all contour lines.



5. Select the projection plot in the left panel, go to **Surface** tab, Enter *0* as the following image shows to put the projection to the bottom of the graph.

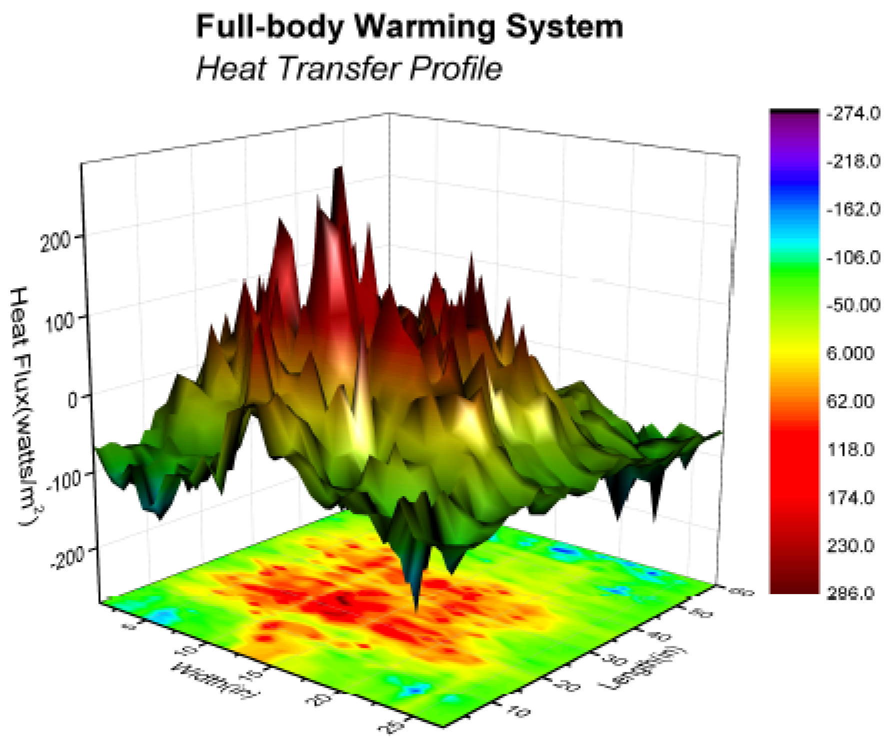


6. Go to the **Mesh** tab, uncheck the **Enable** box to disable the mesh lines. Go to **Colormap/Contours** tab, hide all contour lines like the step4 does.
7. Click Layer 1 in the left panel, and then in the right panel select the **Lighting** tab. Change the setting as following to enable lighting effect:



Then Click **OK** to close the dialog.

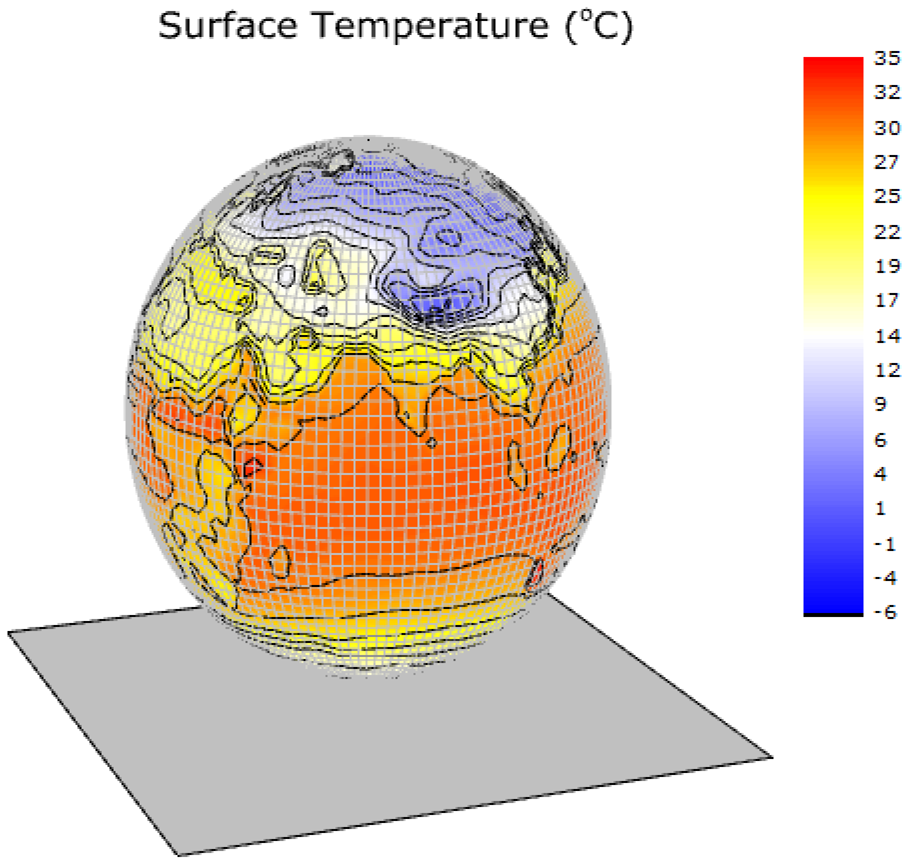
8. Finally, change axis titles and add a graph title as the following graph.



5.8.10 Parametric Surface with Colormap from Data

Summary

In this tutorial a 3D sphere is created using the data from three matrices. And the surface is filled to display the surface temperature contour using the data from another matrix.



Minimum Origin Version Required: Origin 9.0 SR0

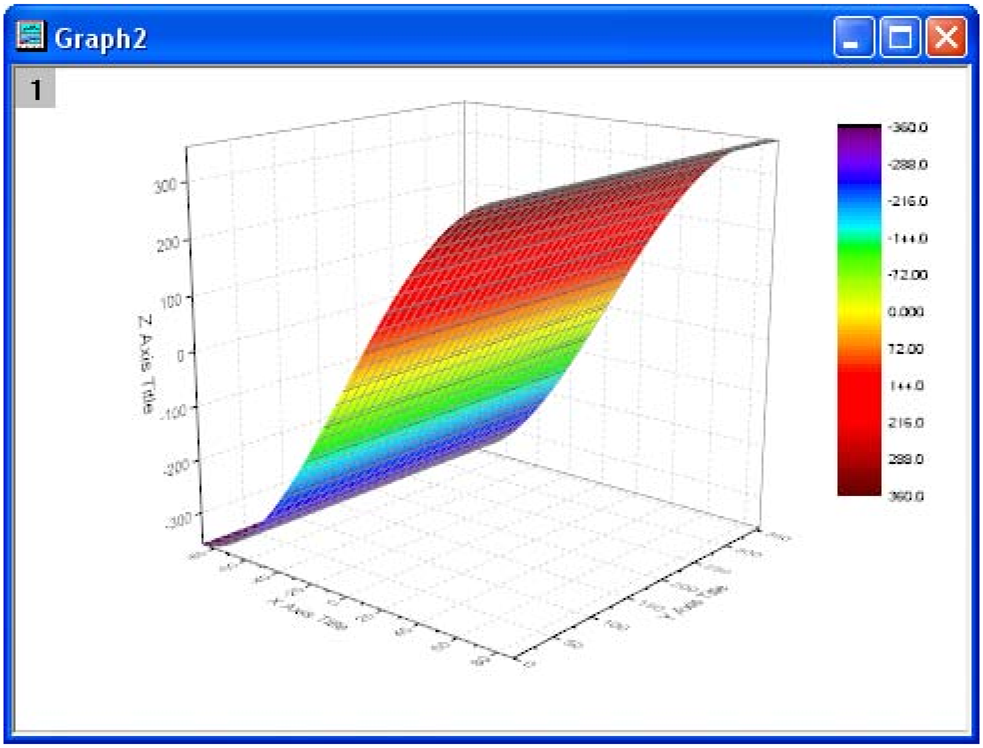
What you will learn

This tutorial will show you how to:

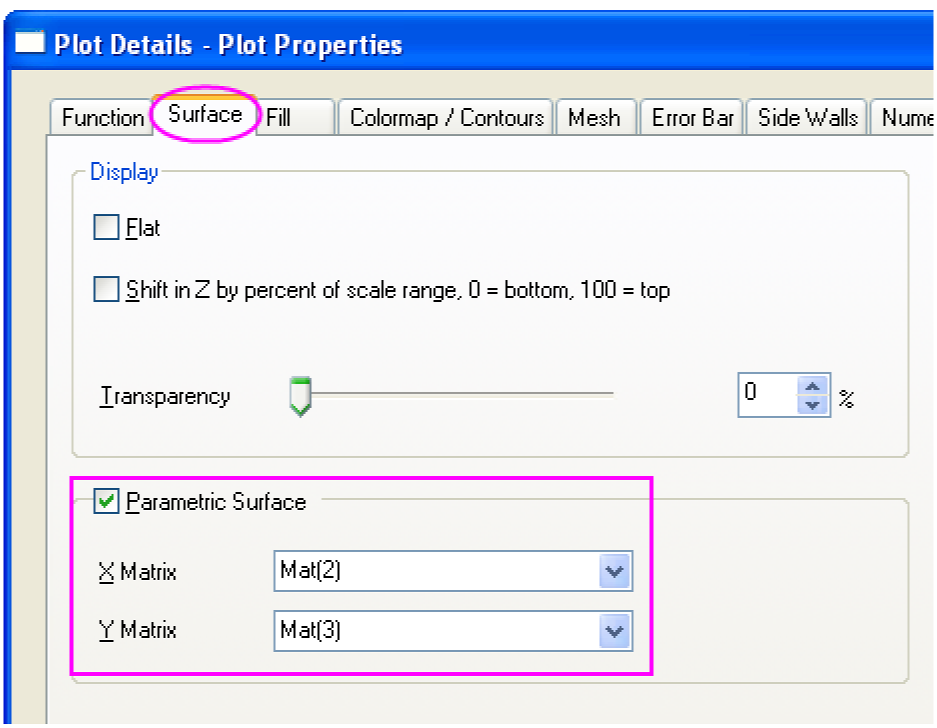
- Create parametric surface from matrix data.
- Set contour fill from another matrix.
- Customize the 3D parametric surface plot.

Steps

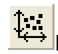
1. Open the 3D OpenGL Graphs project(*\Samples\3D OpenGL Graphs.opj*), go to the *3D OpenGL Graphs: 3D Function Plot: Parametric Surface with Colormap from Data* folder in Project Explorer.
2. Activate the matrix *FUNCA:1/4*, and click the button on **3D and Contour Graph** toolbar to create a colormap surface as below. You can also create this colormap surface by selecting **Plot: 3D Surface: Color Map Surface**.

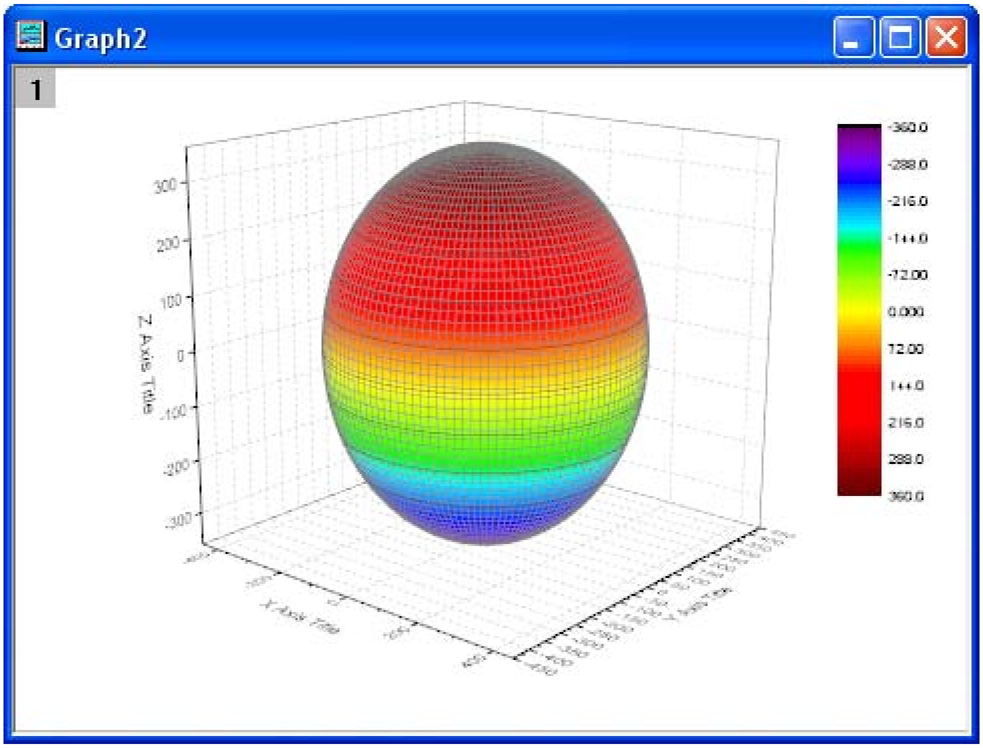


3. Double click on the plot to open the **Plot Details** dialog. Click on the **Surface** tab. Check the box before **Parametric Surface** and set **X Matrix**, **Y Matrix** as **Mat(2)**, **Mat(3)** respectively.

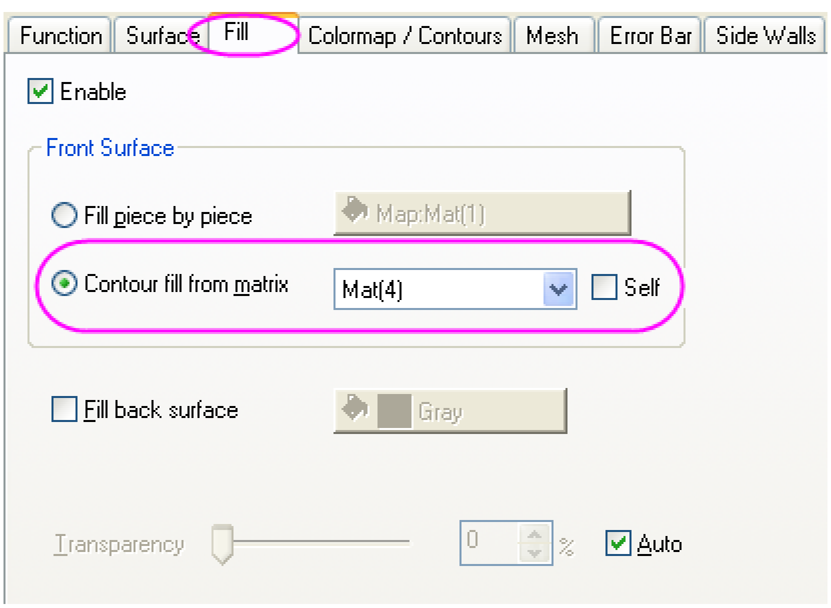


Click **OK** to close the dialog.

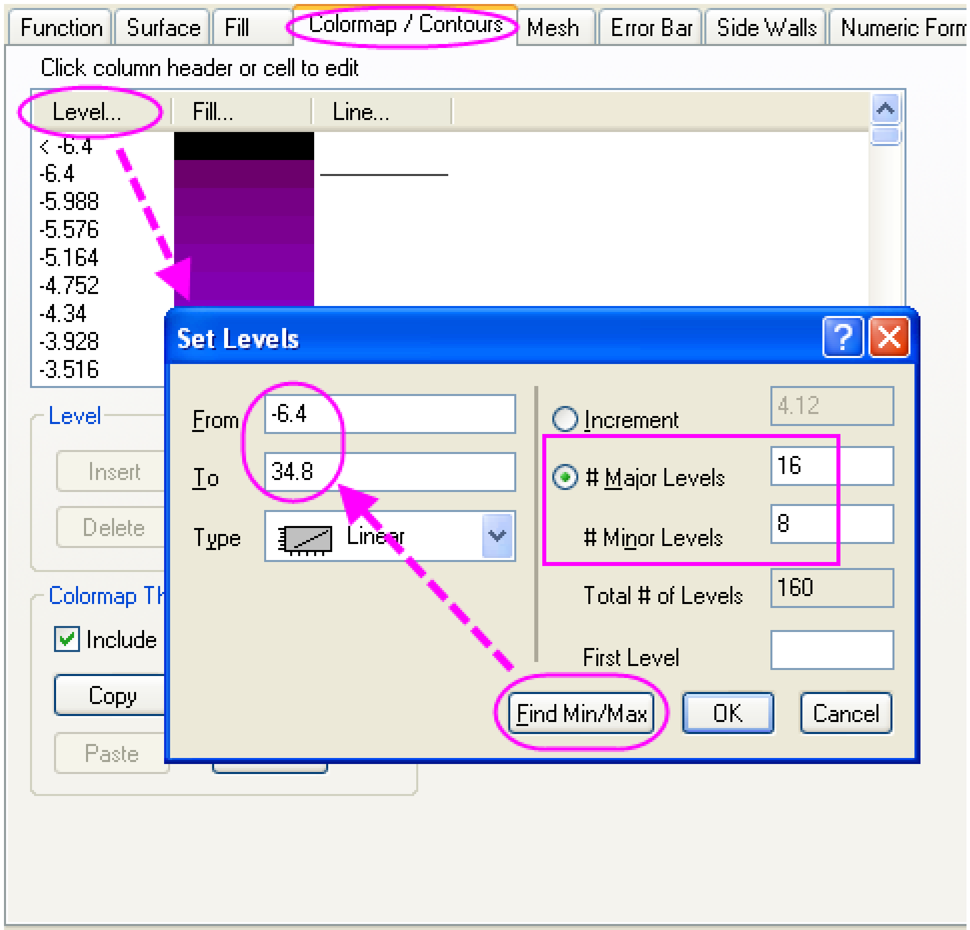
4. In order to show the complete colormap surface click the  button on **Graph** toolbar and the colormap surface should look like the following image:



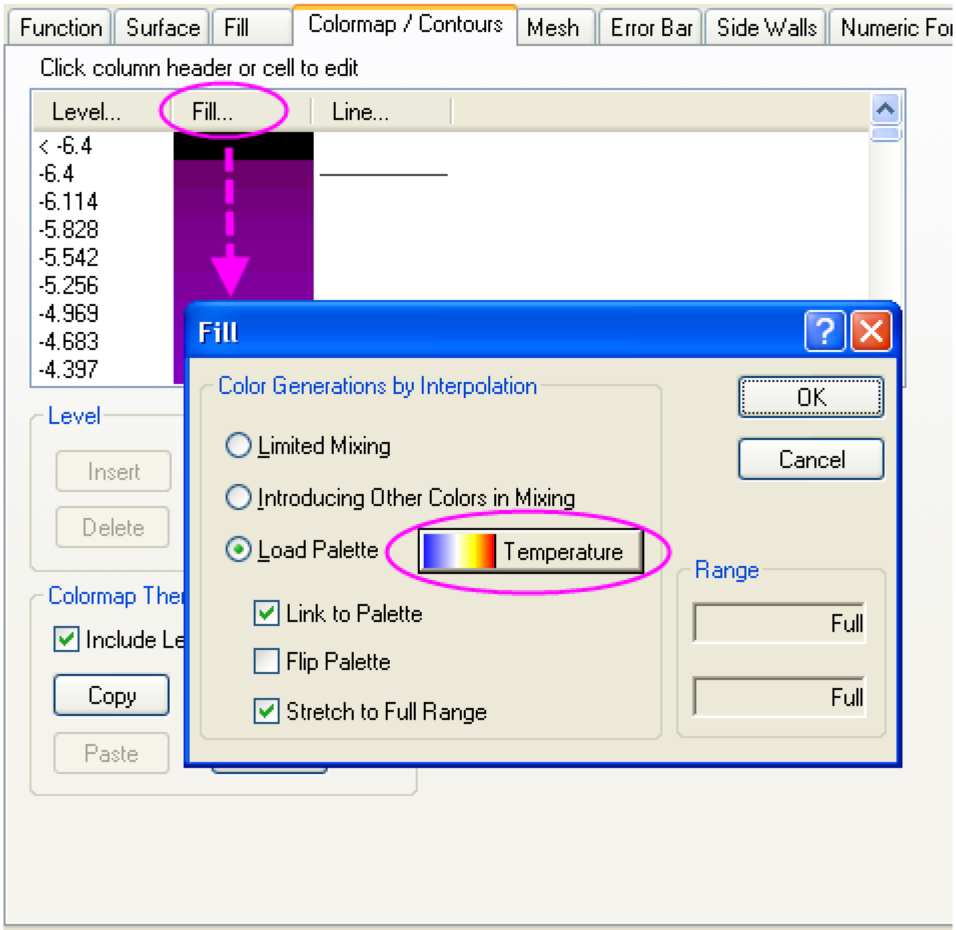
5. Double click on the plot to open **Plot Details** dialog. Go to the **Fill** tab. In Front Surface section uncheck the box before **Self** and set **Contour fill from matrix** as **Mat(4)**. Click **Apply**.



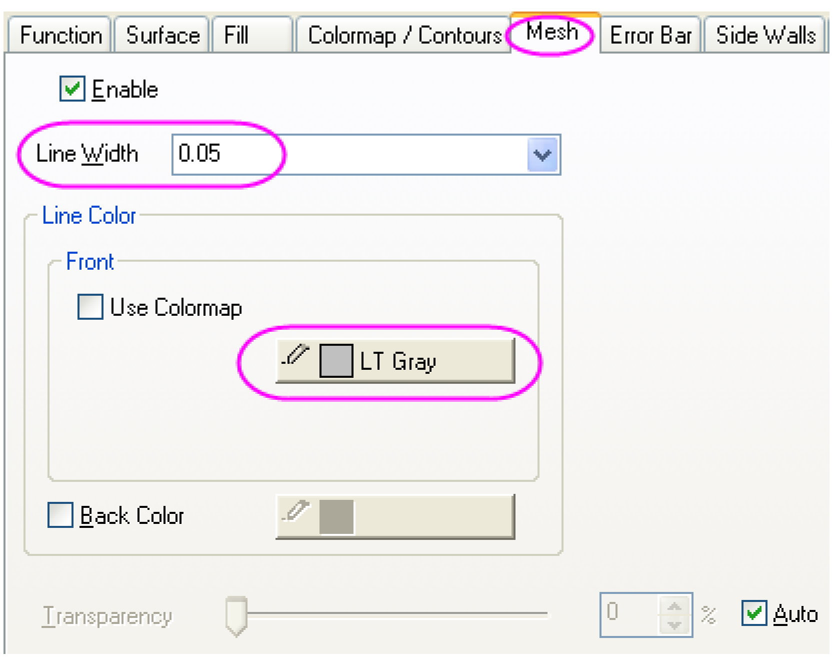
6. Activate the **Colormap / Contours** tab. Click **Level** to open the **Set Levels** dialog. Click **Find Min/Max** and set **Major Levels**, **Minor Levels** as **16**, **8** respectively. Click **OK**.



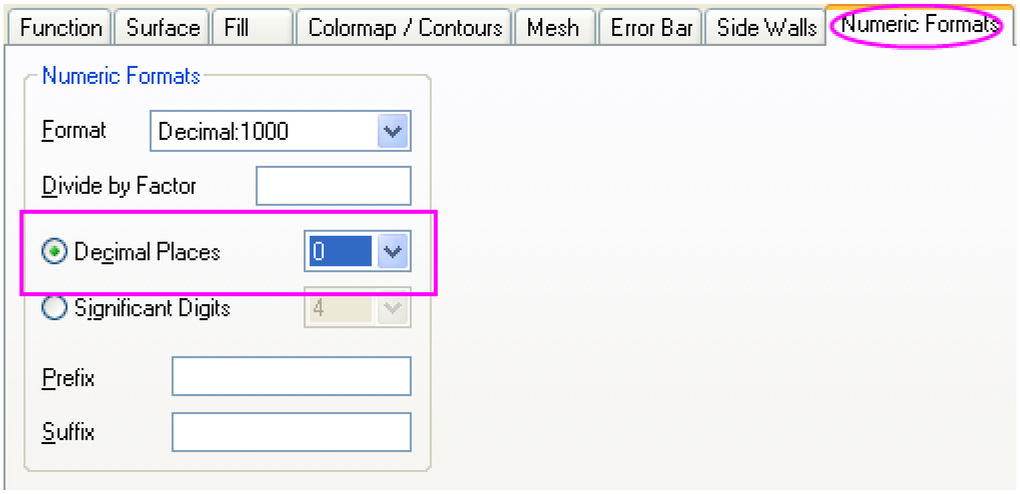
7. Click **Fill** to open the **Fill** dialog. Set **Load Palette** as **Temperature**. Click **OK**.



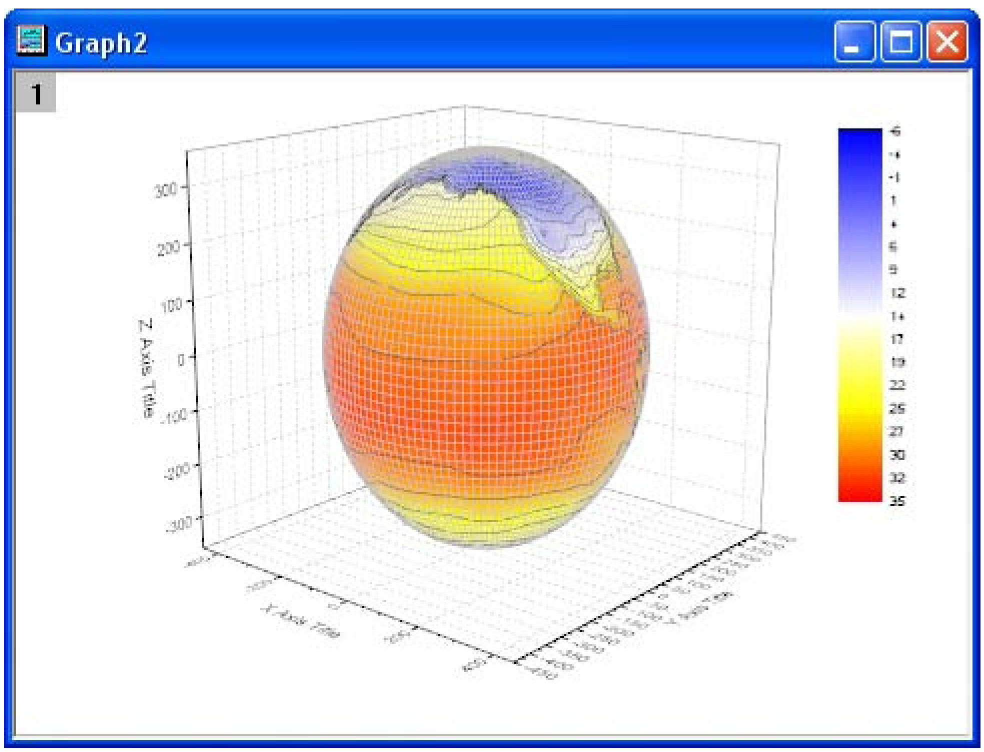
8. Click on the **Mesh** tab. Set **Line Width** as **0.05** and **Line Color** in Font section as **LT Gray**. Click **Apply**.



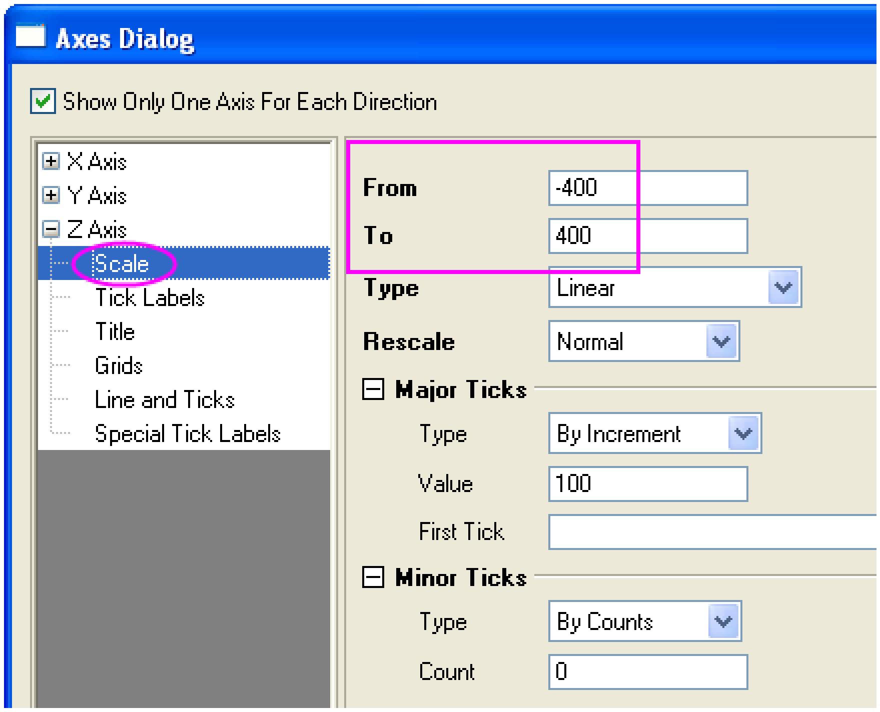
9. Click on the **Numeric Formats** tab. Choose the **Decimal Places radio** button and set its value as **0**.



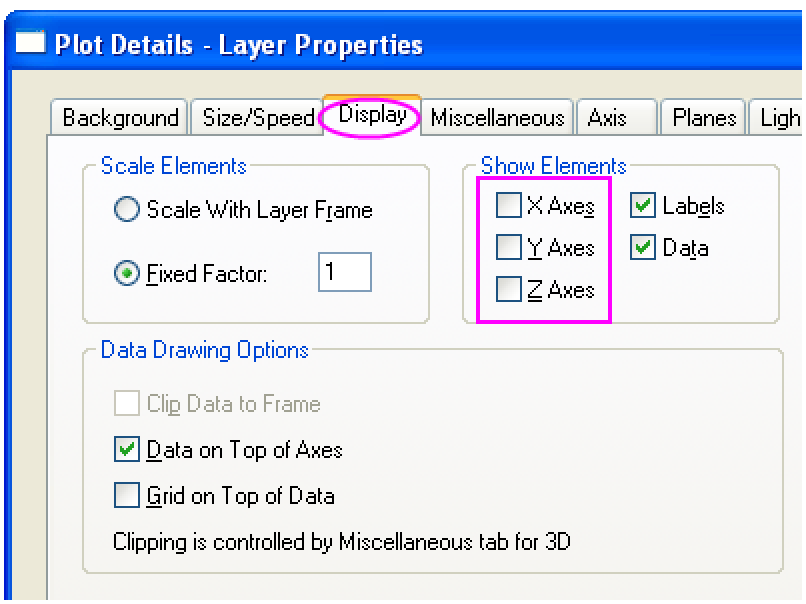
Click **OK** to apply the settings and close the **Plot Details** dialog. The graph should look like the following image.



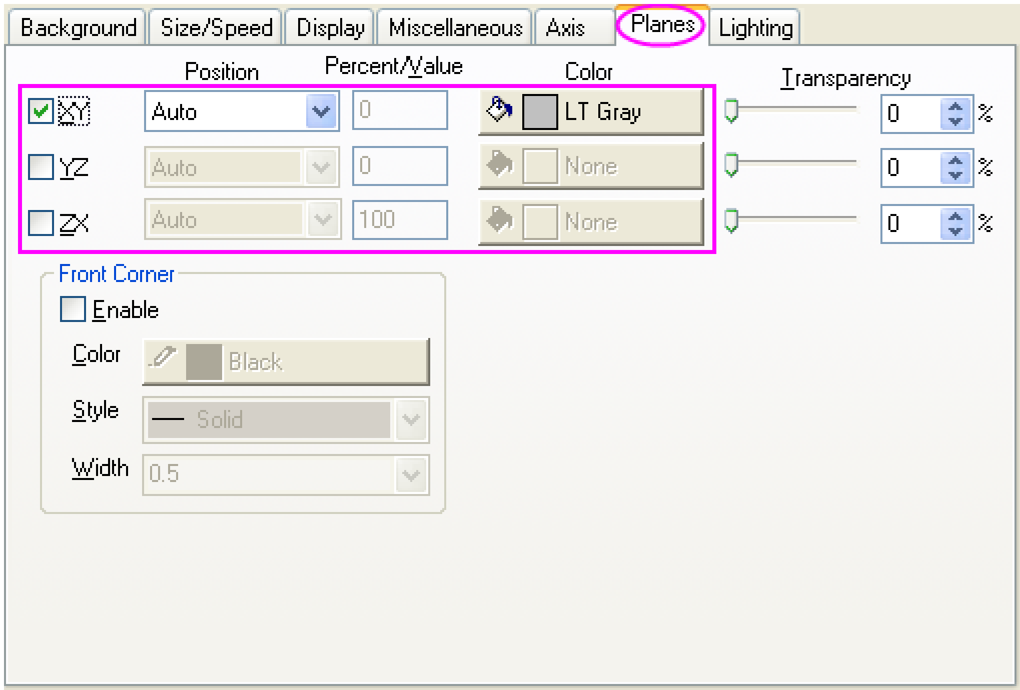
10. Double click on Z axis to open **Axes Dialog**. On the **Scale** node, set the value of **From**, **To** as **-400**, **400** respectively. Click **OK**.



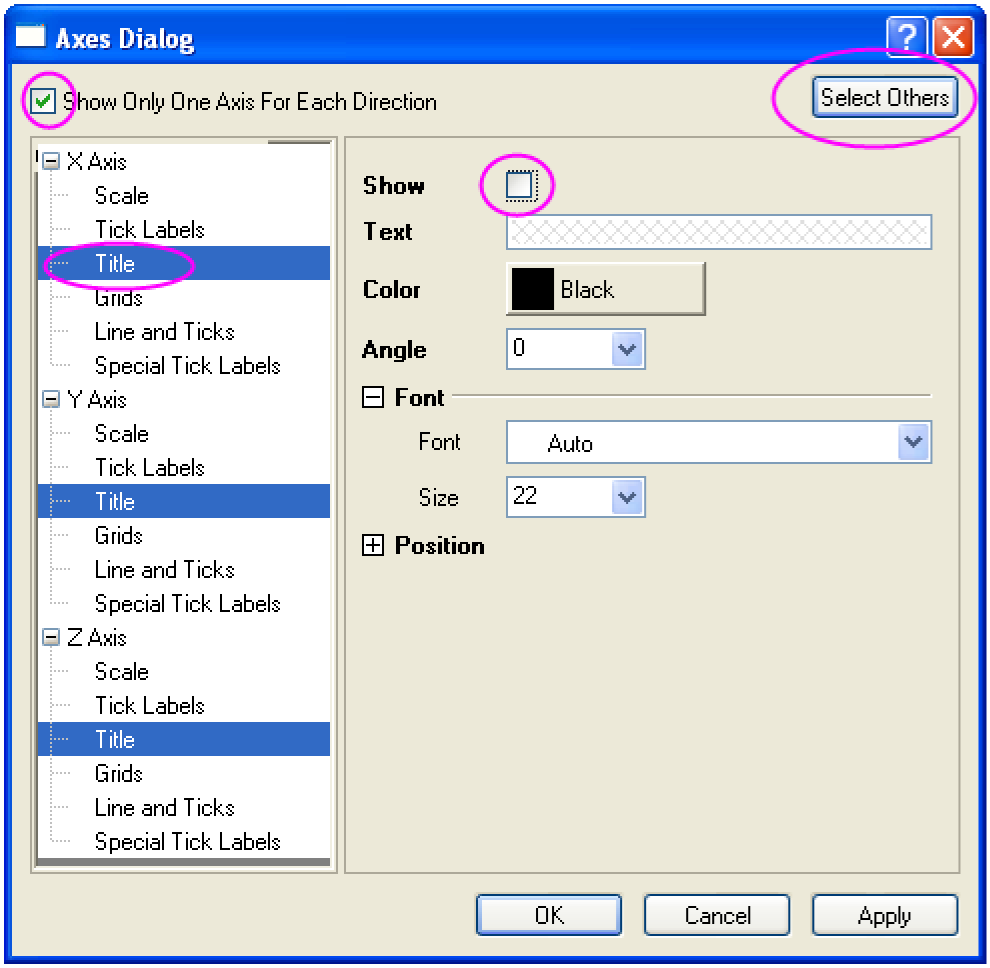
11. Double click on the XY Plane to open **Plot Details - Layer Properties**. Click the **Display** tab, and uncheck the box before **X Axes**, **Y Axes**, **Z Axes** in Show Elements section to hide the axes.



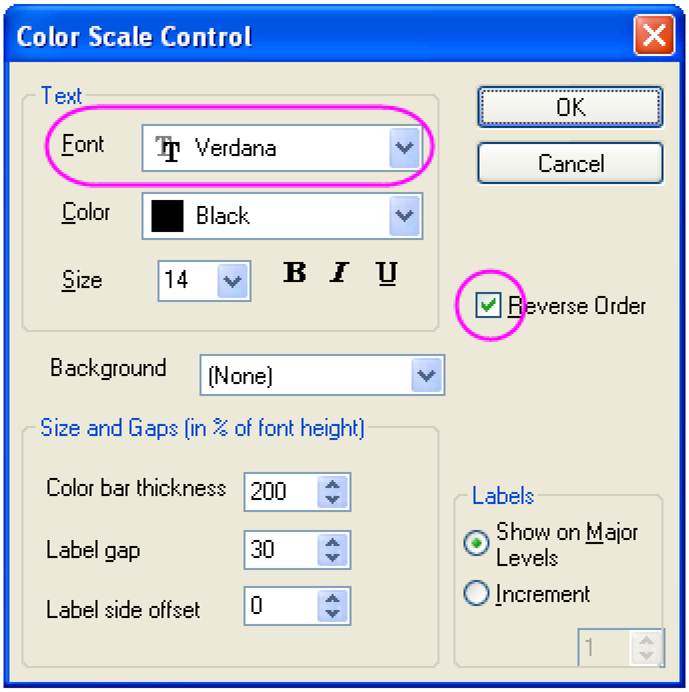
12. Click on the **Planes** tab, and uncheck the boxes before **YZ**, **ZX** to hide YZ and ZX planes. Set **Color** of **XY** as **LT Gray**. Click **OK** to close the dialog.



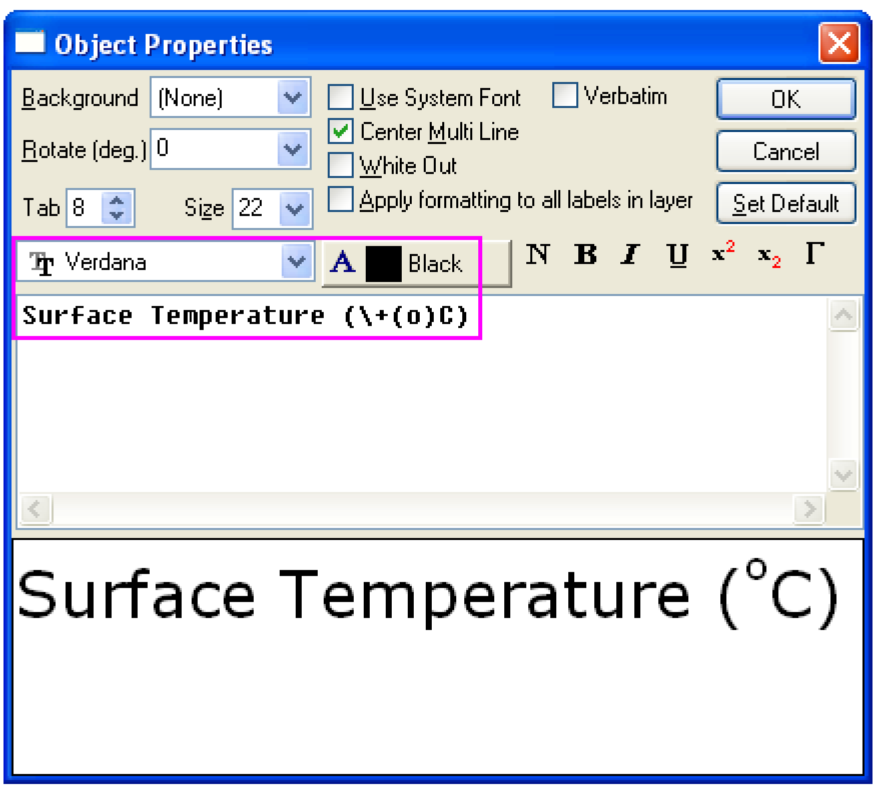
13. Go back to the graph, select **Format:Axes Titles:X axis titles** to open the **Axes** dialog with the **Title** node selected. Click the **Select Others** button. Uncheck the box after **Show** to hide axis title for all axes.



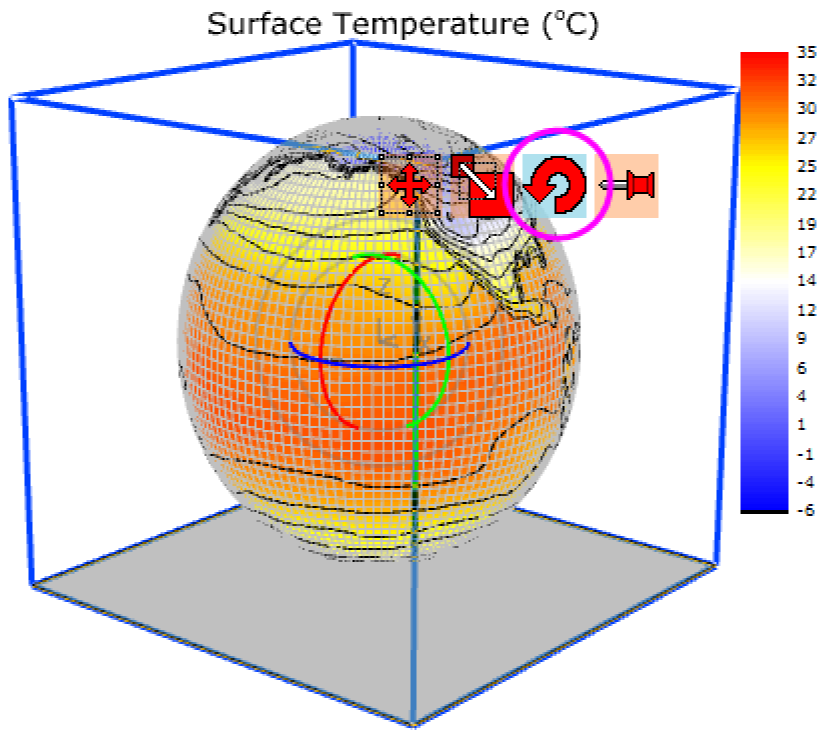
14. Double click on the color scale to open the **Color Scale Control** dialog. Set Text Font as **Verdana**. Check the box before **Reverse Order**. Click **OK** to apply the setting and close dialog. Move the color scale object to a proper place.



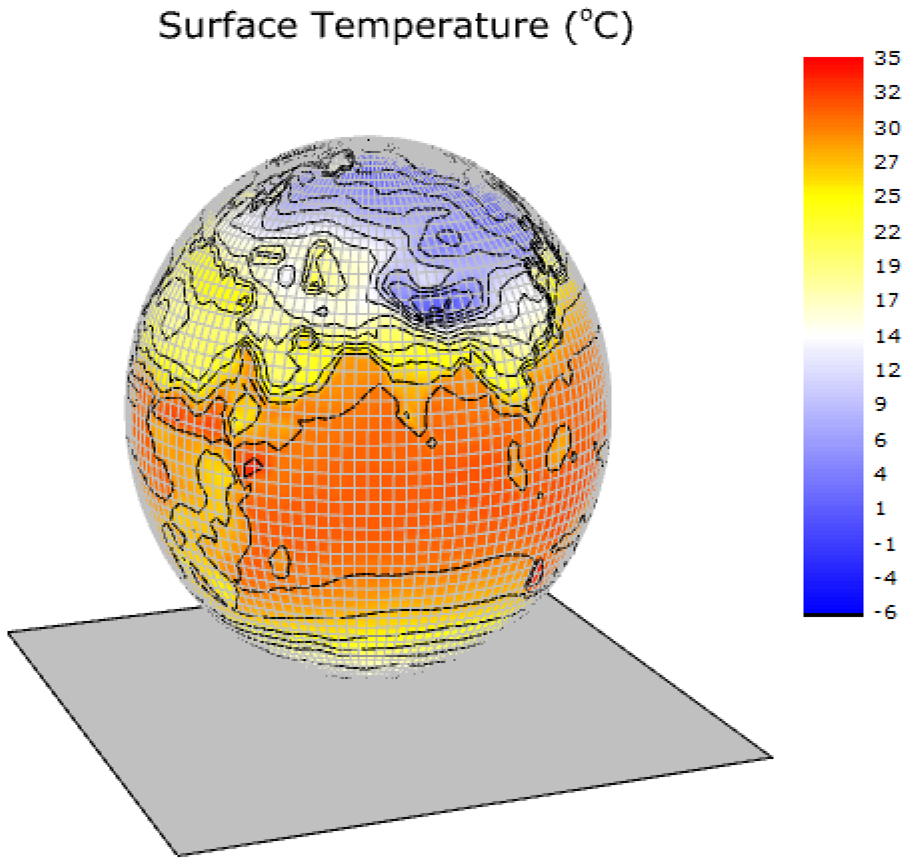
15. Right click on the white area of the graph layer to bring up a context menu and choose **Add / Modify Layer Title**. Select the text object added just now, right-click on it and select **Properties...** on the shortcut menu to open the **Object Properties** dialog. Set text font as **Verdana** and type **Surface Temperature (\+(o)C)** in the content table. Click **OK**.



16. Click on the graph layer within 3D frame (not the data plot), and click the **Rotate** button as shown in the following to activate rotation mode.



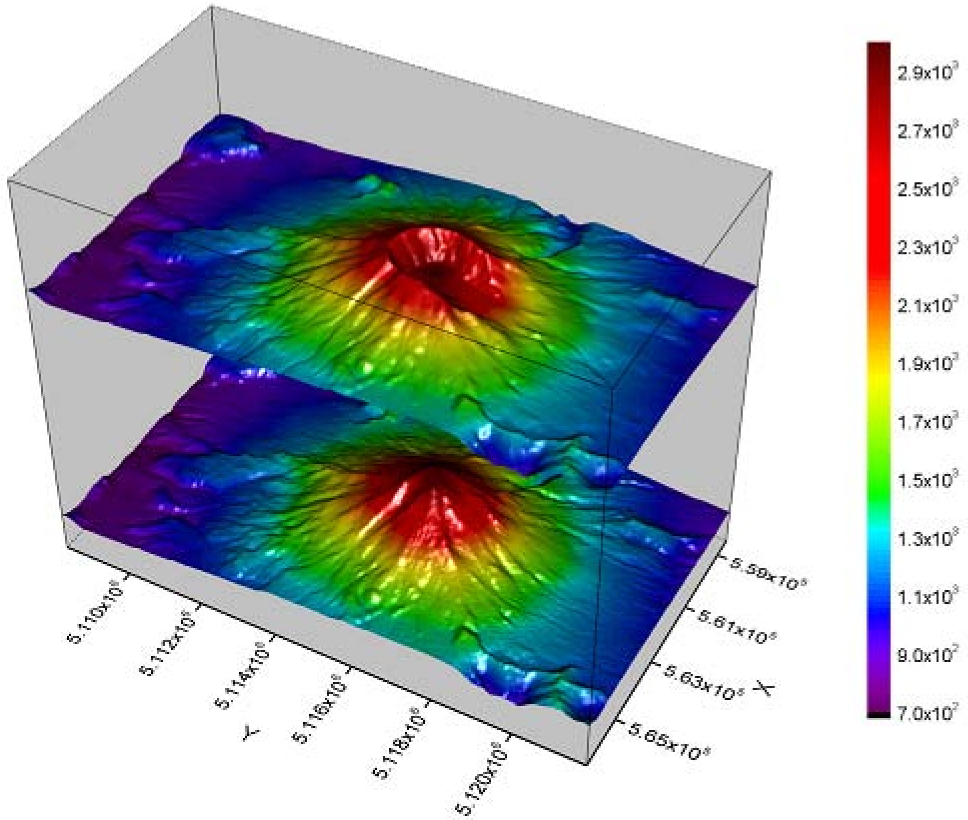
Rotate the plot to get a better view. The graph might look like the following.



5.8.11 Stacked 3D Surface Plots

Summary

This tutorial shows how to create stacked 3D colormap surfaces from different matrix objects. The surfaces in the plot display the topology before and after volcanic eruption. And a graph animation is generated from LabTalk script for the plot rotation.



Minimum Origin Version Required: Origin 9.0 SR0

What you will learn

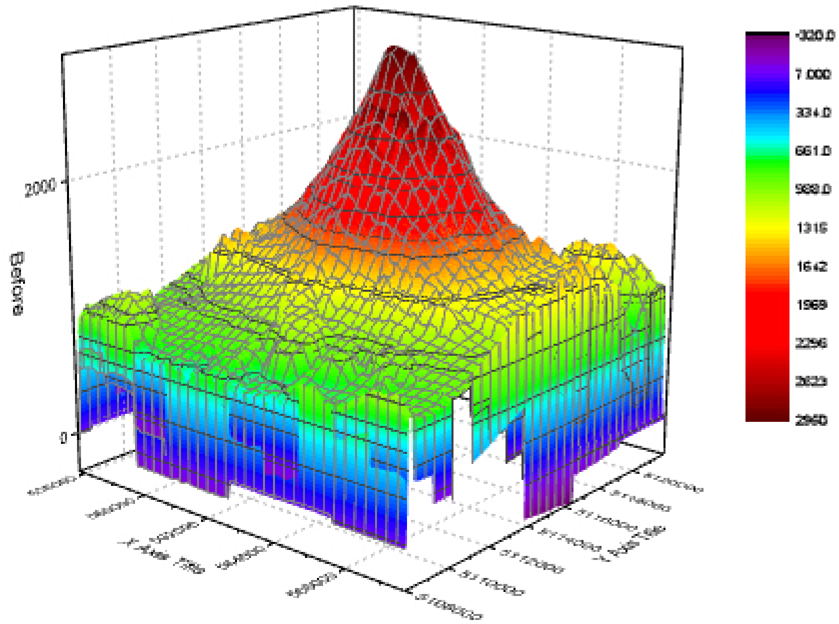
This tutorial will show you how to:

- Create stacked 3D colormap surfaces.
- Customize axes display and layer properties.
- Resize and rotate a 3D plot.

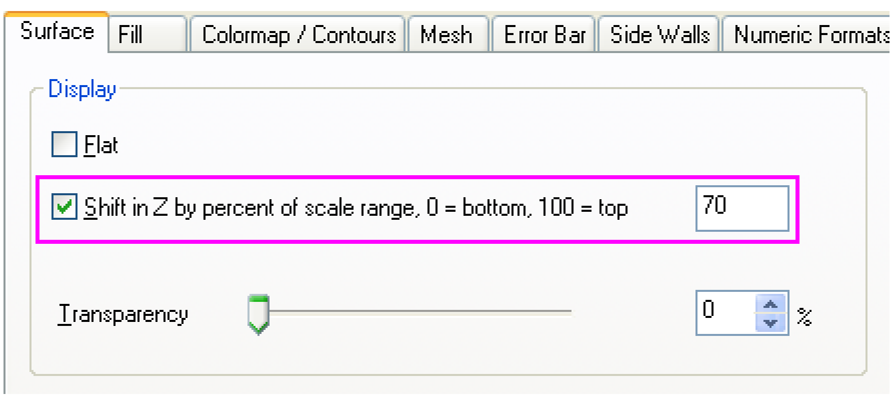
Steps

Create Multiple Colormap Surfaces

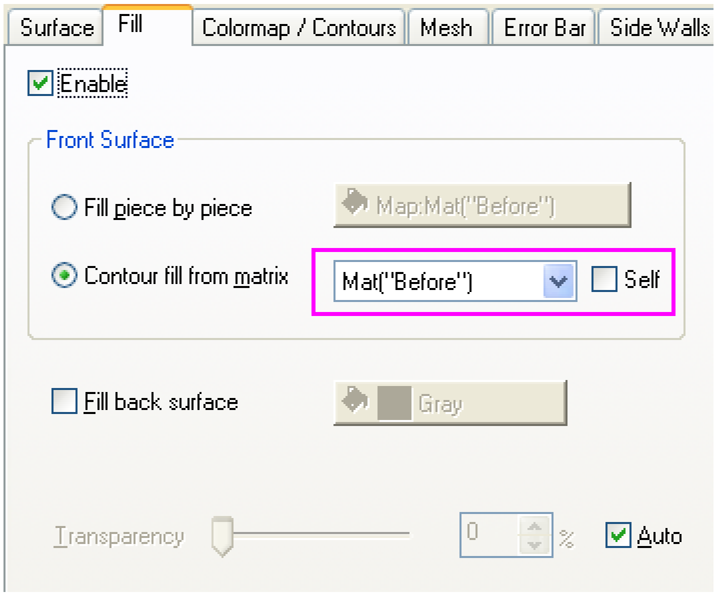
1. Click **File: Open Sample Projects: 3D OpenGL Graphs** from the **Menu** bar to open 3D OpenGL Graphs project. Go to the 3D OpenGL Graphs: 3D Surface: Stacked 3D Surface Plots folder in Project Explorer.
2. Activate the matrix book **Mbook1** which contains two matrix objects, then click **Plot: 3D Surface: Multiple Colormap Surfaces** to create two 3D surfaces from these two matrix objects.



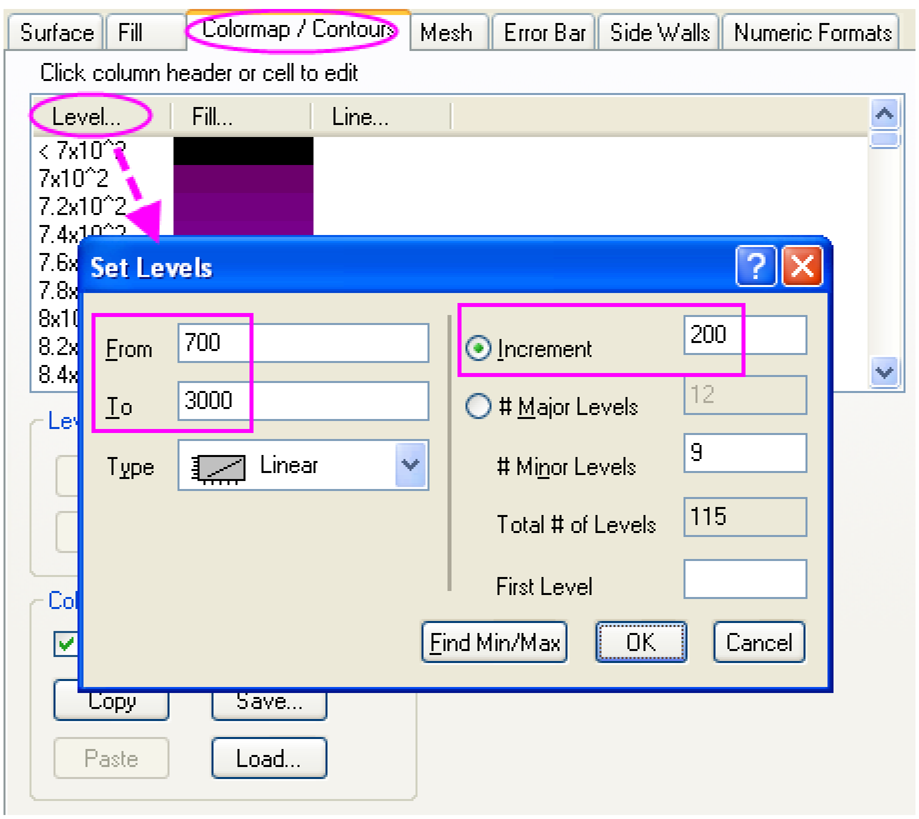
3. Double click on the plot to open the **Plot Details** dialog, you can see that there are two surfaces under the **Layer1** node on the left panel. To shift the "After eruption" surface in Z axis, activate the second plot under **Layer1** on the left panel, and in the right panel, select the **Surface** tab. Then check the box before **Shift in Z by percent of scale range**, and enter **70** in the text box.



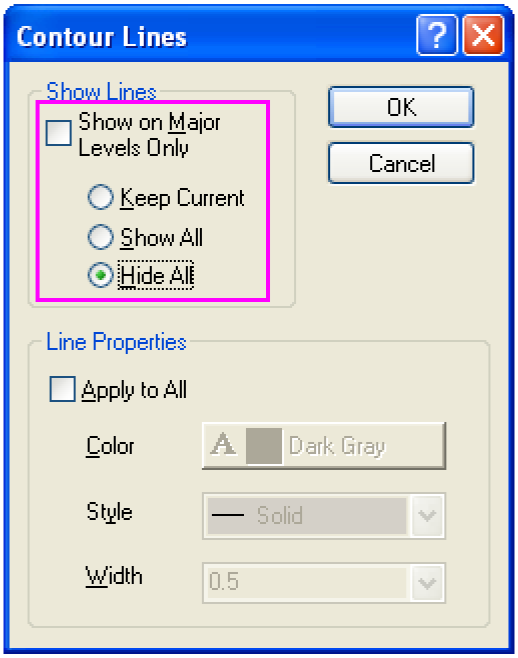
4. Go to the **Fill** tab. In the **Front Surface** section, uncheck the box before **Self** to fill contour by the same matrix object (Mat "Before") as the other surface used.



5. Select the **Colormap / Contours** tab. Click **Level..** to bring up the **Set Levels** dialog. Set the parameters as shown in the following graph and click **OK**.

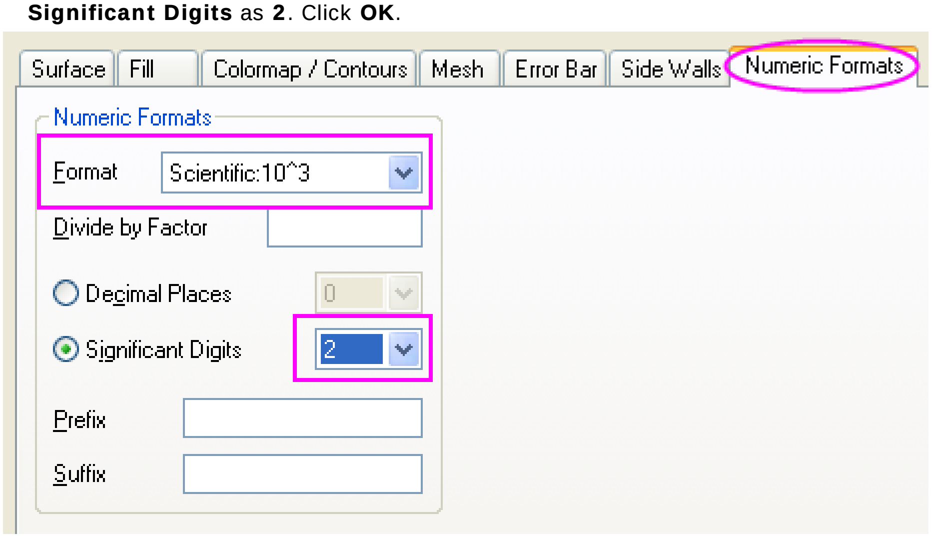


Click **Line...** to open the **Contour Lines** dialog. Uncheck the box before **Show on Major Levels Only** and select **Hide All**. Click **OK**.

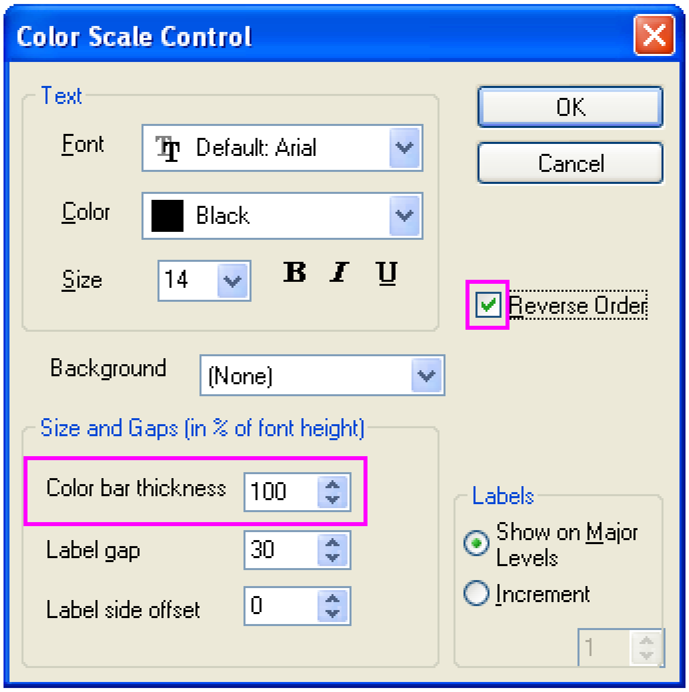


6. Go to the **Mesh** tab, and uncheck the box before **Enable** to disable the mesh line.
7. Repeat steps 5 to 6 for the first plot under the **Layer1** node.
8. In this project the two surfaces use the same matrix as contour fill, so they can share one color scale.

To set the numeric format of the color scale, activate the first plot on the left panel of the **Plot Details** dialog. Then select the **Numeric Formats** tab on the right panel. Select **Scientific: 10^3** from the drop down list next to the **Format**, and set **Significant Digits** as **2**. Click **OK**.



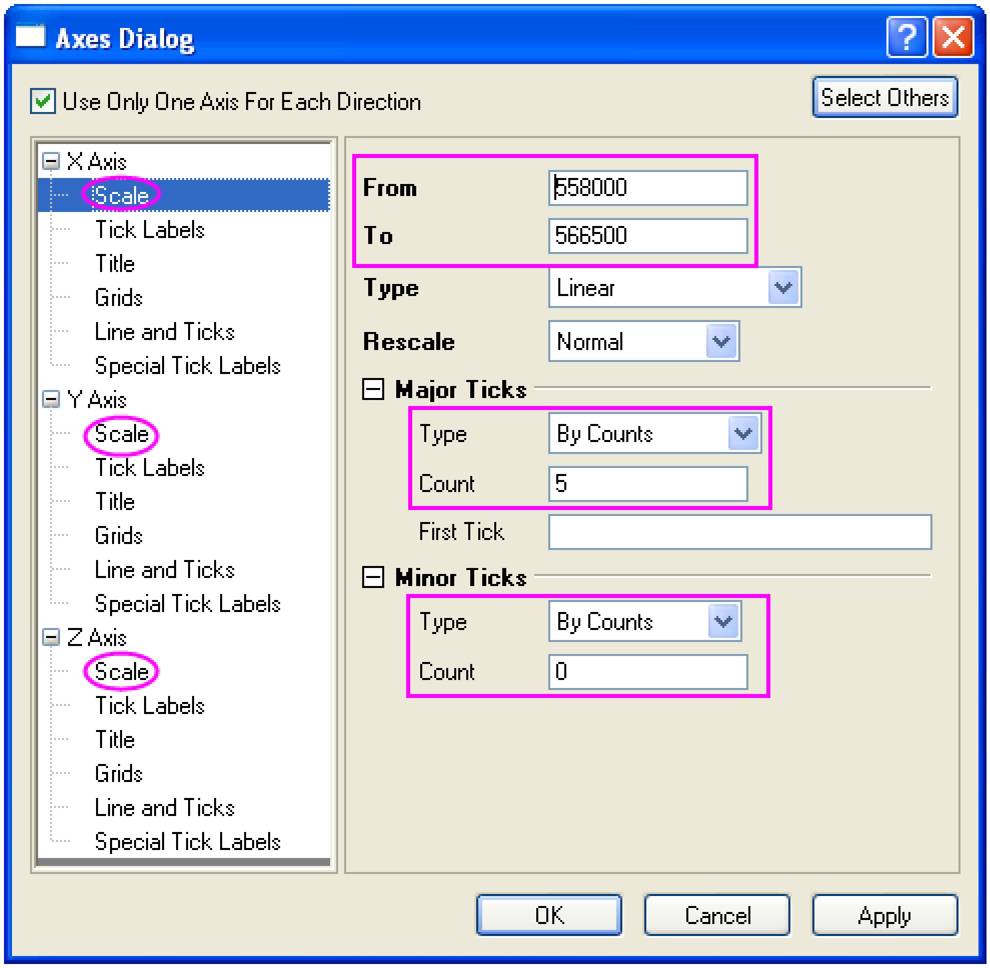
To customize the color scale, double click on the color scale to open the **Color Scale Control** dialog. Check the box before **Reverse Order** and set **Color bar thickness** as **100**. Click **OK**.



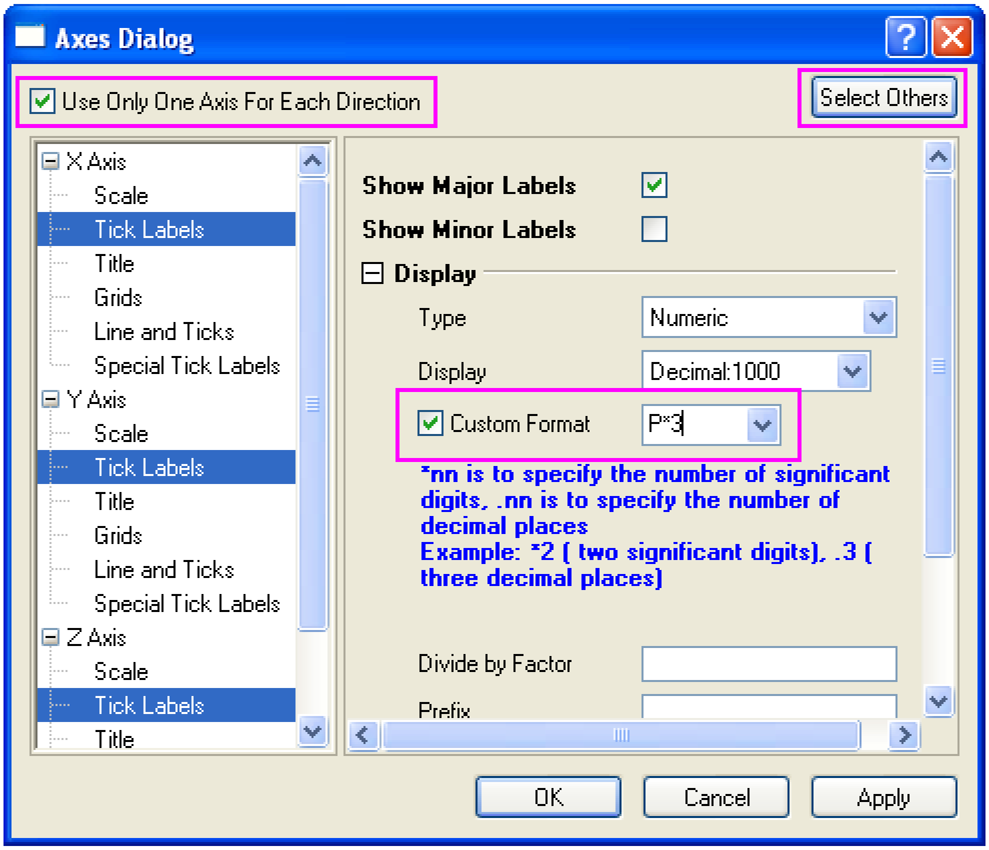
Customize Axes Display

In the **Axes** dialog, you can change the axes scale and tick labels' format. To open this dialog, click **Format: Axes: X Axis...**

1. Settings on the **Scale** node.
   - Set scale from **558000** to **566500** for X Axis, from **5108200** to **5121800** for Y Axis, and from **0** to **10000** for Z axis.
   - For X axis, set **Type** of **Major Ticks** as **By Counts** and set **Count** as **5**. For Y and Z axis, set **Type** of **Major Ticks** as **By Increment** and set **Value** as **2000**. To hide all minor ticks, set **Count** of **Minor Ticks** as **0** for all axes.

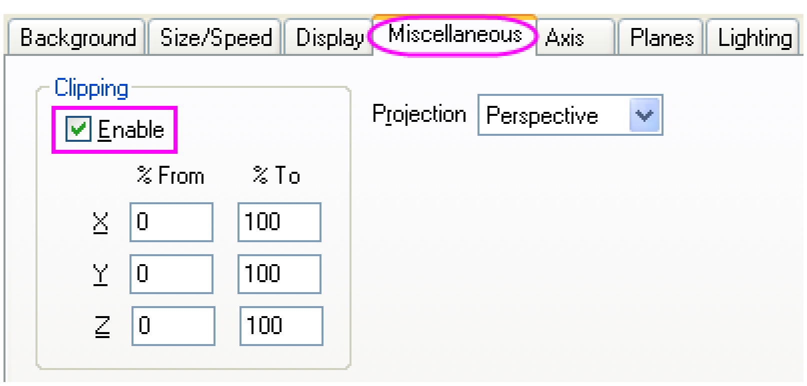


2. Customize the **Tick labels** and **Title**.
   - First, ensure that the **Use Only One Axis for For Each Direction** is enabled in the top of the **Axis** dialog. Therefore, only one axis is listed under each direction in the tree panel. That combined with the **Select Others** button allows you to quickly customize all axes with the same settings.
   - Select the **Tick Labels** node under X Axis. Click the **Select Others** button to select the tick labels of other axes. Check the box before **Custom Format** and select **P*3** from the drop down list to show the tick label as base-10 scientific notation with 3 significant digits. Click **OK**. For more information about the options in this drop down list, please refer to Custom Display Format.

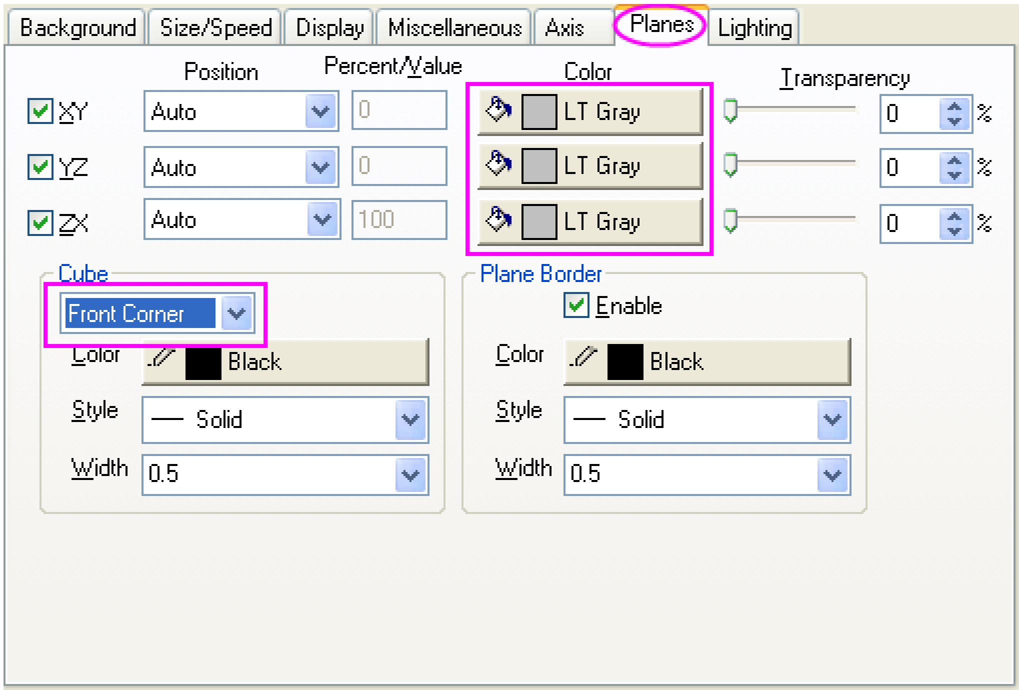


Customize Layer Properties

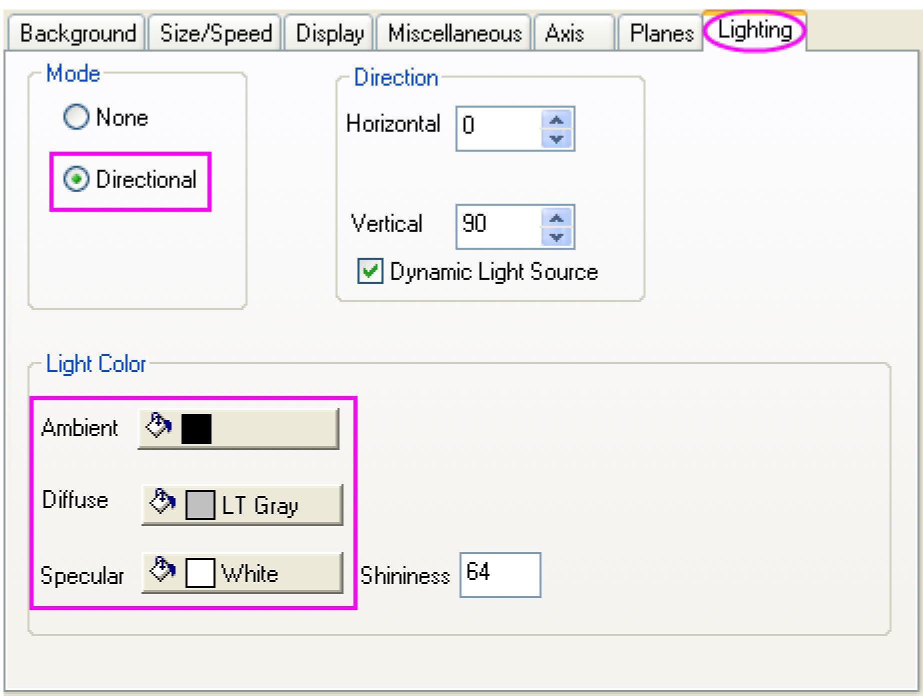
1. Double click on the blank space outside the plots or click **Format: Layer Properties...** from the menu to open **Plot Details - Layer Properties** dialog.
2. Activate the **Miscellaneous** tab on the right panel. Check the box before **Enable** in the **Clipping** section, which will clip the image outside the axes area according to the settings in the **Clipping** section.



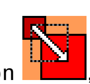
3. Go to the **Planes** tab. Set **Color** as **LT Gray** for all planes. And select **Front Corner** from the drop-down list in the **Cube** section to show the cube's border.



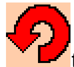
4. Select the **Lighting** tab. In the **Mode** section, choose **Directional** to enable lighting mode. Set **Light Color** as shown in the following graph. Click **OK**.



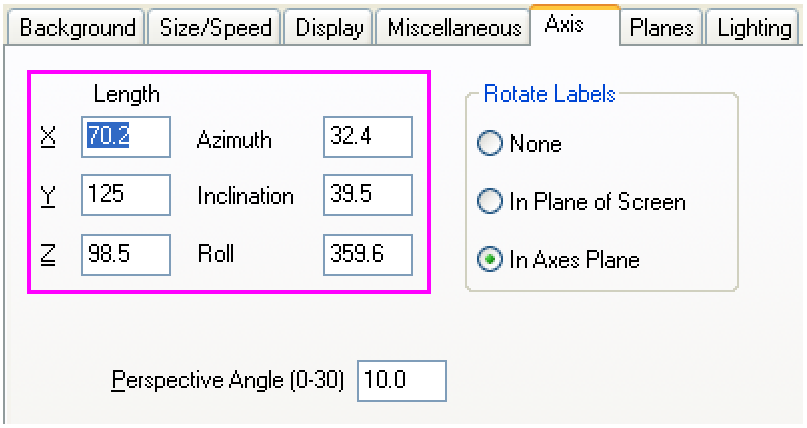
Resize and rotate the plot

1. Click on the cube (not the data plot) to activate the 3D toolbar. Click the Resize button , a 3D Cartesian coordinate will show up. Place the cursor on Y axis, which will then be highlighted,

at this moment drag-and drop the Y axis to stretch the plot in Y axis direction. Do the same to X direction and Z direction.

2. Click the rotate button  to activate rotation mode. A sphere will be displayed at the center of the plot. Rotate the plot to get a better view.

The 3D toolbar allows you to resize and rotate the plot freely. However, you can also achieve the same view as Graph1 in this sample project by setting the value in the **Axis** tab of **Layer Properties** dialog as shown in the following graph.



5.9 Vector

Topics covered in this section:

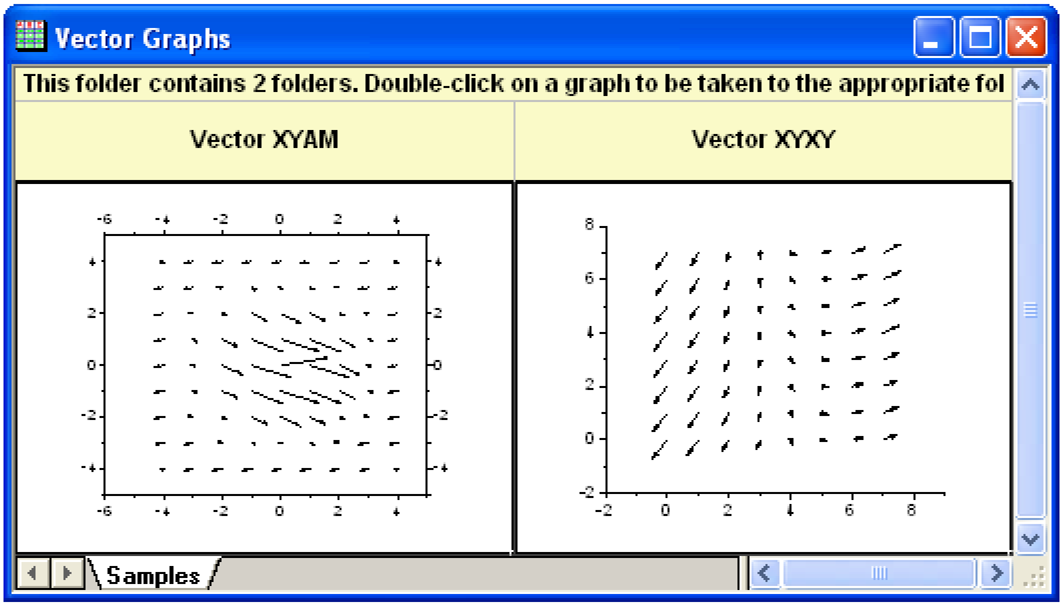
1. Vector Graph
2. 3D Vector

5.9.1 Vector Graph

Summary

A vector plot is a multidimensional graph used in industries such as meteorology, aviation, research, and construction that illustrate flow patterns (e.g. of wind, water, magnetic fields, etc.). Both direction and magnitude are represented in a vector graph. Origin includes two types of vector plots:

1. **Vector XYAM** - takes a starting XY location for the vector tail (by default), an angle and a magnitude.
2. **Vector XYXY** - takes two XY positions and connects them with a vector.



What you will learn

- Two data organizing mode to plot a vector graph.
- Use Plot Setup to assign plot data

Steps

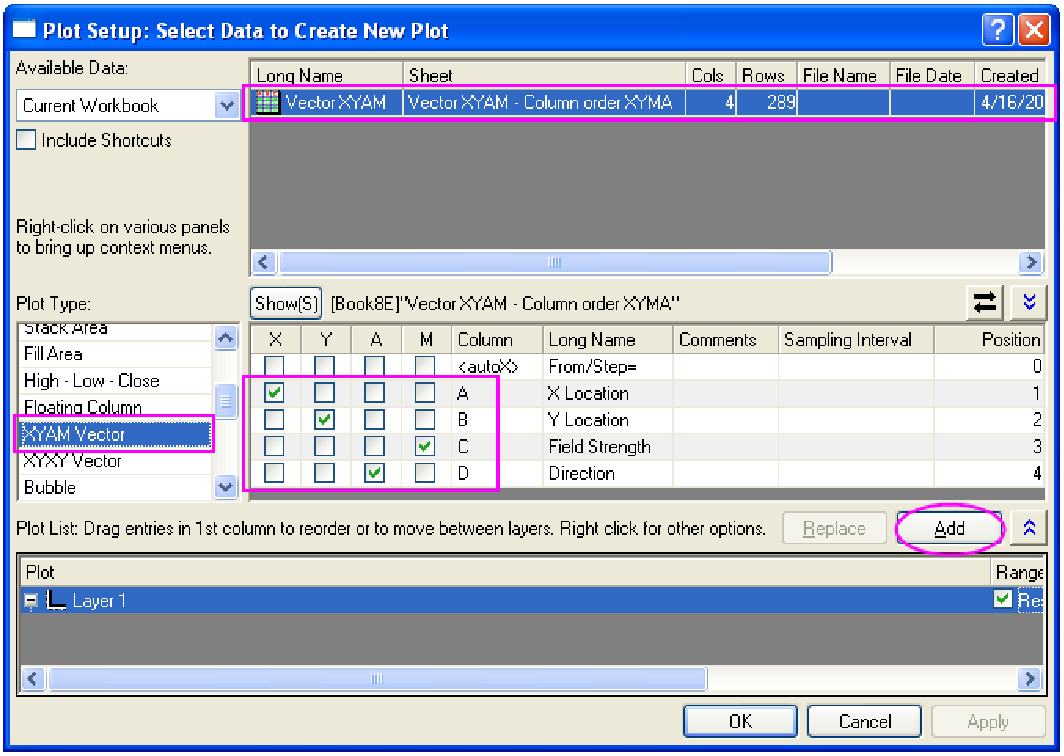
This tutorial is associated with the Statistical and Specialized Graphs project:*\Samples\Statistical and Specialized Graphs.opj*.

To create a vector plot, you need four data columns, and organize your data into two modes:

Vector XYAM

XYAM means X, Y, Angle and Magnitude, respectively. Vectors will start from point (X, Y), and rotate an angle you specified. The following example uses XYAM data mode to create a vector graph.

1. Browse the folder *Statistical and Specialized Graphs: Specialized: 2D Vector* in the **Project Explorer**. Active the data worksheet **Book8E**. Without highlighting any columns, select **Plot: Specialized: Vector XYAM** to open the **Plot Setup** dialog. And assign column A, B, D, C to X, Y, A, M as below, then click **OK** to plot the graph.



2. Select **Format:Plot Properties...** to go to the plot level of the **Plot Details** dialog. Go to the **Vector** tab and change the **Magnitude Multiplier** in **Vector Data** group to *75*. Click **Ok** button to close **Plot Details** dialog.
3. Double-click X or Y axis to open the **Axis** dialog. Active the **Title & Format** tab, and check/enable the **Show Axis & Ticks** option for the **Right** axis. And set the **Major Ticks** and **Minor Ticks** to **Out**. Show the **Top** axis either in this tab.
   Go to **Tick Labels** tab, and check the **Show Major Labels** box for both **Right** and **Top** axis.
   Go to the **Scale** tab, set **Increment** as **2** for **Horizontal**. Click **OK**.
4. Right click on the legend box to bring up the context menu. Select **Properties...** to open the **Object Properties** dialog. Type **\l(1) Field Strength** in the text box and click **OK**.
5. Add layer title as **Spot Write Effectiveness**.

Vector XYXY

Another data organizing mode is XYXY, where the first X, Y is the vector starting point, while the last X, Y is the ending point.

To create such vector graph, active the *Statistical and Specialized Graphs:Vector: 2D Vector* folder in the OPJ, highlight all of the four data columns of **Book9E**, and select **Plot: Specialized: Vector XYXY** from menu. Delete the axis titles if necessary.

5.9.2 3D Vector

Summary

Origin can plot 3D Vectors from two different data structures: **XYZ XYZ** and **XYZ dXdYdZ**.

XYZ XYZ defines the tail and head data for the vector, while XYZ dXdYdZ provides the vector's tail data and the distance between tail and head projections on X, Y, and Z axes.

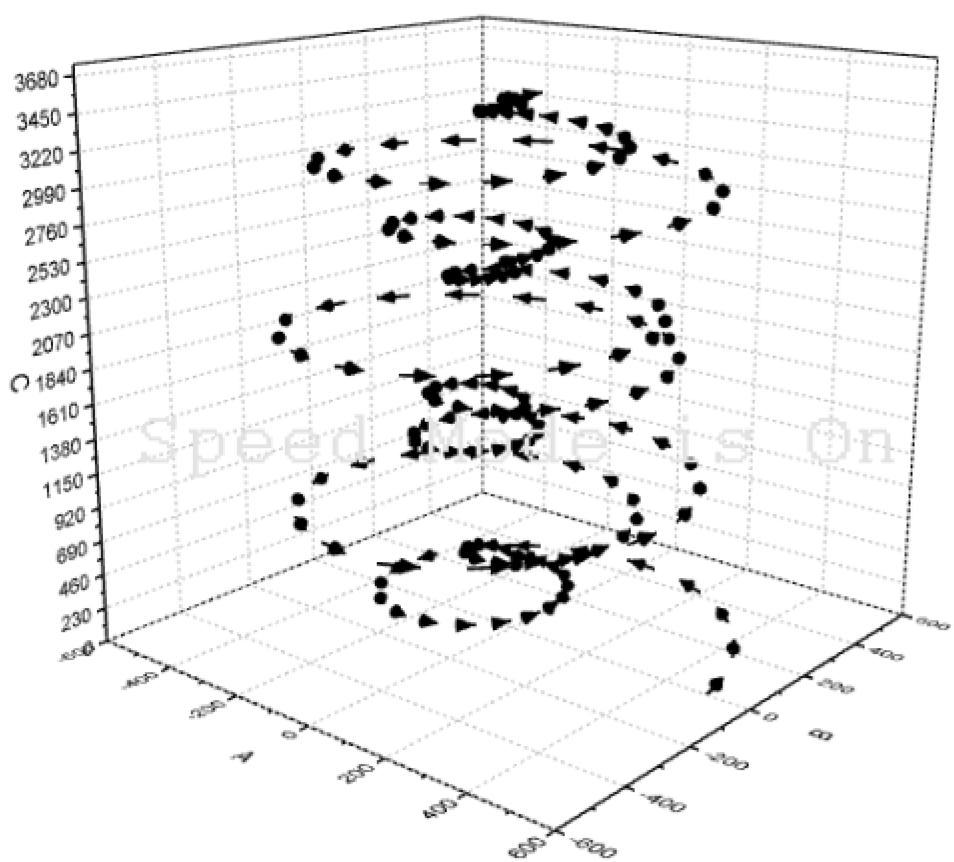
What you will learn

- Create a 3D Vector from XYZ XYZ worksheet data.
- Customize the 3D Vector

Steps

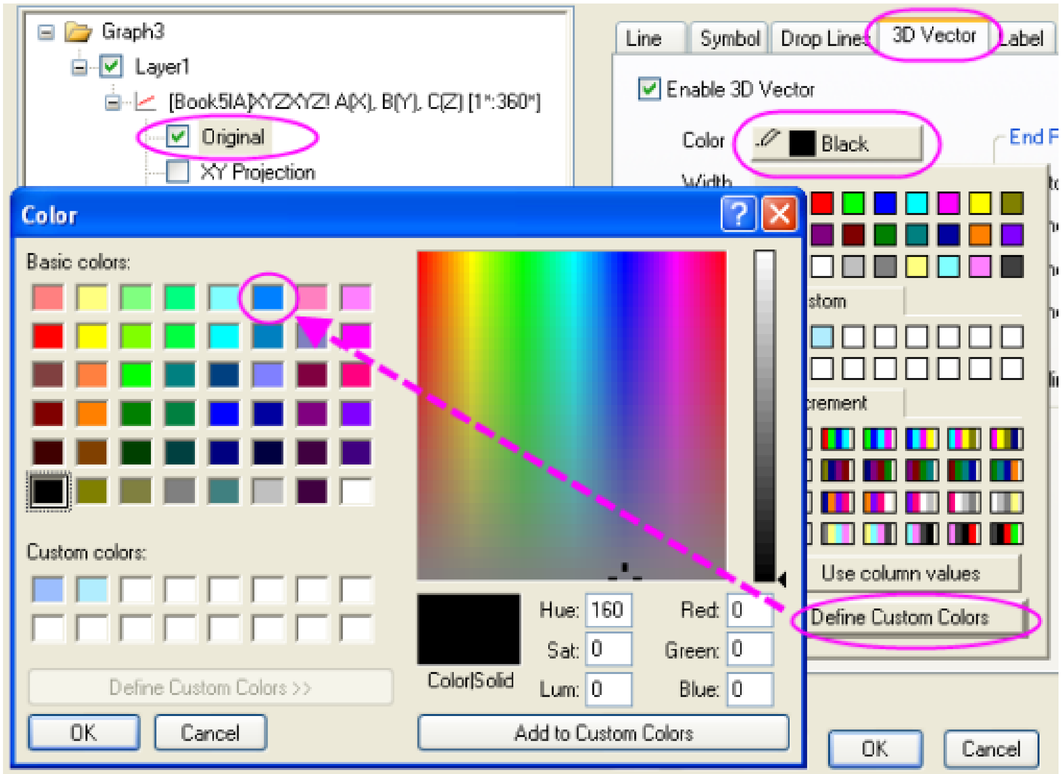
This tutorial is associated with the **3D Vector** folder in the *3D OpenGL graphs* project (\Samples*3D OpenGL graphs.opj*).

1. In the **Project Explorer** (usually at the left of the screen), browse to *\Statistical and Specialized Graphs\Vector\3D Vector*. Select workbook **Book51A**.
2. Click in the upper left corner of the worksheet to select all data.
3. On the main menu, select **Plot**, then point to **3D Symbol / Bar / Vector**, and then click **3D Vector XYZ XYZ**. This will create a 3D Vector graph with our built-in template.

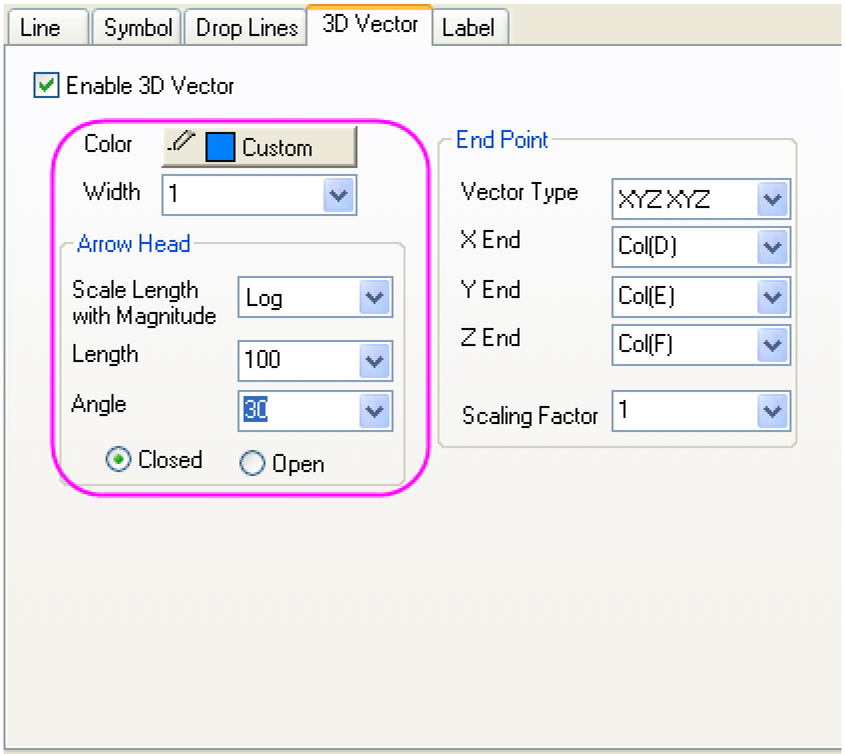


Note that speed mode is on.

4. Double-click on the vector plot to open the Plot Details dialog box. Ensure that the **Original** vector plot has been selected in the left panel of the dialog box. Select the **3D Vector tab**.
   - Click the **Color** button and select **Define Custom Colors** from the menu. Define a custom color in the Color dialog box. (We chose a custom shade of blue.)



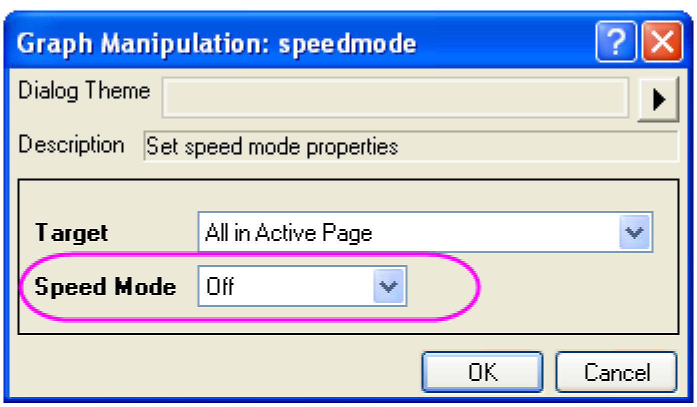
   - In the **3D Vector** tab of **Plot Details** dialog box, set the parameters as shown in the following graph.



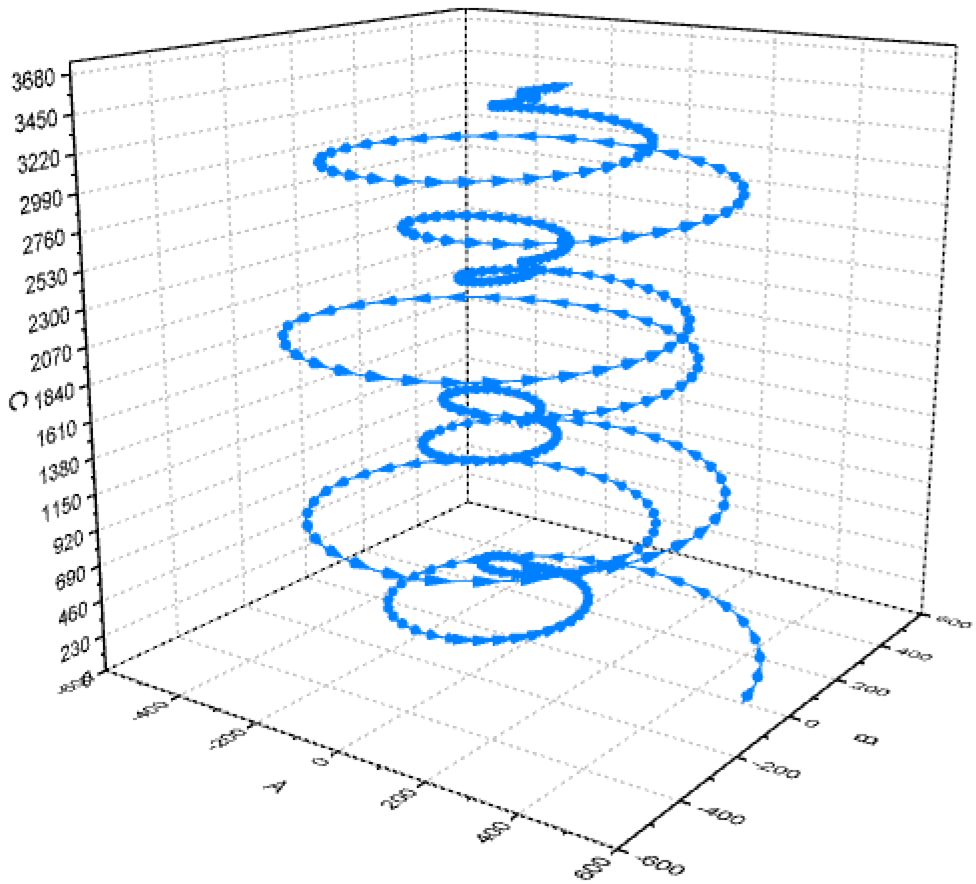
Click **OK** to apply these settings and close the dialog box.

5. Click the button in the **3D rotation** toolbar several times.

6. On the main menu, select **Graph**, point to **Speed Mode...**, then click **Open Dialog...** to open the **speedmode** dialog box. Select **Off** from the Speed Mode dropdown menu. Click **OK** to turn off speed mode and close the dialog box.



The final graph should look something like this:



5.10 Ternary

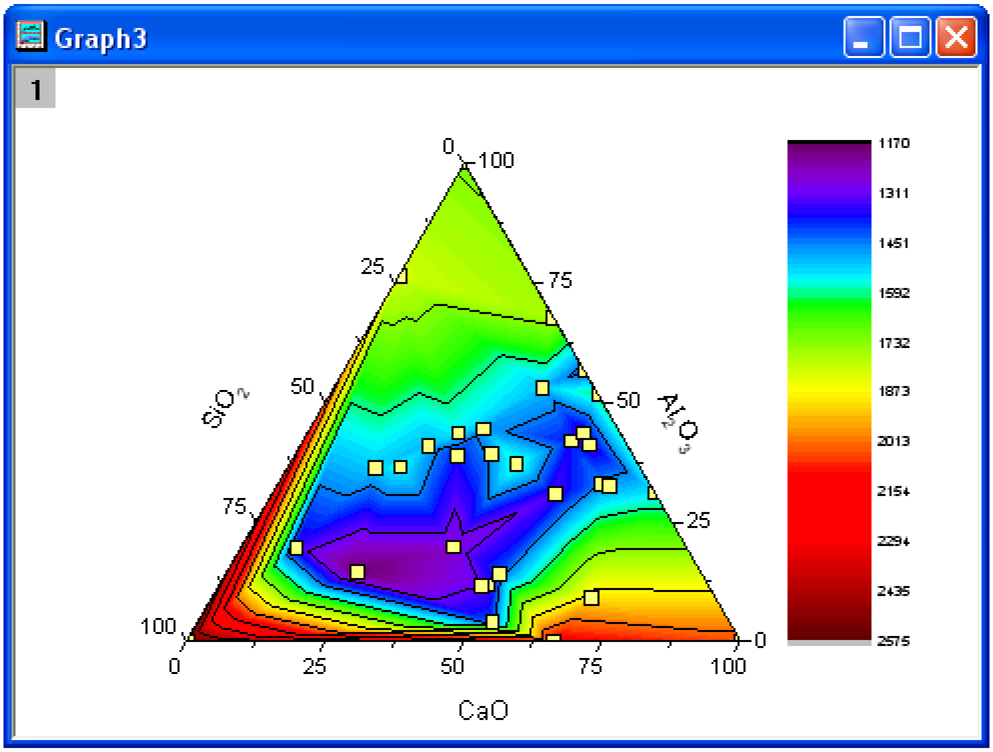
Topics covered in this section:

1. Ternary Contour

5.10.1 Ternary Contour

Summary

This tutorial will show you how to create a Ternary Contour graph and add scatters overlay.



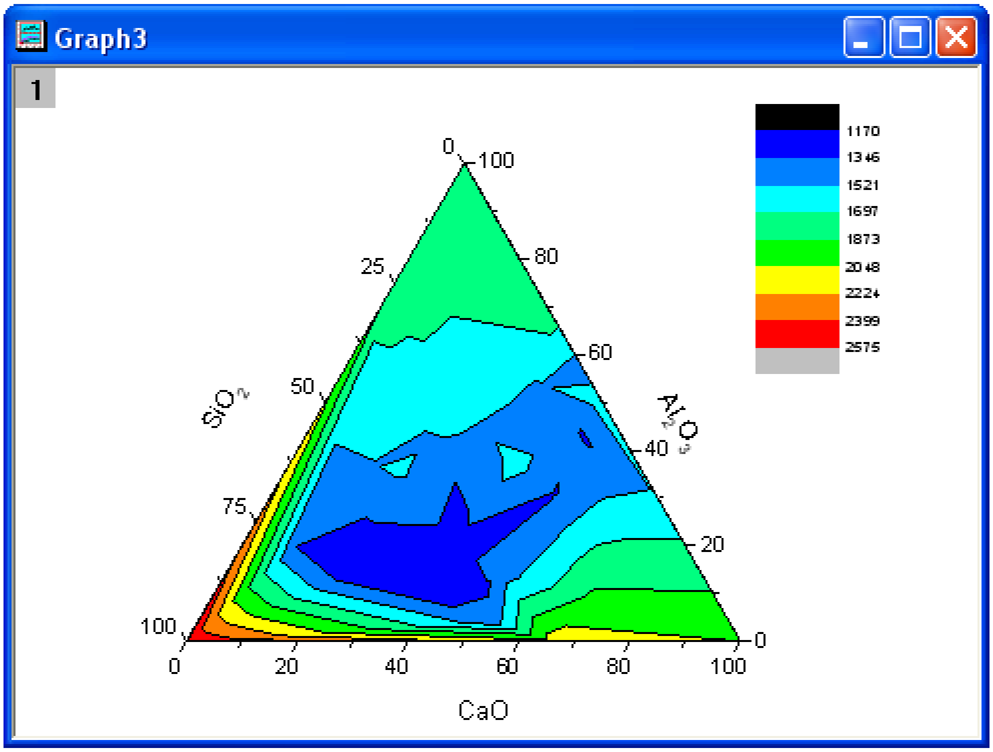
What will you learn

- Create the Ternary Contour graph
- Add scatters overlay with the Plot Setup dialog.
- Customize the graph by using the Plot Details dialog

Steps

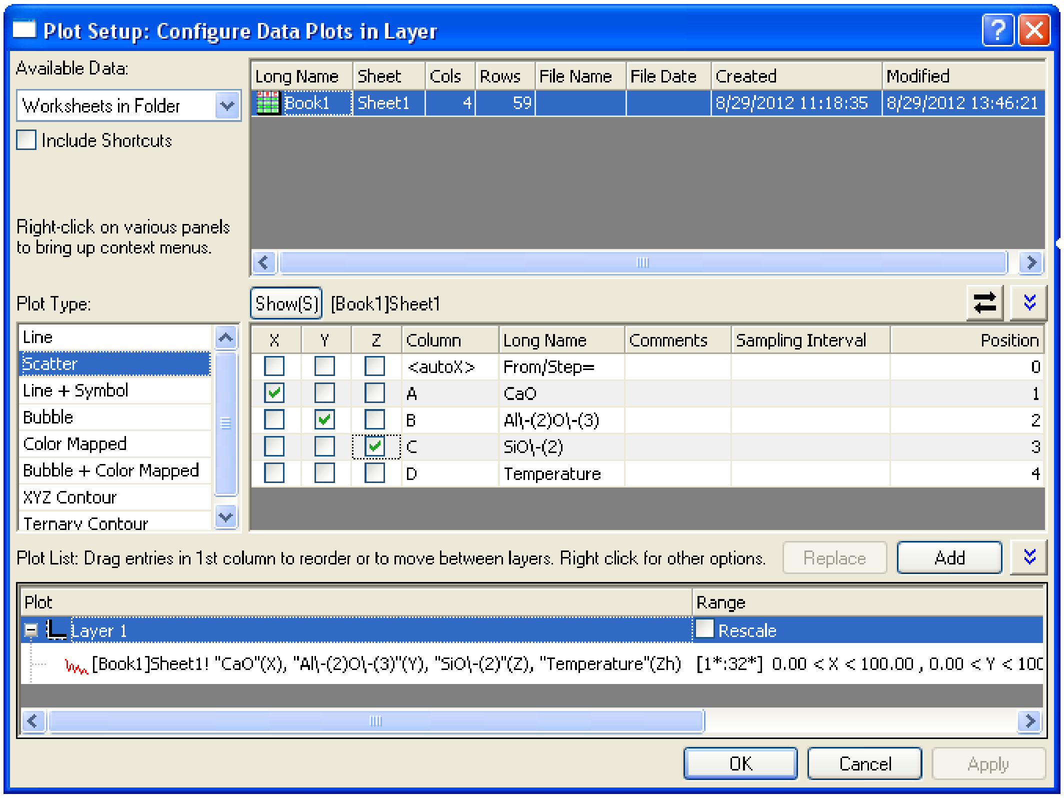
This tutorial is associated with the 2D and Contour Graphs project: *\Samples\2D and Contour Graphs.opj*.

1. Open the *2D and Contour Graphs: Contour: Ternary Contour* folder in the **Project Explorer**. Activate **Book1**,highlight entire worksheet and select **Plot: Contour: Ternary Contour** to create a Ternary Contour graph. The graph should look like:

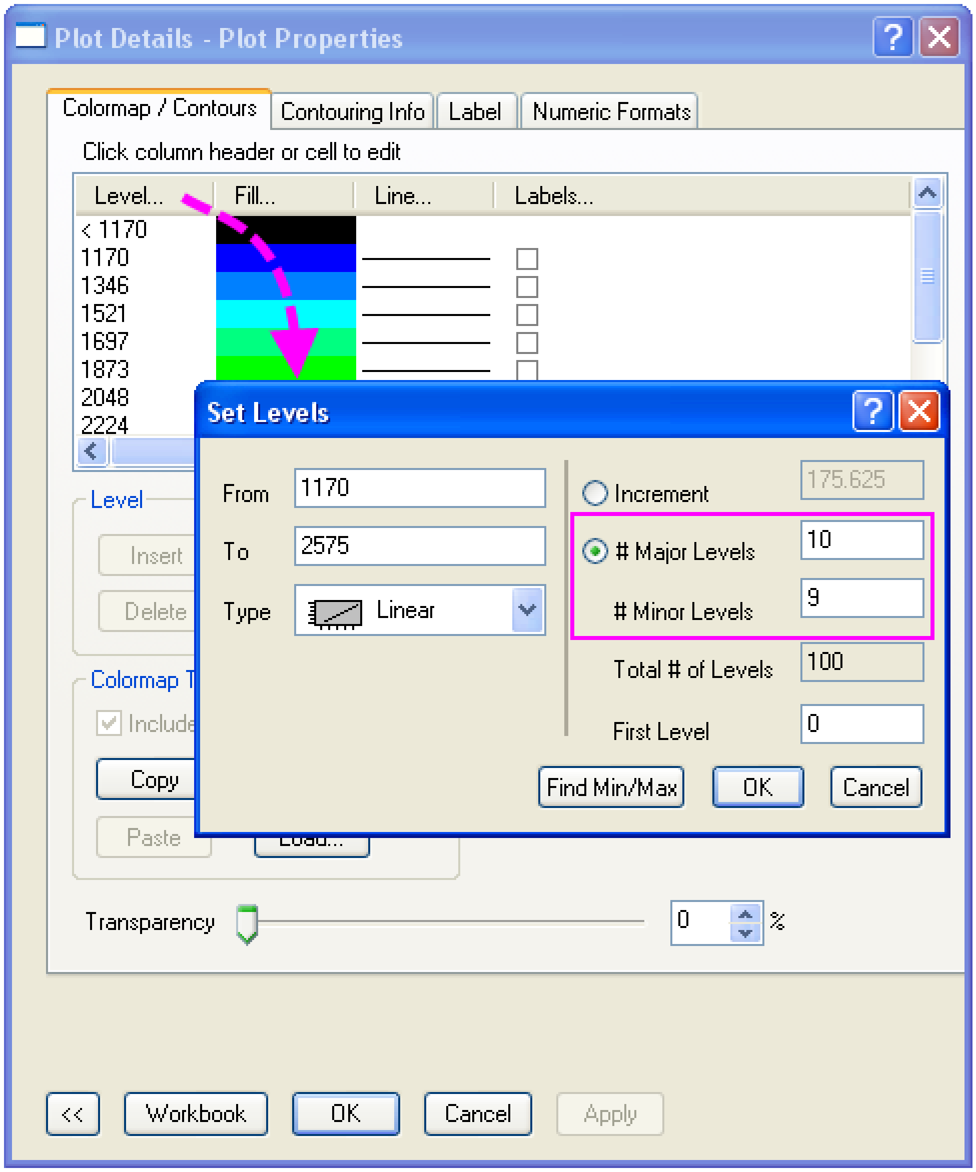


2. Now we will add overlay scatters to the plot. Right click on the layer icon and select **Plot Setup** in the context menu to open the **Plot Setup** dialog.

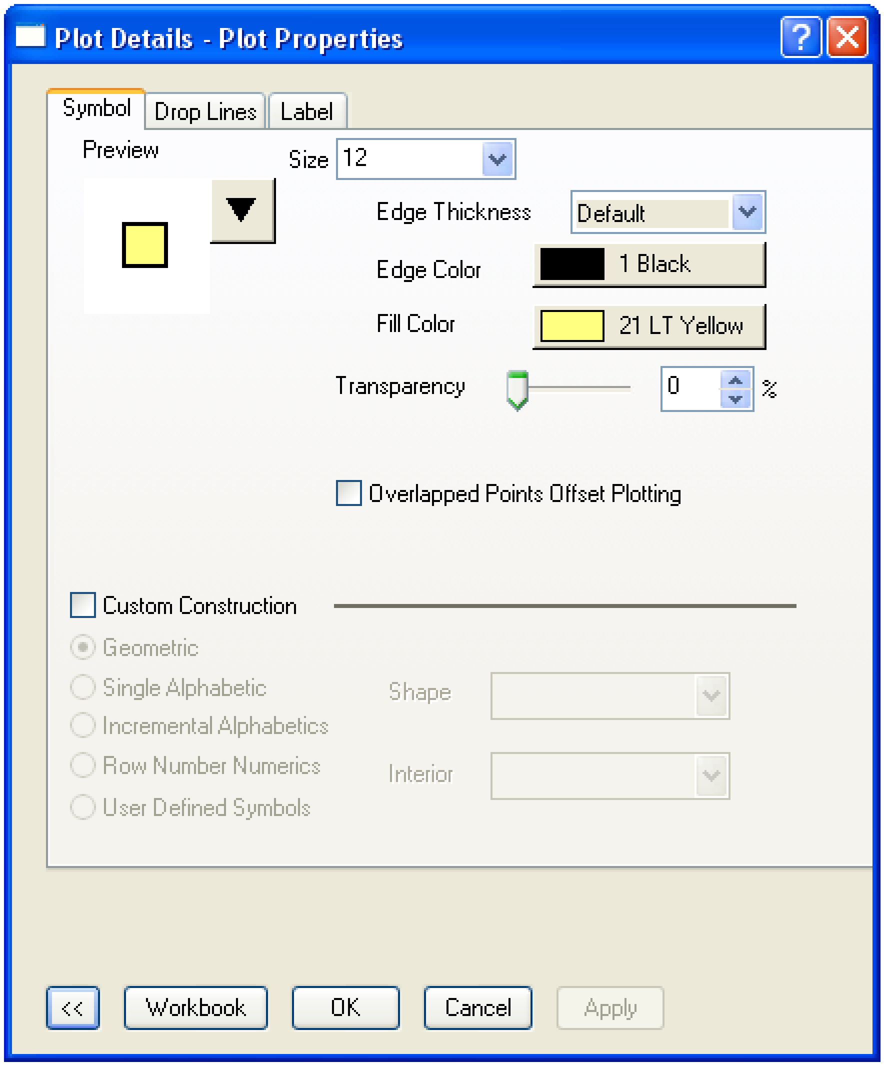
3. Set **Worksheets in Folder** as **Available Data**, select **Book1**, choose **Scatter** as **Plot Type** and column A, B, C as X, Y, Z respectively. Click **Add** to add the scatter plot to the ternary contour plot. The settings should be as following:



4. Now we will customize the contour graph and the scatter. Double-click on the contour plot to bring up the **Plot Details** dialog.
   - select the **Color Map/Contours** tab and click the **Level** heading, then set the dialog as the follwing graph shows.

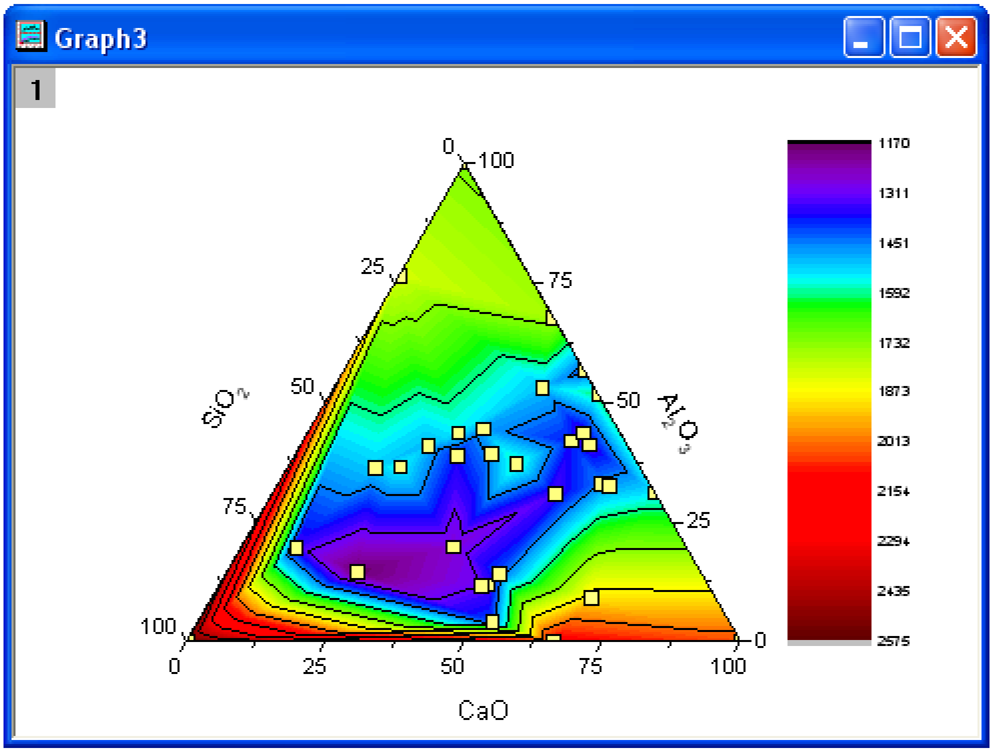


- Click OK to close the **Set Levels** dialog then click the **Fill** heading to open the **Fill** dialog. Select **Load Palette** and then select **Rainbow** from the **Palette** list and click OK.
- In the left panel of **Plot Details** dialog, select the scatter plot and customize the symbol as following:



- Click OK to close the **Plot Details** dialog.
- Drag and drop the color scale to resize and move it to a desired place

The graph should look like



5.11 Waterfall

Topics covered in this section:
1. Waterfall 2. 3D Waterfall Graph

5.11.1 Waterfall with Y and Z Color Mapping

Summary

Origin's **Waterfall** plot can use a dataset stored in a parameter row to set the Z offset and color map with Y or Z values.

What you will learn

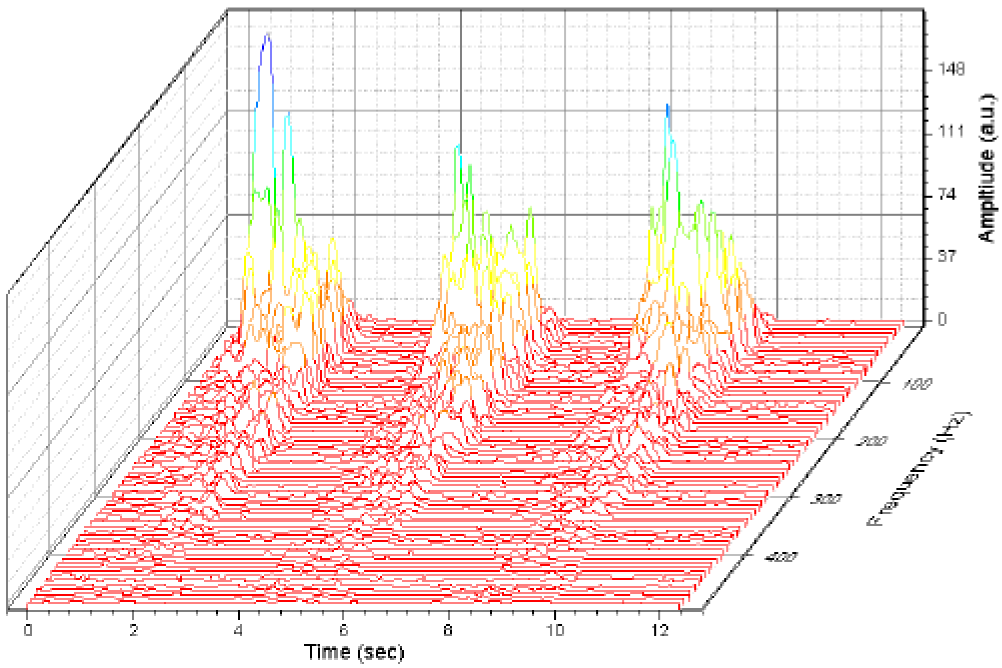
- Create a Waterfall color map with Y or Z values
- Customize colormap levels and palette

Steps

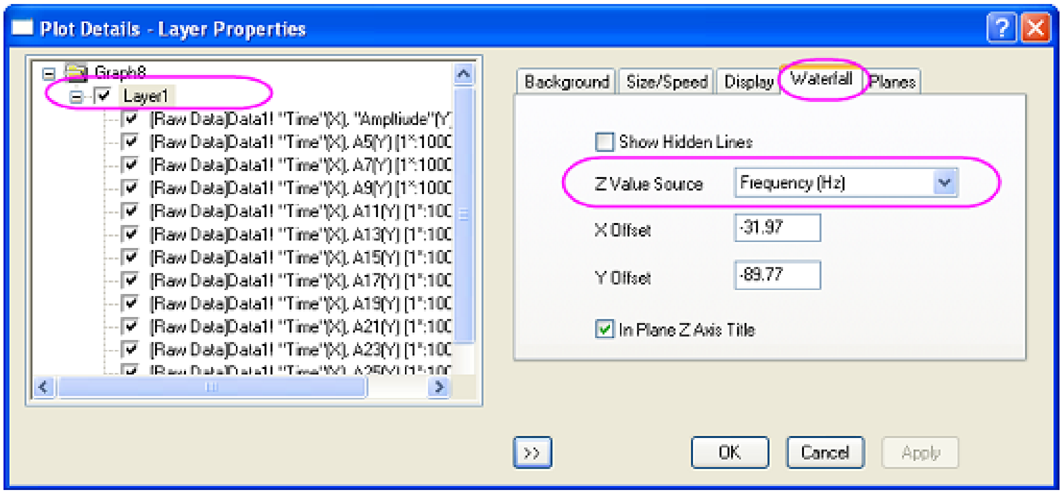
This tutorial is associated with the **Waterfall** folder in the **2D and Contour Graphs** project(\Samples\2D and Contour Graphs.opj).

Create Waterfall with Y Colormap

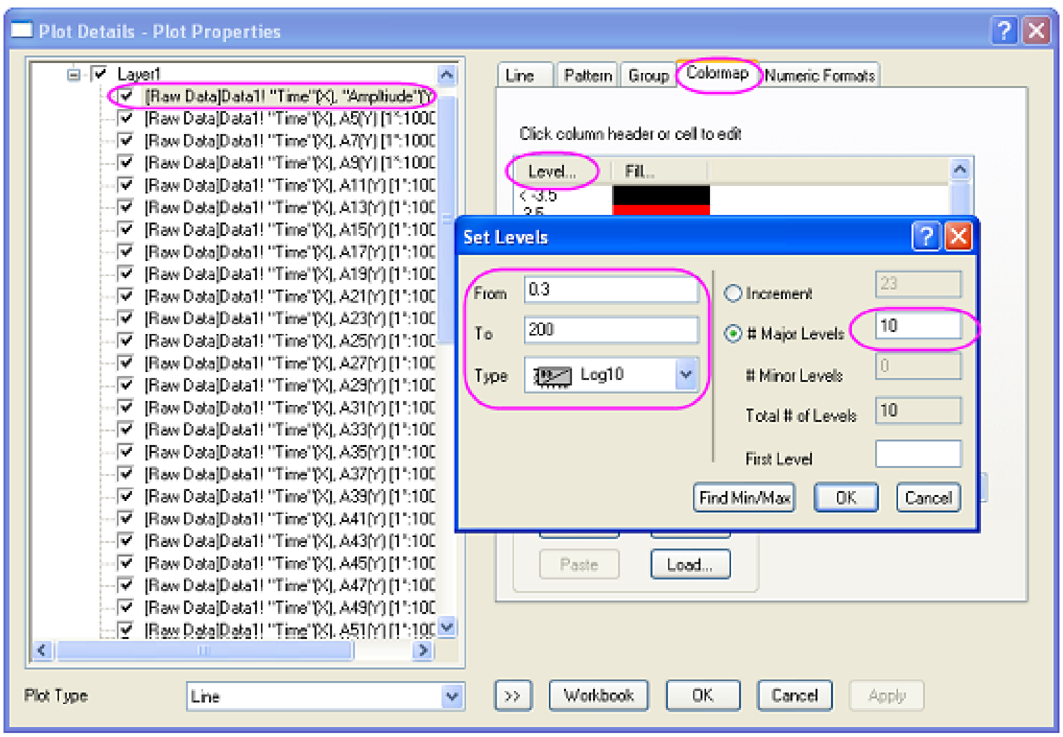
1. In the Project Explorer (usually at the left of the screen), browse to **\2D and Coutour Graphs\Multi Axis and Multi Panel\Waterfall**. Select the **Book4G** window to activate. Click in the upper left corner of the worksheet to select all data.
2. Right-click and select **Plot** from the menu, then point to **Multi-Curve**, and then select **Waterfall Y:Color Mapping** to create a Waterfall with Y color mapping. (Alternatively, select the **Waterfall Y:Color Mapping** button from the 2D Graphs toolbar.)



Note that the values stored in the user-defined parameter row **Frequency (Hz)** have been automatically picked up as the Z Value Source. Confirm this by double-clicking the plot to open the **Plot Details** dialog box and selecting **Layer1** in the left panel. Select the **Waterfall** tab and observe that the **Z Value Source** is set to **Frequency (Hz)**.

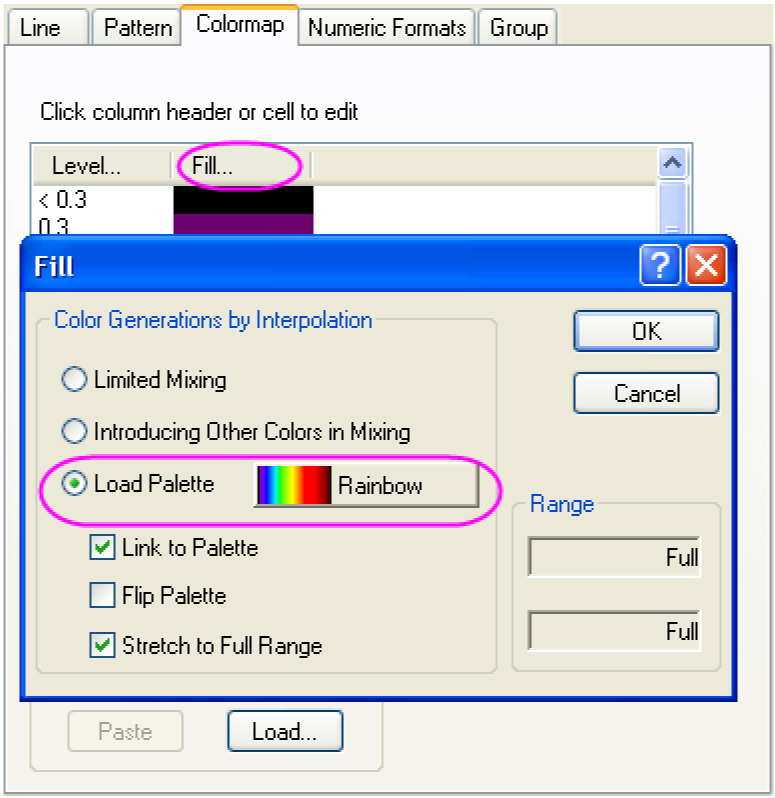


3. With the Plot Details dialog box still open, in the left panel, select the first line plot under Layer1. Select the **Colormap** tab in the right panel. Click on the **Level?** column header to open the **Set Levels** dialog box. Set levels as shown below:



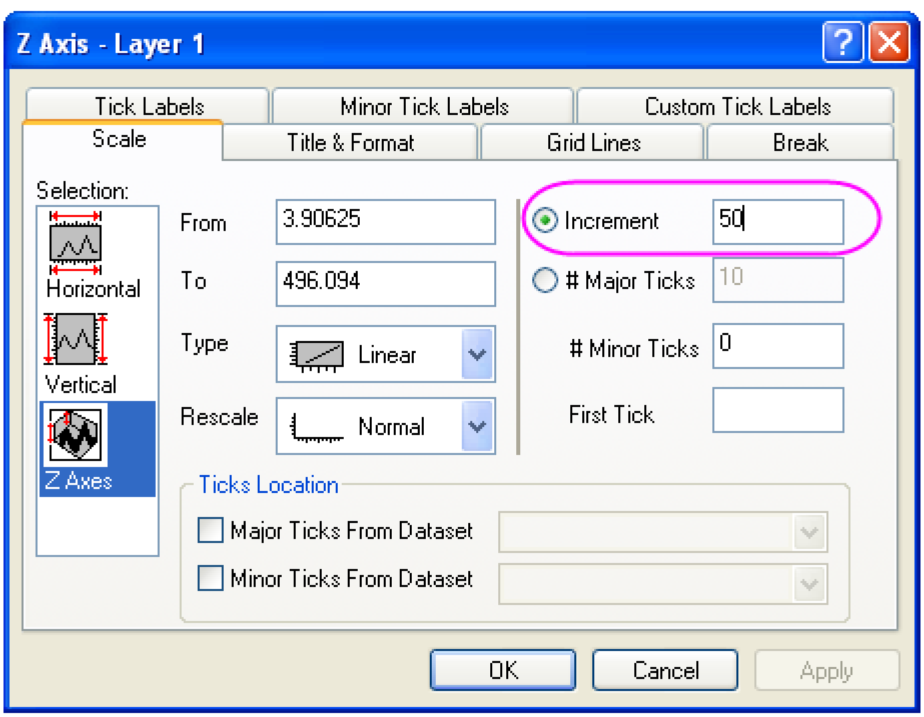
Click **OK** to close the Set Levels dialog box.

4. Click on the **Fill?** column header to open the **Fill** dialog box. Select **Load Palette**, and choose the **Rainbow** palette from the palette list. Click **OK** button to close the **Fill** dialog box.

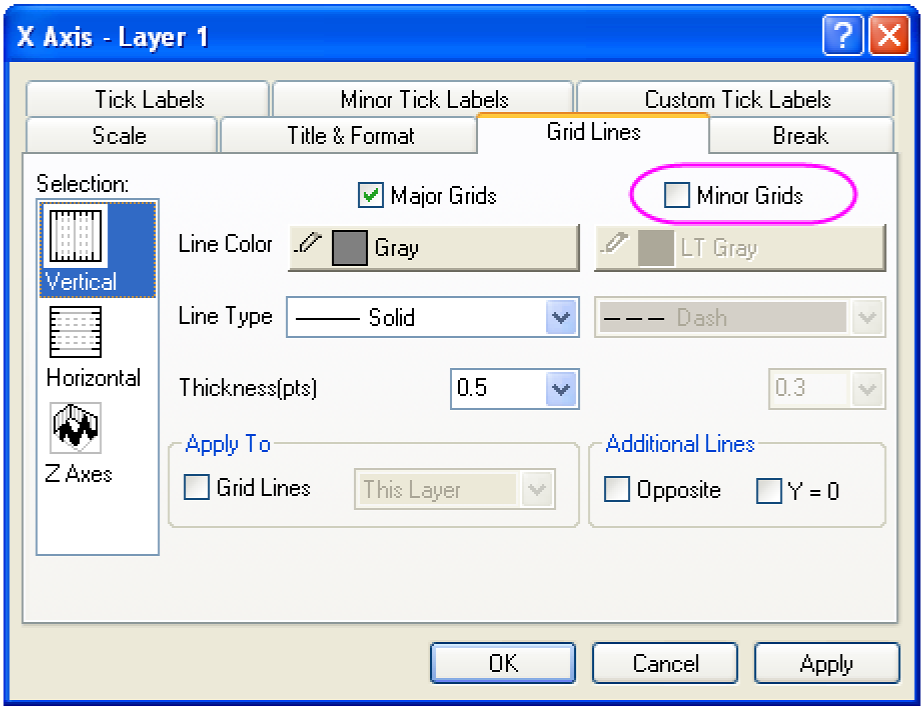


Click **OK** to apply these settings and close the **Plot Details** dialog box.

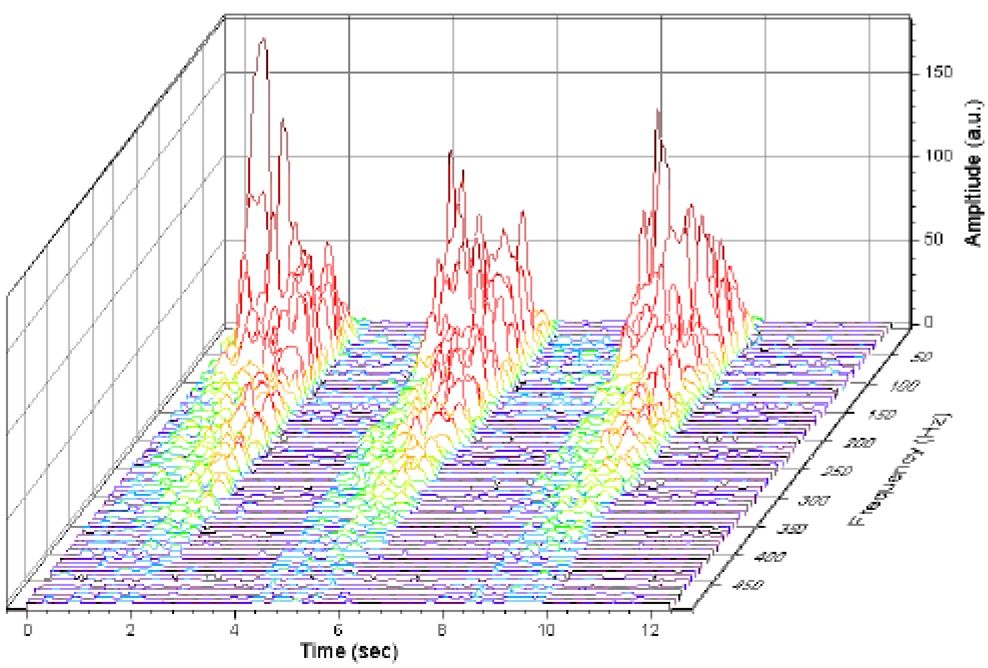
5. Double-click the plot's Z axis to open the Axis dialog box. Set the **Increment** to **50**. Select **Vertical** in the **Selection** box and set the vertical **Increment** to **50**.



Select the **Grid Lines** tab. Select **Horizontal** from the Selection box, and clear the Minor Grids check box. Select **Vertical** from the **Selection** box, and clear the **Minor Grids** check box.

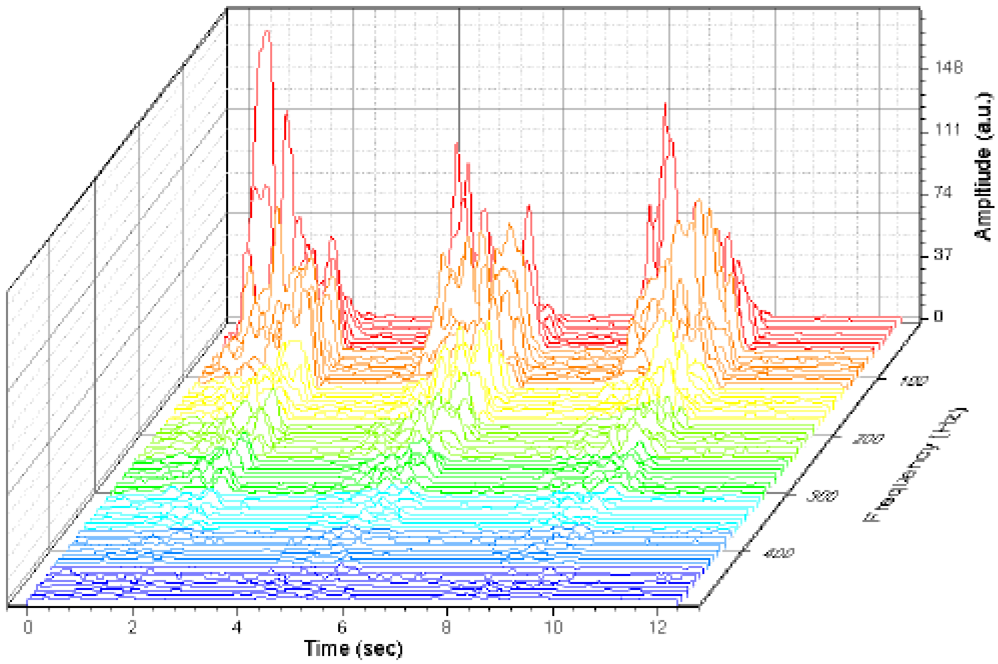


Click **OK** to apply changes and close the dialog box. The final graph should look like this:

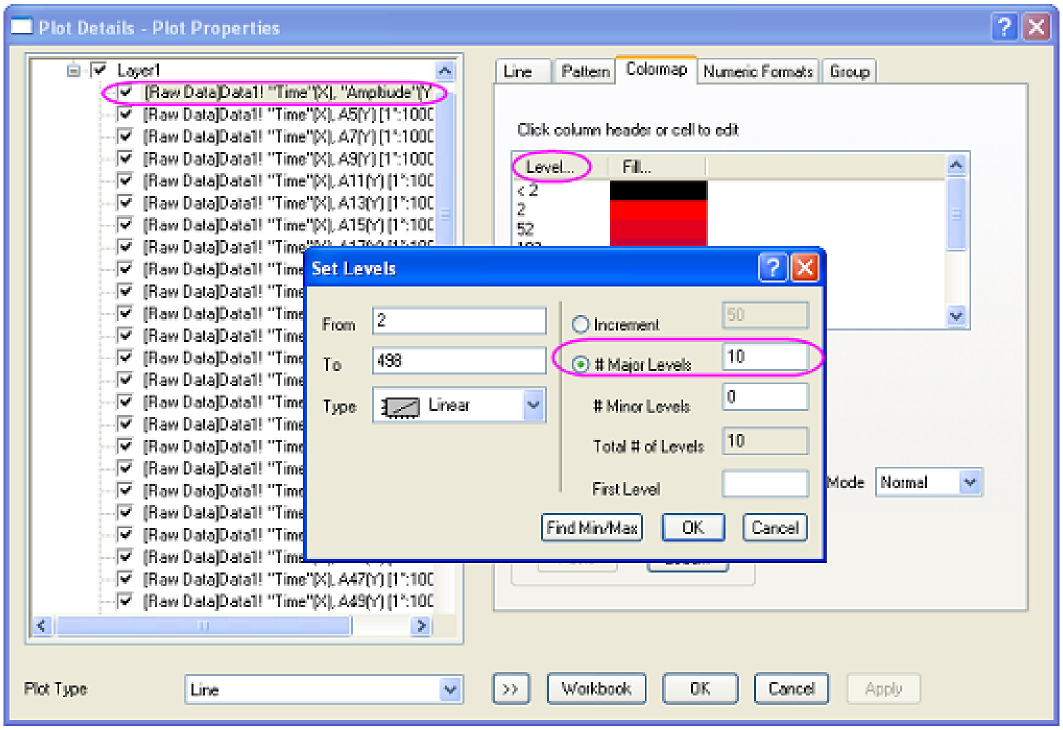


Create Waterfall with Z Colormap

1. In the Project Explorer (usually at the left of the screen), browse to **\2D and Coutour Graphs\Waterfall**. Select the **Book4G** window to activate. Click in the upper left corner of the worksheet to select all data.
2. On the main menu, click **Plot**, then point to **Multi-Curve**, and then select **Waterfall Z:Color Mapping** to create a Waterfall with Z color mapping. (Alternatively, select the **Waterfall Z:Color Mapping** button from the 2D Graphs toolbar.)

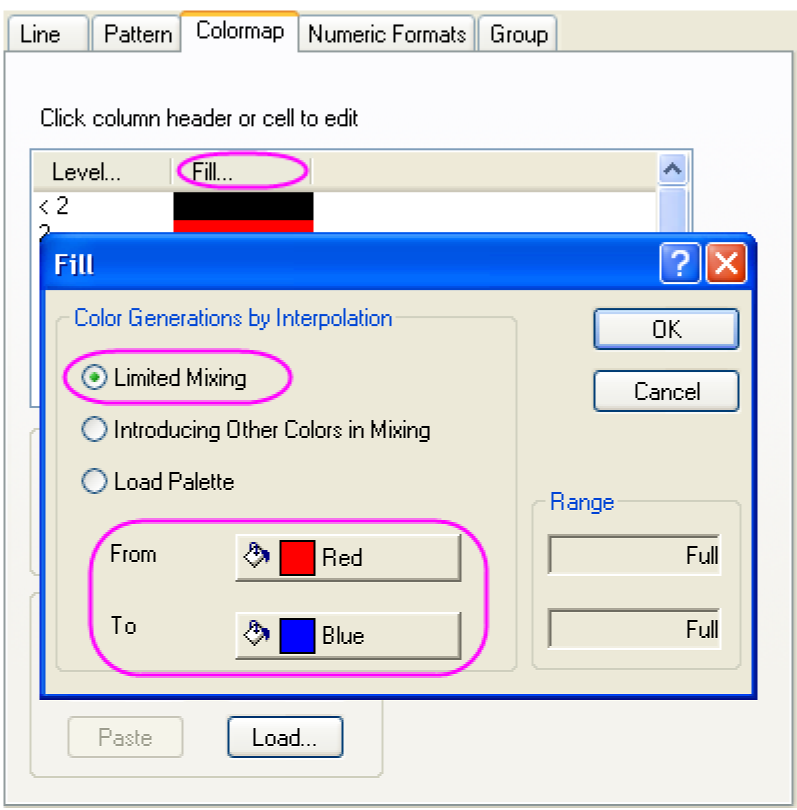


3. Double-click the plot to open the Plot Details dialog box. In the left panel, select the first line plot under Layer1. Select the **Colormap** tab in the right panel. Click the **Level?** column header to open the Set Levels dialog box. Set **Major Levels** to **10**.



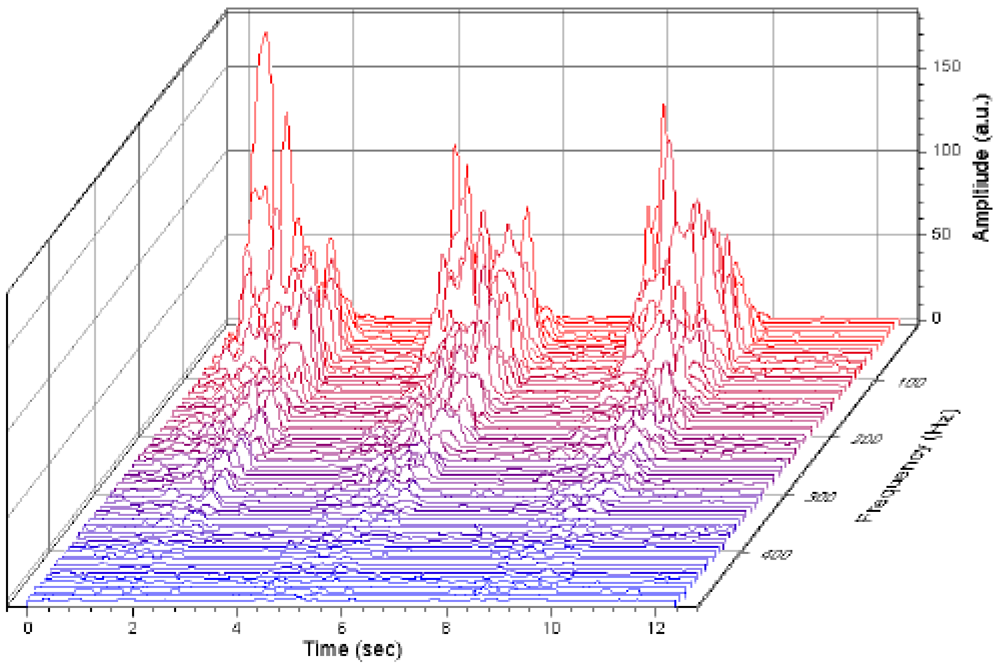
Click **OK** to close the Set Levels dialog box.

4. Click the **Fill?** column header to open the Fill dialog box. Select **Limited Mixing** and set the **From** color to **red** and the **To** color to **blue**. Click **OK** to apply these settings and close the dialog box. Click **OK** to close the Plot Details dialog box.



5. Double-click the Y axis to open the **Axis** dialog box. Set the vertical **Increment** to **50**. Select the **Grid Lines** tab. Select **Horizontal** from the **Selection** box, and clear the **Minor Grids** check box. Select **Vertical** from the **Selection** box and clear the **Minor Grids** check box. Click **OK** to close the Axis dialog box.

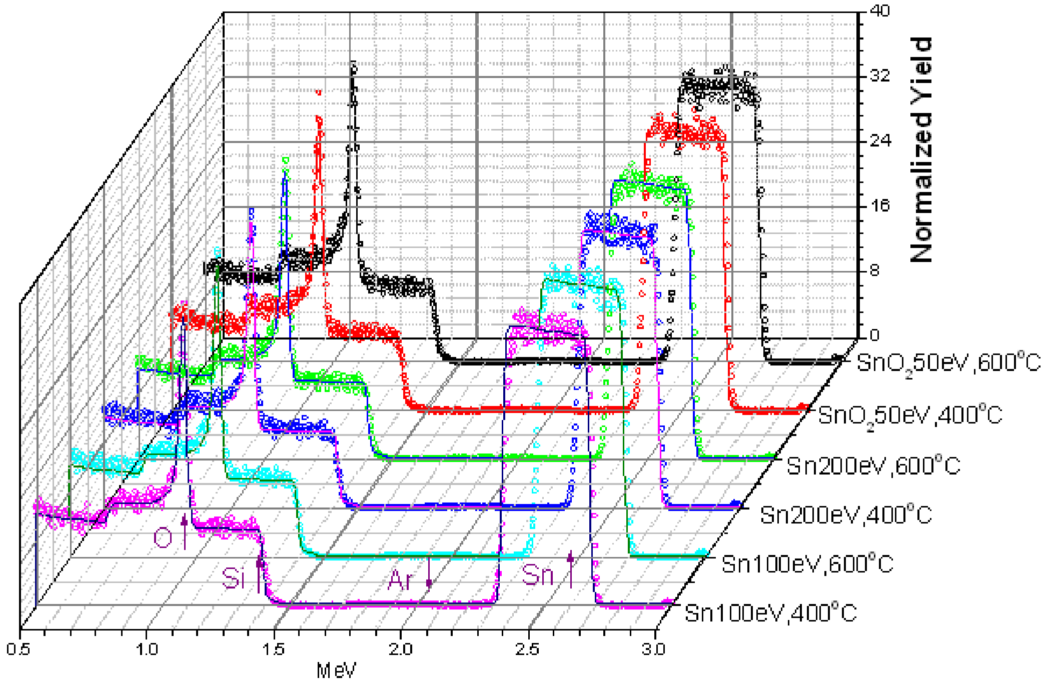
Your final graph should look like this:



5.11.2 3D Waterfall Graph

Summary

The Origin 2D Waterfall graph plots one or more Y columns, or a range from one or more Y columns, as a series of line plots that "recede" into the page. Such graphs are effective for comparing datasets collected under conditions where some parameter is varied incrementally.



Sample of Waterfall graph in Multi-Curve category

What you will learn

- Create waterfall plot
- Change symbol/color for grouped plots
- Merge graphs

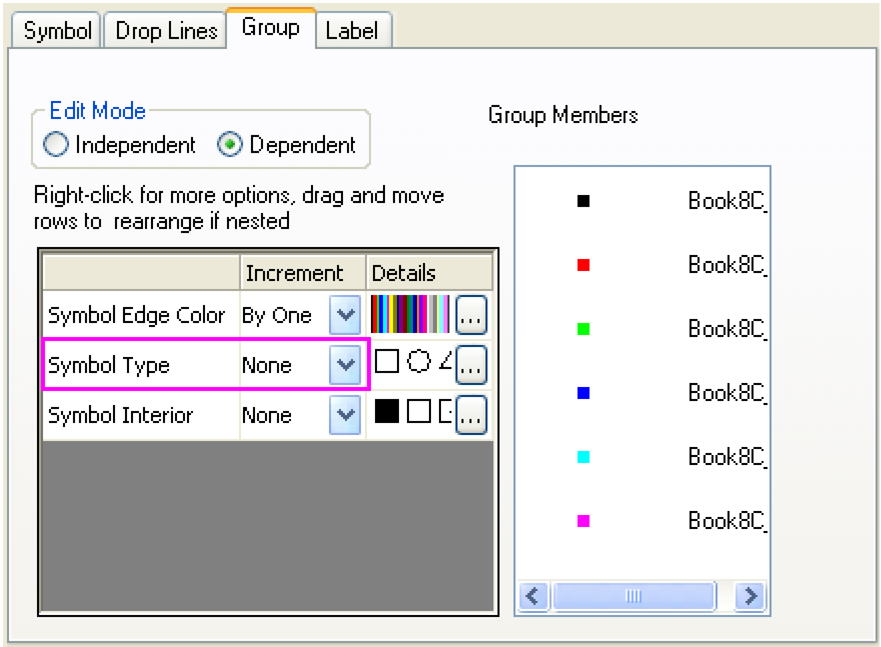
Steps

This tutorial is associated with this graph in Graph Galley. Download the project file in this graph galley page.

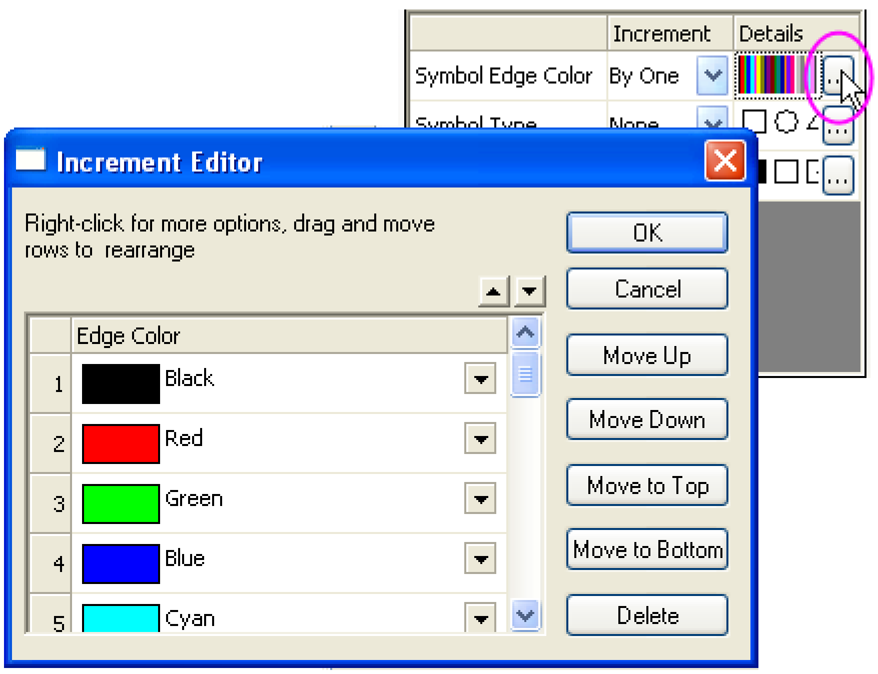
There are two worksheets and one graph in this folder. This example illustrates how to create such a waterfall plot.

1. For each worksheet, highlight all columns and plot as Waterfall graph from **Plot: Multi-Curve: Waterfall**, and use *Measured* and *Simulated* as graph long name, respectively.
2. Double-click the X or Y axis of each plot and adjust axes scales (X Axis From = 0.5, To = 3.0 and Y Axis From = 0, To = 40).
3. Double-click any plot in the *Measured* plot to bring up **Plot Details** dialog at the data level, and change the **Plot Type** drop-down in the bottom left corner to **Scatter** and click **Apply**.

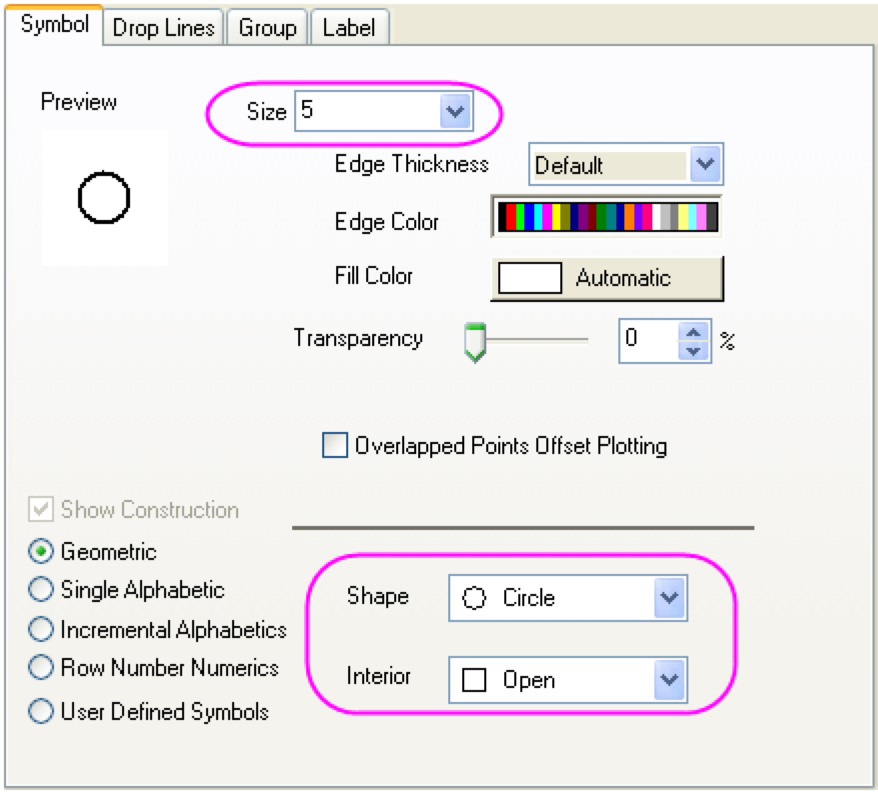
   In the **Group** tab of the **Plot Details** dialog, change the **Symbol Type** to **None** and click **Apply**.



Click in Details next to **Symbol Edge Color** and click the Browse button that appears. Note how the color increment is Black, Red, Green, Blue, Cyan and Magenta.

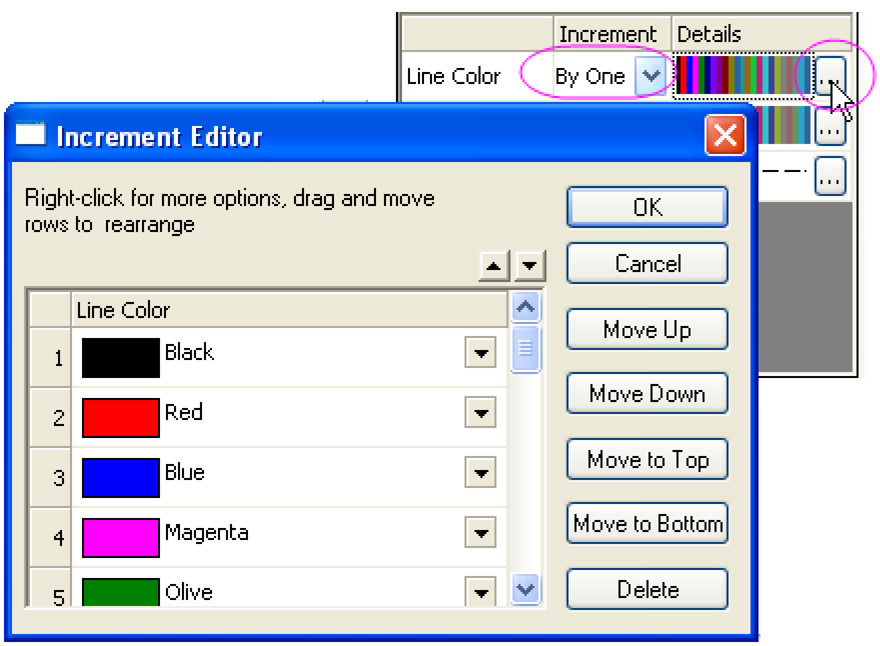


Switch to the **Symbol** tab. Set symbol **Size** to 5, and use **Open** and **Circle** symbol.



Click **OK** to close **Plot Details** dialog.

4. Double-click on any plot in the *Simulated* graph to bring up **Plot Detail** dialog at the data level for this plot. In **Group** tab, change the **Line Color** increment type to **By One**, and change the color Details for **Line Color** to Black, Red, Green, Blue, Cyan and Magenta.



Click **OK** to close **Plot Details** dialog.

5. Now, we will merge these two graphs. Select **Graph: Merge Graph Windows**. Choose **Specified** from **Merge** drop-down list, and select the two graphs you just created in **Graphs** edit box. Arrange the graph by using the following settings:

   Arrange Settings:

Number of Rows:	1
Number of Columns:	1
Linked Layers:	Check/Enable

6. Click the **OK** button to merge the graphs.

7. In the sample we have added labels and changed axis titles.

5.12 Specialized

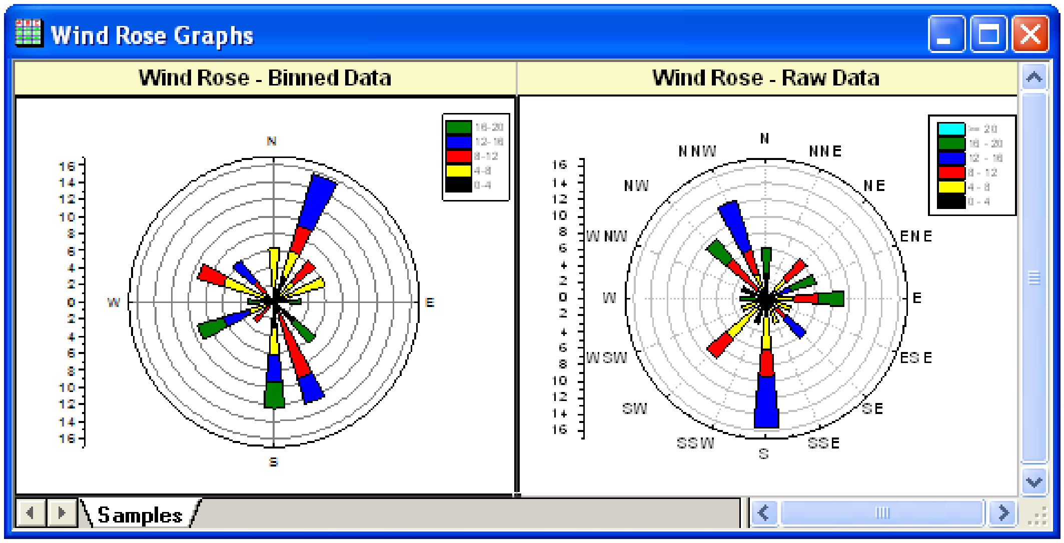
Topics covered in this section:

1. Windrose Graph
2. Plot Functions with Parameters Defined in a Worksheet
3. Open-High-Low-Close-Volume Stock Chart

5.12.1 Windrose Graph

<u>Summary</u>

Windrose graphs are used to present wind speed data and wind direction data collected over some time at a particular location. This tutorial will show you how to create a windrose graph with Origin 9 (post-SR5 builds are required. To create windrose graph with *Raw Data* mode, Origin 8.1 is required).



Minimum Origin Version Required: Origin 8.5.1

What will you learn

This tutorial will show you how to:

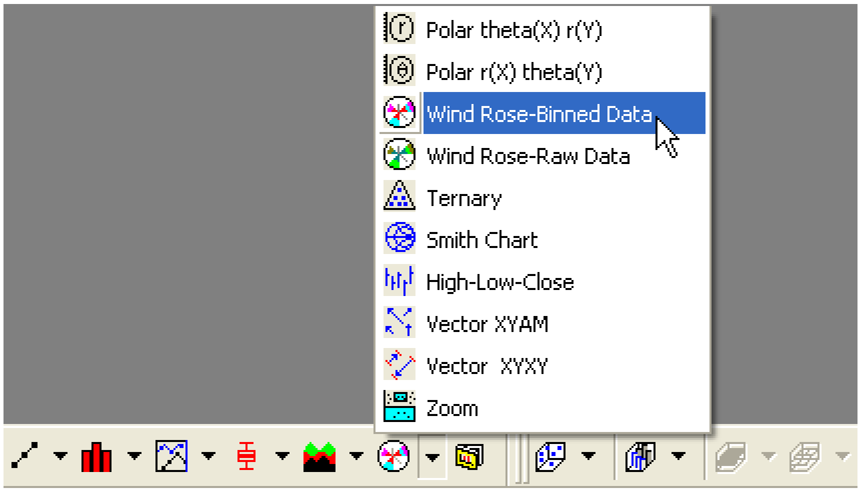
- Create a Windrose graph from binned data.
- Customize the direction tick labels of the Windrose graph.
- Create a Windrose graph from raw data.

Steps

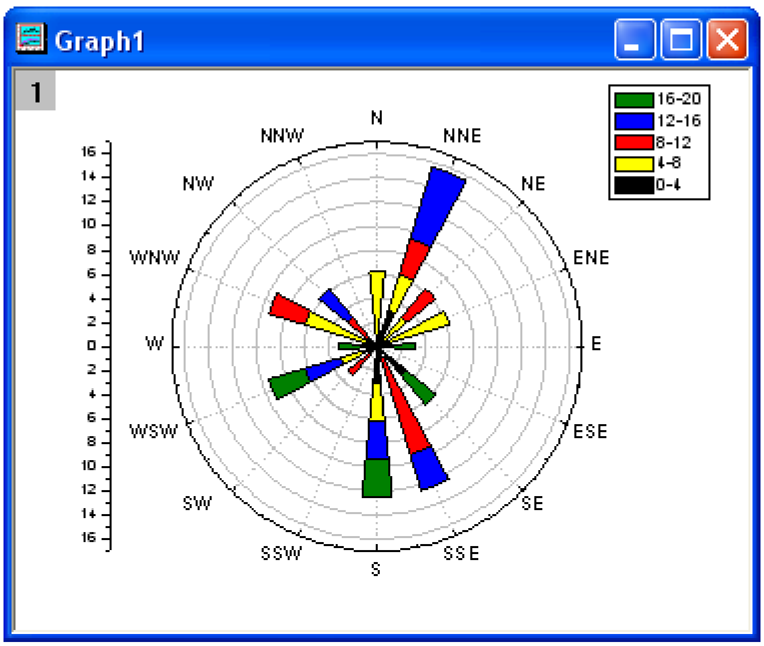
This tutorial is associated with the Statistical and Specialized Graphs project: *\Samples\Statistical and Specialized Graphs.opj*.

Part 1: Create a Windrose Graph from Binned Data

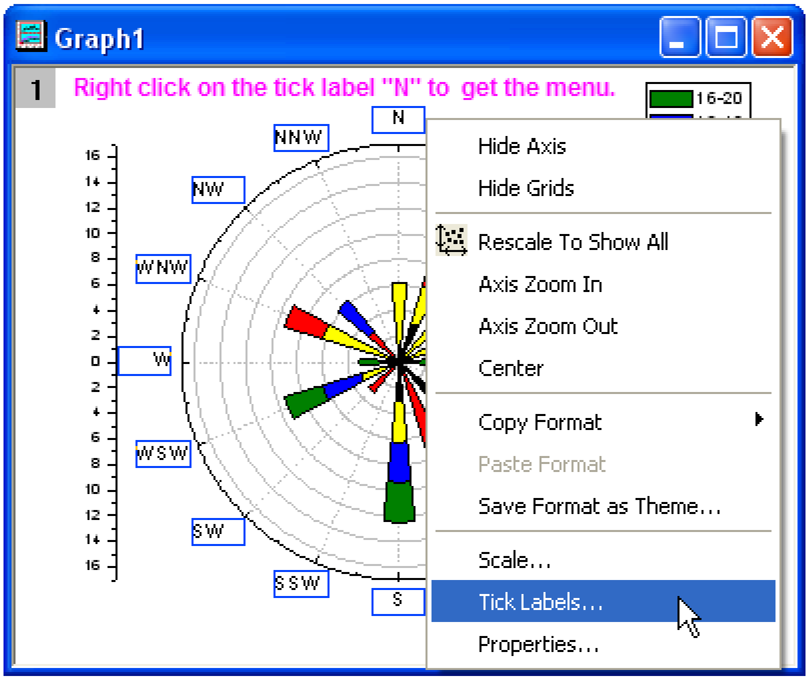
1. Go to the project folder *Statistical and Specialized Graphs: Specialized: Wind Rose* and activate the *Windrose Binned Data* workbook. (If you don't have the project file, please import the sample data from here).
2. Highlight all the columns and click the **Wind Rose - Binned Data** button from the **2D Graphs** toolbar.



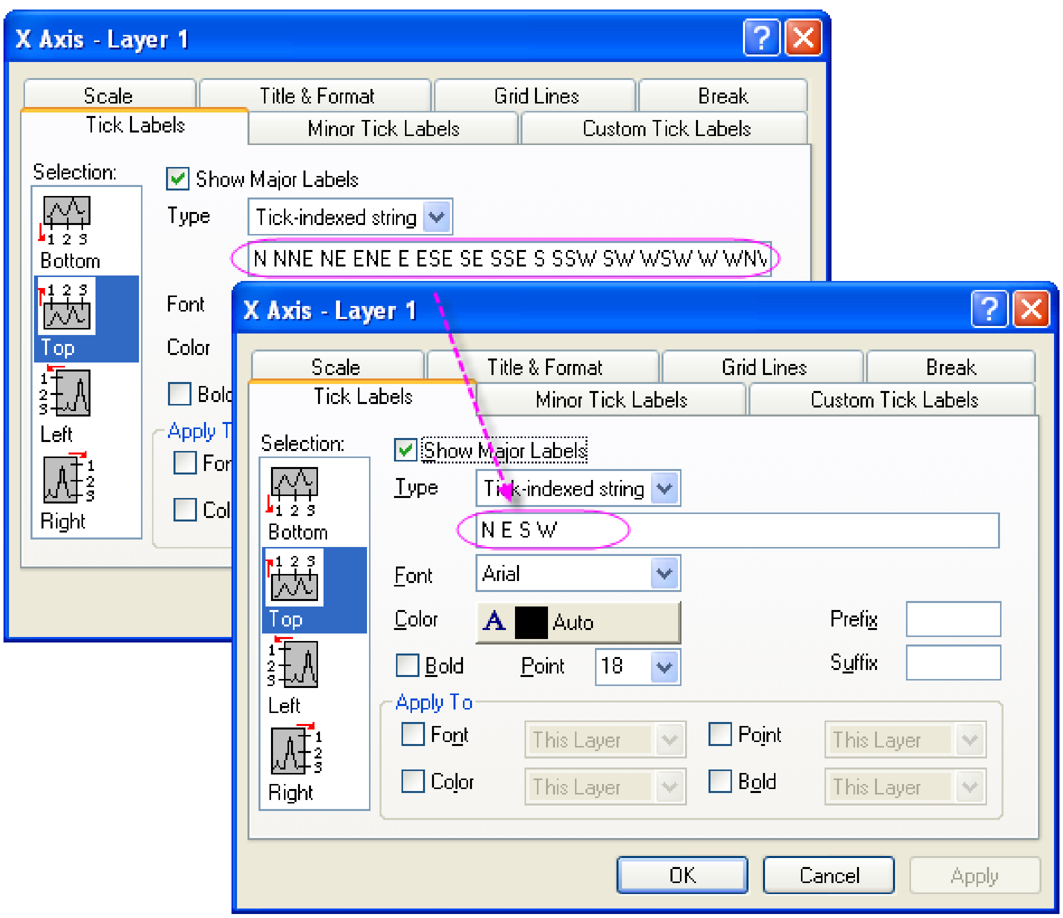
The Windrose graph should look like the following:



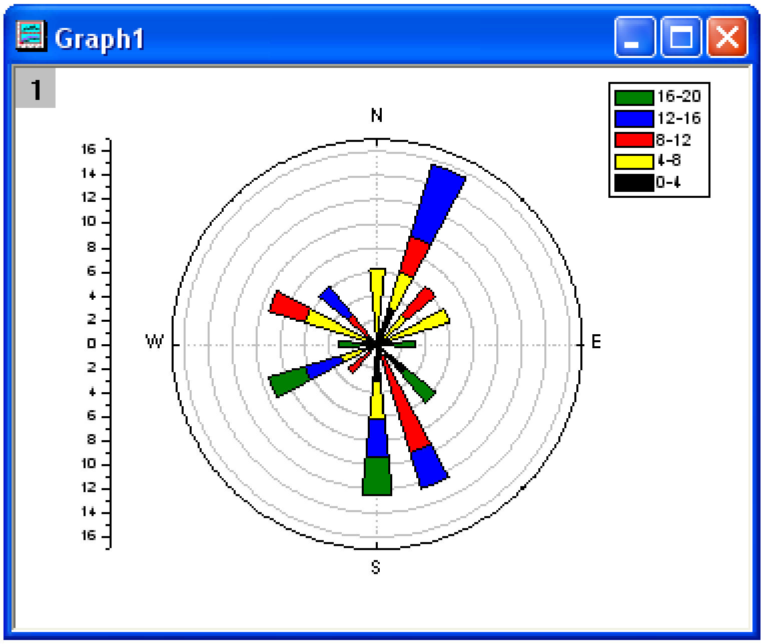
3. Now, change the wind direction to show only N,E,S,W. Right-click on the tick label **N** and then select **Tick Labels** from the short-cut menu. This opens the **X Axis** dialog.



Change the options as the following screenshot.

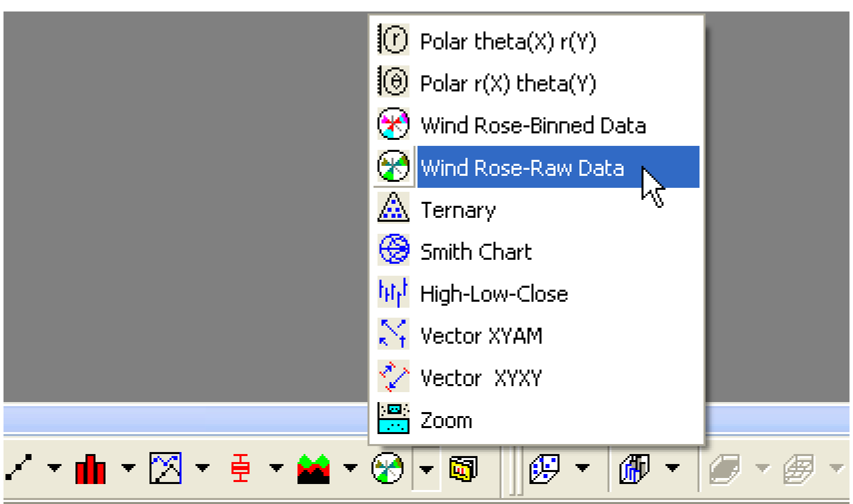


Activate the **Scale** tab. Enter **90** in the **Increment** edit box and then click the **OK** button. The graph should look like:

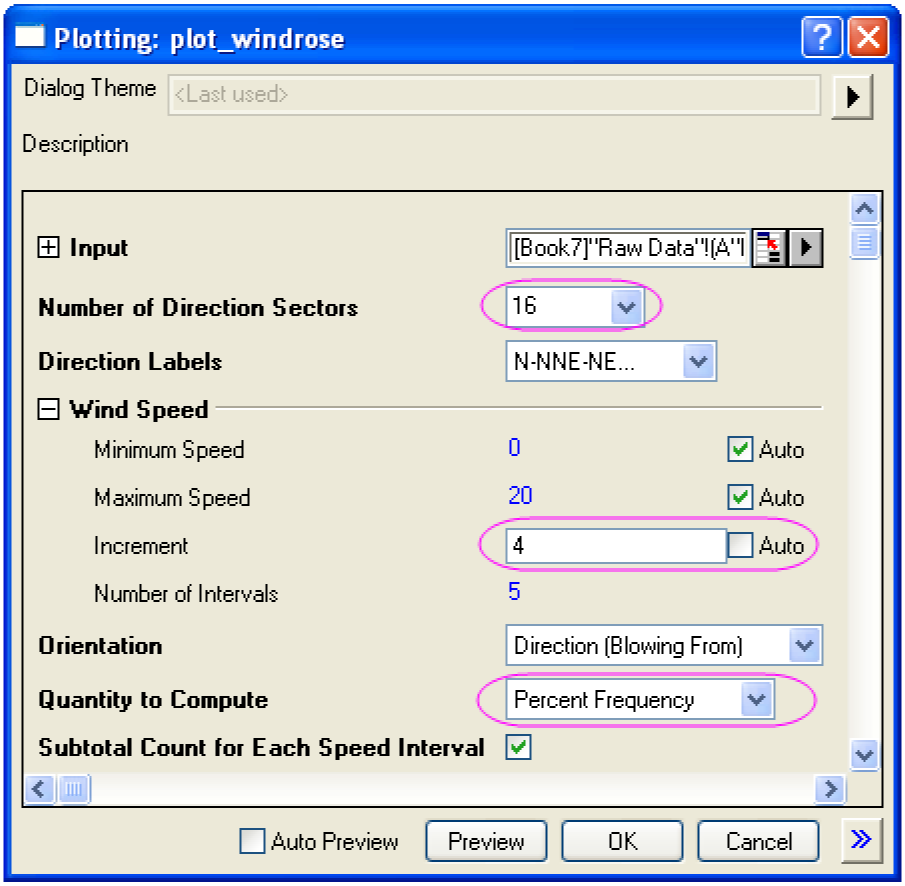


Part 2: Create a Windrose from Raw Data

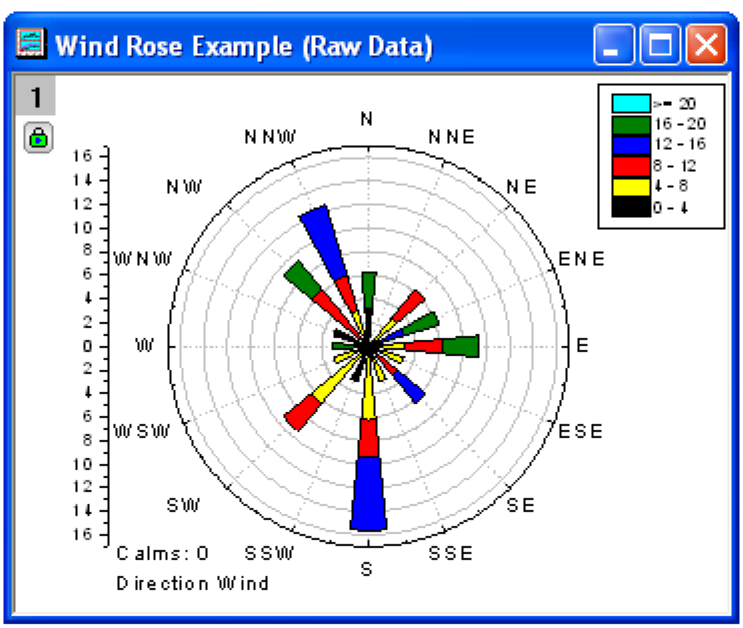
1. Go to the project folder *Statistical and Specialized Graphs: Specialized: Wind Rose* and activate the *Windrose Raw Data* workbook. Go to the first sheet (the Raw Data worksheet). (If you don't have the project file, please import the sample data from here).
2. Highlight columns A & B. Then click the **Wind Rose - Raw Data** button on the 2D Graphs toolbar.



3. Use the settings below to create the Wind Rose graph:



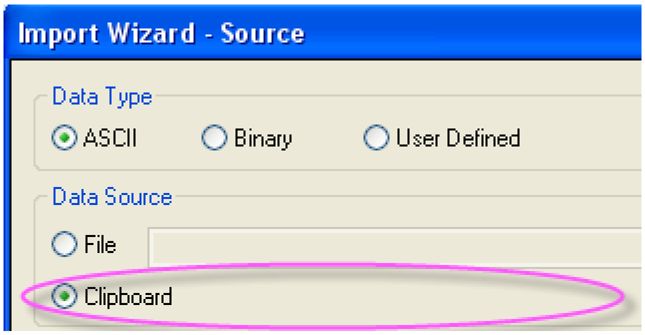
The final graph should look like:



Sample Data

Binned Data

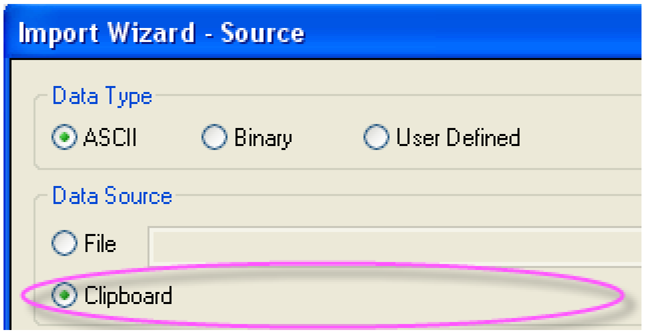
To import **Binned Data** to worksheet, you can copy the **Binned Data 1** (including the heading) and then select **File: Import Wizard**. Select the **Clipboard** checkbox in **DataSource** group and then click **Finish** button to import the data into Origin.



Direction	0-4	4-8	8-12	12-16	16-20
22.5	3.125	3.125	3.125	6.25	0
45	0	3.125	3.125	0	0
67.5	0	6.25	0	0	0
90	0	0	0	0	3.125
112.5	0	0	0	0	0
135	3.125	0	0	0	3.125
157.5	0	0	9.375	3.125	0
180	3.125	3.125	0	3.125	3.125
202.5	0	0	0	0	0
225	0	0	3.125	0	0
247.5	0	3.125	0	3.125	3.125
270	0	0	0	0	3.125
292.5	0	6.25	3.125	0	0
315	0	0	3.125	3.125	0
337.5	0	0	0	0	0
360	0	6.25	0	0	0
382.5	0	0	0	0	0

Raw Data

To import **Raw Data** to worksheet, you can copy the **Raw Data** (including the heading) and then select **File: Import Wizard**. Select the **Clipboard** checkbox in **DataSource** group and then click **Finish** button to import the data into Origin.

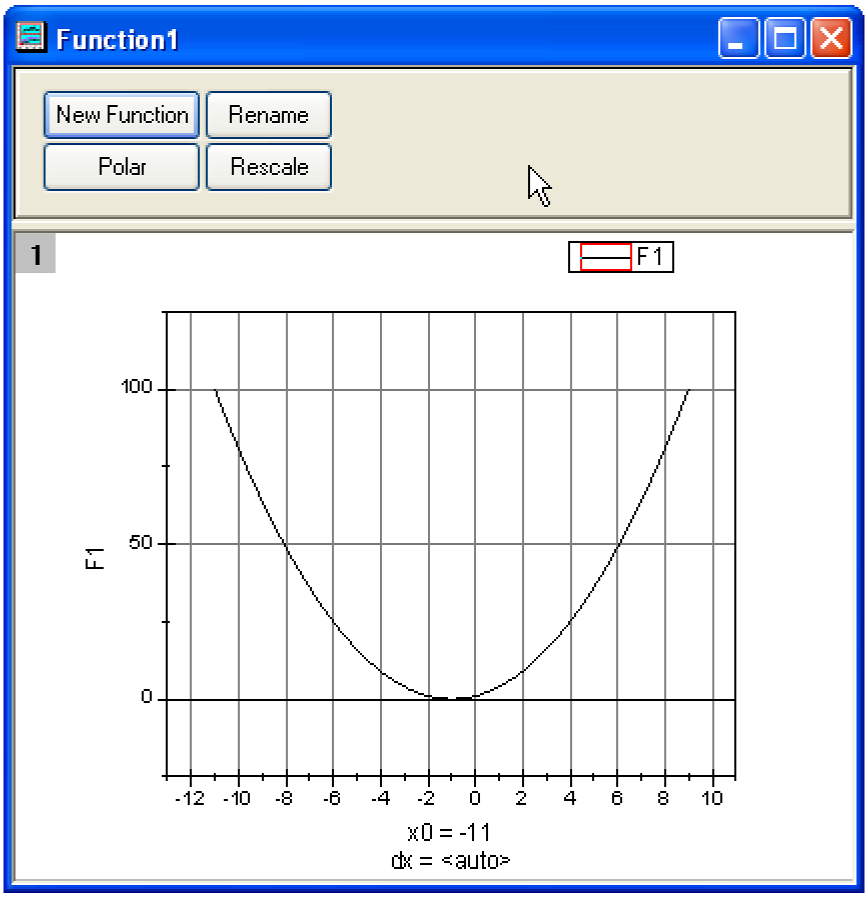


Direction	Speed
311.5	12.75
142.7	11.18
161.6	5.9
277.3	8.24
155.3	13.46
40.8	8.57
43.4	4.38
1.3	10.91
78.8	18.72
237.8	16.22
114.6	0.88
2.1	12.05
290.5	4.6
174.1	3.29
267.6	16.64
8	5.6
213.9	8.04
134.8	17.26
137.6	11.87
46.1	5.48
4.5	13.47
311.2	10.17
154.4	11.17
176.2	18.91
348.1	4.11
225.2	6.65
236.4	12.87
11.7	1.07
278.9	4.36
356.5	5.01
58.9	7.3
161.8	15.6

5.12.2 Plot Functions with Parameters Defined in a Worksheet

Summary

Origin can plot functions. It also can plot functions with parameters defined in a worksheet. The function graph can be updated automatically as the parameters in the worksheet change.



Minimum Origin Version Required: Origin 8.0 SR6

What you will learn

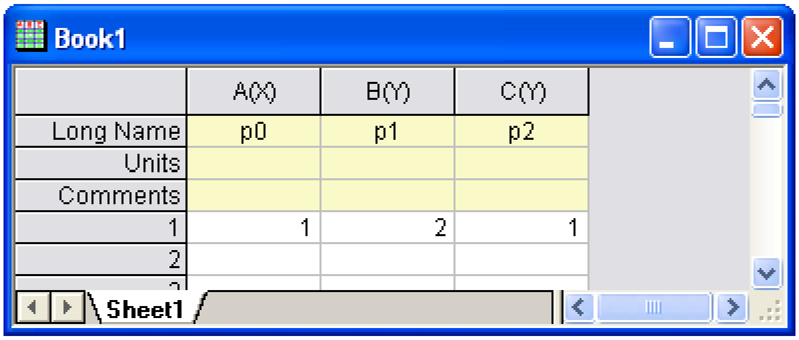
This tutorial will show you how to:

- Define variables from a worksheet in the **Set Values** dialog box.
- Plot a function graph with parameters.
- Update a graph automatically when parameters are changed.

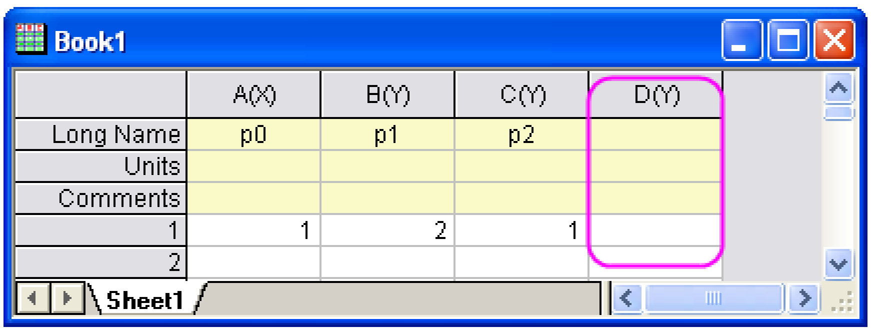
Steps

Let us use this function as an example: `y=p0+p1*x+p2*x^2`

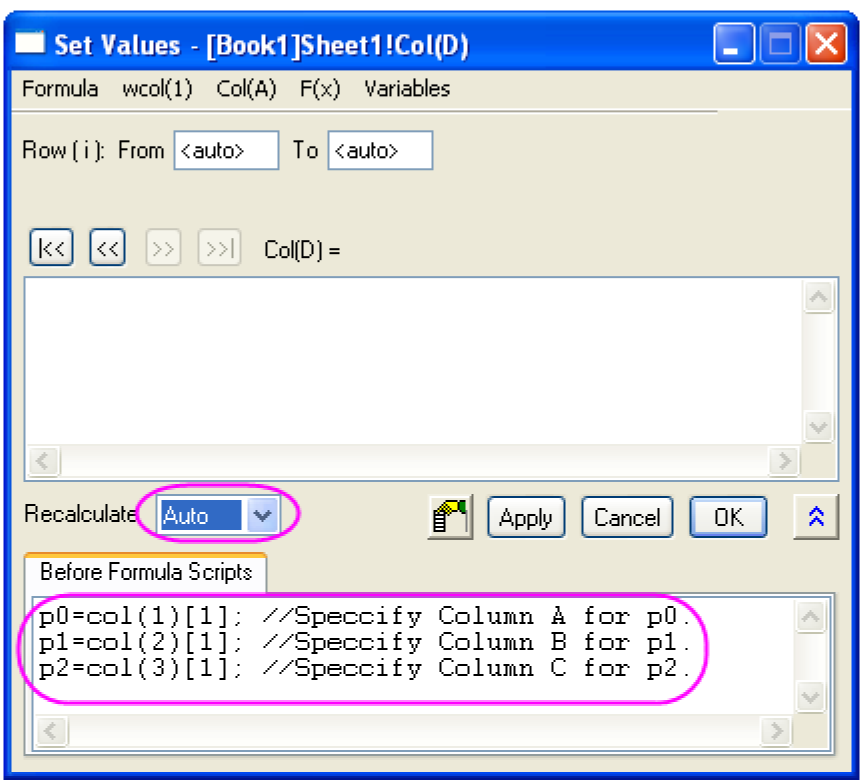
1. Set up a worksheet with three parameters p0, p1, p2 stored in Column A, Column B, Column C as shown below.



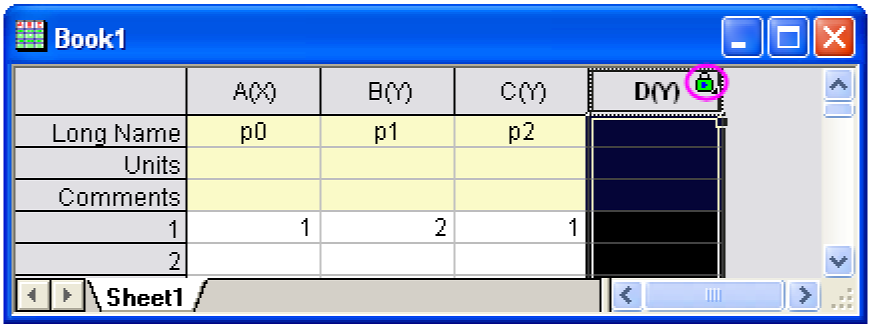
1. Click on the **Add New Columns** button  on the **Standard** toolbar to add a new column to the worksheet.



2. Highlight Column D and then select **Column: Set Column Values**. Select **Auto** from the **Recalculate** drop-down. Type the script shown below to define the parameters in the **Before Formula Scripts** edit box. Click the **OK** button to close the dialog box.

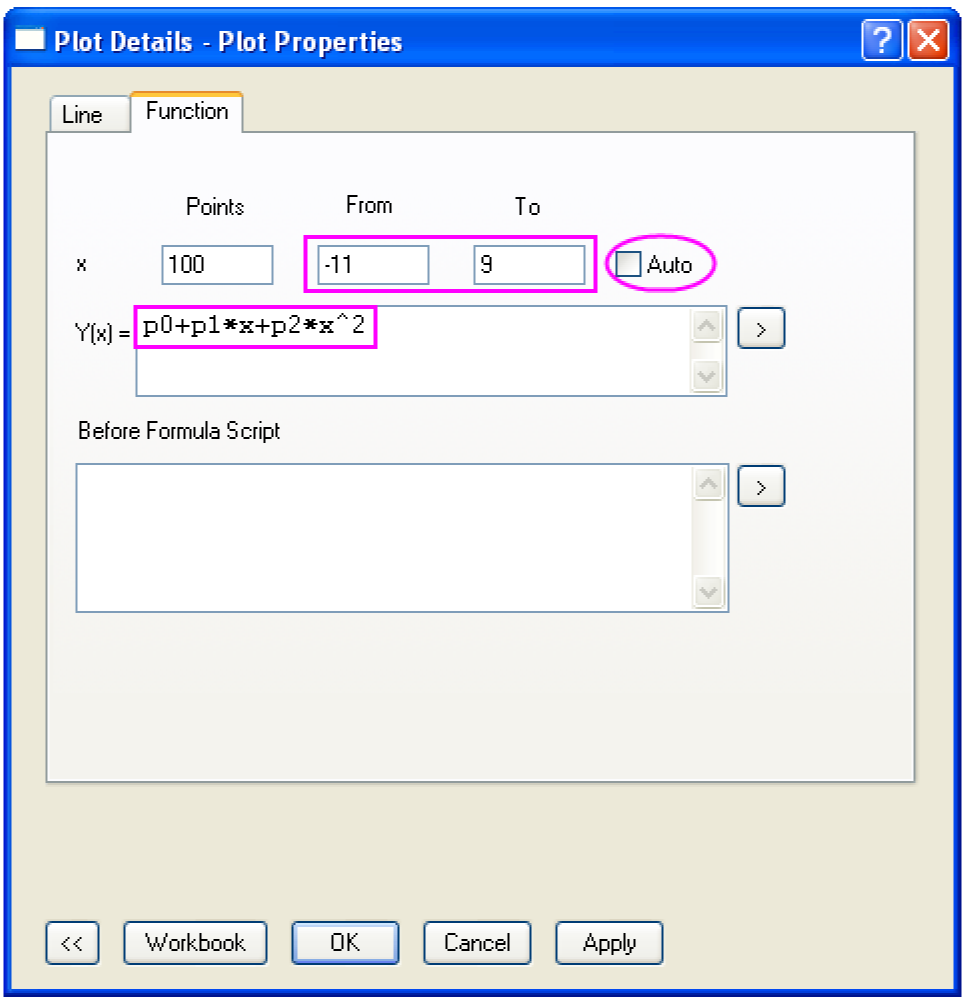


Note that there is a green lock icon on the top right corner of Column D which indicates that the Recalculate Mode is Auto.

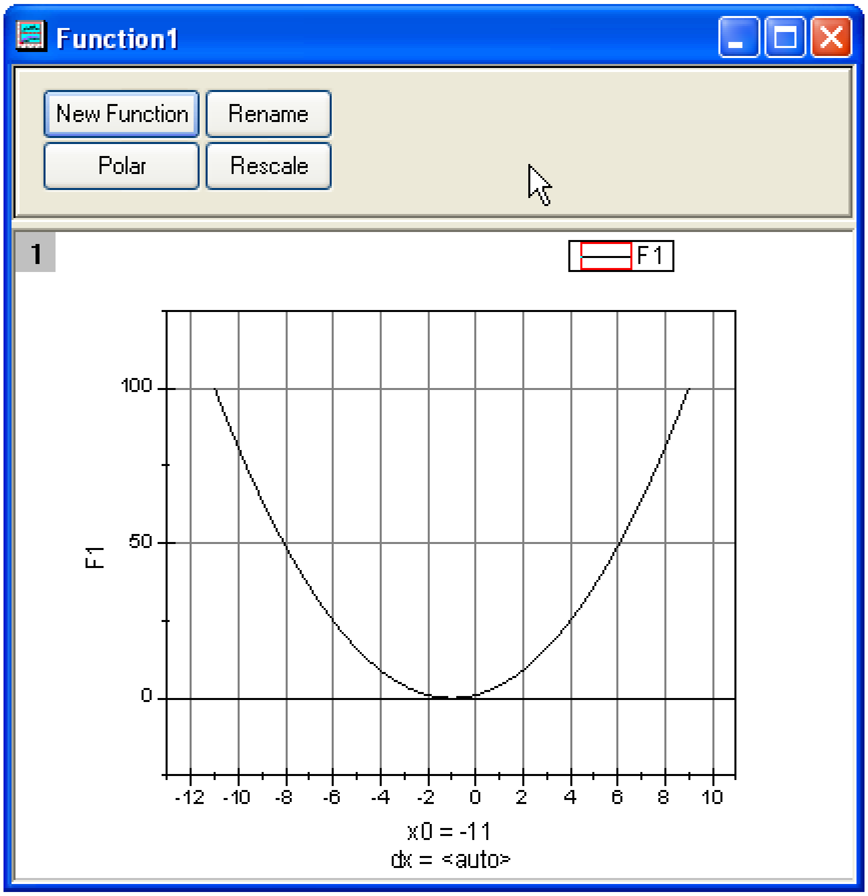


3. Click on the **New Function** button on the **Standard** toolbar. The **Plot Details** dialog opens.

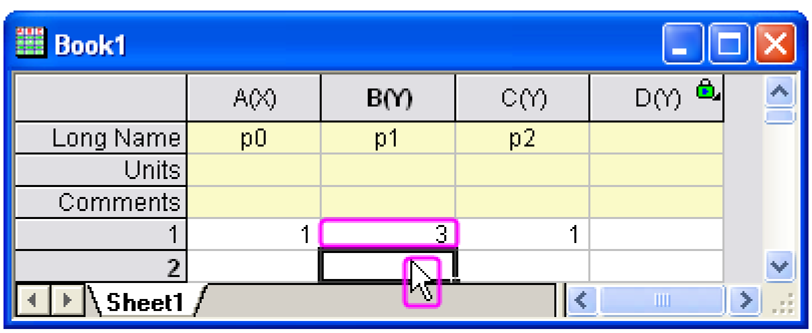
4. In the **Plot Details** dialog, set the options as follows and click the **OK** button to close the dialog box.



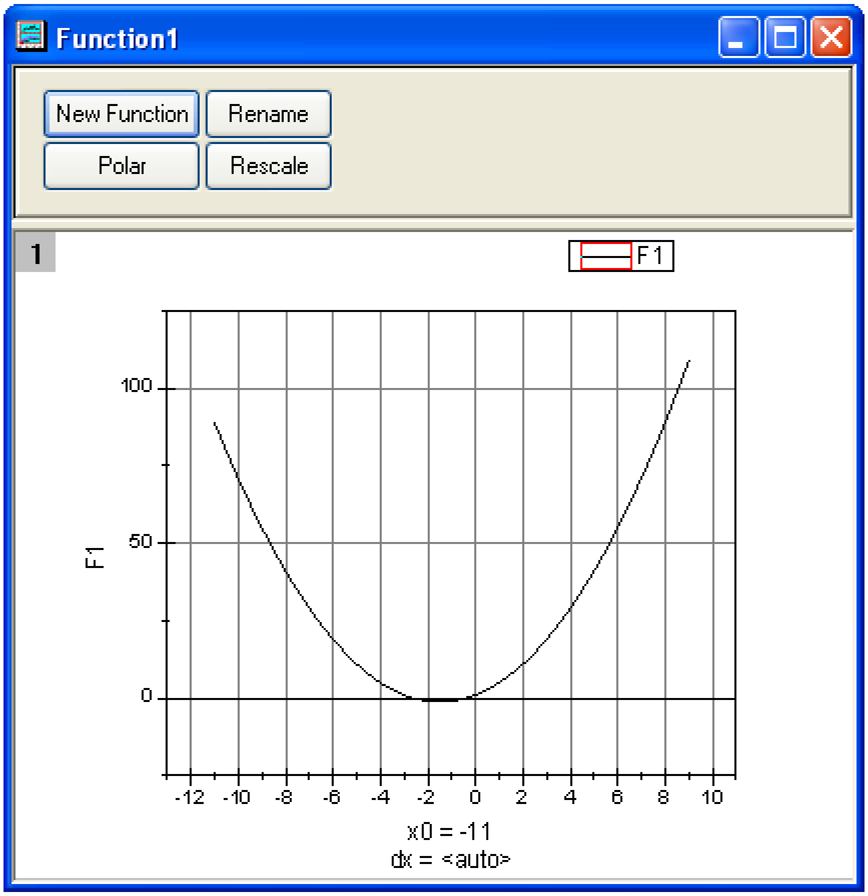
Click on the **Rescale** button on the **Graph** toolbar to adjust the graph's scales.



5. The function graph can be updated automatically when the parameters in the first row of the worksheet change. Change the value in the first row of Column 2 from **2** to **3**. Click outside this cell to finish editing.



The function graph updates to reflect this change.



Scripts

The script used in the **Before Formula Scripts** edit box of the **Set Values** dialog box is:

```
p0=col(1)[1];  //Specify Column A for p0.
p1=col(2)[1];  //Specify Column B for p1.
p2=col(3)[1];  //Specify Column C for p2.
```

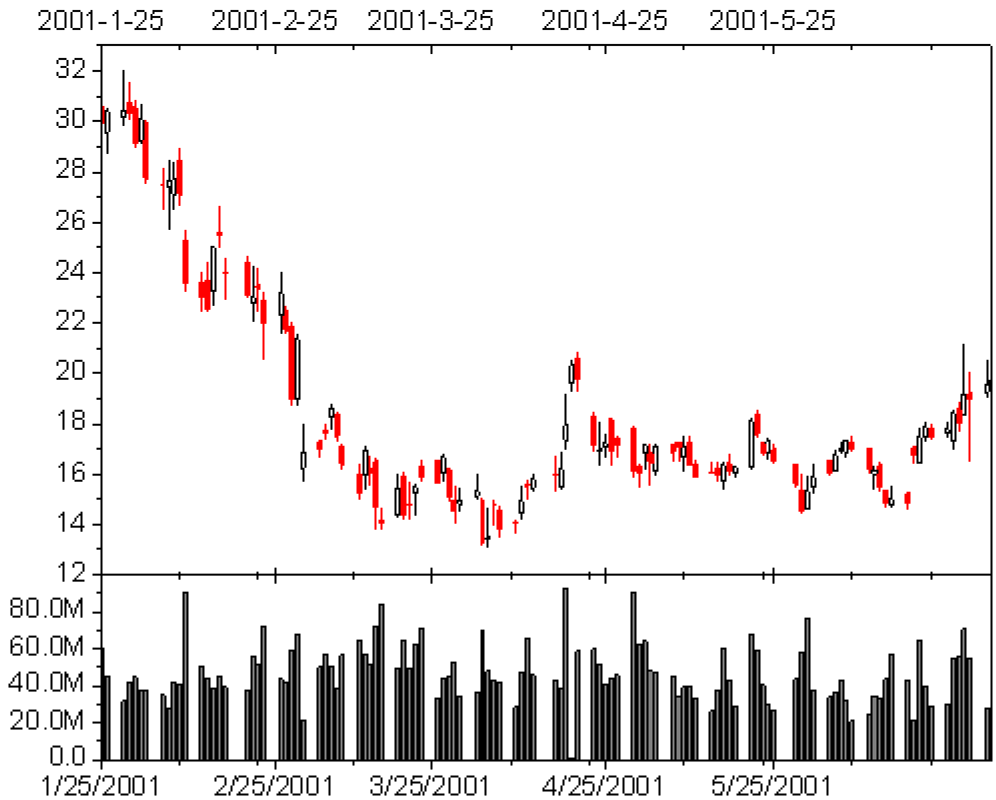
The function in the **F1(x)** edit box of the **Plot Details** dialog box is as follows:

```
p0+p1*x+p2*x^2
```

5.12.3 Open-High-Low-Close-Volume Stock Chart

Summary

This tutorial will show you how to create a stock chart to display open, high, low, and close prices, together with trading volume.



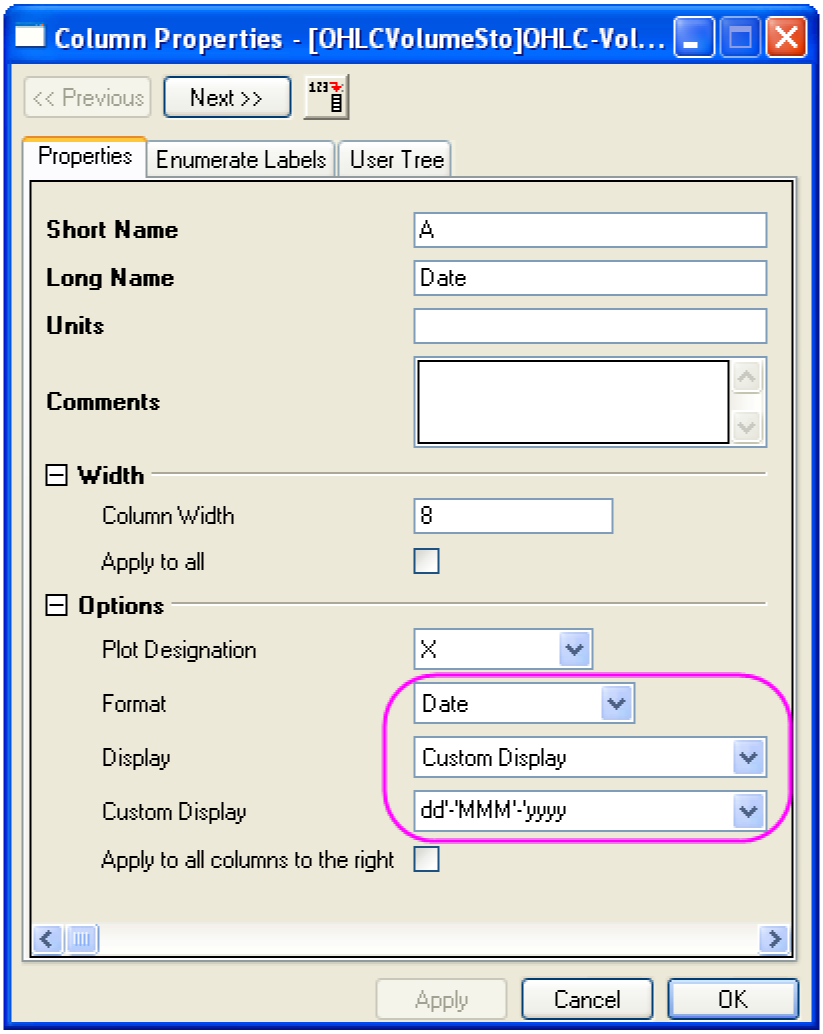
Minimum Origin Version Required: Origin 8.5 SR0

What you will learn

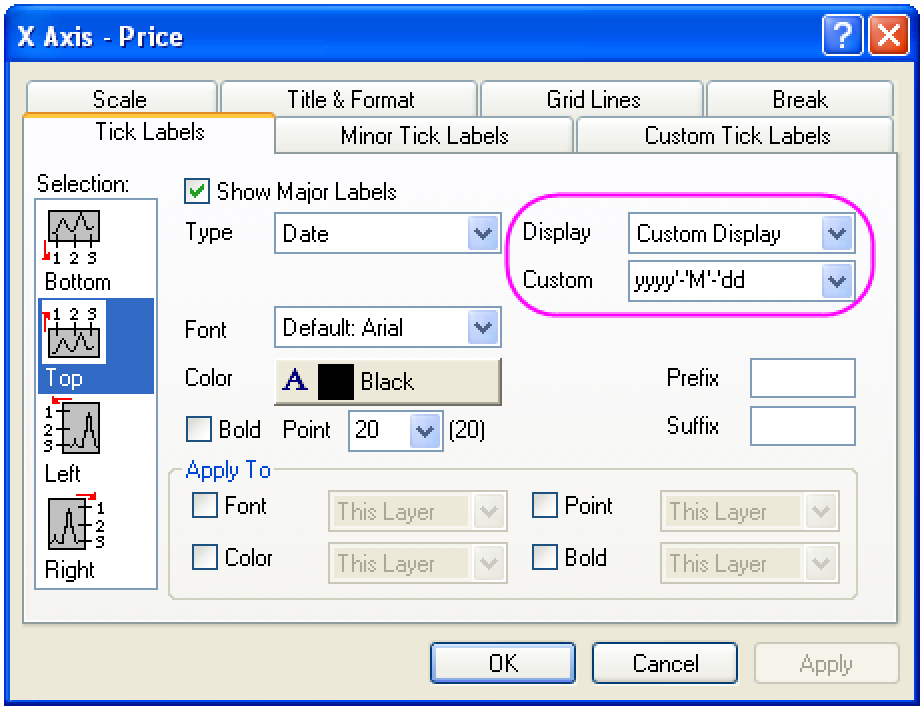
- Set a column format
- Create a stock chart with open, high, low, and close prices, and trading volume
- Set date format for display label of axis

Steps

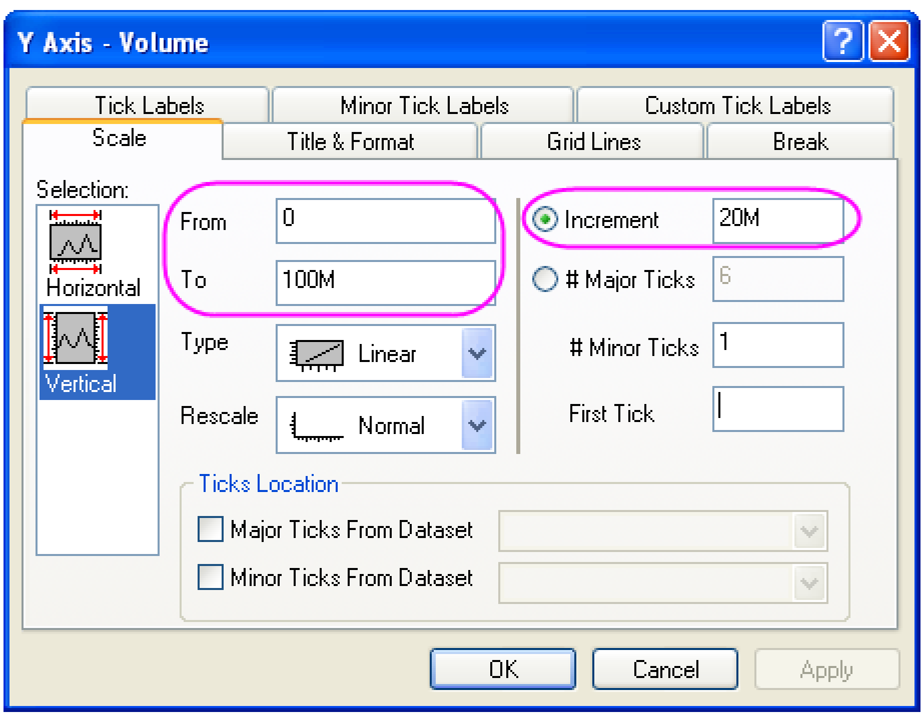
1. Download the source data file from our ftp and import it into Origin. When import this file, keep the system default settings.
2. Double click on the header of column A to bring up the **Column Properties** dialog. Activate the **Properties** tab, under the **Options** branch, Select *Date* from the **Format** drop-down list, and then select *Custom Display* from the **Display** drop-down list. Type *dd'-'MMM'-'yyyy* in the **Custom Display** drop-down list. Click the **OK** button to finish setting column A as Date column.



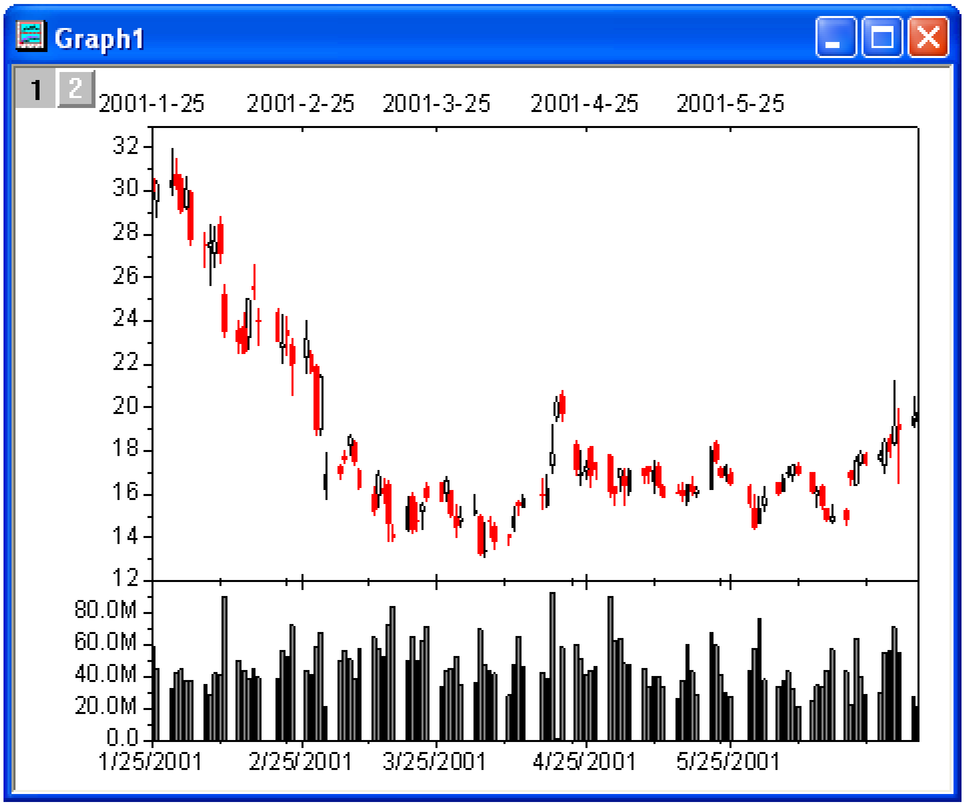
3. Highlight all columns in the worksheet, and then choose menu item **Plot: Stock: OHLC-Volume** to create a graph.
4. Double click on the top axis of layer 1 in the graph to open the **X Axis** setting dialog. Switch to the **Tick Labels** tab, and then choose **Custom Display** from the **Display** drop-down list. Type *yyyy'-'M'-'dd* in the **Custom Display** drop-down list. Click the **OK** button



5. Double click on the Y axis of layer 2 to open the **Y Axis** setting dialog. Go to the **Scale** tab, and set **From**, **To**, **Increment** as **0**, **100M**, **20M** respectively. .



6. Click the **OK** button to finish the axis setting and get the stock chart of open, high, low, and close prices, and trading volume.



6 Customizing Graphs

Topics covered in this section:
1. Themes (Tutorials) 2. Color (Tutorials) 3. Layers (Tutorials) 4. Tick Labels (Tutorials) 5. Customizing a Graph 6. Add remove or reorder data plots

6.1 Customizing a Graph

6.1.1 Summary

Editing your Origin graph is very easy. Any graph element can be selected and with its associated dialog, the properties can be modified. In fact, you can customize your graph right down to a single data point.

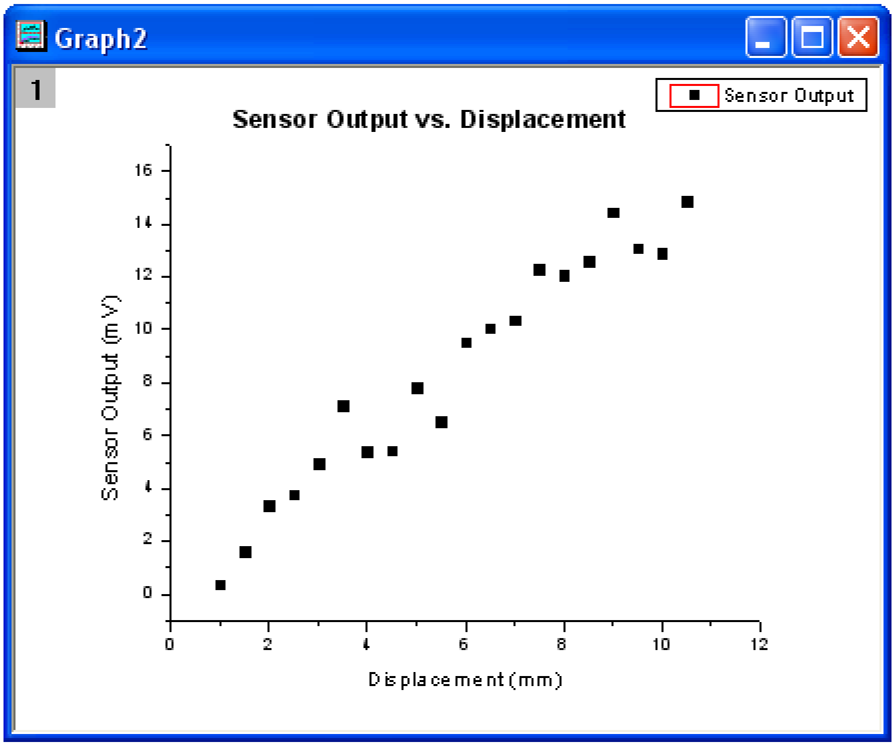
This tutorial will show you how to:

- Resize a Layer
- Add a Layer Title
- Customize and Save a Template
- Customize Axes
- Apply a Graph Theme
- Change Plotting Order in a Layer
- Customize a Point
- Customize a Grouped Plot
- Create a Color-mapped Waterfall Plot

6.1.2 Resize Layer

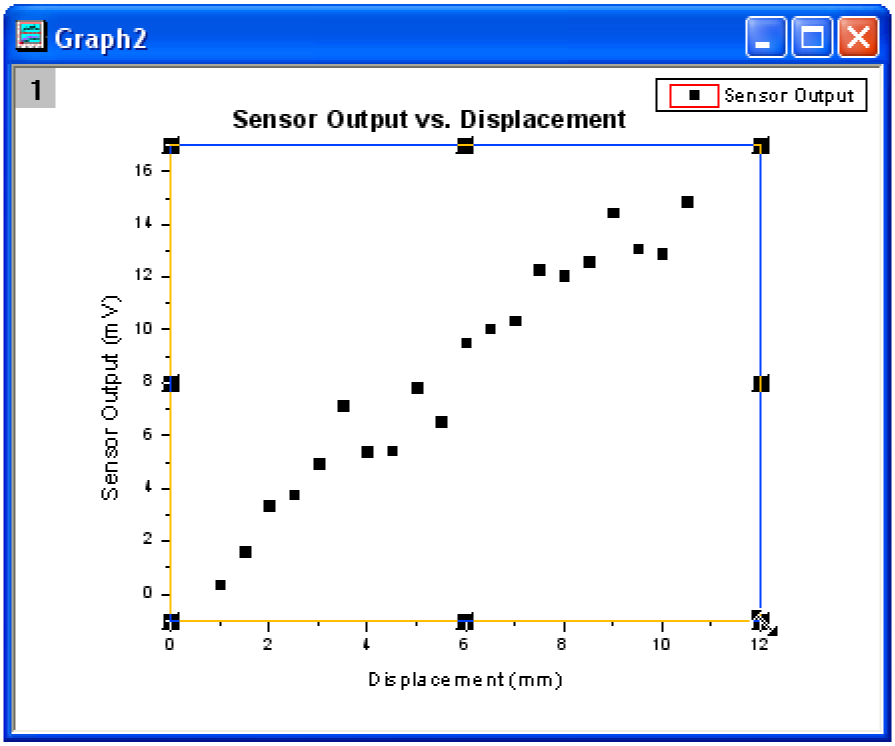
1. Open **Customizing Graphs.OPJ** from the **\Samples\Graphing** folder and select the **Resize Graph and Customize Symbol** folder from the **Project Explorer** window.

2. Make **Graph2** active and right-click inside the layer, above the data points to select **Add / Modify Layer Title** from the context menu that opens. Add a title, as the following graph shows.

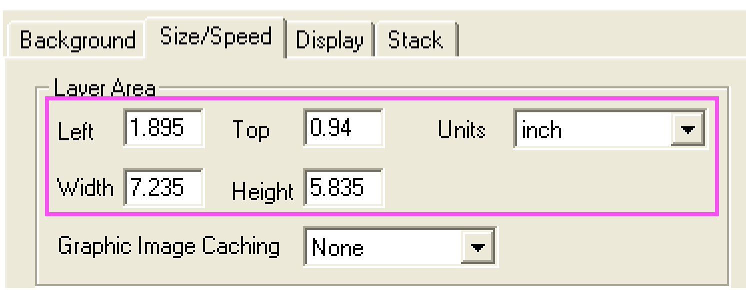


3. The layer can easily be resized graphically by dragging the sizing handles. To resize it, single click inside the layer, but not on a data point. You can left-click in the same space you right-clicked when adding the Layer Title. The layer will become selected as seen below, and then you can drag one of the 8 anchor points to resize the layer. Note: If you hold the Ctrl key down while

dragging, the aspect ratio will be maintained.

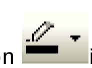


4. You can also use the **Plot Details** dialog to input the size of the layer, in order to resize it precisely. Double-click inside the layer (in the same spot you left-clicked above) to open the **Plot Details** dialog. Go to the **Size/Speed** tab, and set the layer area to the values shown in the image below:



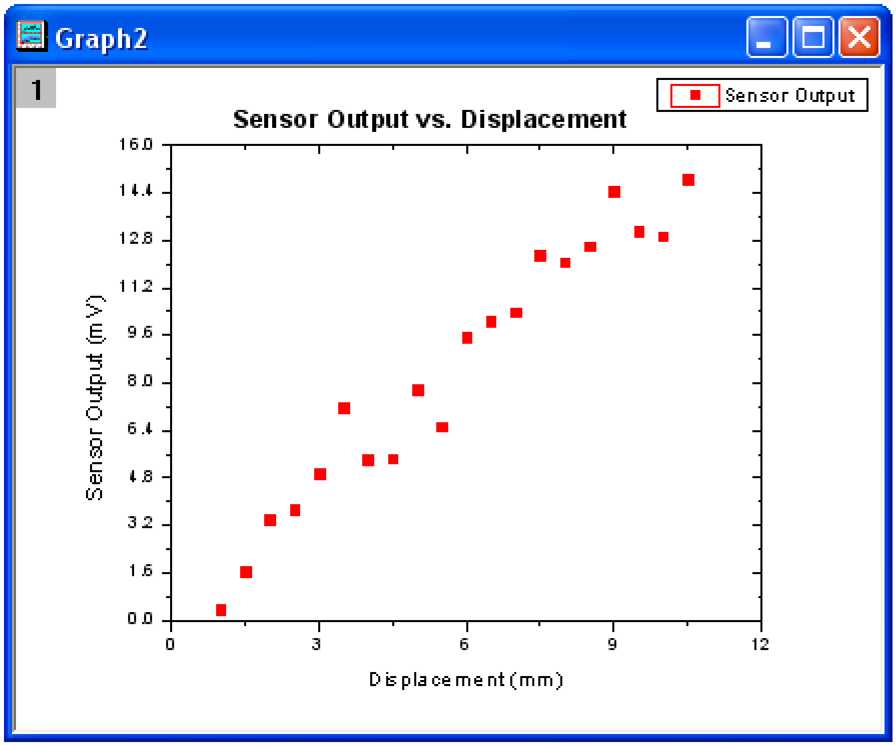
6.1.3 Customize Data Plot and Axes

In this section, we will show you how to change the color of a data plot, and how to change the properties of the axes.

1. Click on one of the data points of **Graph2** to select the entire data plot, and then change the color of the data points to **red** by using the **Line/Border Color** button  in the **Style** toolbar.
2. Next we will use the **Axis** dialog to customize the axes. Double-click on the X-axis to open the dialog and set the controls as follows:
   - On the **Scale** tab, choose the **# Major Ticks** radio button and input **5** into it. Then select **Vertical** from the **Selection** list box. The name of the dialog changes to **Y Axis**

when you do this. Input **0**, **16** and **11** into the **From**,**To** and **# Major Ticks** edit boxes, respectively.

- On the **Title&Format** tab, select **Top** from the **Selection** list and check the **Show Axis&Ticks** check box to add the top X-axis. Then select **Right** from the **Selection** list and again check the **Show Axis&Ticks** check box to add the right Y-axis. Now the graph should look like:

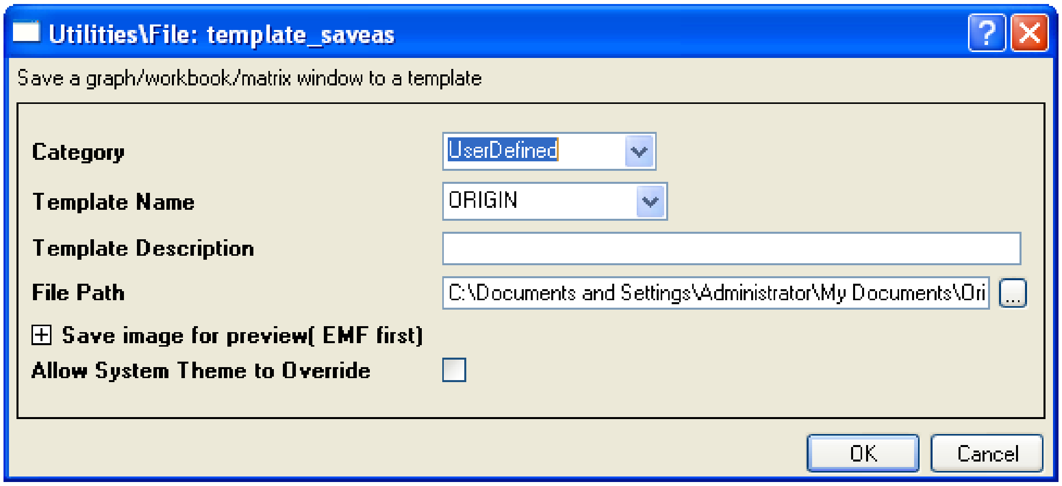


6.1.4 Save and Reuse a Template

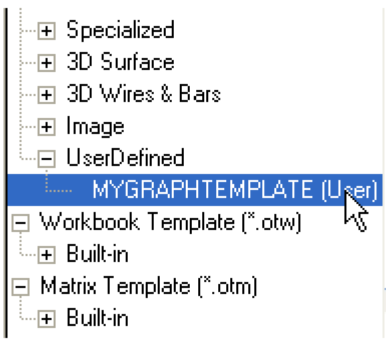
In this section, we will show you how to save the graph above as a template and reuse it.

1. In the menu, select **File: Save Template As** (alternatively, right-click on the graph window title and choose **Save Template As** from the context menu) and rename the template as

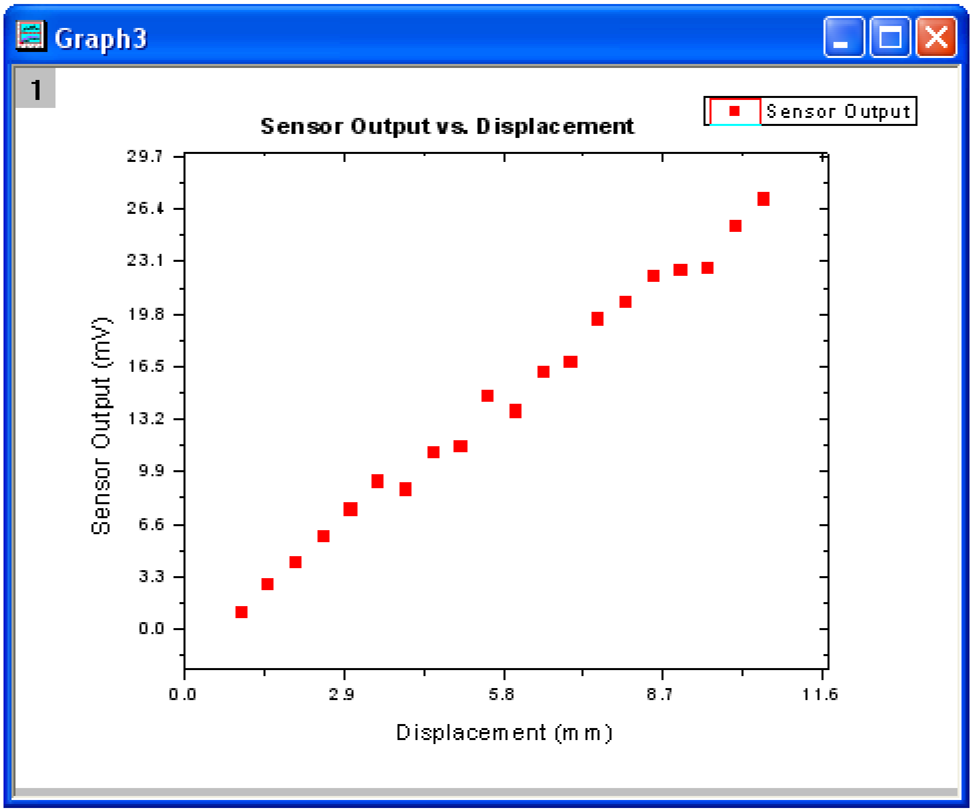
MyGraphTemplate. Then click the **OK** button to save it.



2. Create a new workbook and import the data file **\Samples\Curve Fitting\Sensor2.dat**, by using **File: Import: Single ASCII**. Highlight column B and select **Plot: Template Library**. Then select **MyGraphTemplate** and click the **Plot** button to create a graph.



The graph should look like:

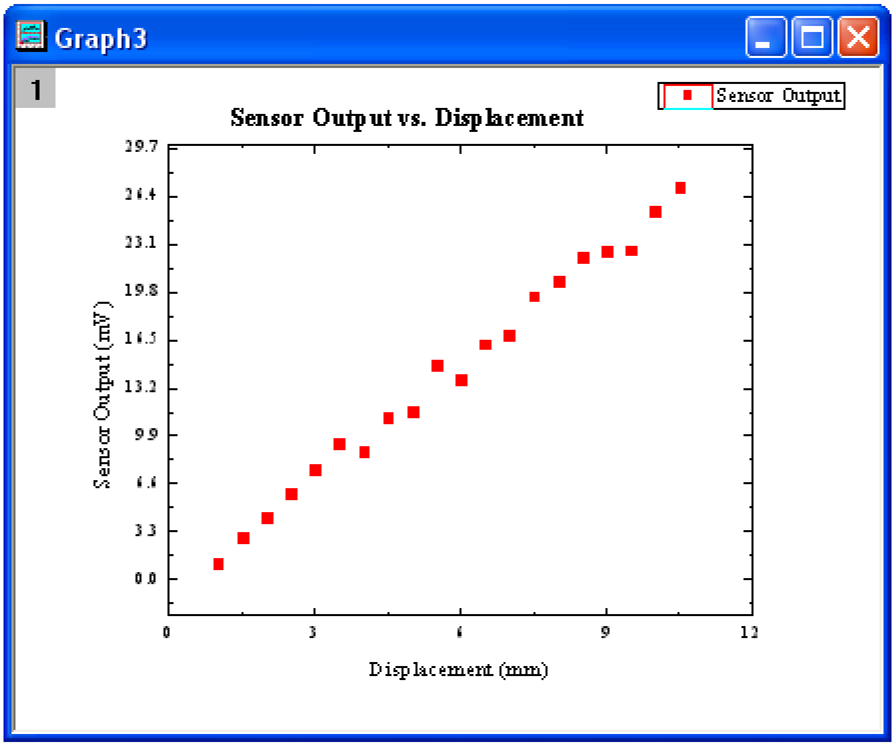


6.1.5 Changing graph using theme

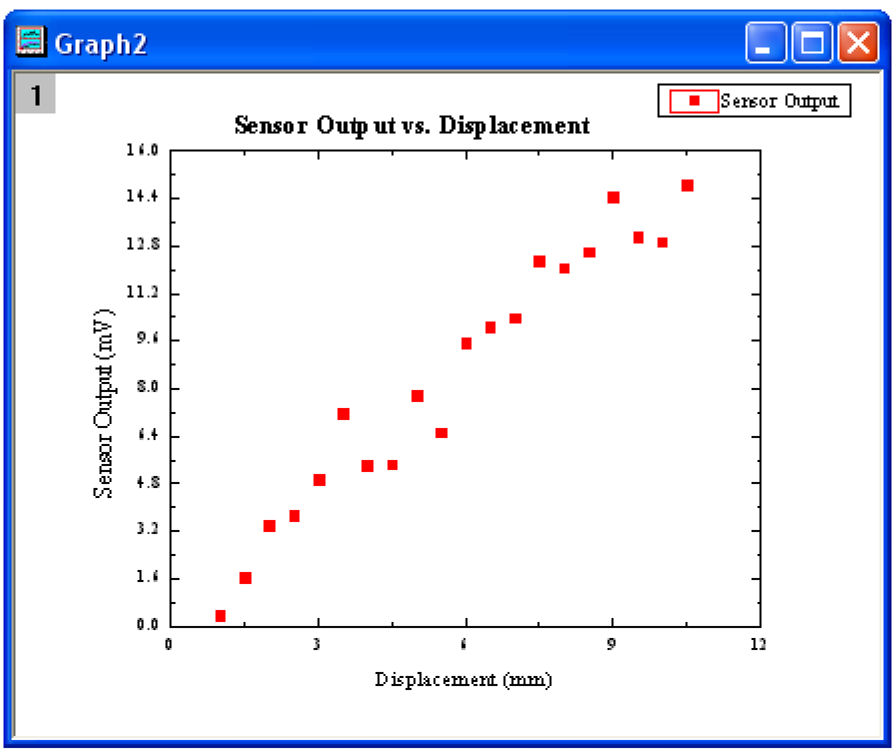
Origin stores the properties of a graph in a theme file. In this section we will show you how to customize a graph by using a theme.

1. With **Graph3** active, select **Tools: Theme Organizer** to open a dialog. Apply the themes **Ticks All In** and **Times New Roman Font**. Then click the **Close** button to close the dialog. The graph

should look like:



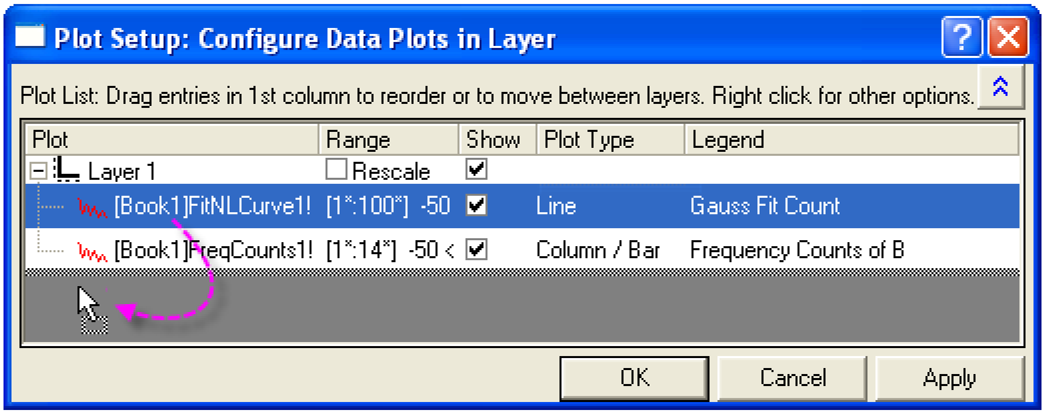
2. Then we will copy the formatting of the current graph and paste it to **Graph2**. Right-click to the right of the layer, anywhere on an empty white space, or even on the gray area, and select **Copy Format: All Style Formats**. Then activate **Graph2**, right-click in a similar spot and select **Paste Format**. Then **Graph2** should look like the image below:



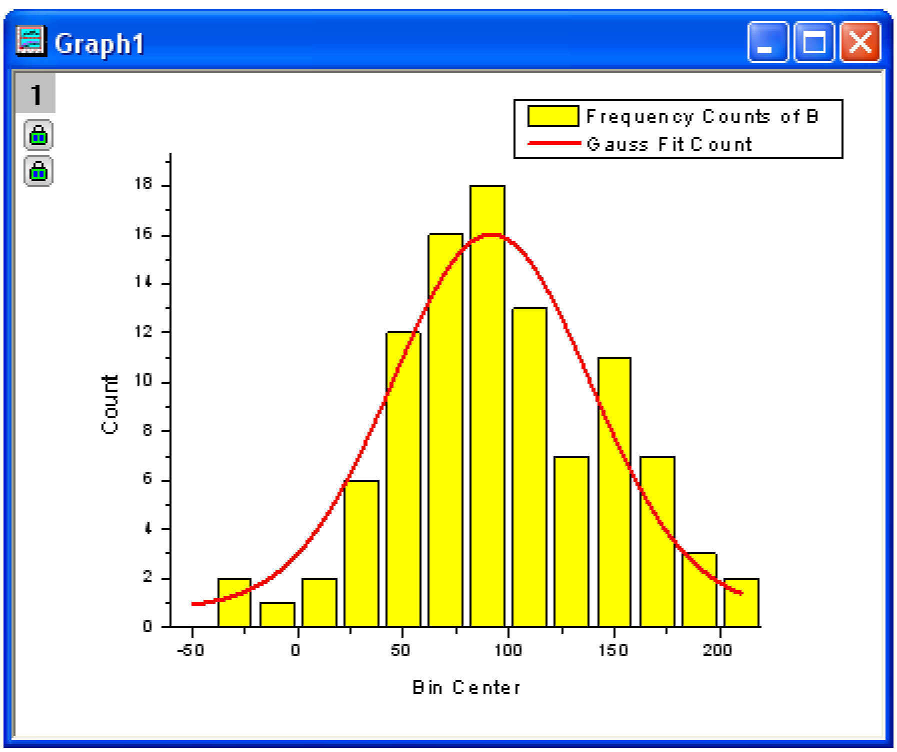
6.1.6 Plotting Order

In this section, we will show you how to change the plotting order by using the **Plot Setup** dialog.

1. Go to the **Plotting Order** folder and activate **Graph 1**. Select **Graph: Plot Setup** from the main menu to open the **Plot Setup** dialog. (Alternatively, Right-click on the layer 1 and select Plot Setup, which also will open the **Plot Setup** dialog.)
2. In the **Plot List** panel, drag the line plot and drop it beneath Column/Bar Plot Type.



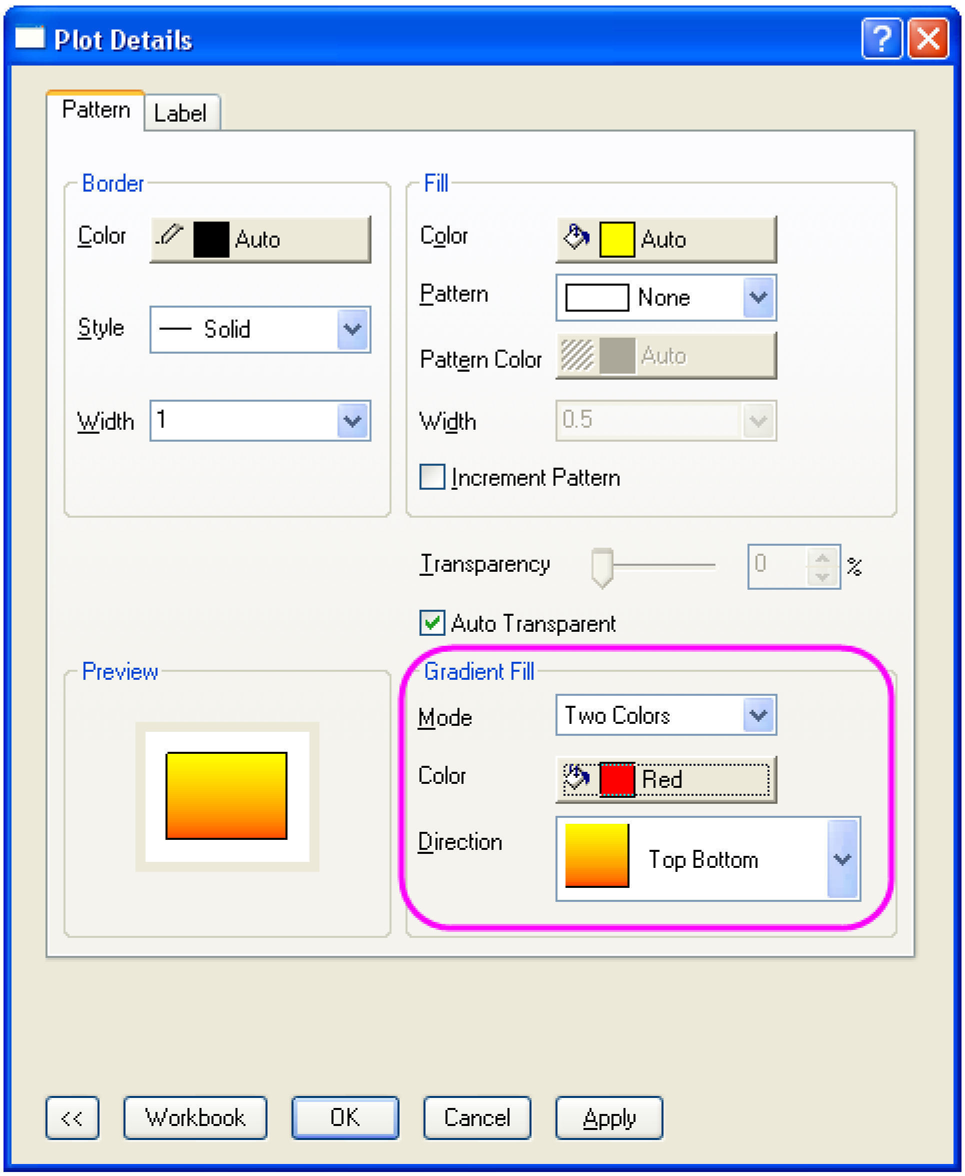
3. Click the **OK** button and you will see that the red curve is now drawn on top. Notice the legend reflects the new plotting order as well.



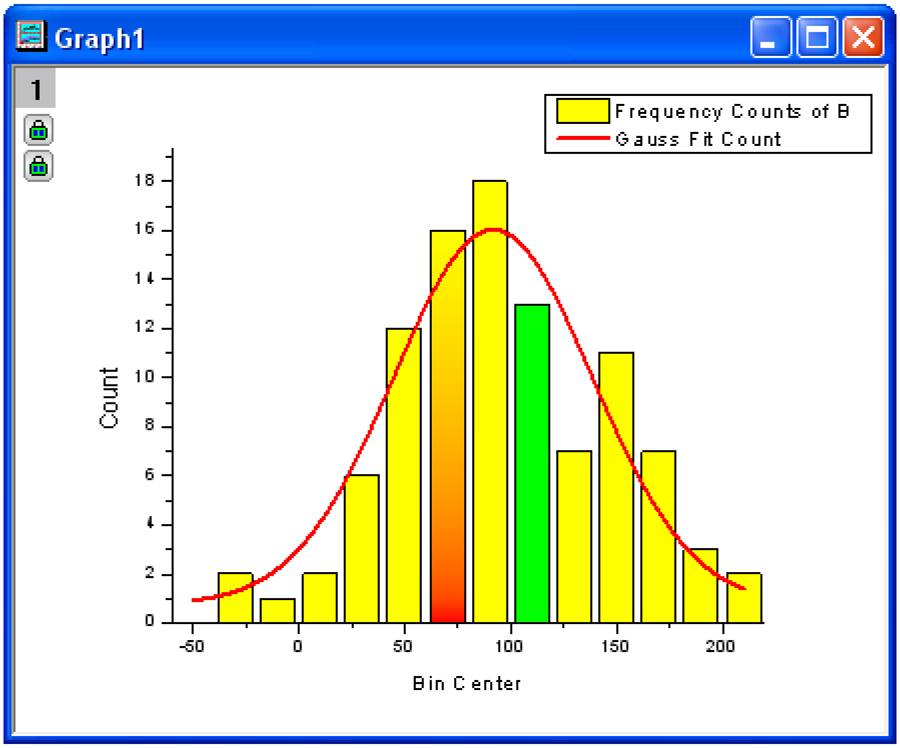
6.1.7 Customize points

In this section, we will show you how to customize a single data point.

1. Continue viewing the contents of the **Plotting Order** folder and make sure **Graph 1** is active. Click to select one of the columns and all columns become selected. Click again on one of the columns to select just that one column. Then change the color to **green** by using the **Fill Color** button  on the **Style** toolbar.
2. You can also use the **Plot Details** dialog to customize a single column. Hold down the Ctrl key while double-clicking on one of the columns to open the **Plot Details** dialog. You'll be editing the properties of just that single data point. Notice the index number for that point in the Plot Details dialog. Set the **Gradient Fill** group as follows to change the column color from yellow to red, gradually.

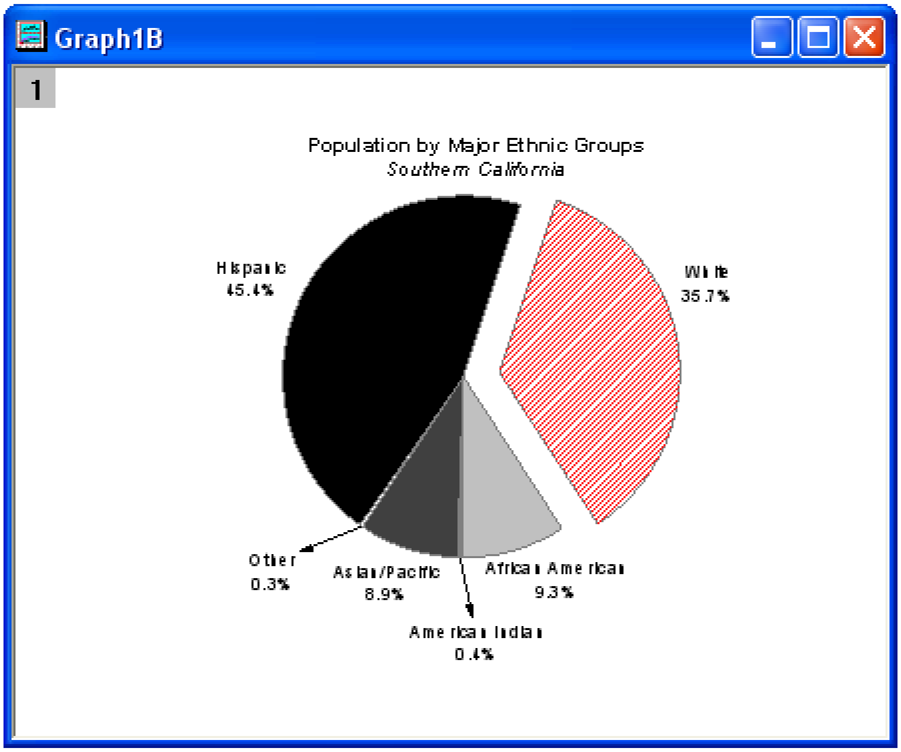


Then the graph should look like:

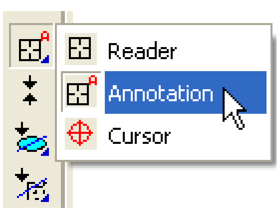


3. You can customize a slice of a pie graph in the same way. Go to the **Edit Single Data Point** folder and activate the pie graph. Hold down the Ctrl key and double-click on one of the slices to open the **Plot Details** dialog. Under the **Pattern** tab, set the **Fill Pattern** to **Dense**, from the drop-down list, and set the **Pattern Color** to **Red**. (Alternatively, you could have clicked once and then clicked again on a slice to select just that one point, and then used the Style toolbar to

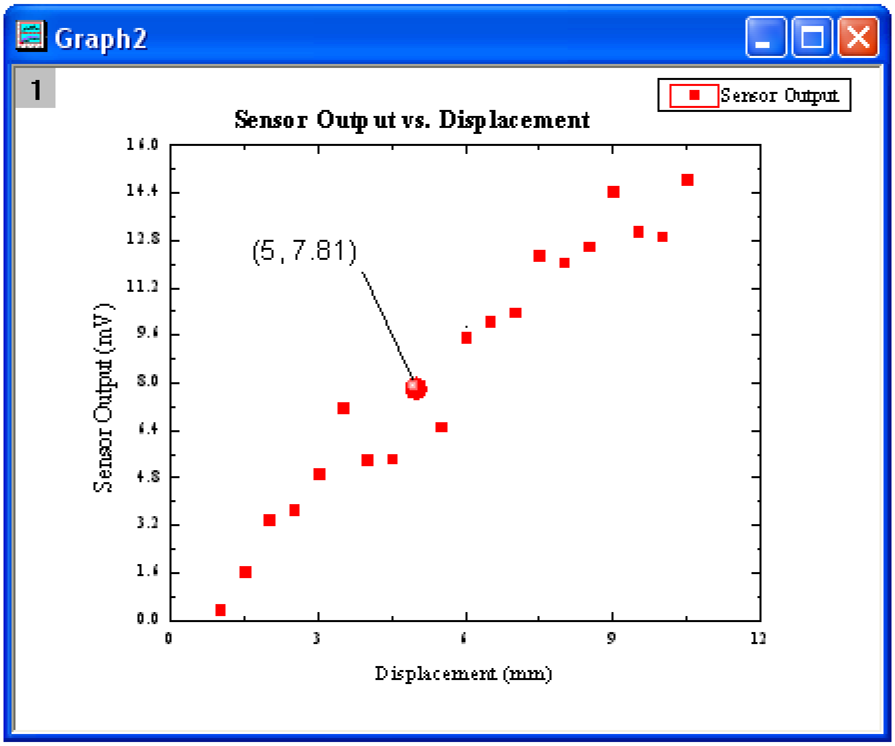
customize that point.) The graph should look like:



4. You can customize a single data point and add an annotation to it. Go back to the **Resize Graph and Customize Symbol** folder. Hold down the Ctrl key and double-click on one of the scatter points to open the **Plot Details** dialog. Under the **Symbol** tab, click the triangular **Preview** button to open the symbol gallery, and then select **Sphere** for the symbol. Increase the **Size** to **18**, and click the **Ok** button to close the dialog.
5. Now you can use the annotation tool to add the X and Y values that correspond to your customized data point. Select the **Annotation** button from the **Tools** toolbar.



Then move the annotation cursor to your customized data point and double-click on it. Origin will automatically add a text object. Hit the ESC key or click on the Pointer button to stop annotating. You can customize the text label by double-clicking on it to enter Edit mode. You can also change its position by clicking and dragging - a line will automatically be drawn connecting the point and the label.

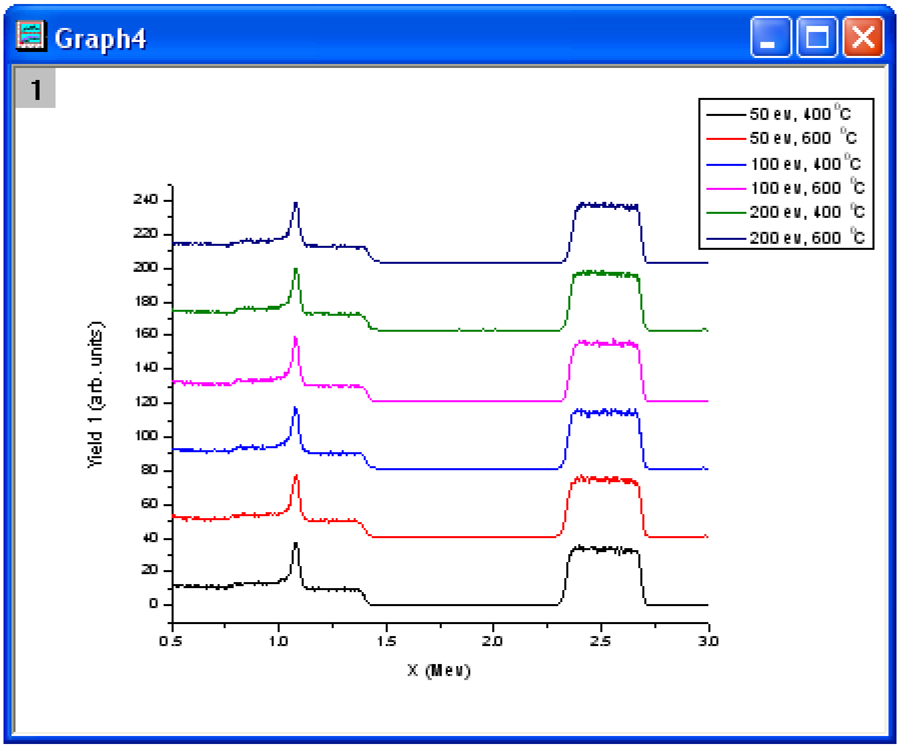


6.1.8 Group plots

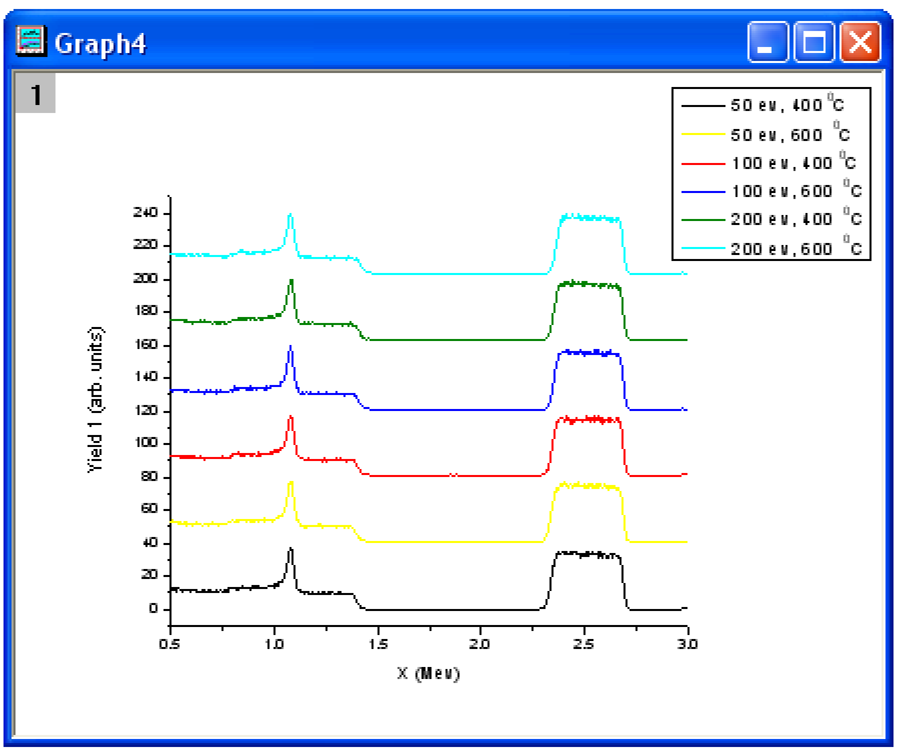
In this section, we will show you how to customize a grouped data plot.

1. Select the **Grouped Data** folder. **Book3** will be active. Highlight the entire worksheet and click the line button  on the **2D Graphs** toolbar to create a line graph.
2. Double-click on the X-axis to open the **Axis** dialog. Input **0.5** into the **From** edit box and **3.0** into the **To** edit box. Select **Manual** from the **Rescale** drop-down list (you will need to scroll up in the list). This prevents the **From** and **To** values from changing while rescaling. Click **OK** to apply these settings.
3. Double-click somewhere inside the layer, on a blank place above the line plots, to open the **Plot Details** dialog. On the **Stack** tab, select **Auto** in the **Offset** group. Click **OK** to close the dialog.
4. Select **Graph: Rescale to Show All**. The Y scale of the graph automatically rescales, while the X scale doesn't change because the option was set to manual. Resize the layer and move the

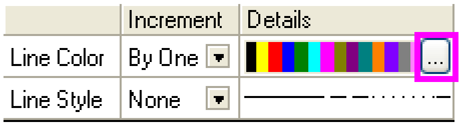
legend as you see fit.



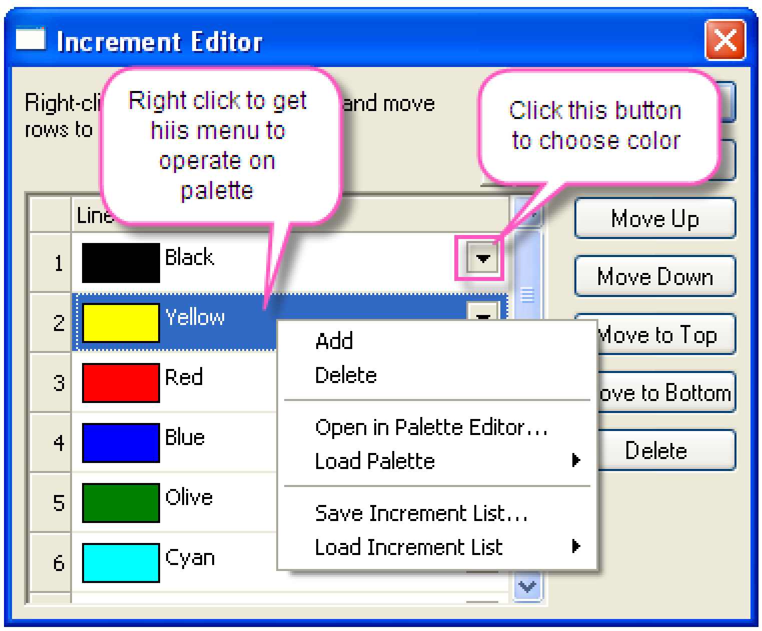
5. Click on one of the data plots to select the entire group, and then use the **Line/Border Color** button  on the **Style** toolbar to change the colors. In **Incr. List**, you can select the second-to-last increment list . The graph should look like:



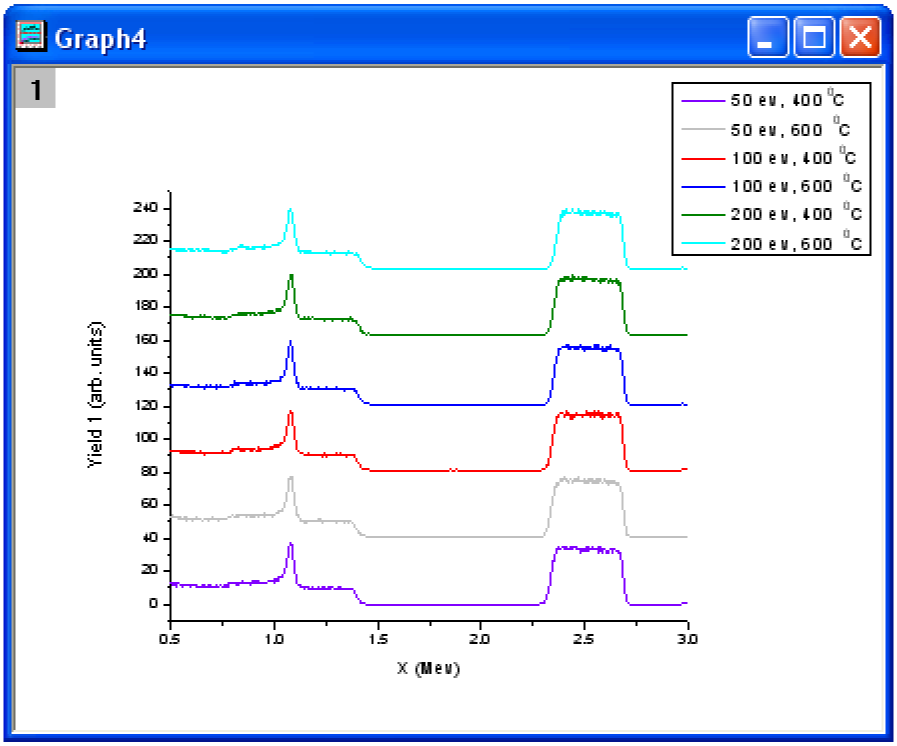
6. Though these data plots have been grouped, you can also customize each of them individually by clicking on the data plot twice. For example, you can click the yellow data plot twice (click once, pause and click again) and then click the **Line/Border Color** button  on the **Style** toolbar to change the color to another color, such as **Olive**.
7. You can also customize the graph by double-clicking on one of the data plots to open the **Plot Details** dialog. Under the **Group** tab, you can click the browser button



to open the **Increment Editor** dialog. In this dialog, customize the graph as shown in the following image. Please note that you can also drag the index of a row and move it to change the order of the color list.



The graph should look like:

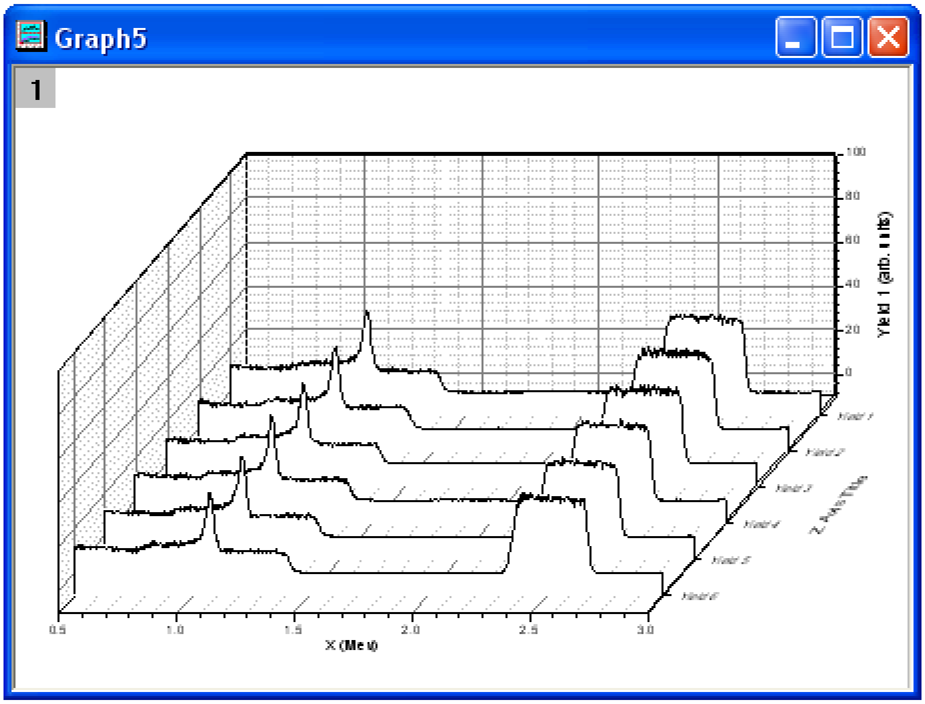


8. Right-click on the inner position of this dialog and select **Save Increment List** to save it as **MyPalette** for future use. Then you can right-click to get the context menu and select the **Load Increment List** option. You will find that **MyPalette** appears as the first entry in the fly-out menu, as the following image shows.

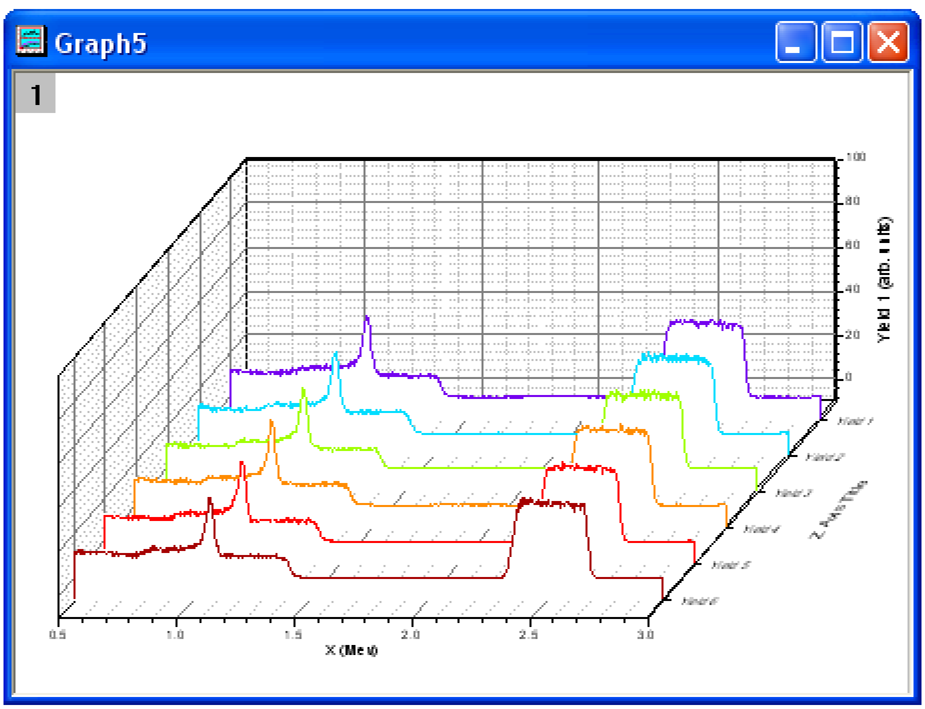
In the following section, we will show you how to use **Palettes** to set colors for a group of data plots.

1. Activate **Book3** and highlight all the columns. Select **Plot: Multi-Curve: Waterfall** to create a graph. Double-click on the X-axis to open the **Axis** dialog, and set **From** as **0.5** and **To** as **3**. Select **Vertical** from the **Selection** list, and set the **From**, **To**, and **Increment** edit boxes to **-**

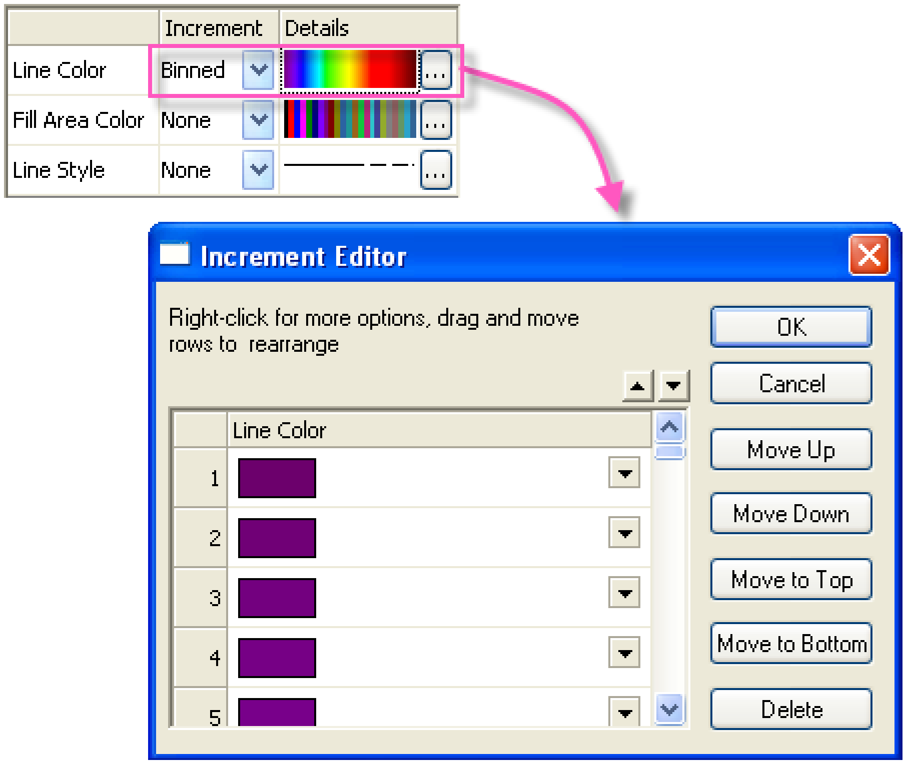
10, **100** and **20**, respectively.



2. Click on one of the data plots and then use the **Line / Border Color** button  on the **Style** toolbar to change the colors. You can select the **Rainbow** palette  under **Palettes**. The graph should look like:

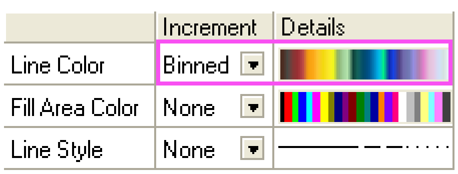


3. You can double-click on one of the plots to get the color list, as the following image shows:

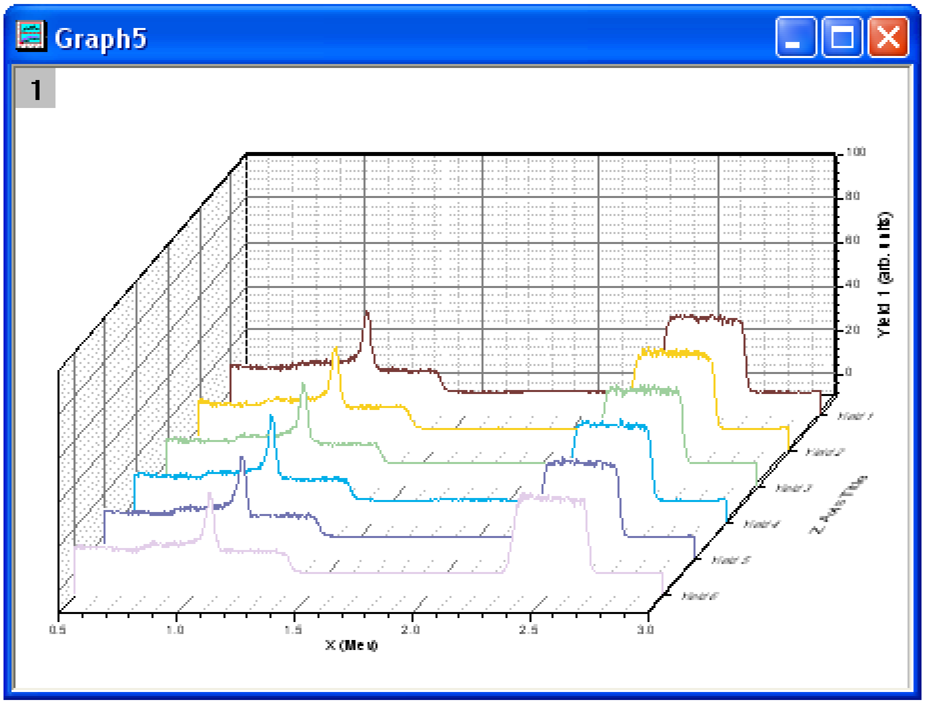


You can find that the **Increment** has been set to **Binned**, so colors are picked from the 256 available in the **Rainbow** palette.

4. We will change the palette to **Reef** by clicking on the **Browser** button to open the **Increment Editor** dialog. Then right-click on the inner position of this dialog and select **Load: Reef.PAL**. The color list in the **Plot Details** dialog should be:



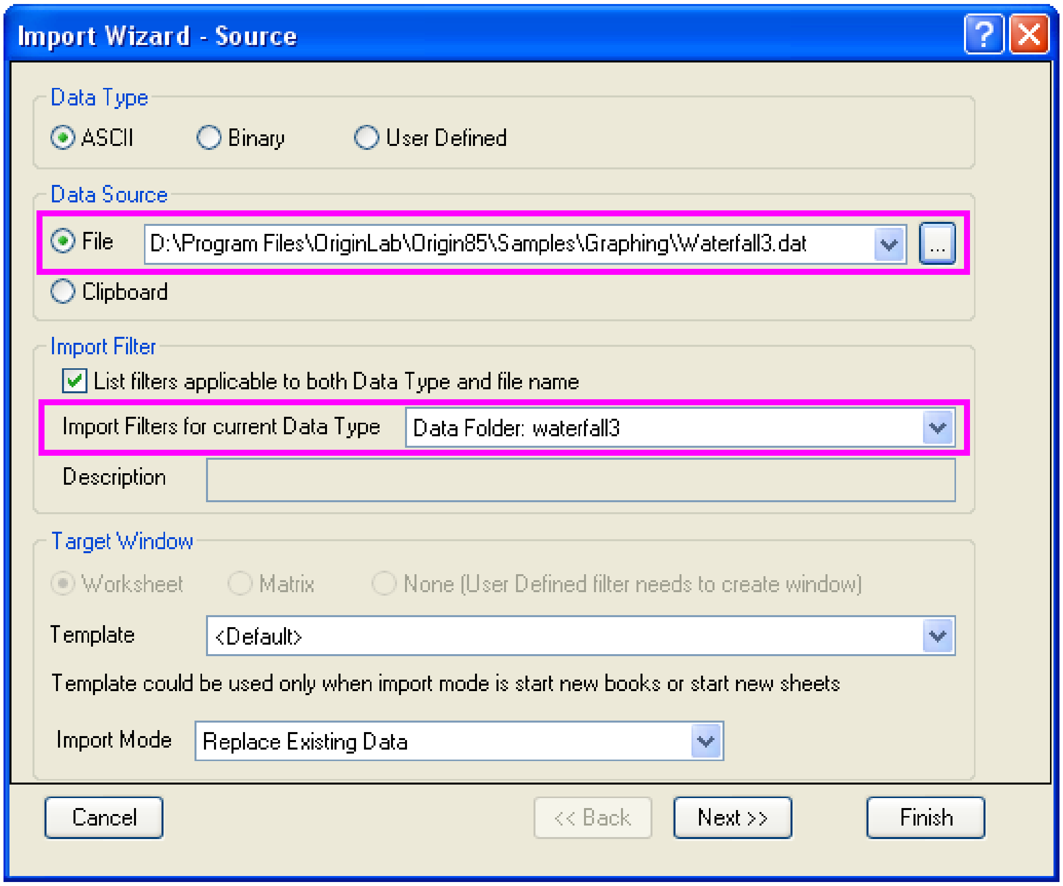
And the graph should look like:



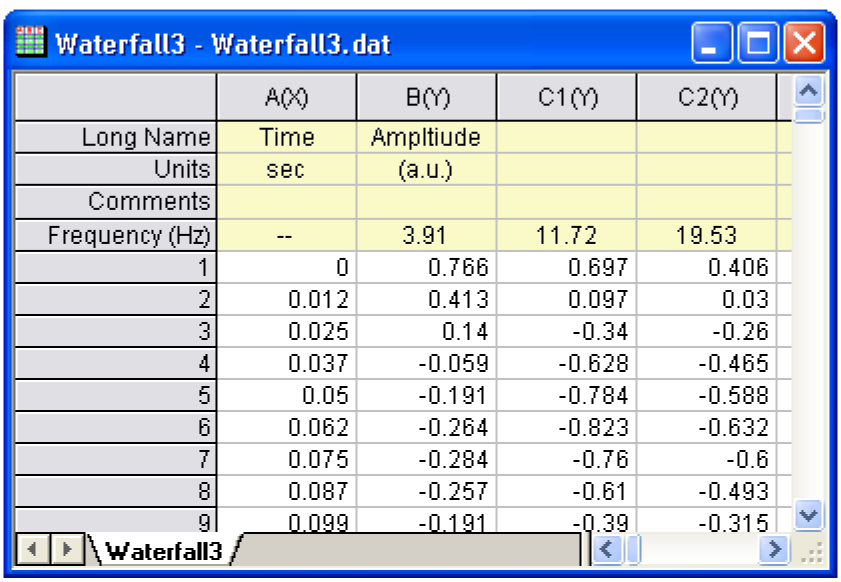
6.1.9 Plot a Color-mapped Waterfall

In this section, we will show you how to create a color-mapped waterfall and use the desired column label row as the Z-axis.

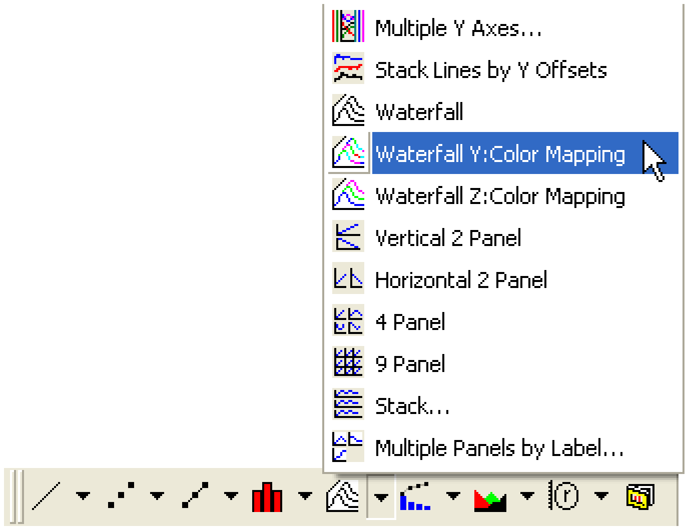
1. Click the **New Workbook** button on the **Standard** toolbar to create a new workbook.
2. Click the **Import Wizard** button on the **Standard** toolbar. This opens the **Import Wizard** dialog. Click the browser button to the right of **File** and select *Waterfall3.dat* from the */Samples/Graphing* folder. Make sure *Waterfall3.oif* is picked up automatically for **Import Filters for current Data Type**.



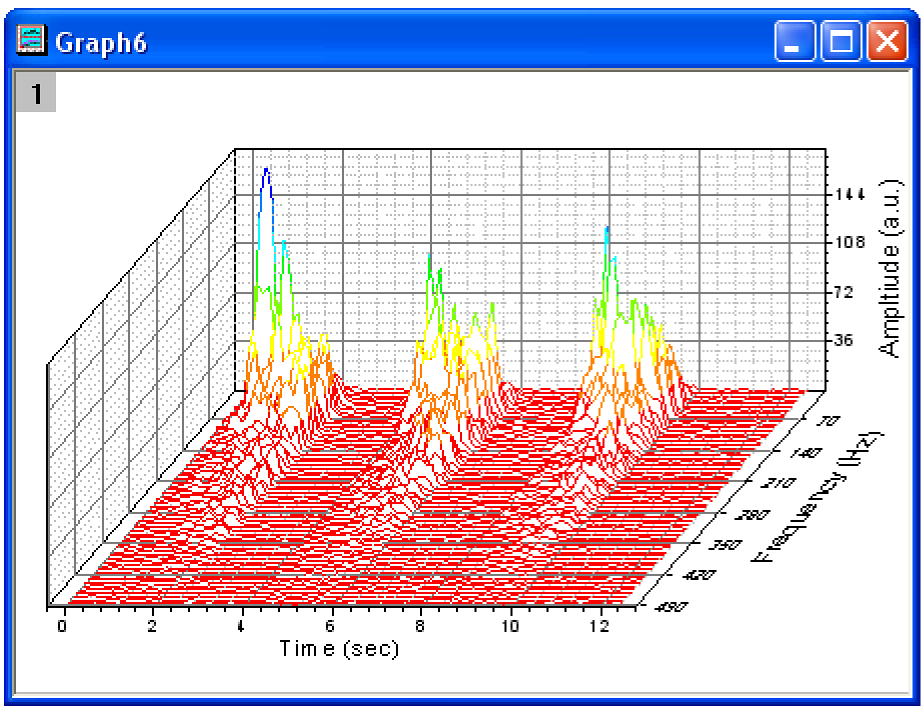
Click **Finish** to import the *Waterfall3.dat* file.



3. To plot a waterfall color-mapped using each line's Y value, highlight the whole worksheet and select **Waterfall Y: Color Mapping** from the **2D Graphs** toolbar (Alternatively, Select **Plot: Multi-Curve: Waterfall Y:Color Mapping** from the main menu).

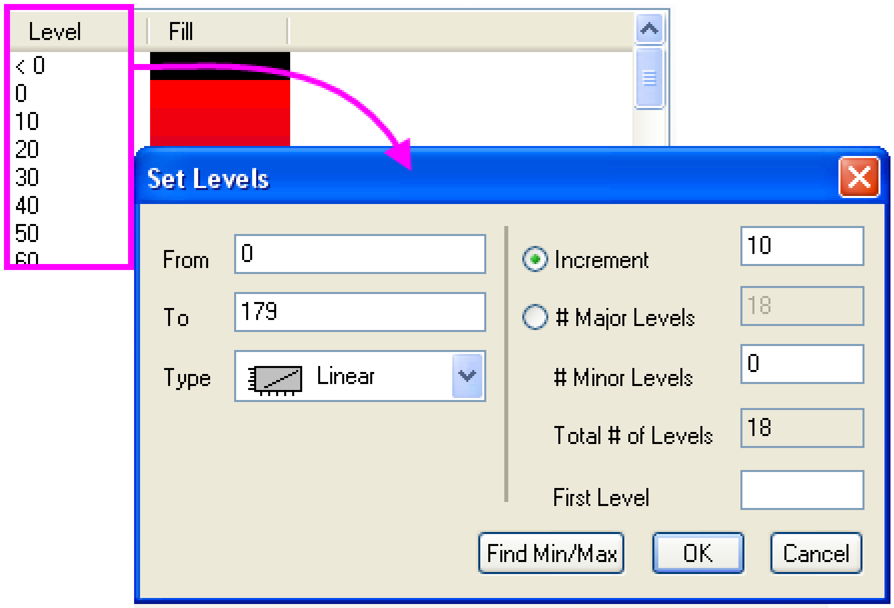


The graph should look like the picture below:

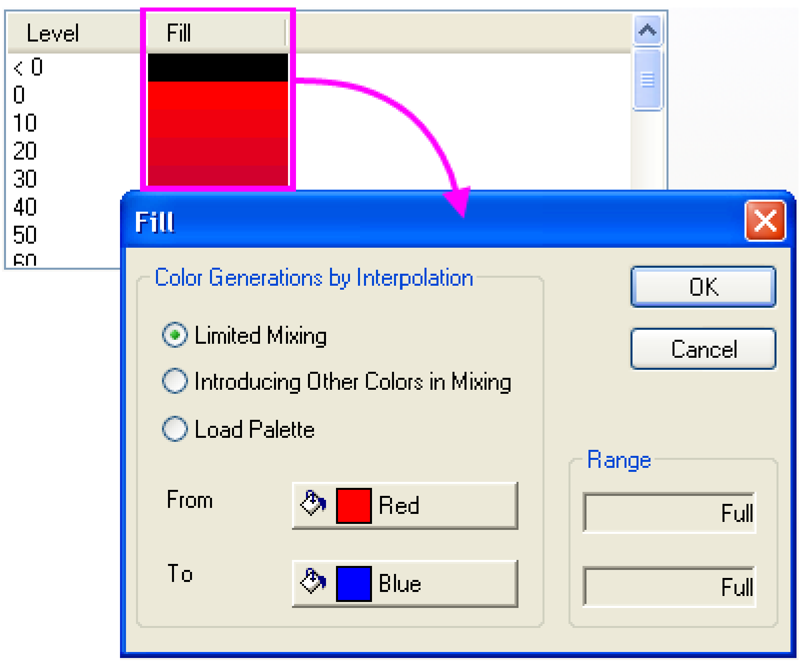


4. Double-click on the waterfall plot to open the **Plot Details** dialog. Go to the **Color Map** tab. On this tab, you can:

- Click on the **Level** column header to change the color levels

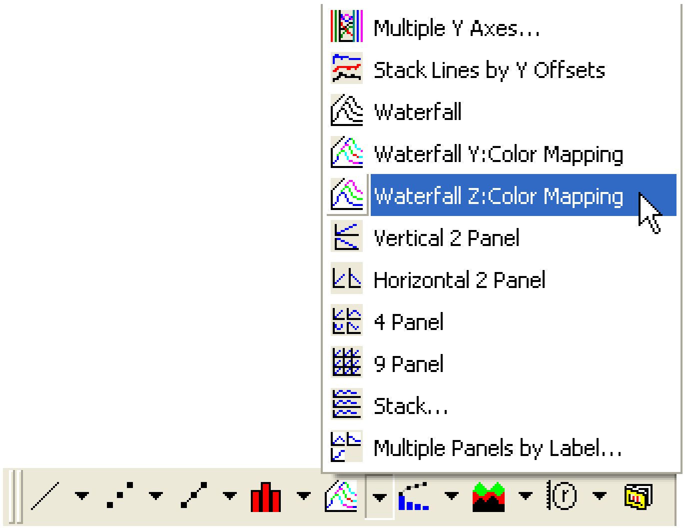


- Click on the **Fill** column header to load a color palette or change the filled color list:

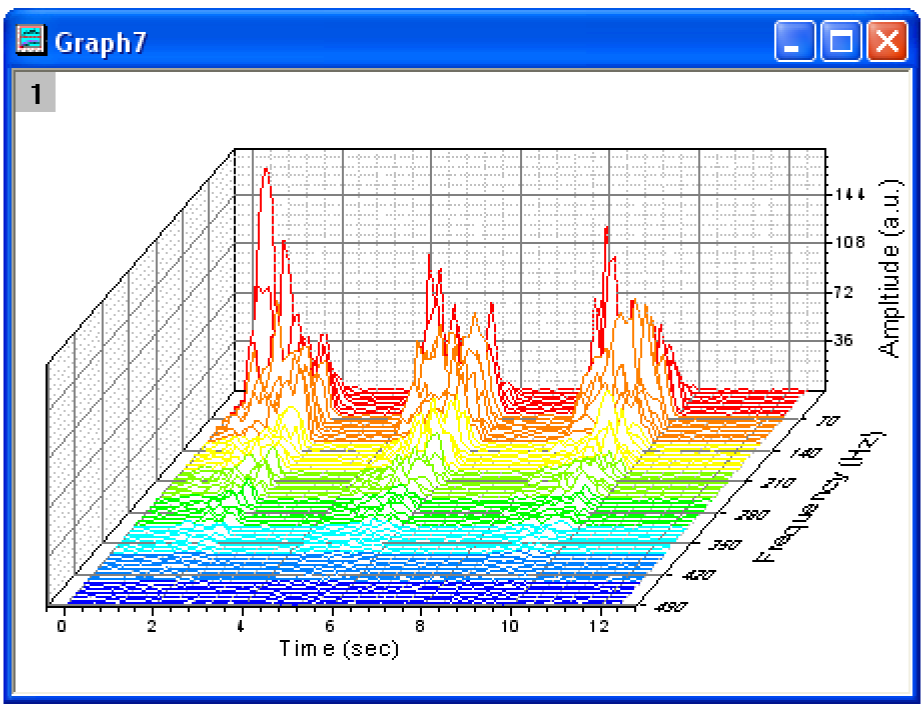


Please note that you can also click inside one single cell to change that level individually.

5. To plot a waterfall color-mapped with Z values, highlight the **waterfall** worksheet in **Book 2** and select **Waterfall Z: Color Mapping** from the **2D Graphs** toolbar (Alternatively, Select **Plot: Multi-Curve: Waterfall Z:Color Mapping** from the main menu).

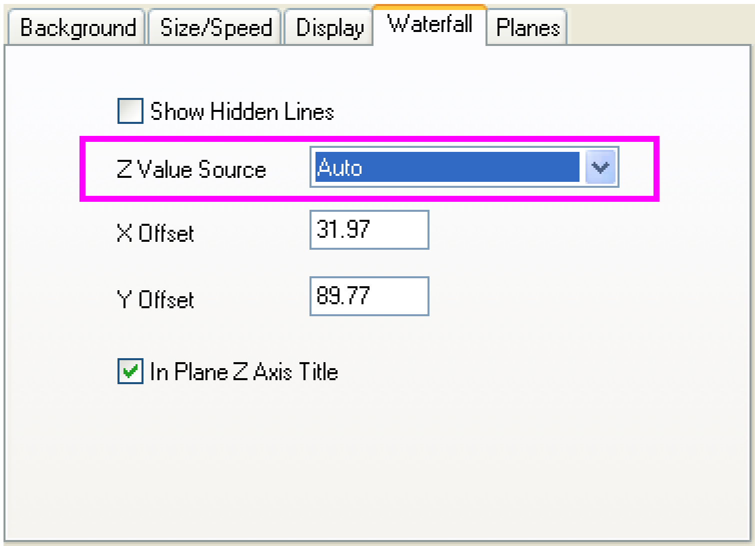


The graph should look like the picture below:

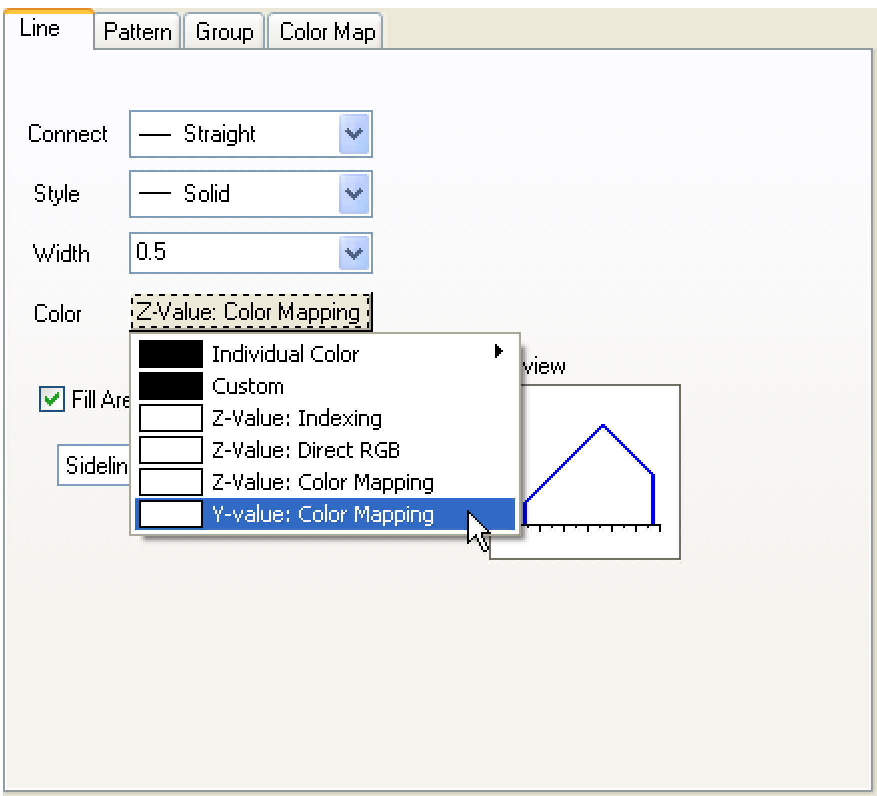


6. Note that the user-defined parameter "Frequency (Hz)" is used as the Z axis automatically. To use another column label as the Z axis, Short Name for example, double-click inside the layer and away from the waterfall lines to open the **Plot Details** dialog at the **Layer** level. Go to the

Waterfall tab, and select **Auto** from the **Z Value Source** drop-down list.



7. To switch between Y and Z color mapping, select the first plot on the left panel of the **Plot Details** dialog and activate the **Line** tab. Select **Y-value: Color Mapping** from the **Color** drop-down list, to switch to Y color mapping. You can also select **Z-Value: Indexing** or **Z-Value: Direct RGB** from this drop-down list.



6.2 Themes

- Copy and Apply a Plot Format to Another Plot

6.2.1 Copy and Apply a Plot Format to Another Plot

Summary

It is possible to copy and paste formatting from one plot to another, so there is no need to spend time recreating identical customizations such as size and color of symbols and lines.

Minimum Origin Version Required: Origin 8.0 SR6

What you will learn

This tutorial will show you how to:

- Copy a plot format (color, size, etc. of the symbol or the line), and apply it to other plot.

Steps

1. Click the **New Project** button on the Standard toolbar to begin with a new project.
2. Select **File: Import: Single ASCII** menu, and import **exponential decay.dat** in the **\Samples\Curve Fitting** subfolder in your Origin program directory.
3. Highlight column B, C and D, and select **Plot: Line+Symbol: Line+Symbol" menu to plot these three datasets.**
4. Double-click on the plot to show the **Plot Details** dialog box.
5. Choose **Group** tab in the dialog, and select **Independent** for the **Edit Mode** -- this makes it easier to customize individual plots.
6. Make sure that the top data plot (Time(X) Decay 1(Y)) is selected in the left panel of the **Plot Details** dialog. If not, select this plot branch in the left panel.
7. Select the **Symbol** tab, and change the Size to "5". (You can also change the shape or the color to others of your choice.)
8. Select the **Line** tab, and change the width to "0.2". (You can also change the style or the color to others of your choice.) Click OK. You will see that the **Decay 1** plot has been customized.
9. Click on **Decay 1** plots to select. Right-click on it and select **Copy format: ALL**. This will copy the plot format of Decay 1 to the clipboard.
10. On the graph, click on the **Decay 2** data plot to select it, right-click and select **Paste Format**. You will see the plot format of Decay 1 copied to Decay 2.

6.3 Color

OriginLab:Plotting Overlapping Data and Setting Transparency

6.3.1 Plotting Overlapping Data and Setting Transparency

Summary

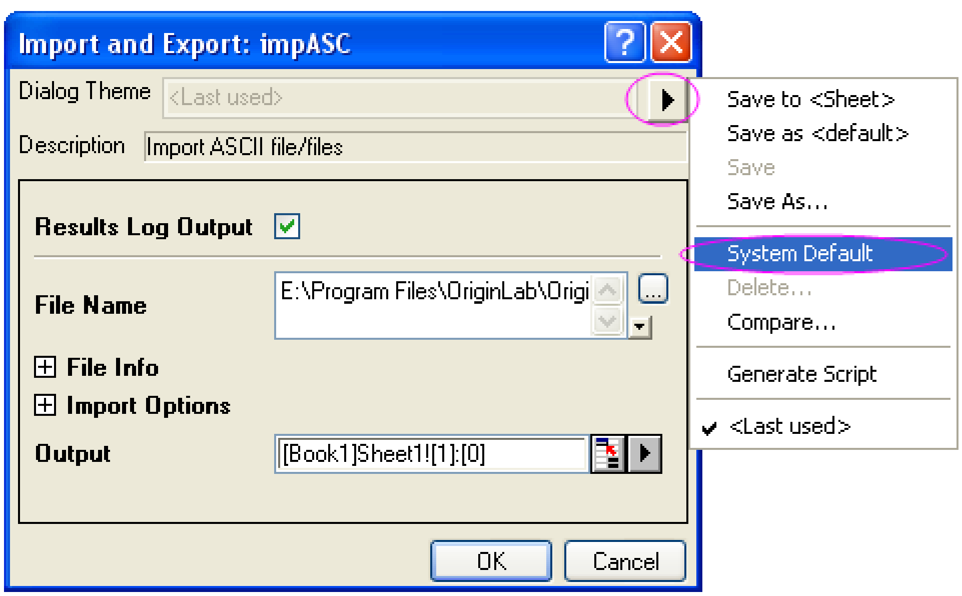
In this tutorial, we will show you how to create overlapping column plots, and then set transparency to make the overlapping parts visible.

This tutorial will show you how to:

- Customize the grouped column plots
- Set Transparency

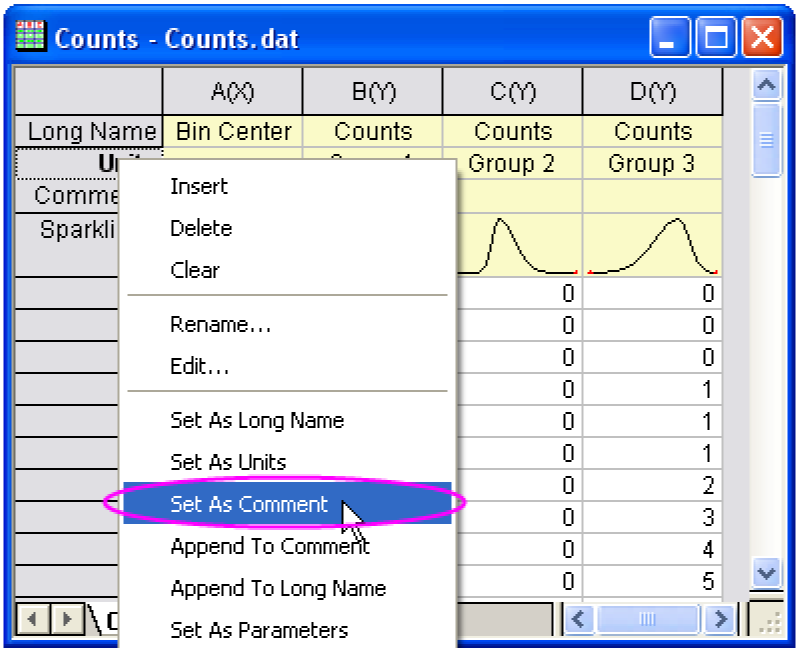
Plot Overlapping Data and Set Transparency

1. Start with a an empty worksheet and from the menu, select **File: Import: Single ASCII** or click the **Import Single ASCII** button to open the file browser.
2. Choose the file *<Origin Installation Directory>\Samples\Graphing\Counts.dat*, check the **Show Options Dialog** check box, and click the **Open** button to bring up the **impASC** dialog.
3. Click the triangle button in the upper right corner, and then select **System Default** from the context menu to apply the default settings to this dialog.

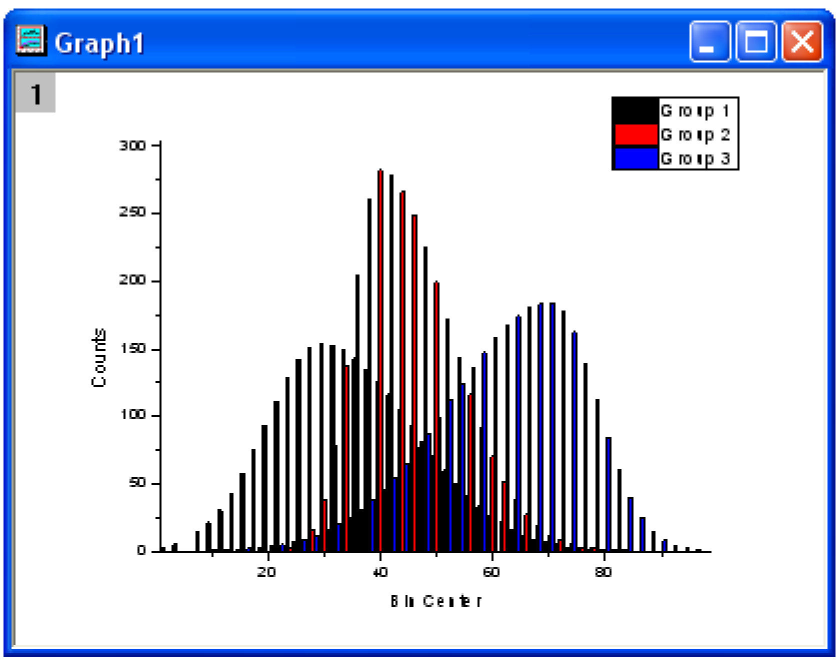


4. Click the **OK** button to import the data to the worksheet.

5. Right-click on the row header of **Units**, and then select **Set As Comment** from the pop-up menu to set the contents to be comments instead of units.

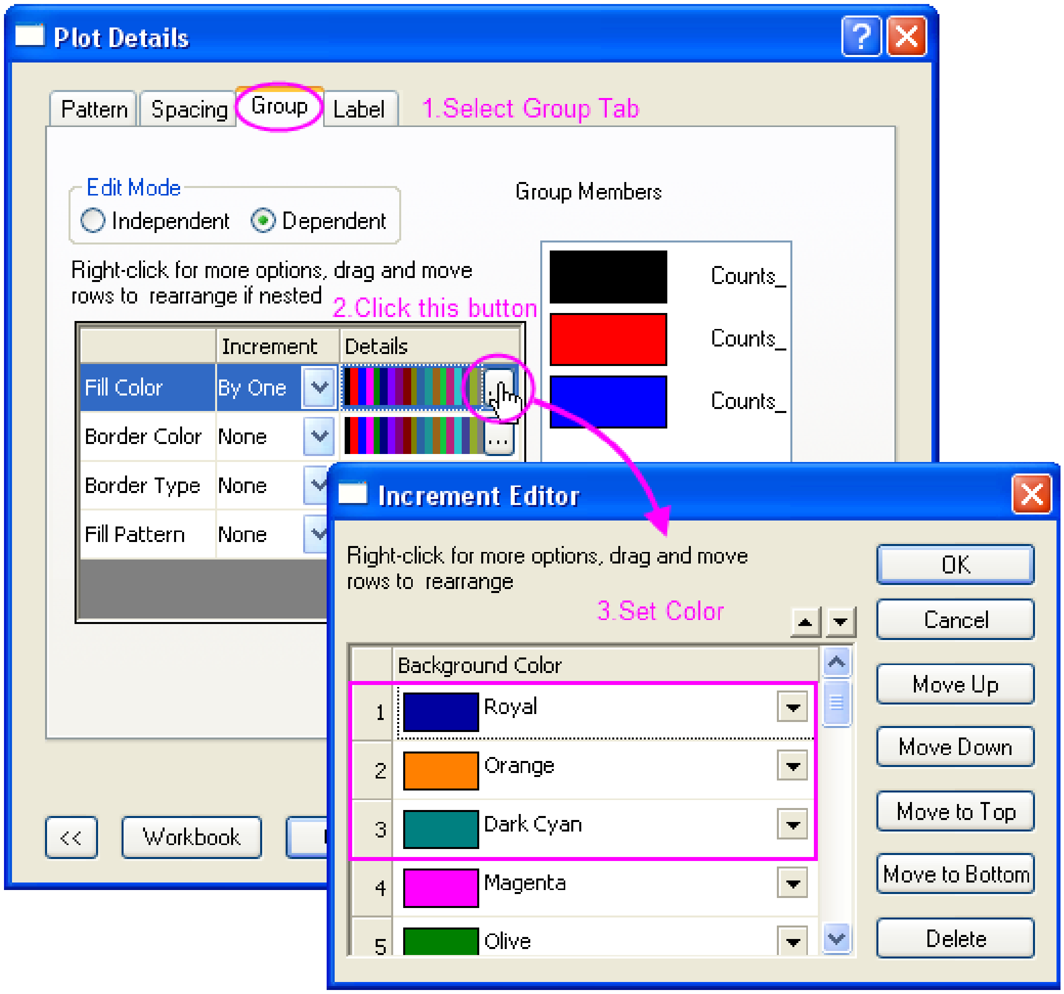


6. Highlight all columns in the worksheet, and then from the menu, select **Plot: Column/Bar/Pie: Column** to create a column graph.

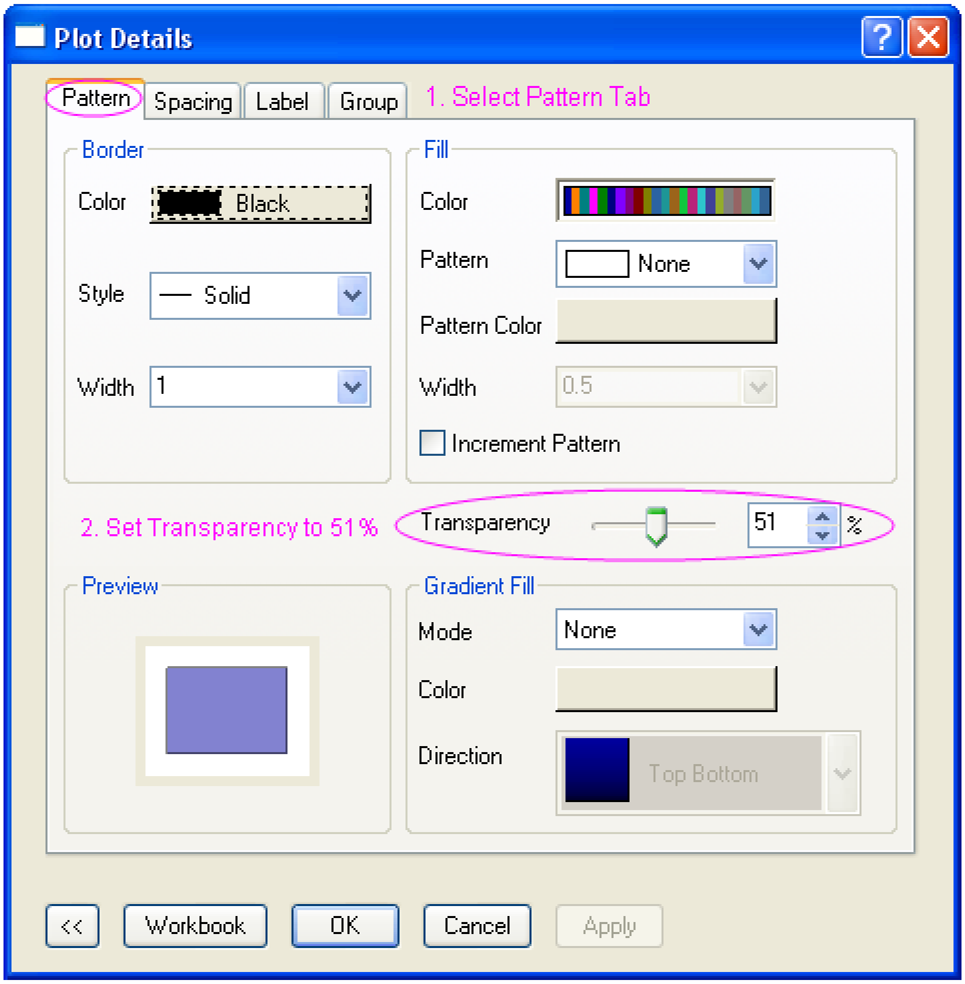


7. Double-click on the plot to open the **Plot Details** dialog. In the dialog, choose the **Group** tab. Click on the color bar under **Details** and click in the **Fill Color** row. Then click on the button that appears to open the **Increment Editor** dialog. In this dialog, you can set the first three colors to

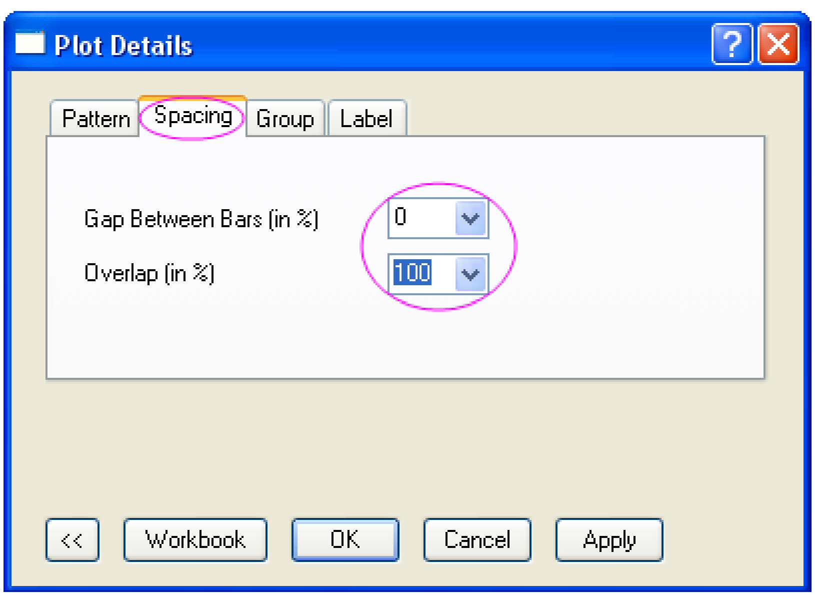
be some specific colors such as Royal Blue, Orange, and Dark Cyan. Click OK to close the dialog.



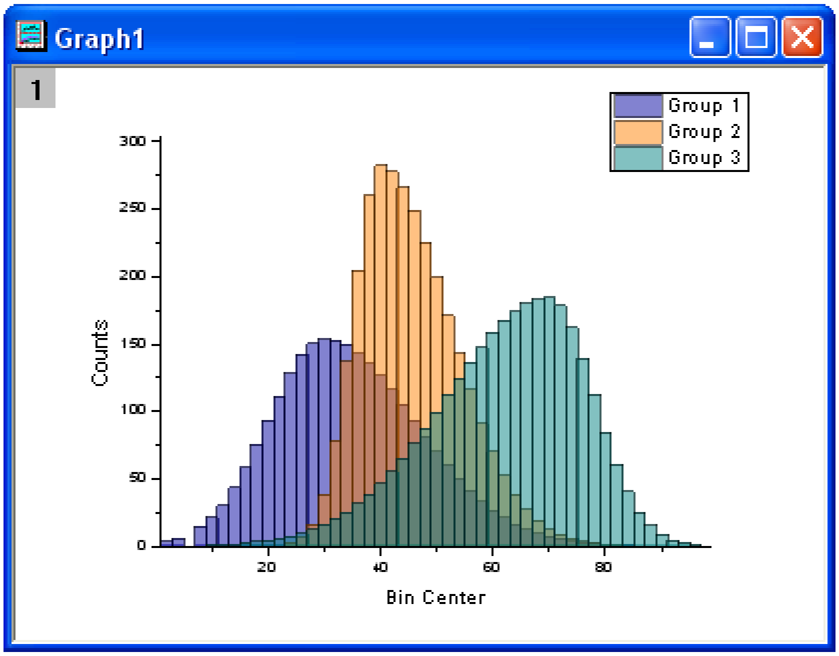
8. Select the **Pattern** tab in Plot Details, and set the **Transparency** control to 50%.



9. Select the **Spacing** tab, and set **Gas Between Bars** to zero and **Overlap** to 100.



10. Click the **OK** button to close the Plot Details dialog. The resulting graph should look like the image below:



6.4 Add remove or reorder data plots

6.4.1 Summary

There are multiple ways to add/remove a data plot in your Origin graph. You can simply drag a set of worksheet data and drop it into a graph window to add it to the active layer. And you can also add data plots by using the **Layer Content** dialog or the **Plot Setup** dialog. The **Layer Content** dialog provides you with further options to reorder data plots and group/ungroup data plots.

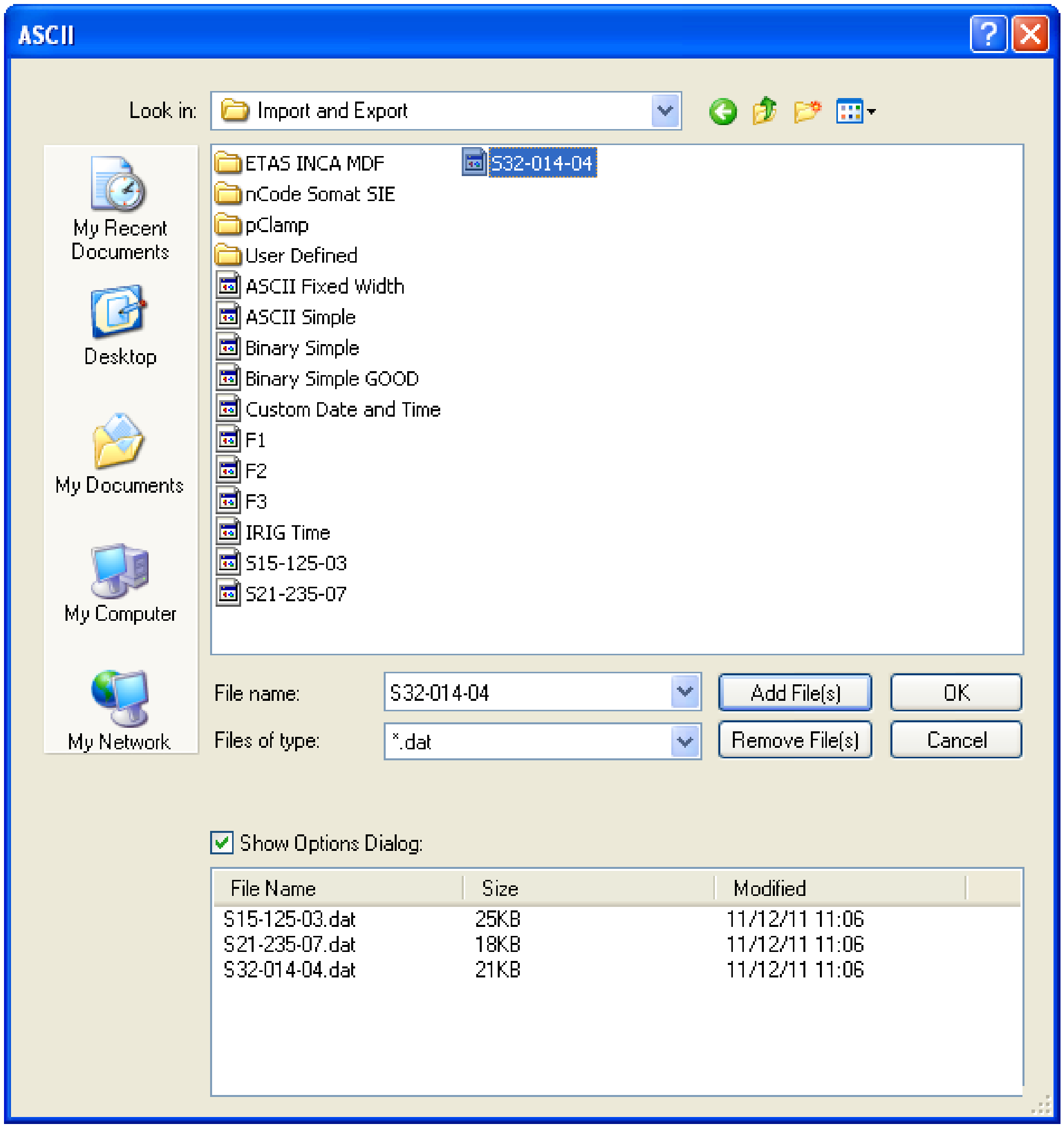
This tutorial will show you how to:

- Add data plot to an existing graph by drag-and-drop.
- Add data plot by using the **Layer Content** dialog.
- Group and ungroup data plots.
- Reorder data plots.

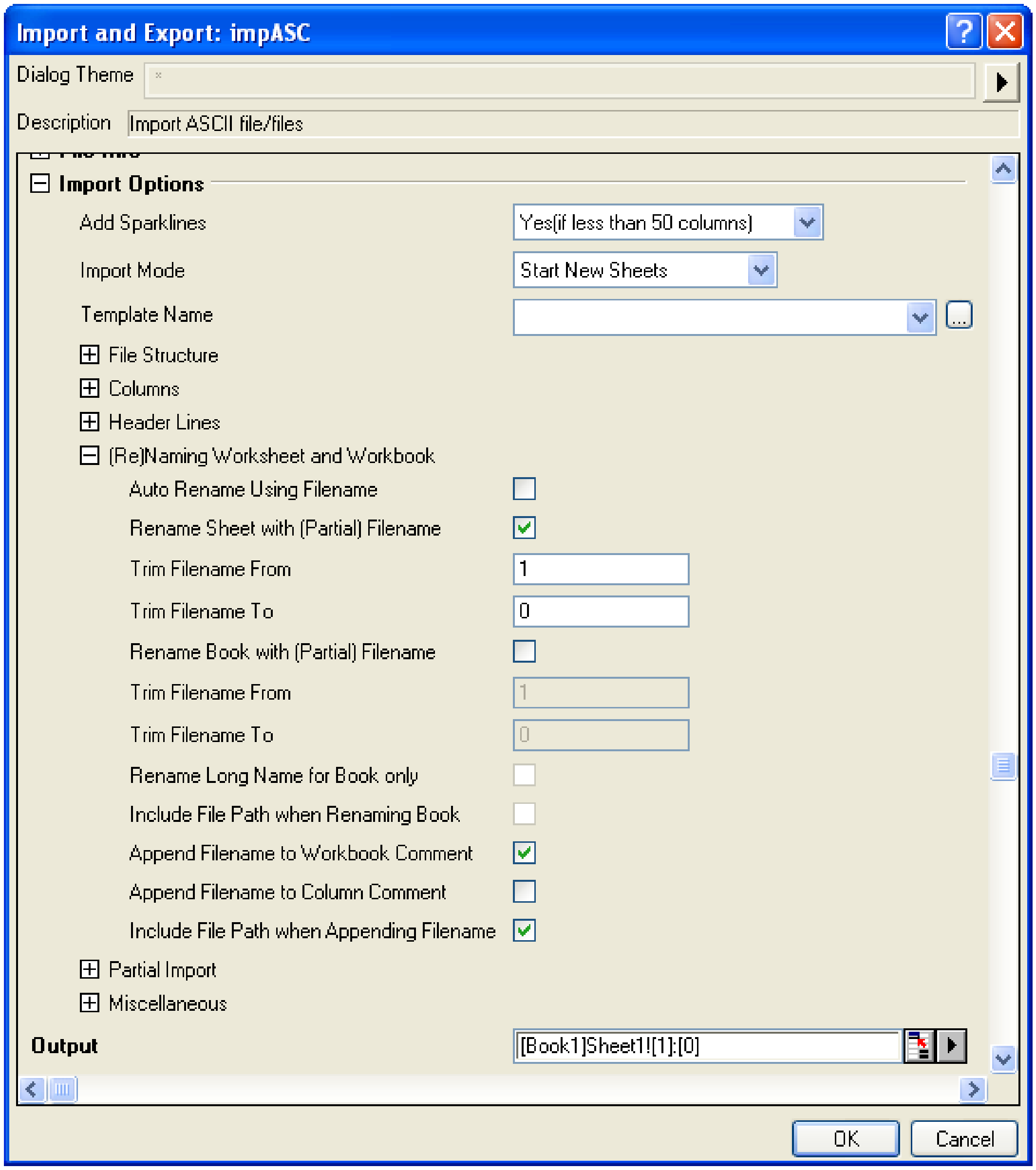
6.4.2 Add data plot to an active graph by drag-and-drop

1. Start with a new project and click the **Import Multiple ASCII** button  on the Standard toolbar.

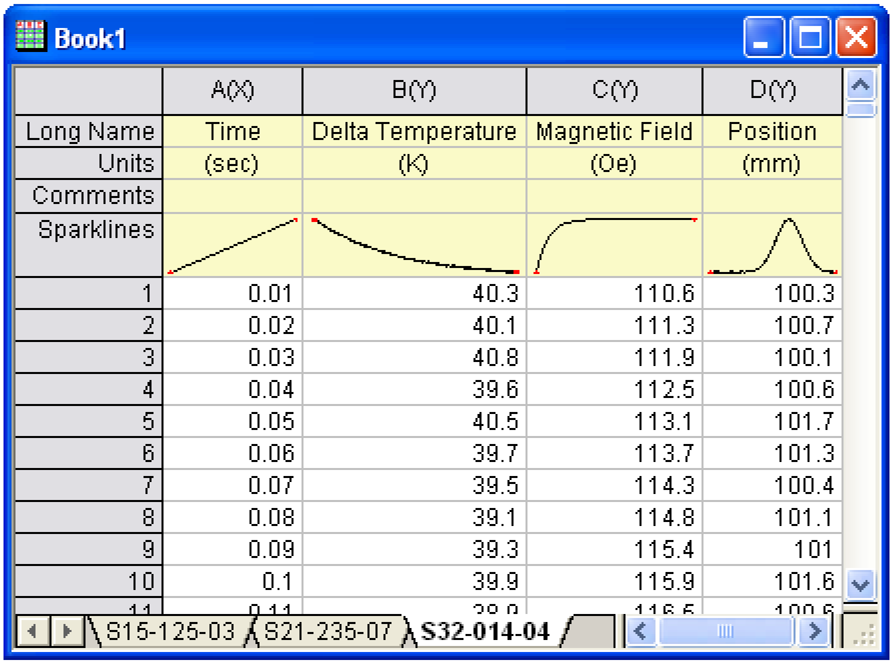
2. Select the files *S15-125-03.dat*, *S21-235-07.dat* and *S32-014-04.dat* from *\Samples\Import and Export* and add them to the lower panel of the dialog. Make sure the **Show Options Dialog** box is checked and click **OK**. This will open a dialog for import settings.



3. Change **Import Mode** to **Start New Sheets**. Expand the **(Re)Naming Worksheet and Workbook** node. Check the **Rename Sheet with (Partial) Filename** check box and uncheck the **Rename Book with (Partial) Filename** so that only the sheet gets renamed.

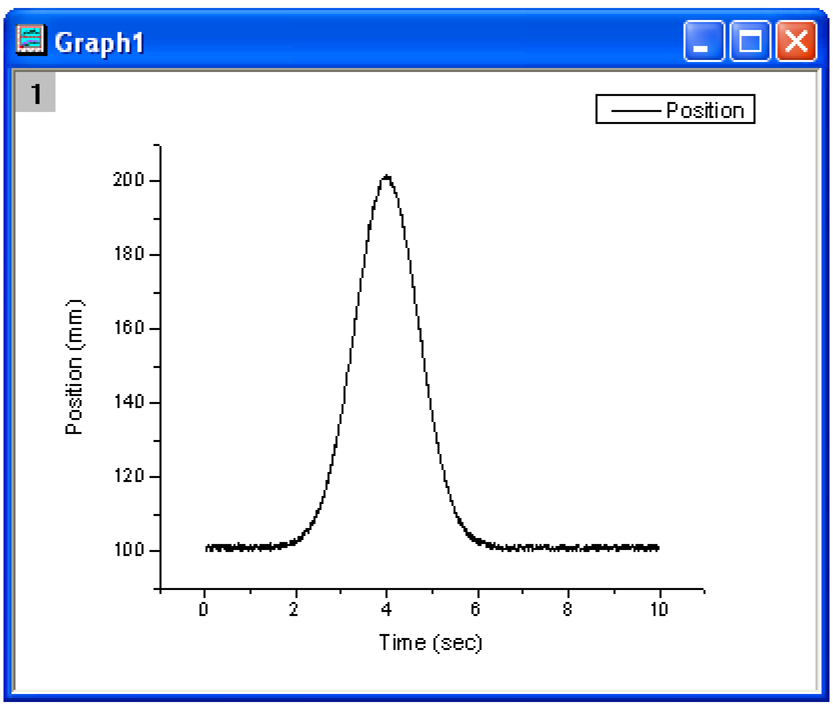


Keep the rest of the settings unchanged and click **OK** button to import these three files.

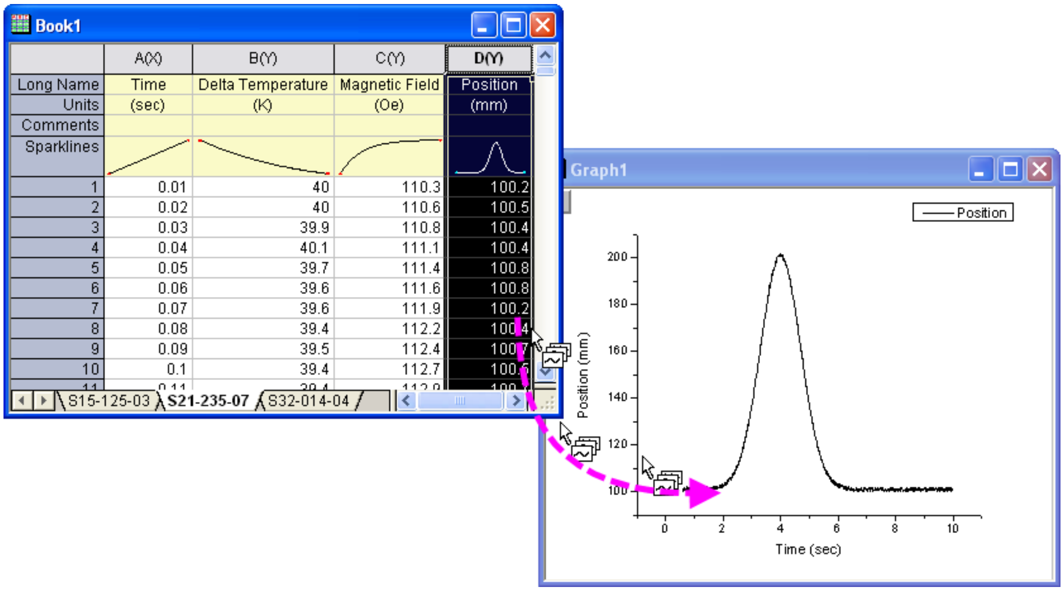


	A(X)	B(Y)	C(Y)	D(Y)
Long Name	Time	Delta Temperature	Magnetic Field	Position
Units	(sec)	(K)	(Oe)	(mm)
Comments				
Sparklines				
1	0.01	40.3	110.6	100.3
2	0.02	40.1	111.3	100.7
3	0.03	40.8	111.9	100.1
4	0.04	39.6	112.5	100.6
5	0.05	40.5	113.1	101.7
6	0.06	39.7	113.7	101.3
7	0.07	39.5	114.3	100.4
8	0.08	39.1	114.8	101.1
9	0.09	39.3	115.4	101
10	0.1	39.9	115.9	101.6

4. Activate Sheet S15-125-03. Highlight Column D. Select **Plot: Line: Line** to create a line plot.

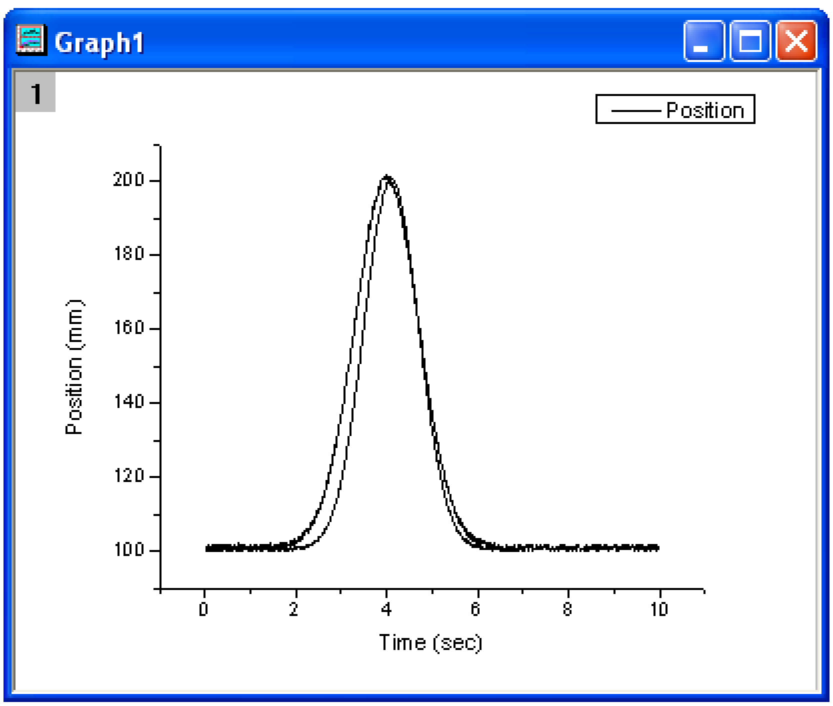


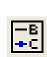
5. Go to Book1. Activate Sheet S21-235-07. Select Column D. Move the mouse to the edge of Column D carefully until the mouse turns to . Hold down the left mouse button to drag Column D to Graph1 window.

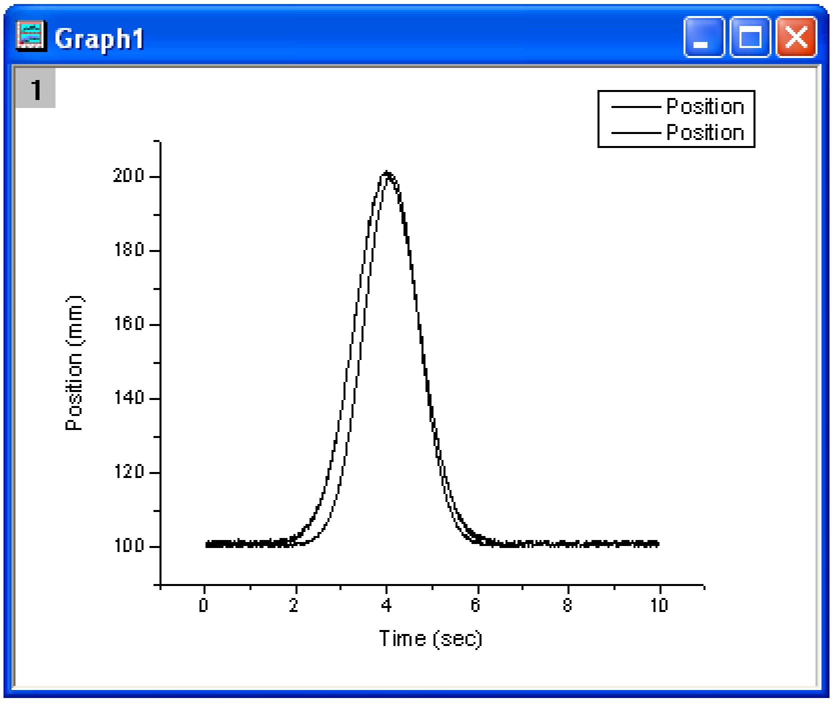


	A(X)	B(Y)	C(Y)	D(Y)
Long Name	Time	Delta Temperature	Magnetic Field	Position
Units	(sec)	(K)	(Oe)	(mm)
Comments				
Sparklines				
1	0.01	40	110.3	100.2
2	0.02	40	110.6	100.5
3	0.03	39.9	110.8	100.4
4	0.04	40.1	111.1	100.4
5	0.05	39.7	111.4	100.8
6	0.06	39.6	111.6	100.8
7	0.07	39.6	111.9	100.2
8	0.08	39.4	112.2	100.4
9	0.09	39.5	112.4	100.7
10	0.1	39.4	112.7	100.[illegible]

6. A Reminder Message dialog box will appear asking you whether or not to rescale the axes. Select **Yes, and do the same in the future, no need to ask again** and click **OK**. The second line plot is added to the graph.

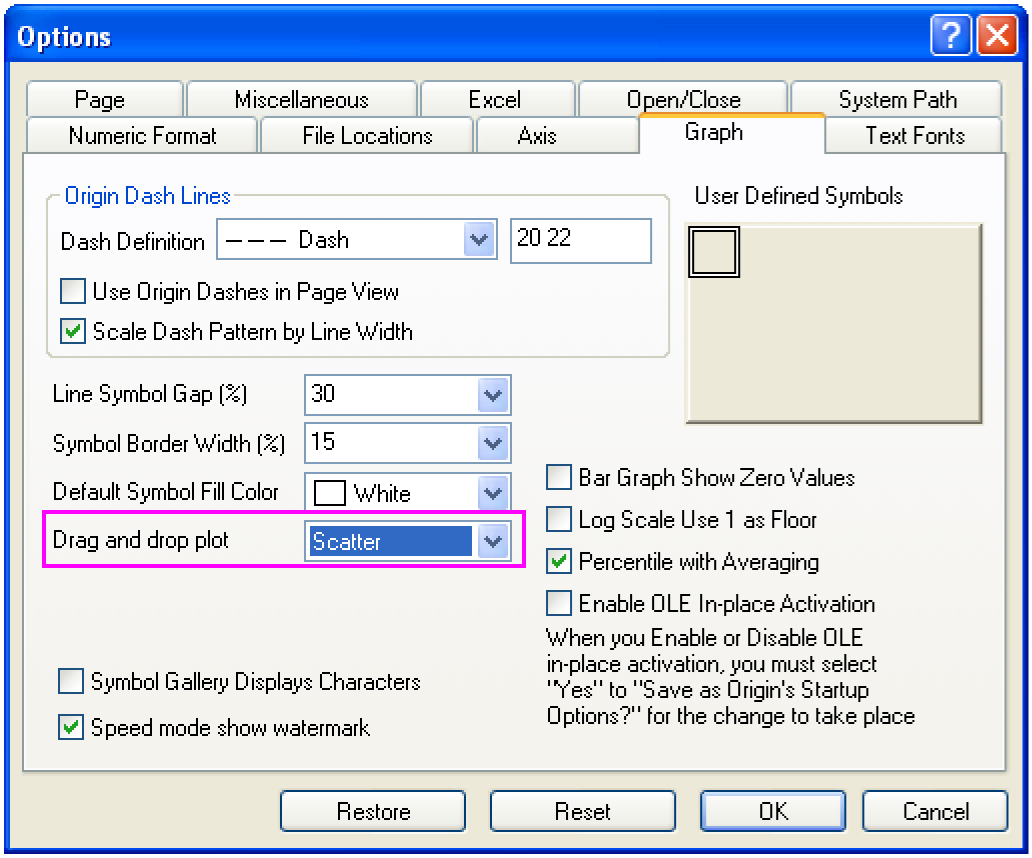


7. Click the **New Legend** button  on the Graph toolbar to rebuild the graph legend.



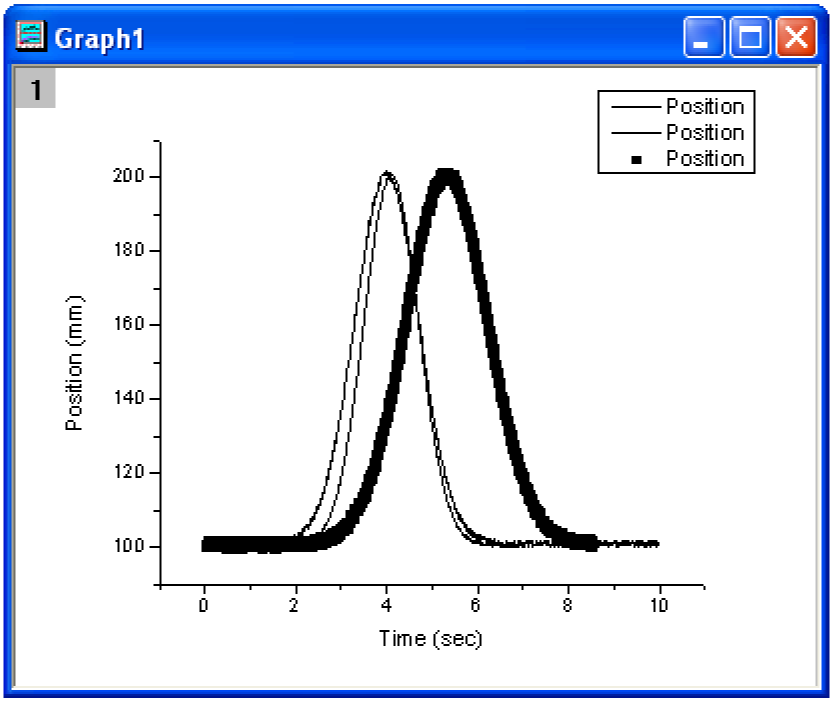
The following steps will show you how to specify the plot type when adding a plot to an existing graph by this drag-and-drop method.

8. Select **Tools: Options** dialog to open the **Options** dialog.
9. Go to the Graph tab. In the middle of this dialog, you will see the **Drag and drop plot** option. This option allows you to specify the plot type, while adding a dataset to an existing graph by drag-drop. The default setting is Current, which means the plot type depends on the current plot's type in the existing graph. Change **Drag and drop plot** to Scatter.



Click **OK** to close the dialog.

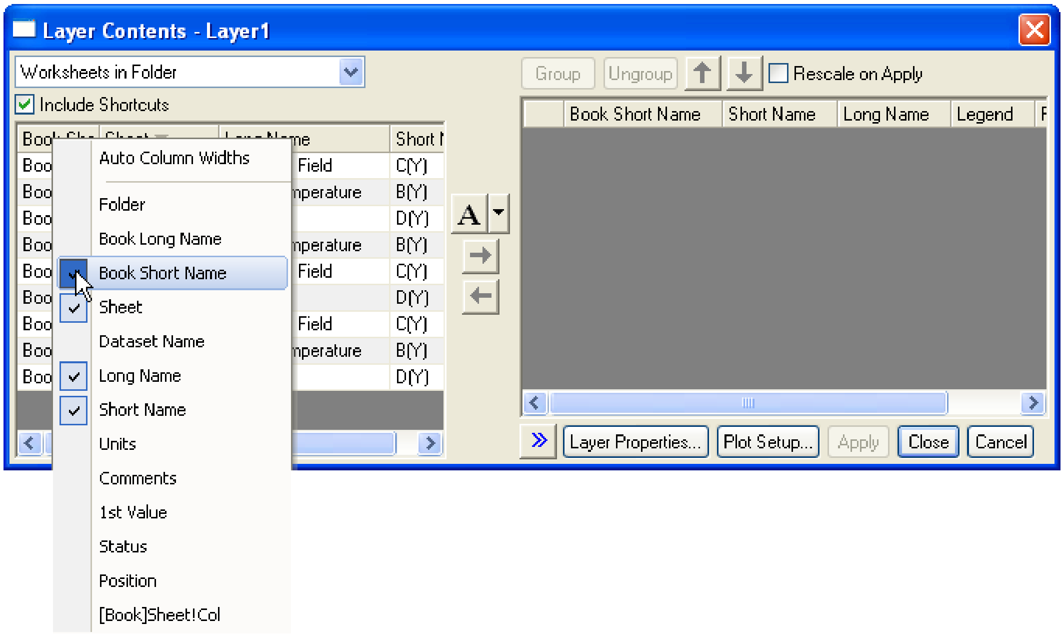
10. Go back to Book1. Activate sheet S32-014-04. Highlight Column D. Drag and drop it to Graph1 window. This time the dataset is added as a Scatter plot.

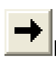


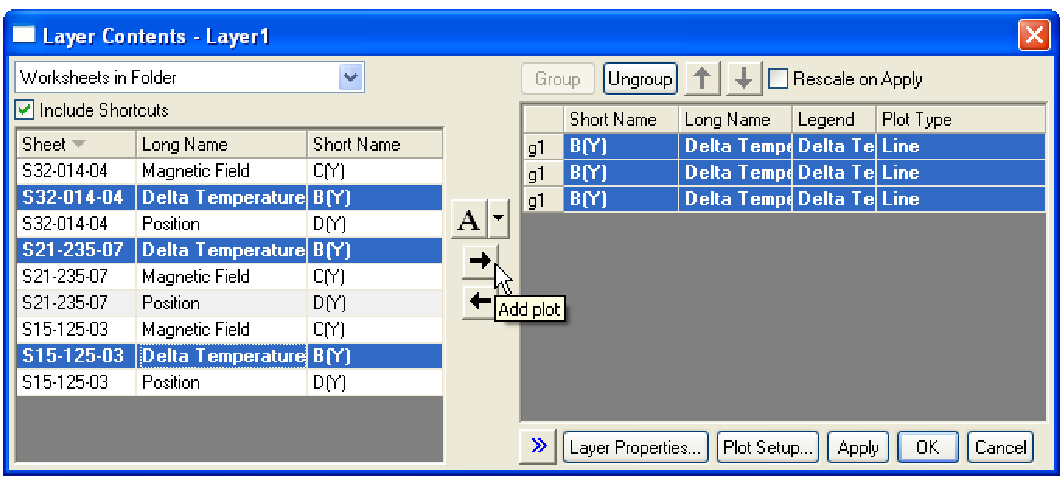
6.4.3 Add data plot to an active graph by drag-and-drop

Continue using data in section 1.

1. Click the New Graph button  on the Standard toolbar. This will create a new graph window without data.
2. Double-click on the Layer 1 icon on the top left corner of the graph window. This will open the **Layer Content** dialog.
3. On the left panel of the **Layer Content** dialog, all the available datasets are listed. You can customize to show the information of datasets. Because all of our data comes from Book1, we do not need to show the Book name. Right-click on the Book column header, uncheck Book Short Name from the context menu. This will hide the book name.



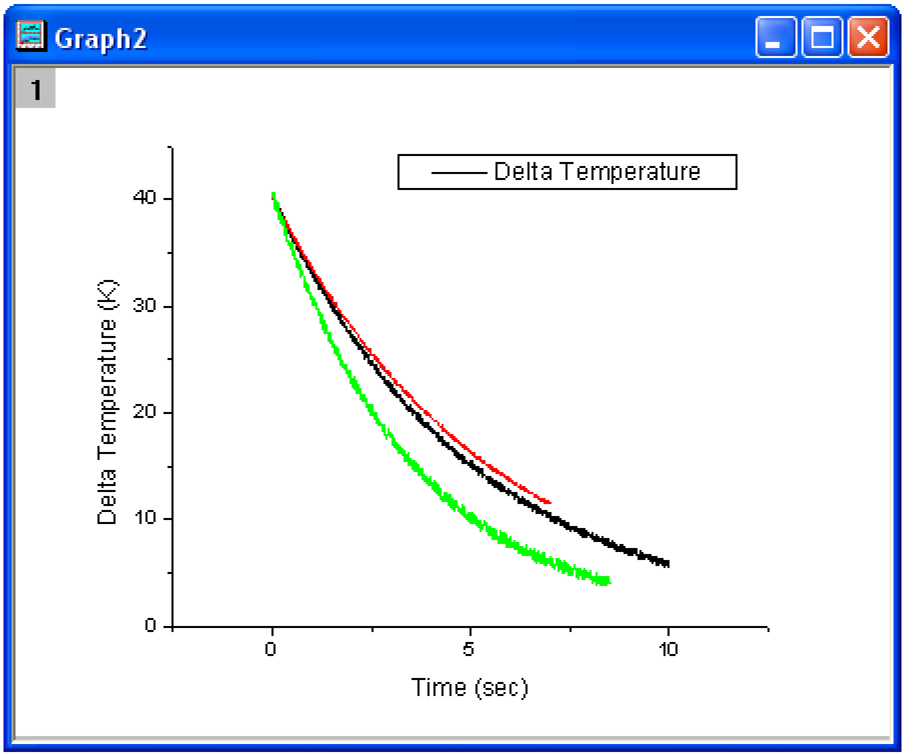
4. To add Column B, Delta Temperature, in three sheets, press Ctrl key on the keyboard and select Column B in worksheet *S32-014-04*, *S21-235-07*, and *S15-125-03*, respectively. Click the  button to add them to the right panel.



The  button shows the plot type of these three datasets. **A** means auto, which is Line by default. If you want to change the plot type, please press the triangle button next to  and select a desired plot type before you click the  button to add them to the right panel.

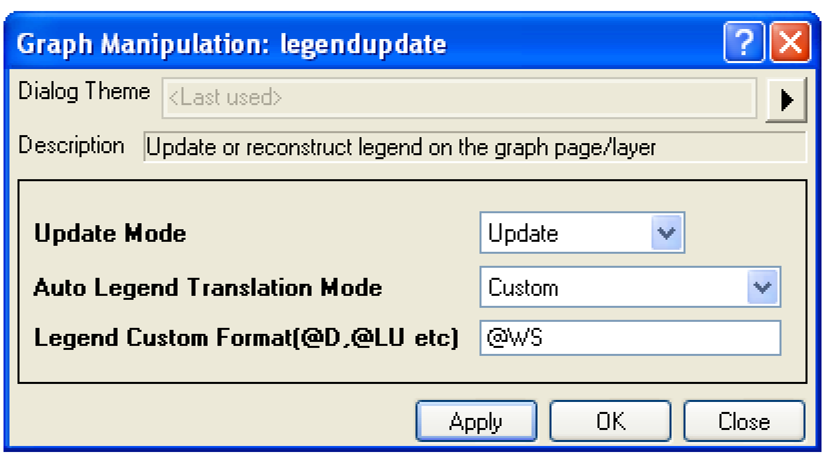
5. The first column on the right panel shows that the added curves are grouped by default. They are marked as group1 (g1). The color of curves in the group will change increasingly. If you want to ungroup them, click the **Ungroup** button.

6. Check the **Rescale on Apply** check box. Click **OK** button to add curves to Graph 2.

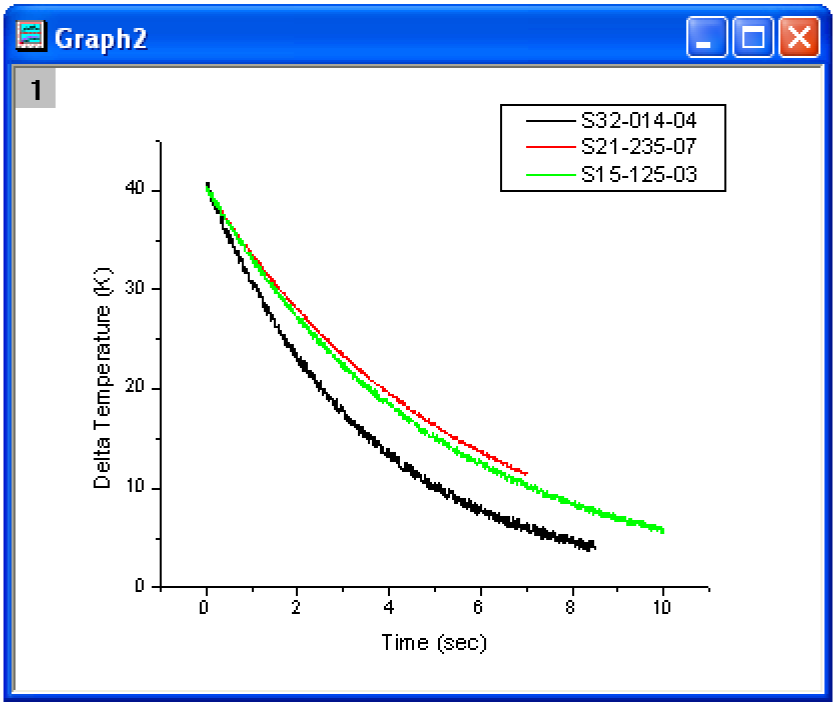


Because all three curves come from three different sheets, we can use the sheet name to distinguish them.

7. Select Graph: Update Legend from menu. This will open the **legendupdate** dialog.
8. Make sure Update has been selected for **Update Mode**. Select Custom for **Auto Legend Translation Mode**. Type "@WS" in the **Legend Custom Format** edit box. This means worksheet name will be used for legend.



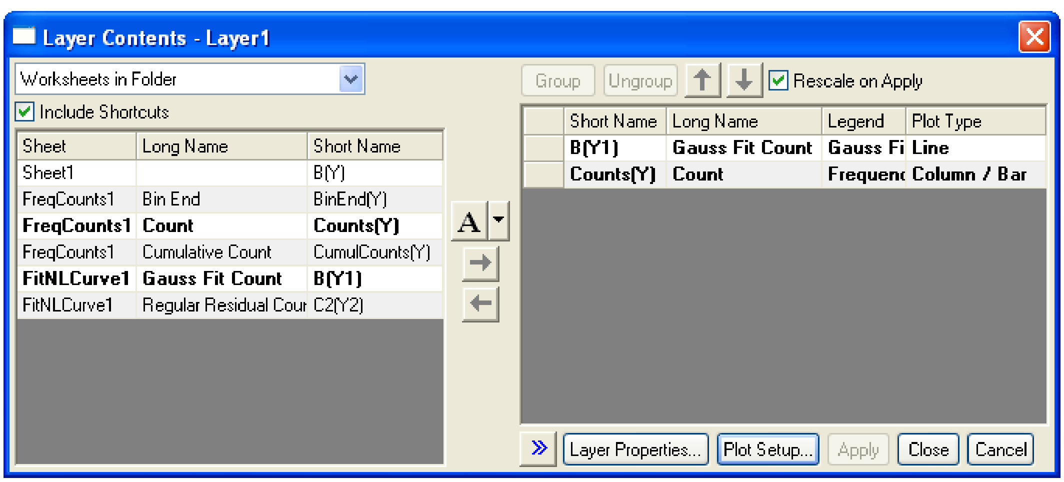
Click **OK** button to apply.



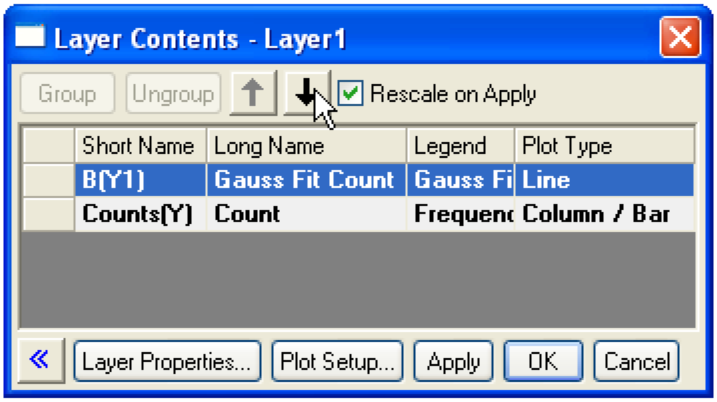
6.4.4 Reorder data plots

1. Open **Customizing Graphs.OPJ** from the *\Samples\Graphing* folder and select the **Plotting Order** folder from the **Project Explorer** window.

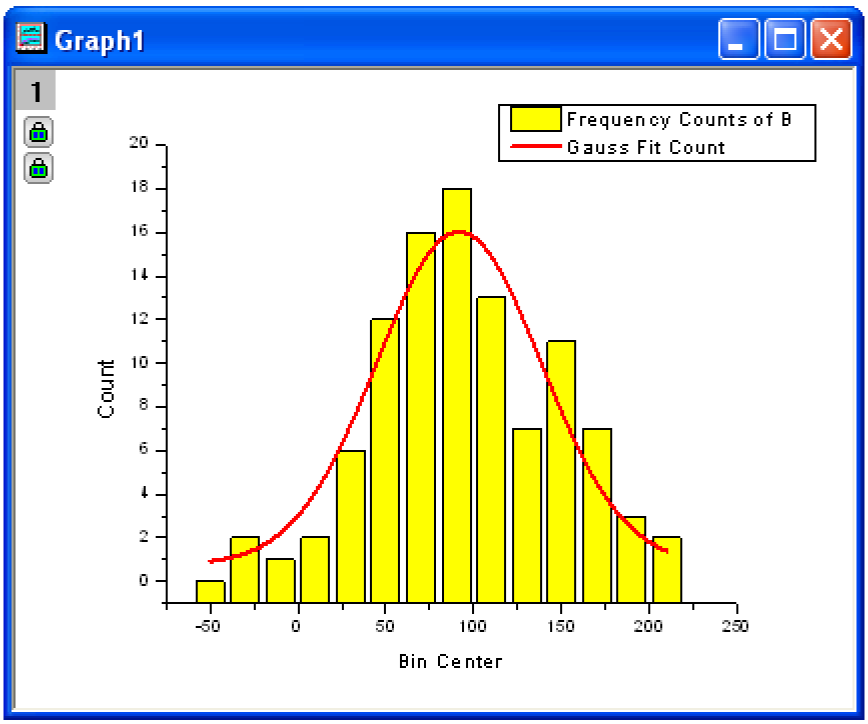
2. Activate Graph 1 window. Double-click on the Layer 1 icon on the top left corner of the graph to open the **Layer Content** dialog.



3. Click the **>>** button to fold the left panel. On the right panel, select the Gauss Fit line plot. Click the down button to move it down.



4. Click the **Apply** button. The Gauss Fit line plot is now on top of the column plot. Click **Close** button to close the dialog.



6.5 Layers

- Adding and Arranging Layers
- Merging and Arranging Graphs

6.5.1 Adding and Arranging Layers

Summary

A typical graph page generally includes three elements: a set of X, Y (and Z) coordinate axes, one or more data plots, and associated text and graphic labels. Origin combines these three elements into a movable, resizeable unit called a layer. While a page can contain as many as 121 layers, only one layer can be active at any one time.

What you will learn

- How to tell how many layers a graph has?
- How to tell what data is in what layer?
- How to swap layers

Steps

Importing data

1. Click the **Import Single ASCII** button, , on the Standard toolbar. The ASCII dialog opens.
2. In the Origin folder, browse to the Samples folder and then the Graphing folder. Select Wind.dat from the list of files.
3. Click **Open**. The data file imports into the worksheet.

Plotting the data

1. Highlight the Speed and Power columns.
2. Click the **Line** button, , on the **2D Graphs** toolbar. A line plot is created. It appears that this data would be better plotted on a double-Y graph, a graph with two controlling Y axes.
3. Click the X to **Close** this window. You will be asked if you want to hide or delete the window. Click the **Hide** button. (If you Delete, you will not be able to Undo, and you will need to re-create the graph. Hiding closes the window from view, but you can later make it visible using the Project Explorer window.)
4. The Speed and Power columns should still be highlighted. Click the **Double-Y Axis** button, , on the 2D Graphs toolbar. This new graph contains 2 layers.

How to tell what data is plotted in a layer?

One way is with the legend:

1. Double-click on Graph1 from the Project Explorer window. The graph opens and becomes the active child window.
2. Select **Format:Page** and go to the **Legends/Titles** tab.
3. Set the **Auto Legend Translation Mode** to *Data Range*.
4. Click **OK**.

A second way is with the status bar:

1. Double-click on Graph2 from the Project Explorer window. Graph2 is now in front and becomes the active child window.
2. Click the Layer 1 icon,  to make layer 1 active.
3. Look in the lower right of Origin's status bar and you'll see [WIND]WIND!Col(Speed)[1:12].
4. Repeat for Layer 2 and you'll see [WIND]WIND!Col(Power)[1:12].

A third way is from the menu:

1. Make Graph2 active.
2. Right-click on the Layer 1 icon. At the bottom of the context menu, you see the data plot list. The one checked is the active data plot.
3. Right-click on Layer 2 to view the list of data that is plotted in it.
4. Note that you can also select the **Data** menu to view the data plot list.

A fourth way is with the **Plot Setup** dialog.

1. With Graph2 still active, Right-click on the layer 1 icon to select **Plot Setup**.
2. In the opened **Plot Setup** dialog, expand the Plot List to see also the data in Layer 2. You can see the advantage here is that you can see data in all layers at once.
3. Uncheck the Show check box for the **Speed** data plot.
4. Click **OK**. This data plot is only hidden which is why the legend still indicates its presence in the layer.

The last way is from the **Plot Details** dialog.

1. Double-click on any of the line+symbol plots in Graph2.
2. Expand the tree so you see contents of both **Layer1** and **Layer2**.
3. Uncheck the **Speed** and **Power** data plots.
4. Click **OK**.

6.5.2 Merging and Arranging Graphs

<u>Summary</u>

The **Merge Graph Windows** dialog allows you to select which graphs you wish to combine, choosing from any graph in the project. It also has controls to specify how you want the individual graphs arranged on the new page.

The **Object Edit** toolbar allows you to quickly align and size multiple layers.

The **Layer Management** dialog lets you add, arrange and link layers on a single graph page.

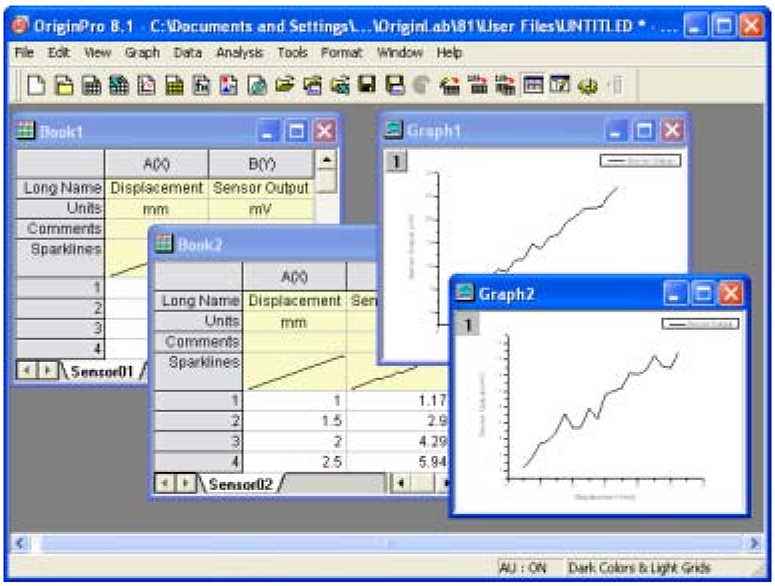
This tutorial will show you how to:

- Merge multiple graphs into one graph

- Resize and align layers quickly
- Add a second axis using a nonlinear relationship with the primary axis
- Use Layer Management for more complex layer positioning and linking

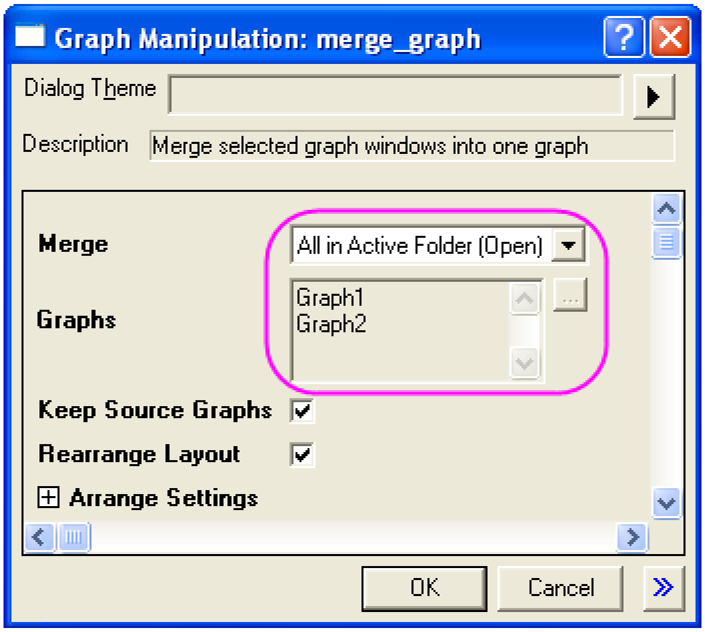
Creating a Merged Graph from Multiple Graphs

1. Import the two files **sensor01.dat** and **sensor02.dat** from ***\Samples\Curve Fitting***, as separate sheets in separate books, using the file names as the sheet names. (Please read the Importing Data tutorial for how to import files.)
2. Select the Y column of one sheet and create a line plot. Repeat with the Y column of the other sheet and create a second line plot. You will have two separate graph windows at this point.



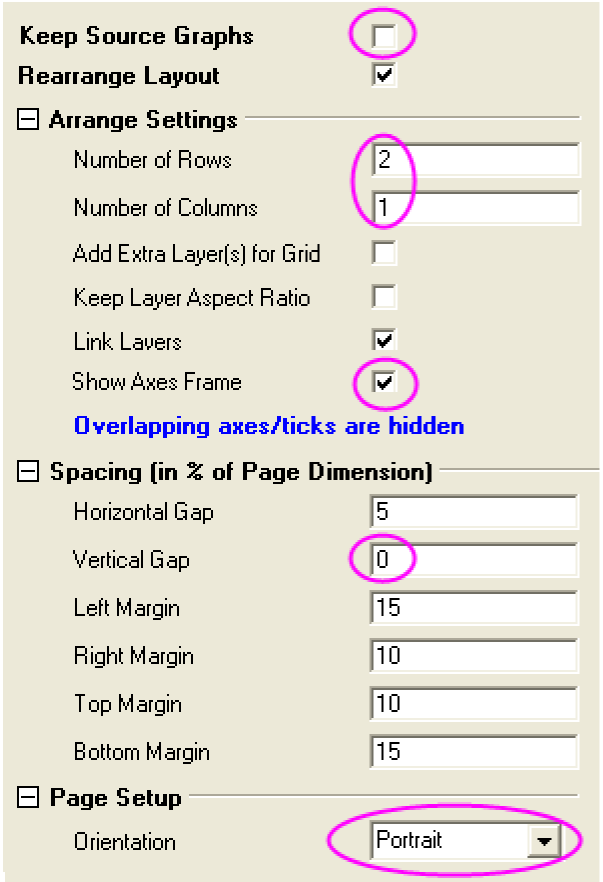
3. Now, to merge the two graphs into one page, bring up the merge graphs dialog from **Graph: Merge Graph Windows**. In this dialog, the default setting for merging graphs is **All in Active**

Folder (Open), so the two graph windows are already listed in the **Graphs** box.

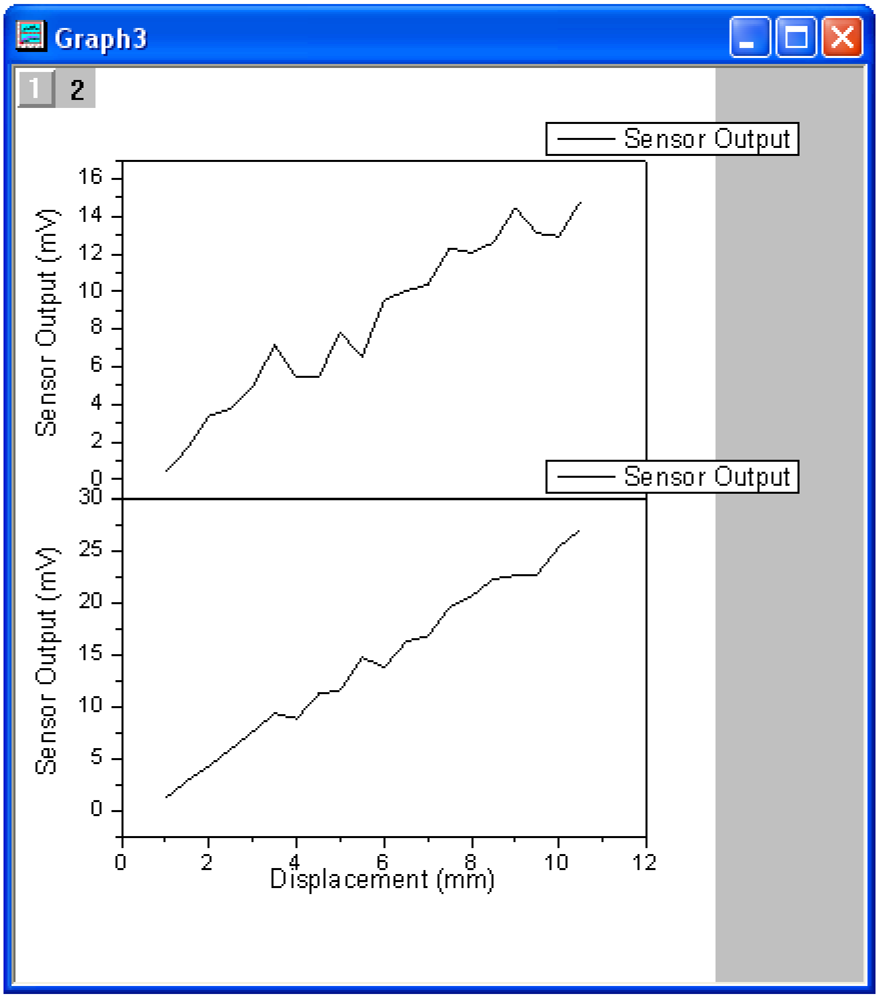


4. Specify the following settings to merge the two graphs: 1), Uncheck **Keep Source Graphs**. This will remove source graphs after merging. 2) Arrange the layers as one column and two rows. 3) Check **Show Axes Frame**. In this example, the two layers share the same X axis range, so we can hide the overlapped X axis. 4) Set the **Vertical Gap** to 0 for the two layers. 5) Change page

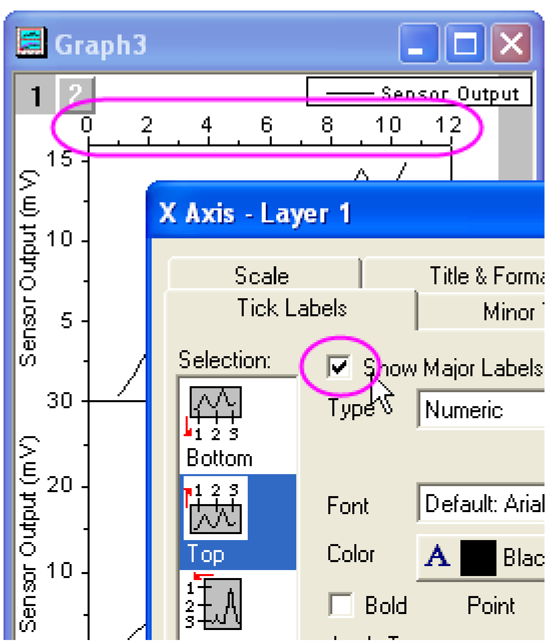
orientation to **Portrait**. Then the source plots will be added as layers in the new graph page.



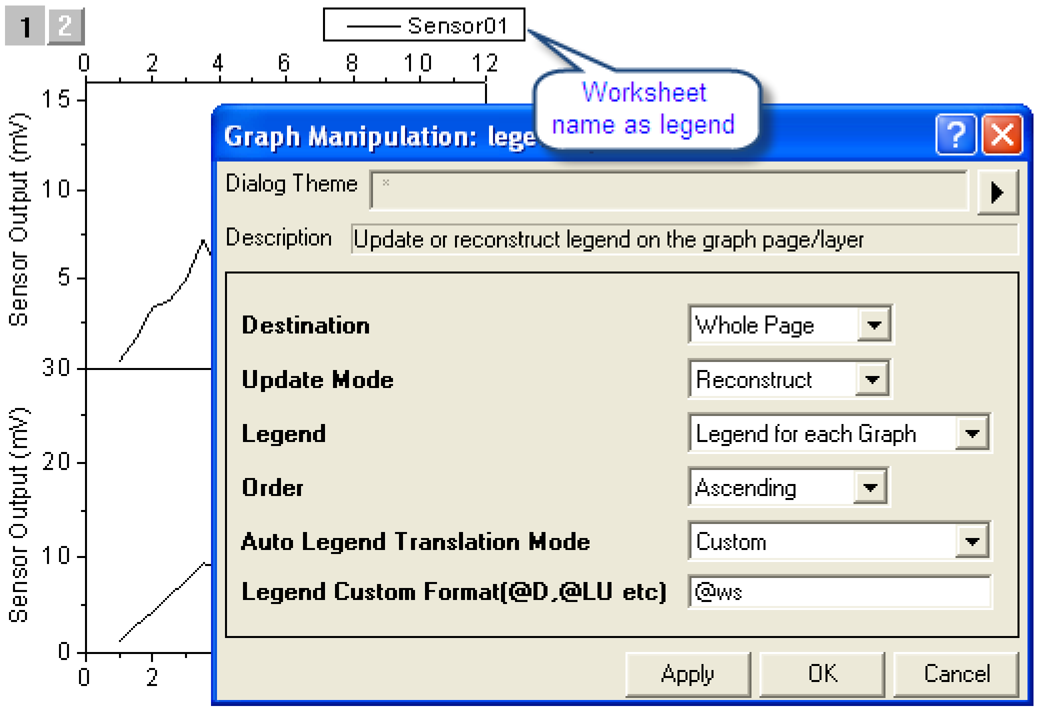
5. Click **OK** to close the dialog, and a new merged graph page is created:



6. Double-click the top X-axis and add tick labels as below:

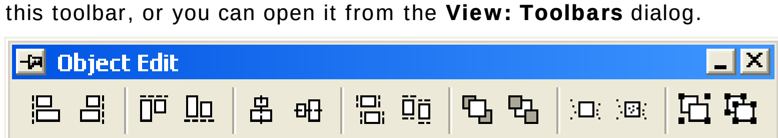


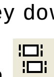
7. The two graph legends are the same in the output page. For the graph legend, Origin uses the worksheet column's **Comments** label row, or if that's empty it will use the **Long Name** if there is one, and if not, the Column name. To modify the legends, select **Graph: Update Legend** from the menu to open the legend update dialog. You will change the legends for the **Whole Page**, and **Reconstruct** them. Select **Custom** from **Auto Legend Translation Mode** and enter **@ws** in the **Legend Custom Format** edit box. Here, @ws means Origin will use the worksheet name as the graph legend.



Aligning Layers using the Object Edit Toolbar

1. Select **File:Open** and open the OPJ ***\Samples\Graphing\Layer Management.opj*** and go to the subfolder ***Arranging Layers***. (If you don't see the subfolders, click **View: Project Explorer** to open the Origin Project Explorer window.)
2. Now we want to use the **Object Edit** tools to rearrange the graph. Make sure you already opened this toolbar, or you can open it from the **View: Toolbars** dialog.



3. On the graph, hold the **Shift** key down and click on all four layers to select them as a group. Then press the **Uniform Width**  and **Uniform Height**  buttons on the **Object Edit** toolbar to make them the same height and width.
4. Click and select the bottom two layers and click the **Bottom**  button in the Object Edit toolbar to align them. Do the same thing for the top two layers, then click the **Top**  button to align them.

5. Now click and select the top and bottom on the left column, and do **Left** align , then repeat for the other two in the right side column by **Right** align button.

The object edit toolbar provides a quick way to align and set the size of layers. The first layer you select is the reference layer and all others will adjust according to that one. The **Layer Management** tool provides many more options, such as reordering and linking layers, in addition to setting size and alignment.

6. Do not save changes to your project, as the same project is used later to demonstrate Layer Management.

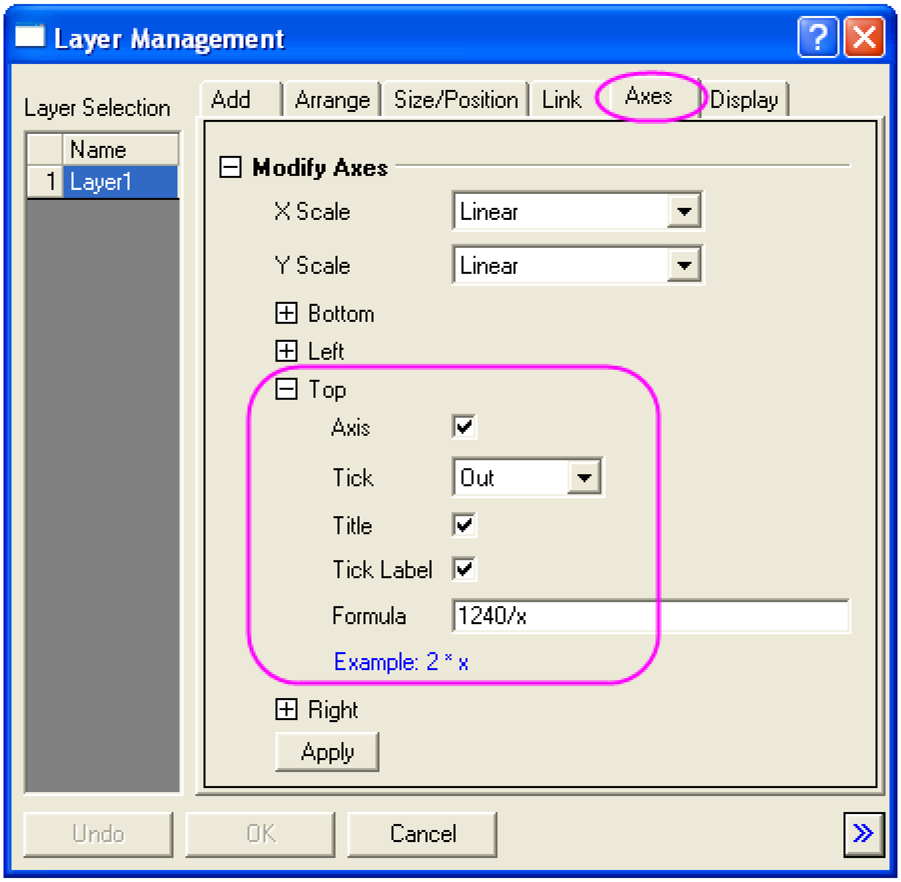
Displaying Opposite Axes with a Nonlinear Formula

An Origin 2D graph layer is a set of X, Y axes, and opposite axes can be turned on for both X and Y. In addition, the opposite axes can also display labels using any user-specified nonlinear formula with respect to the primary axis.

1. Using the Project Explorer window, switch to the subfolder named **Nonlinear Axis**.
2. With the graph active, bring up the Layer Management tool by selecting **Graph: Layer Management**.
3. Switch to the **Axes** tab and expand the **Top** branch and check the **Axis**, **Title**, and **Tick Label** check boxes.
4. We want to display the labels on the top axis in units of Energy and the relationship between wavelength and energy is:

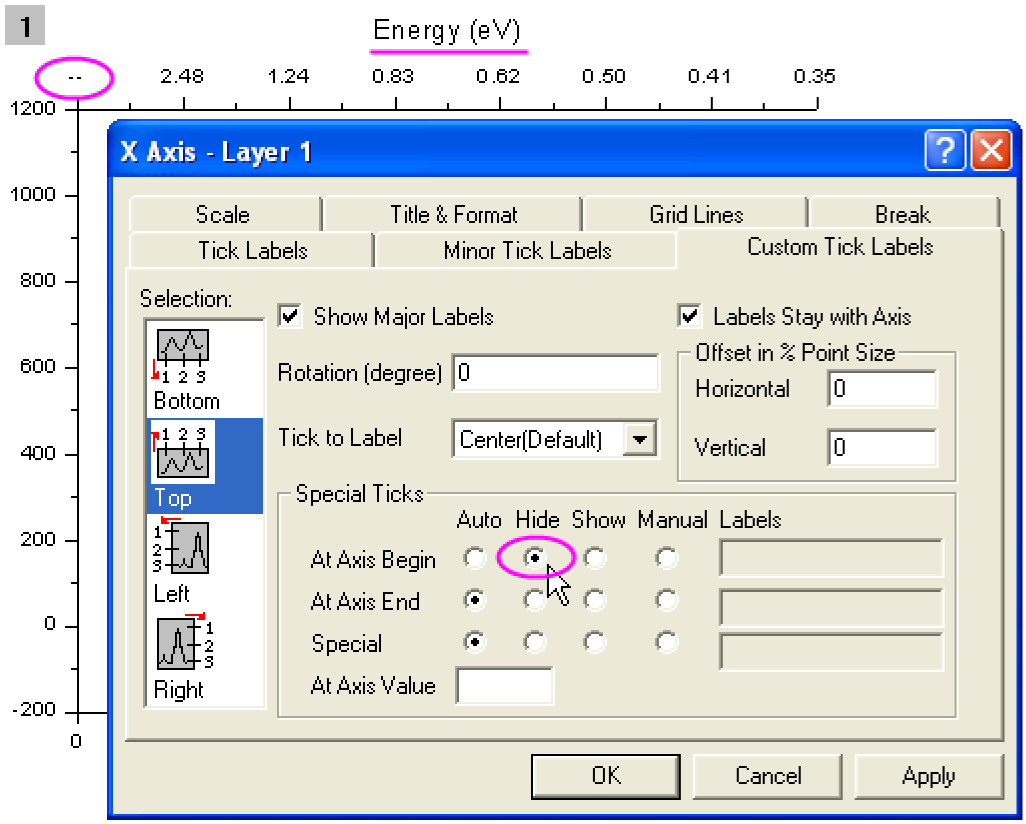
Energy (eV) = 1240/Wavelength (nm)

So in the **Formula** box, enter: 1240/x and click **Apply**, then click **OK** to close the dialog.



5. Double-click and edit the top X-axis title and change it to: Energy (eV)
6. Double-click the top axis labels or axis, to open the X-Axis dialog. Switch to the Custom Tick Labels tab, and then click Hide radio for At Axis Begin, to hide the missing value label for energy

that corresponds to zero wavelength.



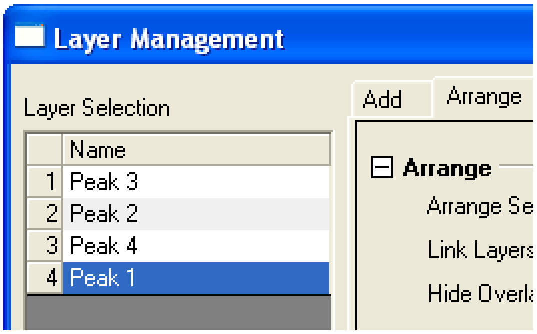
If you are setting up a nonlinear relationship for the right Y-axis, in the **Formula** you still need to use **x** instead of **y**.

Using Layer Management to Link and Position Layers

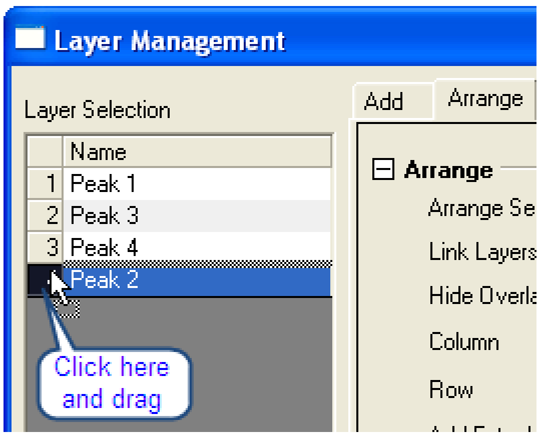
The Layer Management tool can be used to position, resize, swap, and link layers in order to establish the desired relationship between layers, as demonstrated in the following steps:

1. Reopen the project ***\Samples\Graphing\Layer Management.opj*** and then switch to the subfolder named ***Arranging Layers***. You can reopen the project by selecting **File: Recent Projects: Layer Management.opj**. Do not save changes to the project.
2. Bring up the Layer Management tool from **Graph: Layer Management**. Then go to left panel and rename the layers so that they are, top to bottom: Peak 3, Peak 2, Peak 4, Peak 1. The layer names now correspond to the legend for each layer. Note: to rename a layer, double-click on the

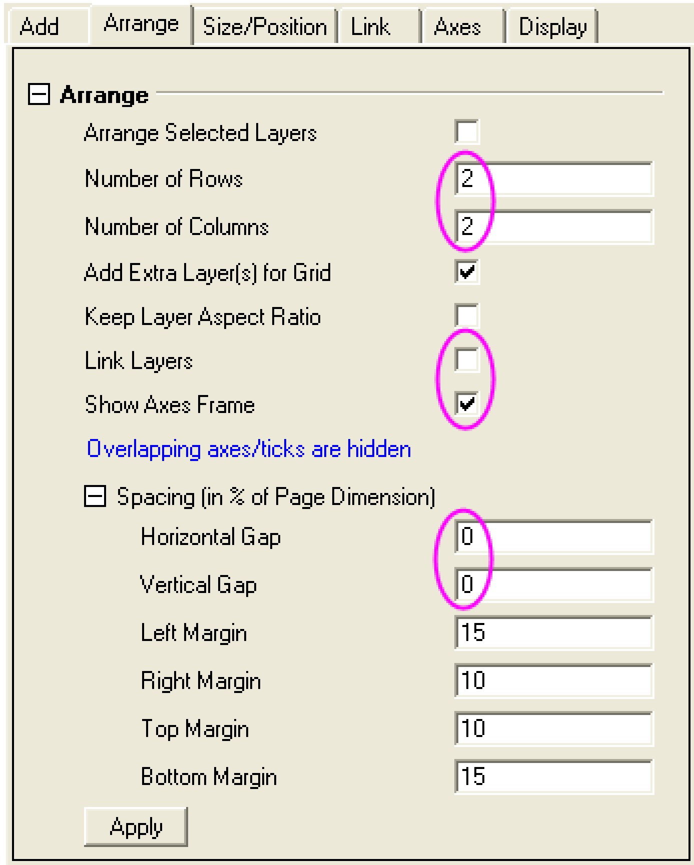
name, as the hint text in the dialog says.



3. Now drag and arrange the list on the left so that they are ordered Peak 1, Peak 2, Peak 3, Peak 4. The layer number and names now match.

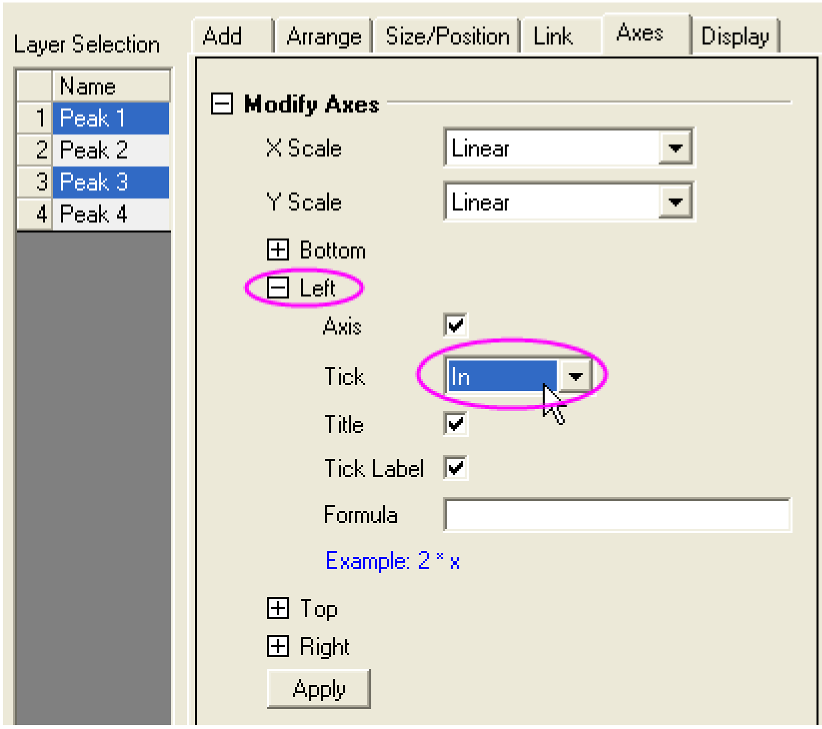


4. Now switch to the **Arrange** tab and do the following steps: 1) Uncheck the **Link Layers** check box. 2) Check **Show Axes Frame**. 3) Leave the **Number of Columns** and **Number of Rows** edit boxes as 2 by 2. 4) Set **Horizontal Gap** and **Vertical Gap** to 0.

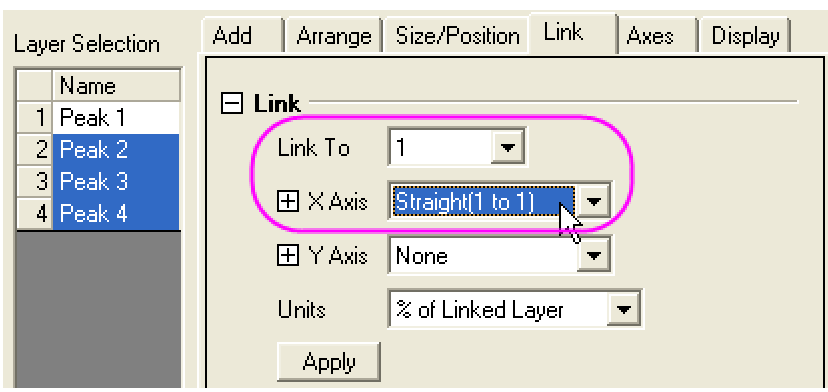


Click **Apply**. This resizes and repositions the layers so they are aligned, and hides ticks and labels where layers overlap.

5. Now go to the Axes tab, hold down the **Ctrl** key and select Peak 1 and 3 on left list, then expand the **Left** branch and set tick direction to **In**, and click **Apply**.

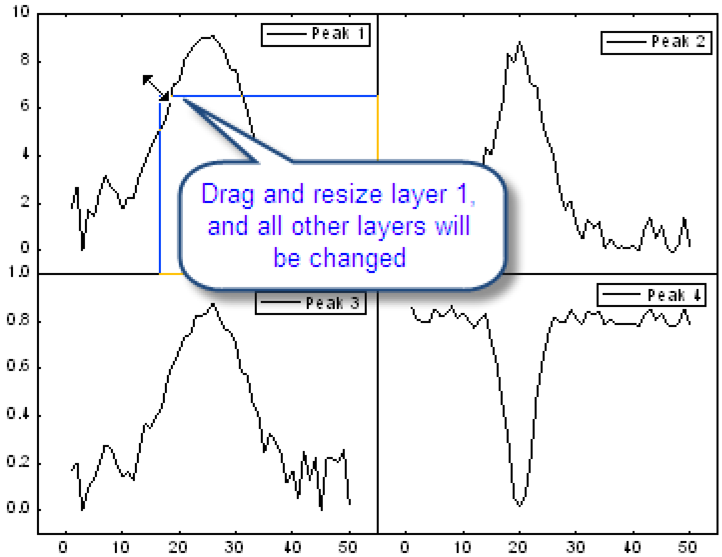


6. Now select Peaks 3 and 4, set the **Bottom** tick direction to **In** and click **Apply**. Then select Peaks 1 and 2, set the **Top** ticks to **In** and click **Apply**.
7. Now go to the Link tab and select Peaks 2, 3, and 4 in the left list and link them to layer 1, making sure you have the X-Axes linked **Straight(1 to 1)**, and click **Apply**.

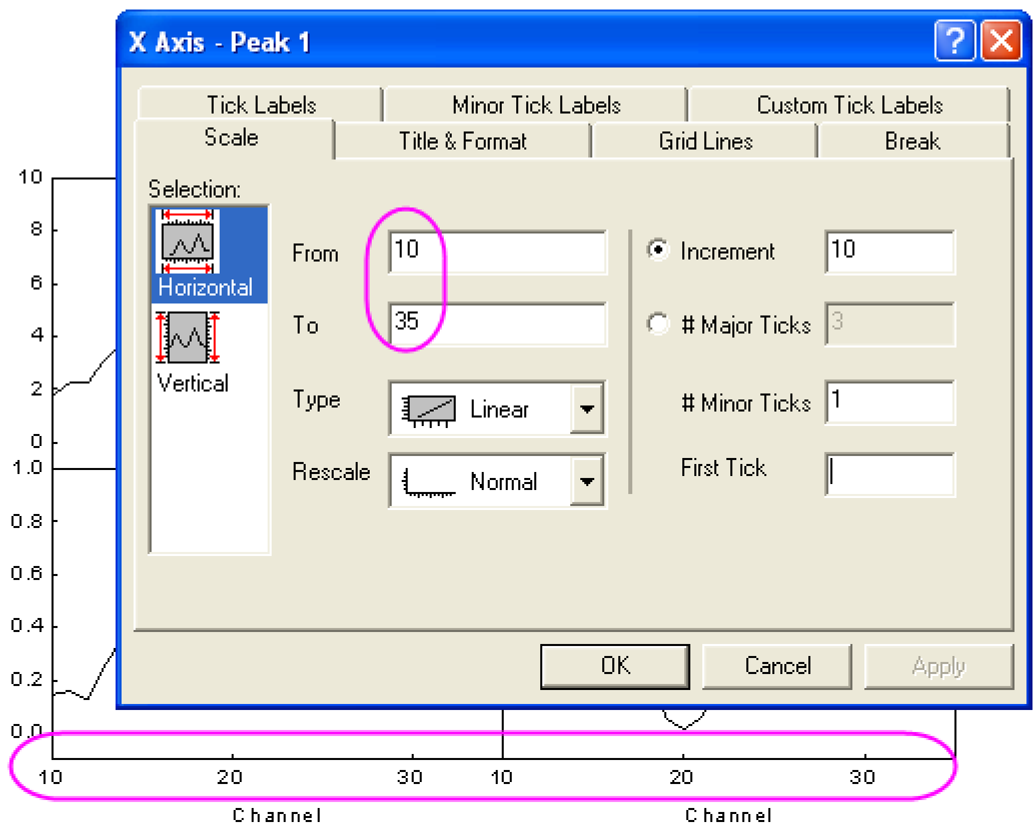


Then click **OK** to close the dialog.

8. Click and select the top left layer, layer 1, then drag and resize the layer and move the layer. Note that the other layers resize and move too, since they are linked by dimension to layer 1.



9. Go to the top left layer (layer 1), and double-click the top X-axis. In the X-Axis dialog, change the X-Axis scale From 10 To 35, and click **OK**. You'll see that all the other layers now show the same new x range, as they are all linked in X.



When arranging linked layers, they are treated as one unit, so you should first unlink them if you need to rearrange and then you can link them again after arranging.

6.6 Tick Labels

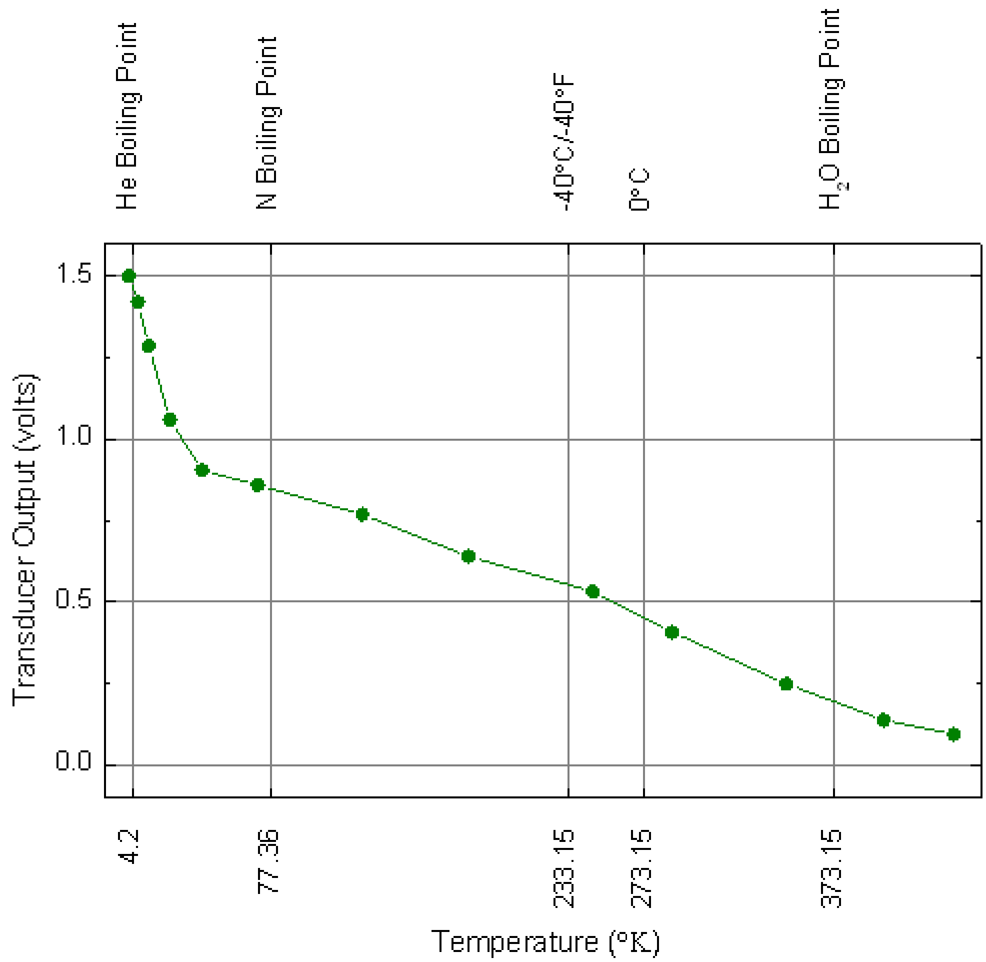
Topics covered in this section:

1. User-Defined Arbitrary Tick Positions
2. Plot and Customize Date Time Data on Graph

6.6.1 User-Defined Arbitrary Tick Positions

Summary

This graph will show you how to specify tick locations using a dataset and show customized tick labels at those locations.



Minimum Origin Version Required: Origin 8.5.1 SR0

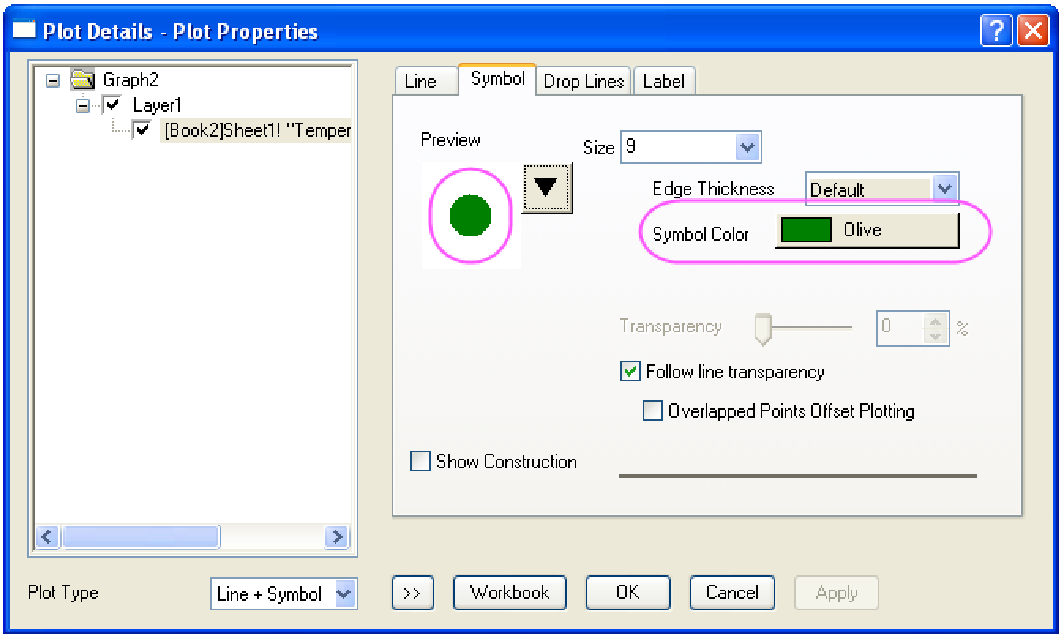
What you will learn

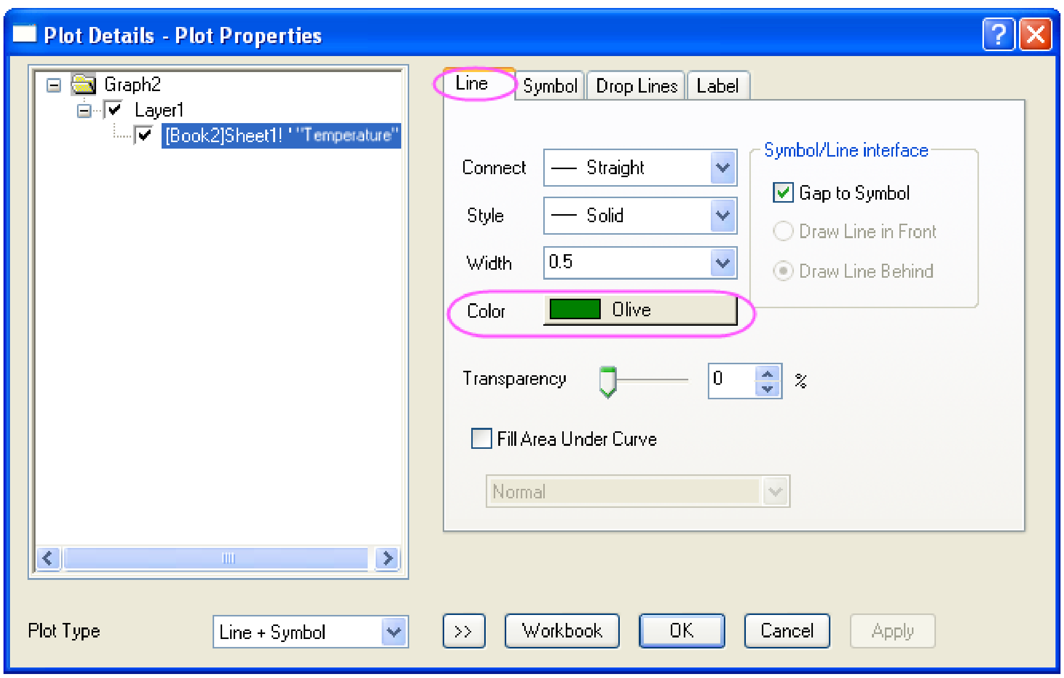
- How to specify the tick location using dataset
- How to show the customized tick labels

Steps

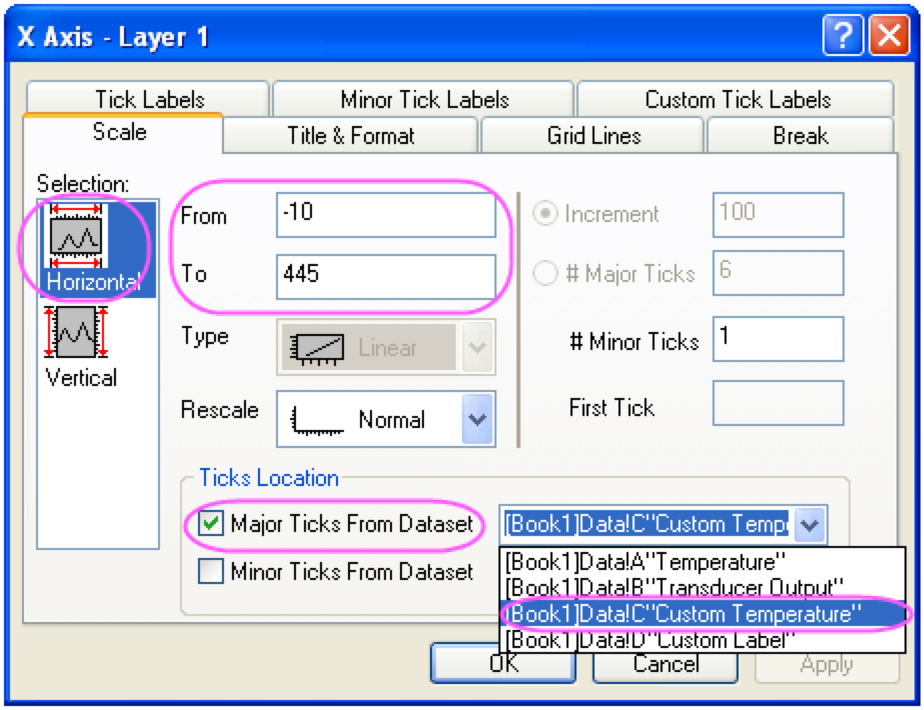
Copy the sample data, and paste it into Origin. Set the first row as Long Name and the second row as Units.

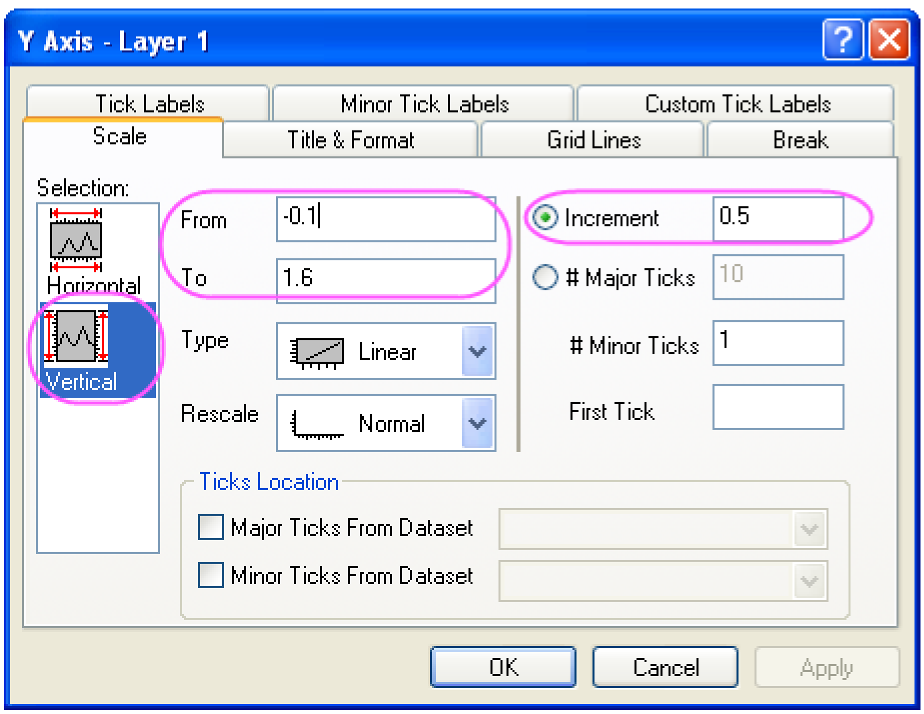
1. Highlight columns A and B. In the main menu, click Plot, then point to Line + Symbol, and then click Line + Symbol. Alternatively, you can simply click the Line + Symbol button on the 2D Graphs toolbar.
2. Double-click the graph to open the Plot Details dialog. On the Symbol tab, change the symbol to a solid circle, and the color to Olive. On the Line tab, change the line color to Olive and click OK to close the dialog and apply the changes.



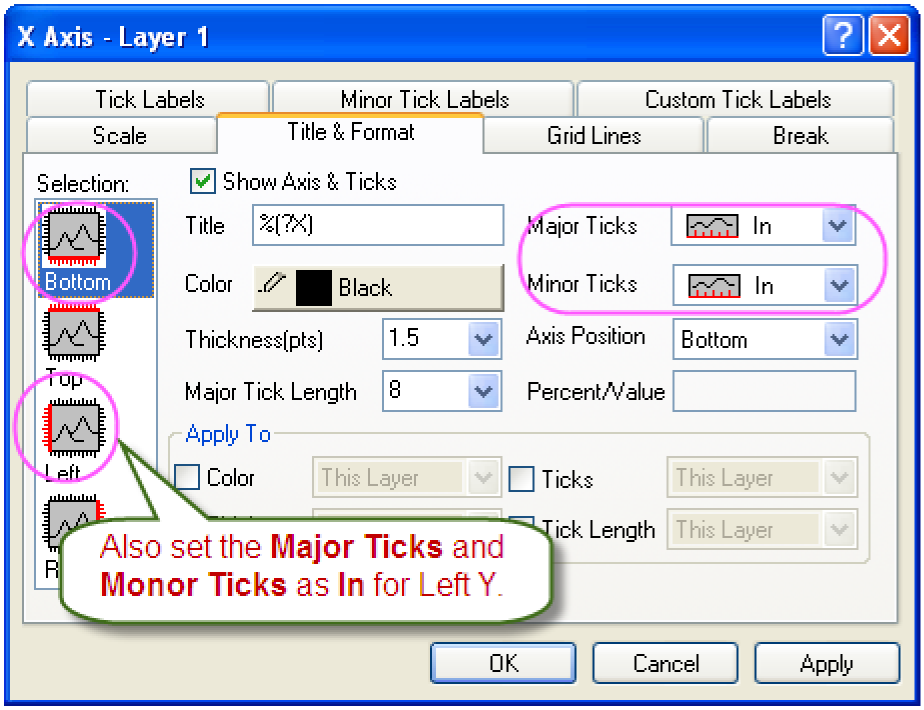


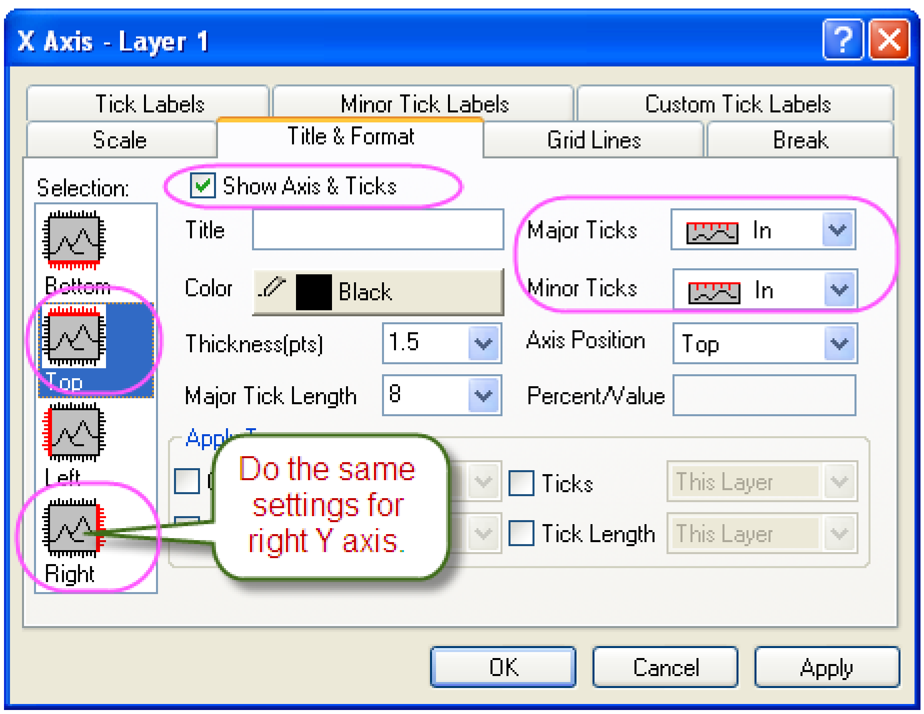
3. Double-click the X axis to open the Axis dialog. On the Scale tab, enter these settings for the X and Y axes:



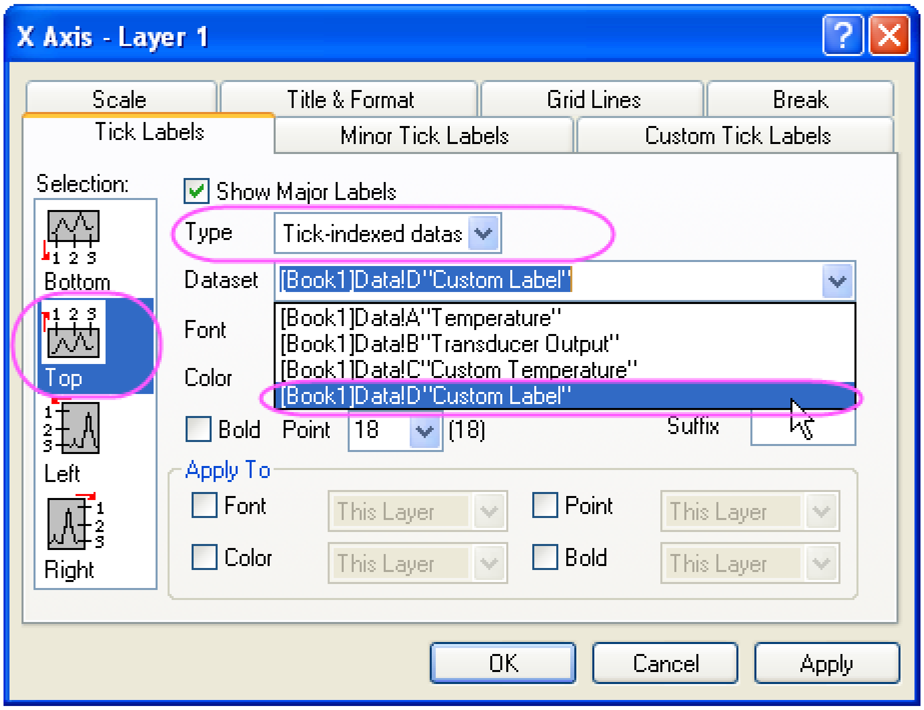


4. On the Title & Format tab, set Major Ticks and Minor Ticks to In for both Bottom and Left axes. Check Show Axis & Ticks for both Top and Right axis, and set Major Ticks and Minor Ticks to In.

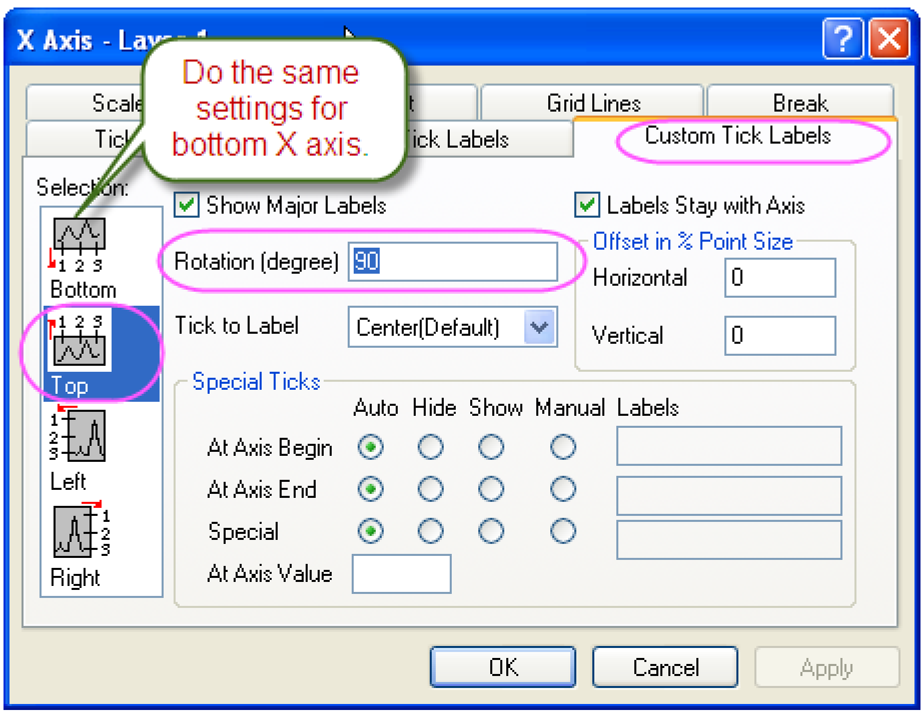




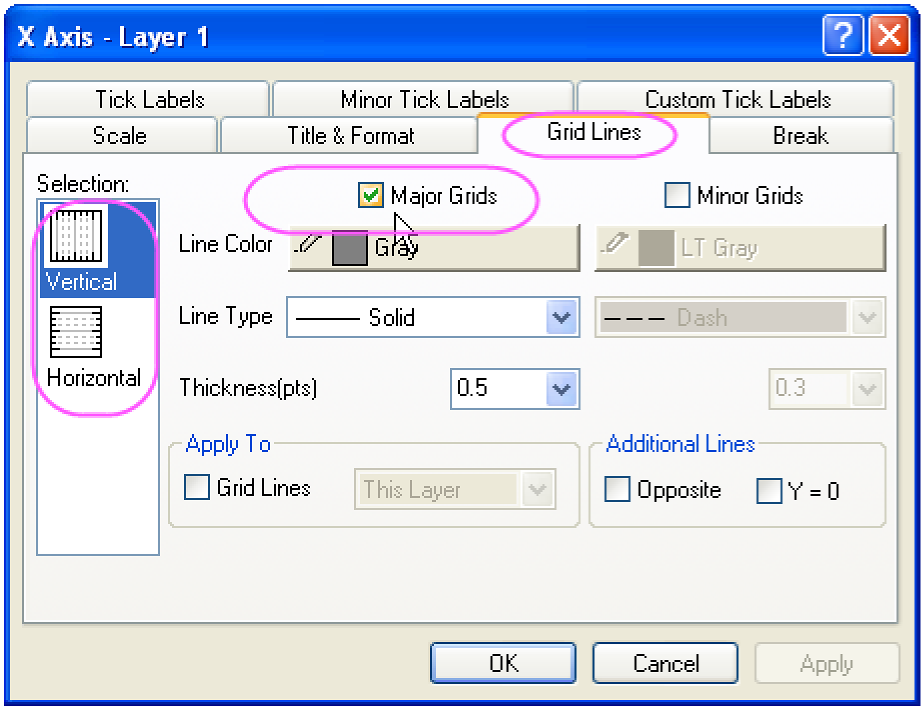
5. On the Tick Labels tab, select Top in the Selection box. Select Tick-indexed dataset from the Type menu and column D from the Dataset menu.



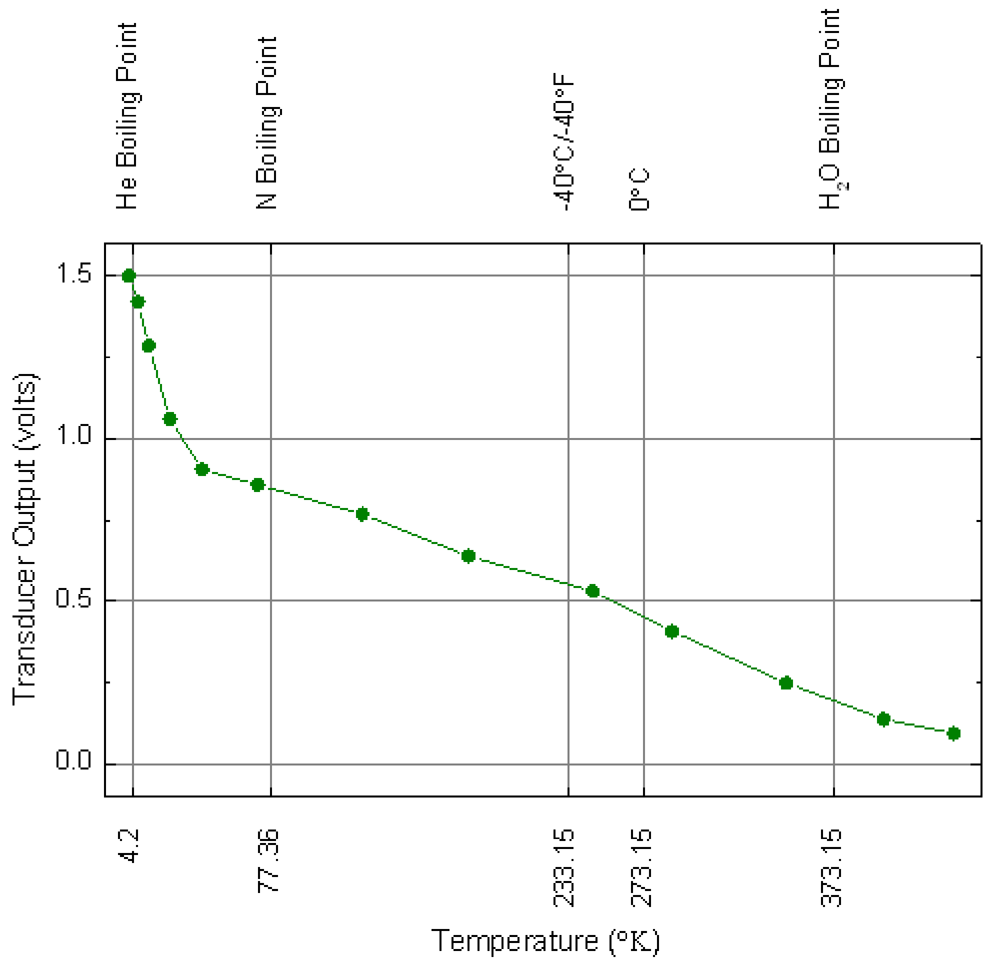
6. On the Custom Tick Labels tab, set Rotation(degree) to 90 for both Bottom and Top axes.



7. On the Grid Lines tab, activate Major Grids for both Vertical and Horizontal.



8. Click OK to close the dialog. Remove the legend from the graph. Your final graph should look like this:



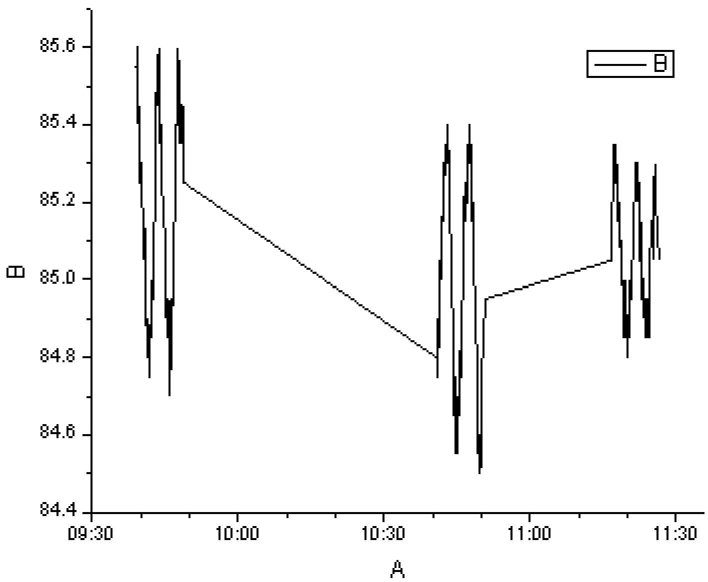
Sample Data

Temperature	Transducer Output	Custom Temperature	Custom Label
°K	volts		
2.5	1.5	4.2	He Boiling Point
2.5	1.5	4.2	He Boiling Point
7.3	1.42048	77.36	N Boiling Point
13	1.28681	233.15	-40°C/-40°F
24	1.06011	273.15	0°C
41	0.90549	373.15	H_2O Boiling Point
70	0.85831		
125	0.7679		
181	0.63948		
246	0.53202		
288	0.40753		
348	0.24898		
399	0.13759		
436	0.09435		

6.6.2 Plot and Customize Date Time Data on Graph

Summary

This tutorial will show you how to manipulate the date and time data, and customize the date and time data on the graph.



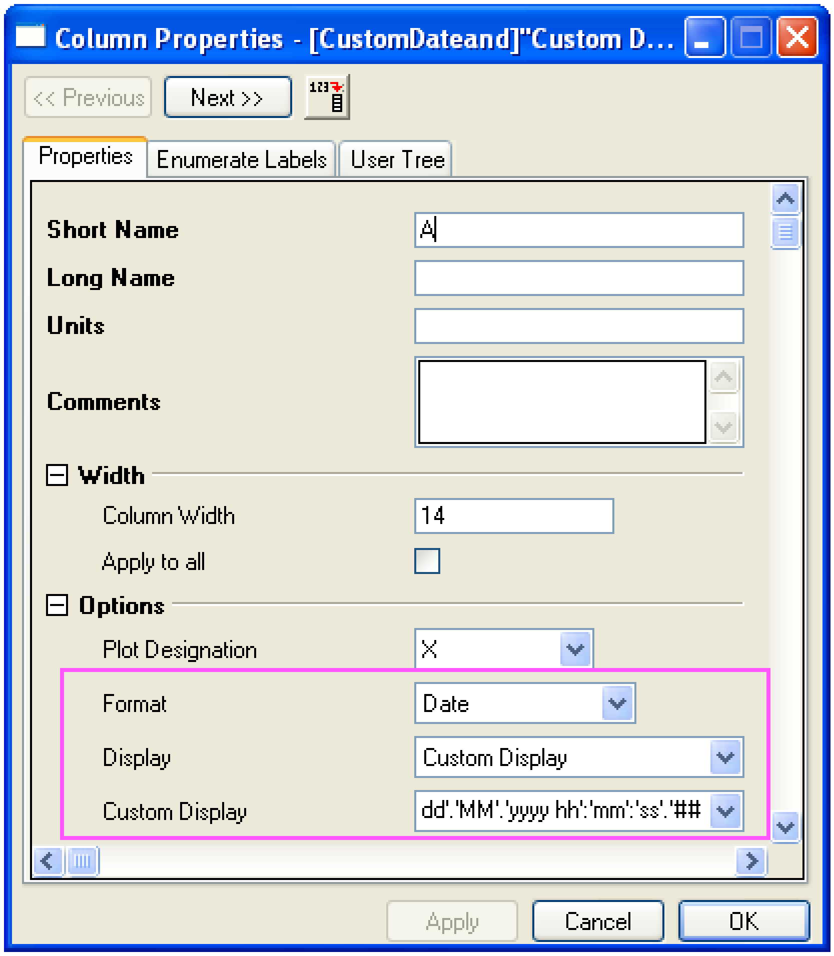
Minimum Origin Version Required: Origin 8.0 SR6

What you will learn

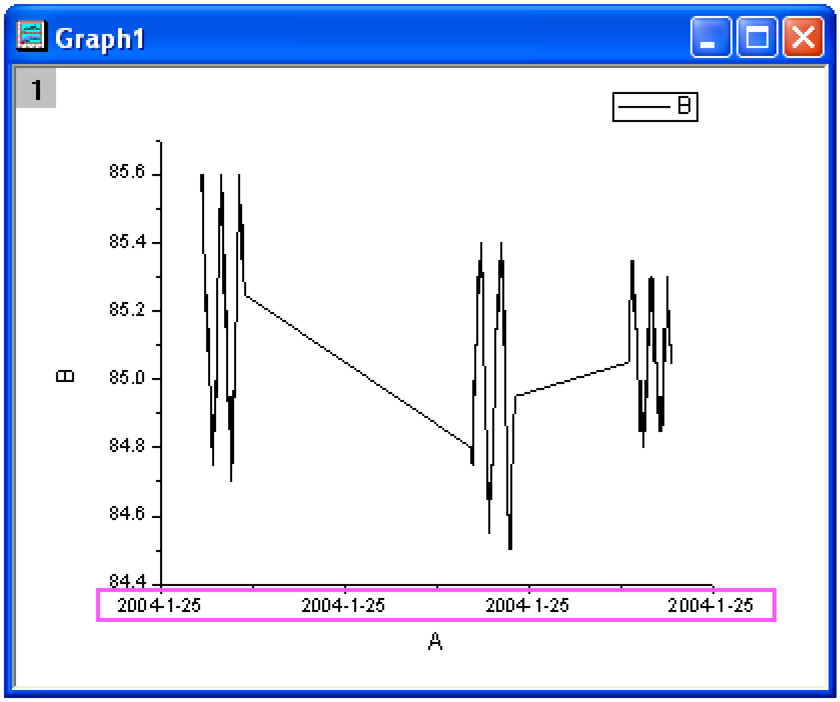
- How to plot the date and time data on the graph.
- How to customize the date and time tick labels.

Steps

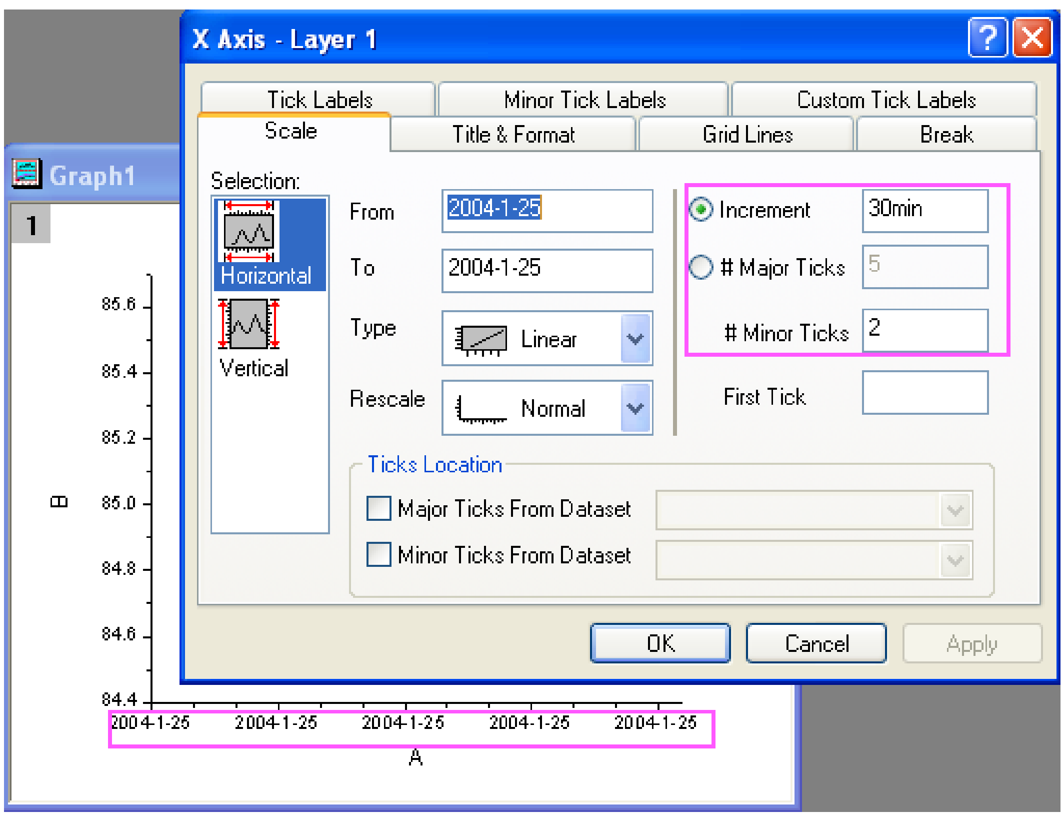
1. Start with a new project and import the data file *Sample\Import and Export\Custom Date and Time.dat* under Origin's program folder.
2. Double-click on A(X) to open the **Column Properties** dialog; set **Format** as *Date*, **Display** as *Custom Display*, and input *dd'.'MM'.'yyyy hh':'mm':'ss'.'##* in the **Custom Display** box as shown below.



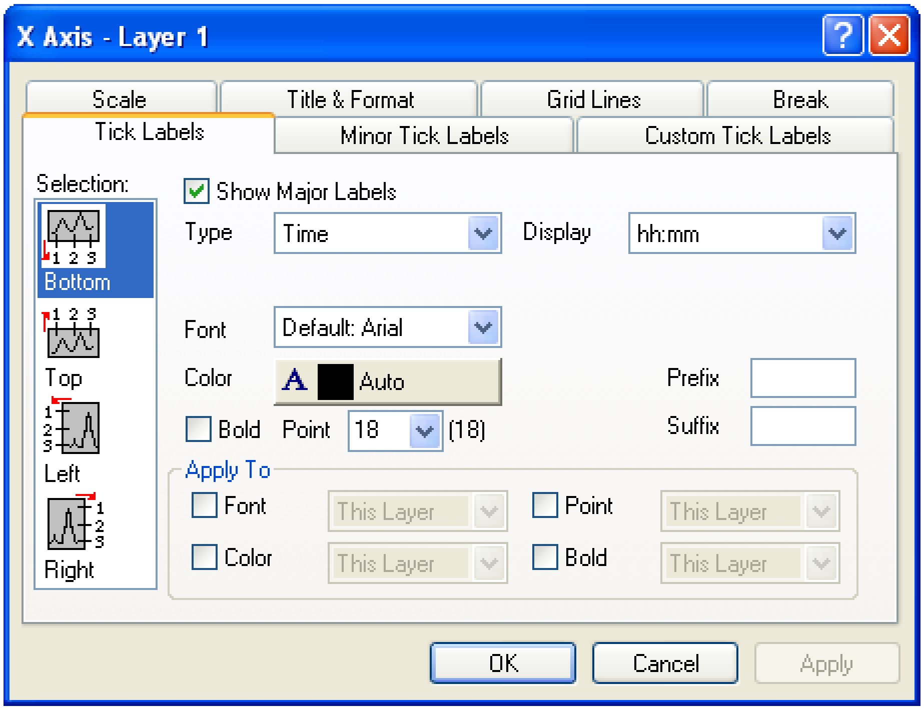
3. Click **OK** button to go back to the worksheet. Highlight col(B) and select **Plot: Line: Line** from the main menu to plot it as a line graph. You will see that the tick labels of X axis are all the same.



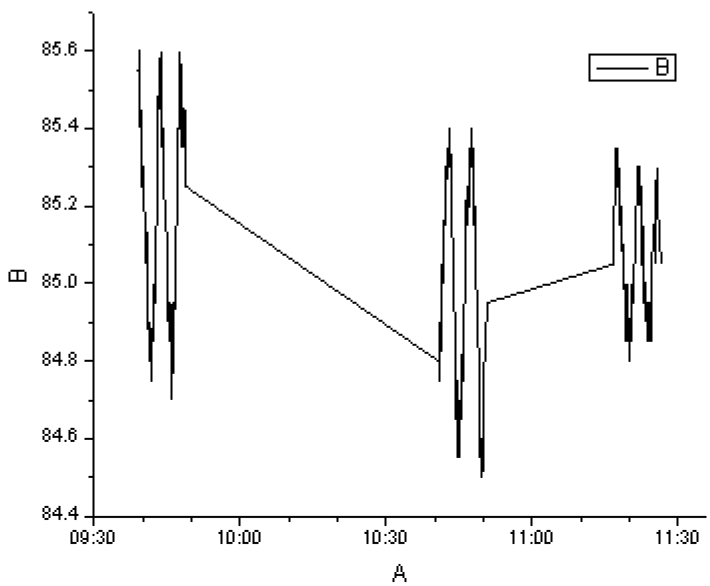
4. Double-click on the X axis (or select **Format : Axes : X Axis** from the main menu) to open the **Axis** dialog. In the **Scale** tab, change **Increment** to *30min* and **Minor Ticks** to *2*. Click the **Apply** button, you will see that the tick labels are still all the same and make no sense.



5. Go to the **Tick Labels** tab in the **Axis** dialog, change **Type** to *Time* and select **Display** to *hh:mm* as shown below.



6. Click **OK** button, you will see that tick labels now show as the corresponding time.



7 Importing Data

Topics covered in this section:

7.1 ASCII

- Single ASCII
- Import Wizard
- Import Time Data
- Post Processing with Import Filter

7.1.1 Single ASCII

Summary

The **File: Import: Single ASCII** menu allows you to automatically import a single ASCII file where the data columns are delimited orderly and it consists of few header lines (maybe just a short description for the file and then names and units for the columns).

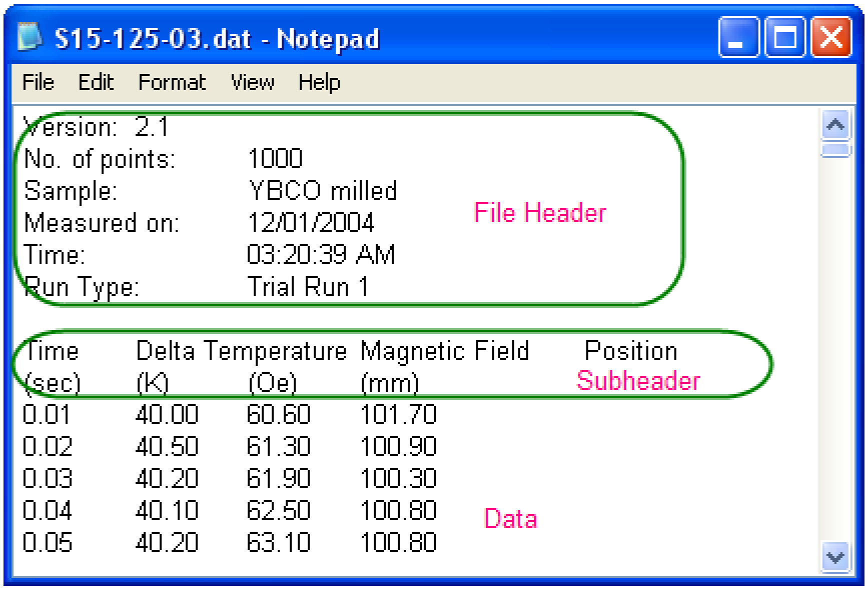
Minimum Origin Version Required: Origin 8.0 SR6

What you will learn

This tutorial will show you how to Import ASCII files.

Steps

Using Windows Explorer, browse to the *Samples**Import and Export* subfolder of the Origin program folder (by default installed in the Program Files folder). Open the file *S15-125-03.dat* in Windows Notepad. You can see that this file includes header lines and data lines. For Single ASCII files, Origin can auto detect file header/subheader and extract this information to the worksheet headers, such as *Long Name*, *Units*, etc.

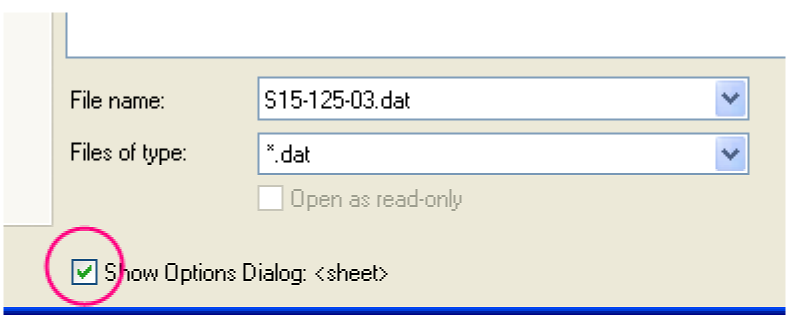


Note: Header lines are lines of text that are not part of the data and do not share the same delimiter formatting as the data. Subheader lines also are not part of the data, but share the same delimiter formatting and therefore correspond to particular columns of data.

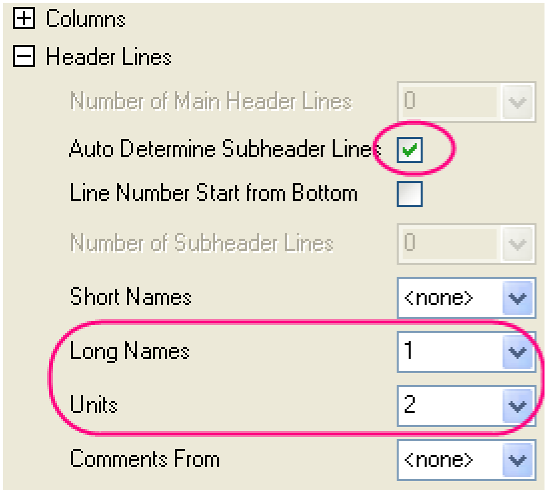
To Import this file

1. Select **File: Import: Single ASCII** from the menu to open the File Import dialog. Browse to the *\Samples\Import and Export* subfolder of the Origin program folder. Highlight the file *S15-125-03.dat*.

2. If you double-click this file or click the **Open** button, Origin will import the file automatically. To view the settings for how Origin will import the file, check the **Show Options Dialog** checkbox at the bottom of the dialog and then click **Open**. This will bring up the *impASC* X-Function dialog.



3. Expand the **Import Options: Header Lines** tree node.



By default, Origin auto detects the subheader, and data will be imported from subheader. In this example, Origin automatically set the first line,

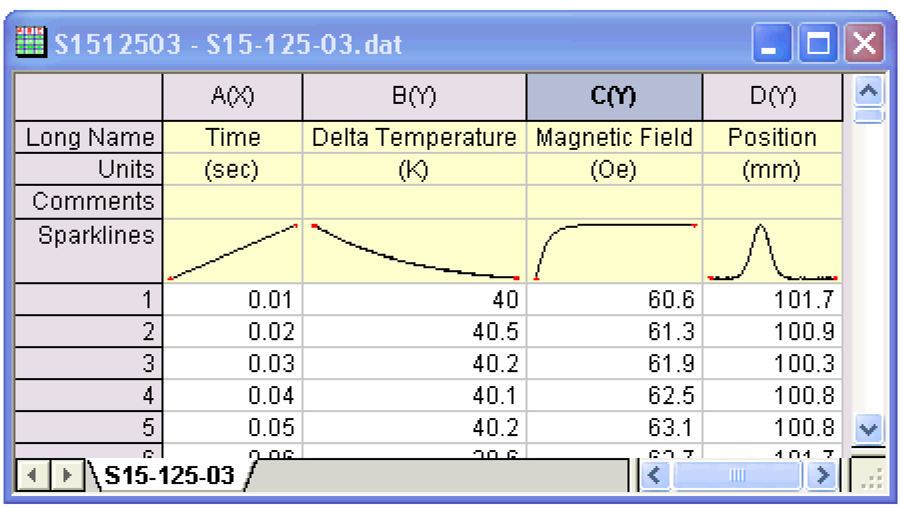
```
Time Delta Temperature Magnetic Field Position
```

as the worksheet column Long Name and the second line,

```
(sec) (K) (Oe) (mm)
```

as Units.

4. Click **OK** to accept these settings and import data into worksheet.



7.1.2 Import Wizard

Summary

The **Import Wizard** allows you to preview your ASCII file. This is especially helpful when you are importing a file with many lines of header, and want to extract variables from both the file name and file headers for use in annotating a graph.

Minimum Origin Version Required: Origin 8.0 SR6

What you will learn

This tutorial will show you how to use the **Import Wizard**.

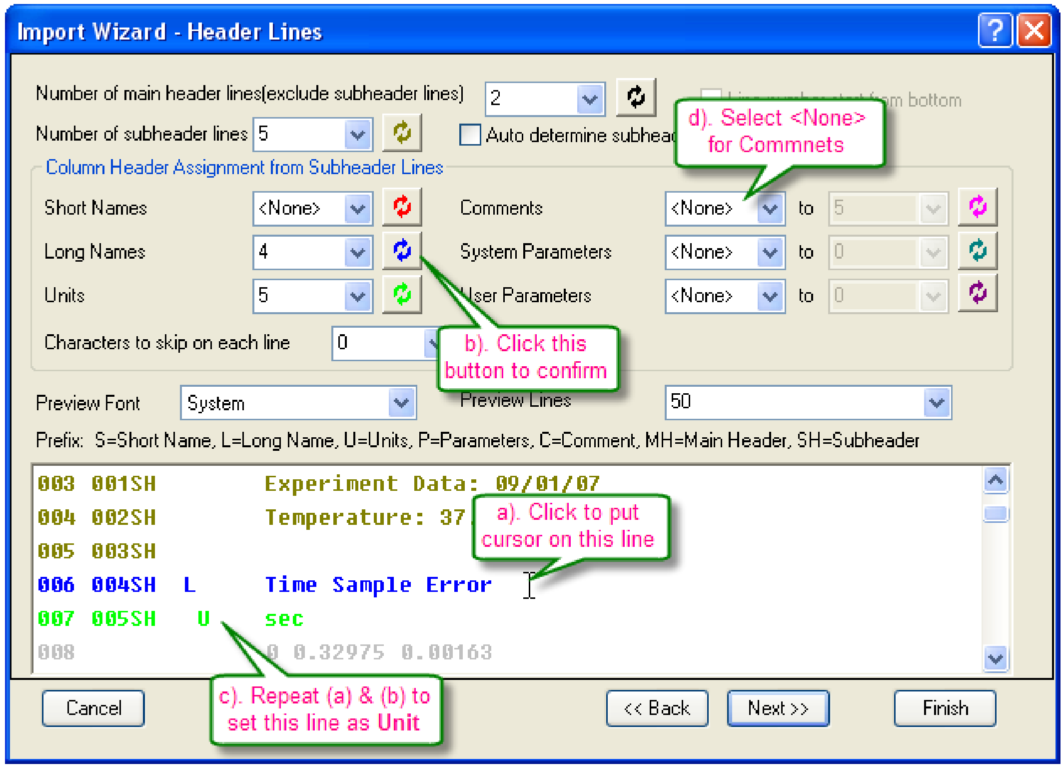
Steps

The **Import Wizard** allows you to import complicated ASCII files, extract variables from the import file name and header (for reuse in Origin), specify custom delimiters and date formats, or handle post-processing of your imported data using a custom LabTalk script. Another chief advantage of using the Wizard to import your data files is that you can save your custom settings to a filter that can be used repeatedly to import the same or similarly-structured files.

1. Create a new workbook. Click **File**, point to **Import**, and then click **Import Wizard** to open the *Import Wizard* dialog. Click the browser button  to the right of the **File** box. Browse to the *\Samples\Import and Export* folder and open the file *F1.dat*. Click **Next** to navigate to **File Name Options** page.

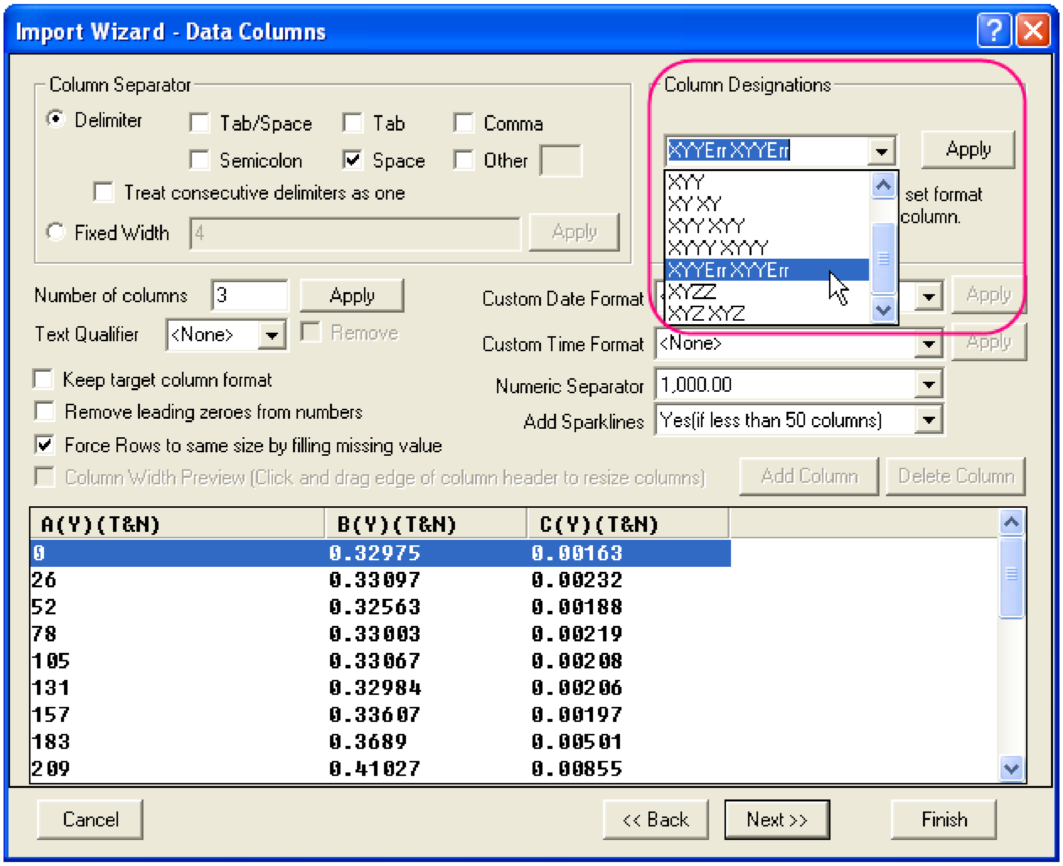
2. Make sure the **worksheet with file name** box is checked to rename worksheet by imported file name. Click **Next** to navigate to **Header Lines** page.

3. This page enables you to easily customize the worksheet headers. For example, to specify the worksheet long name, place your cursor on the relevant line in the lower panel, and then click the button next to *Long Names*.

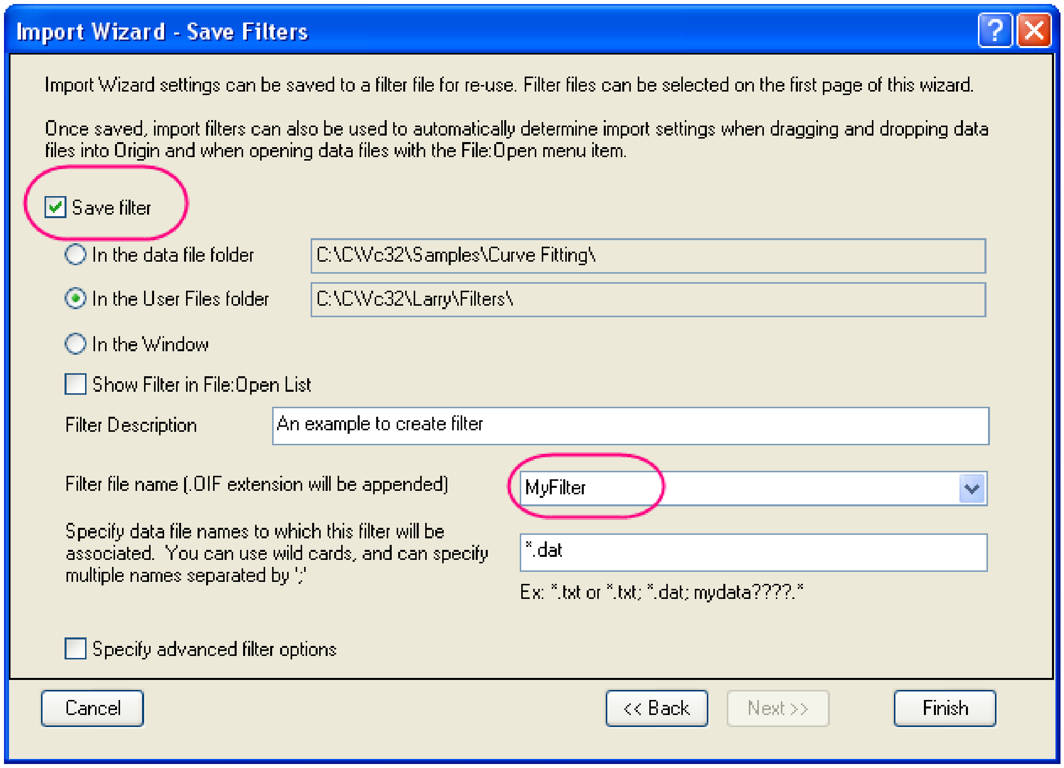


Similarly, specify the *Unit* line and select *<None>* for *Comments*.

4. Click **Next** twice to go to the **Data Columns** page. Select **XYYErrXYYErr** from the *Column Designations* drop-down menu and click **Apply**.



5. Click **Next** twice to go to the **Save Filters** page. To use these settings again, you can save this import procedure as a filter. Select the **Save filter** check box and type a filter name in the **Filter file name** text box (*MyFilter* in this example).



Click **Finish** to import the data.

7.1.3 Import Time Data

Summary

Origin interprets *Dates* based upon the Gregorian Calendar, while *Time* is interpreted in *hours*:*minutes*:*seconds*. When working with Date and Time data, Origin displays these data in different formats, but internally uses underlying numeric values for calculations and certainly plotting operations. This tutorial shows you how to import custom date/time data.

Minimum Origin Version Required: Origin 8.0 SR3

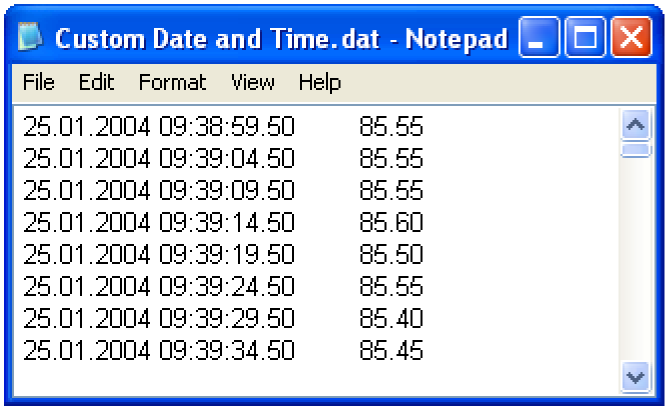
What you will learn

This tutorial covers:

- How to import data using multiple delimiters.
- How to define a custom date/time format
- How to change the display settings for the custom date/time format

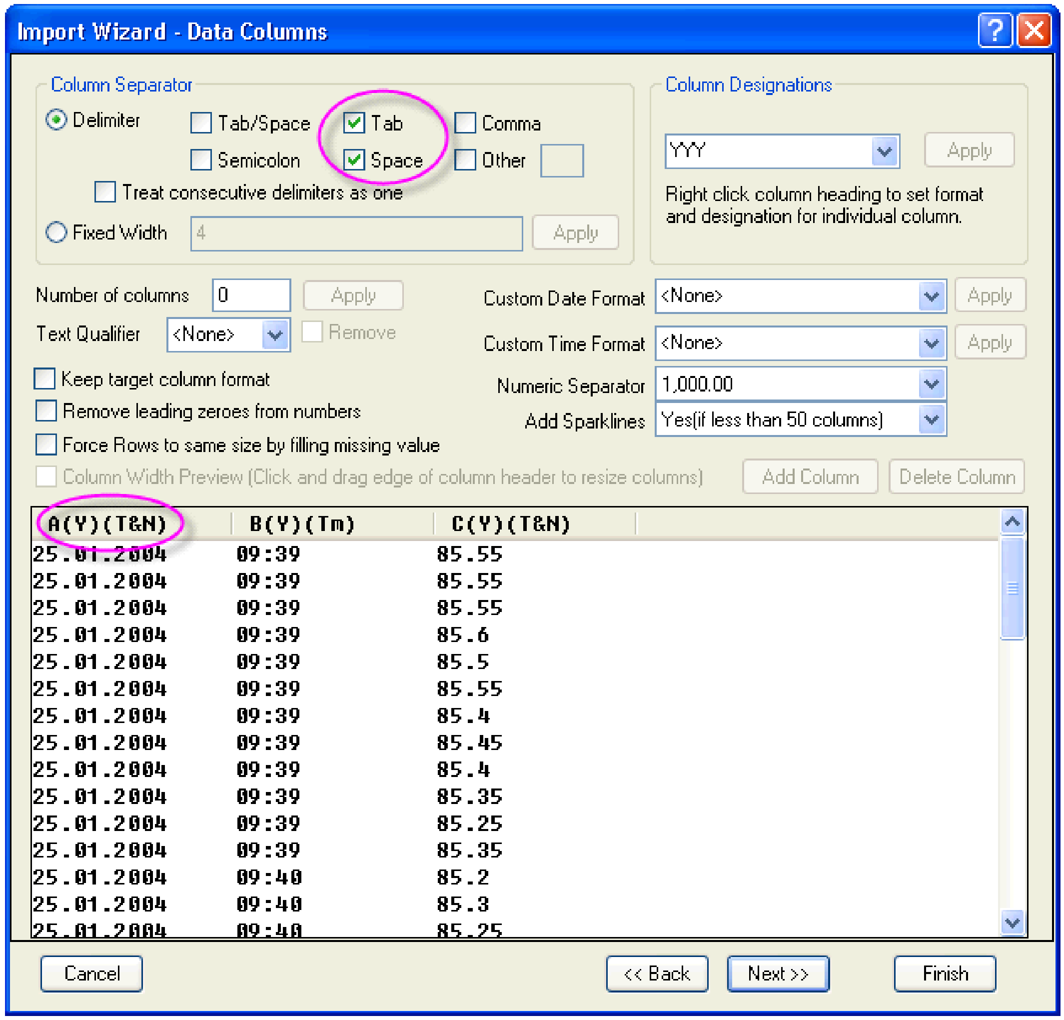
Steps

1. We will import *\Samples\Import and Export\Custom Date and Time.dat* in this tutorial. Before importing the file, let's look at the data structure first.



We can see that there is a space between *Date* and *Time*, and it uses Tab to separate *Time* from the rest of the data. So we will use multiple delimiters to import this data file.

2. Open the file in **Import Wizard**. Accept the default settings on all pages until you get to the **Data Columns** page. Origin will, by default, use Tab to separate the data into two columns. To divide *Date* and *Time*, check the **Tab** and **Space** checkbox in the **Column Separator** group.

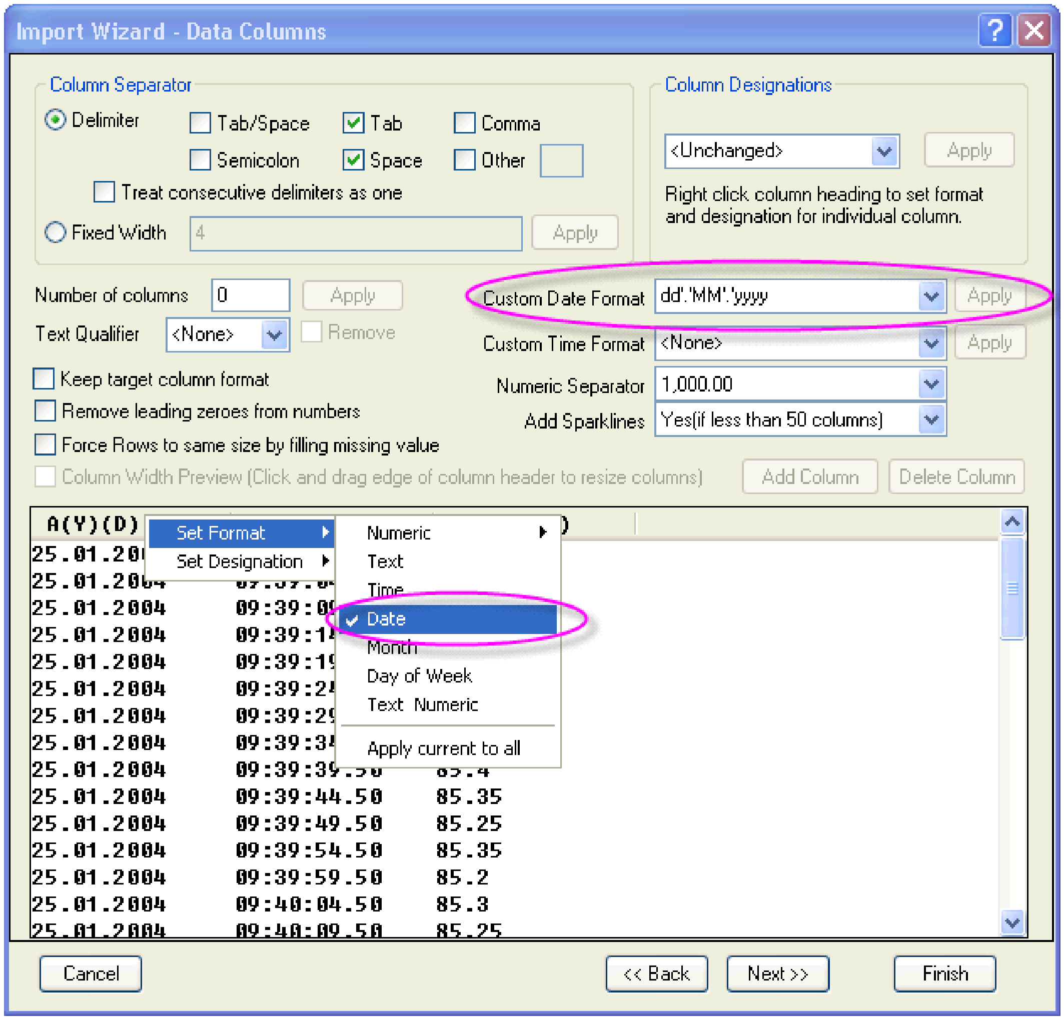


Note in the preview box that the column title is **A(Y)(T&N)**, where **(T&N)** means the data format is *Text & Numeric*. Because the date uses "." to separate day, month and year, Origin by default treats the first column as *Text*. For the second *Time* column, Origin shows the underlying numeric values. To import data correctly, we should change the column properties.

3. In the **Custom Date Format** edit box, enter:

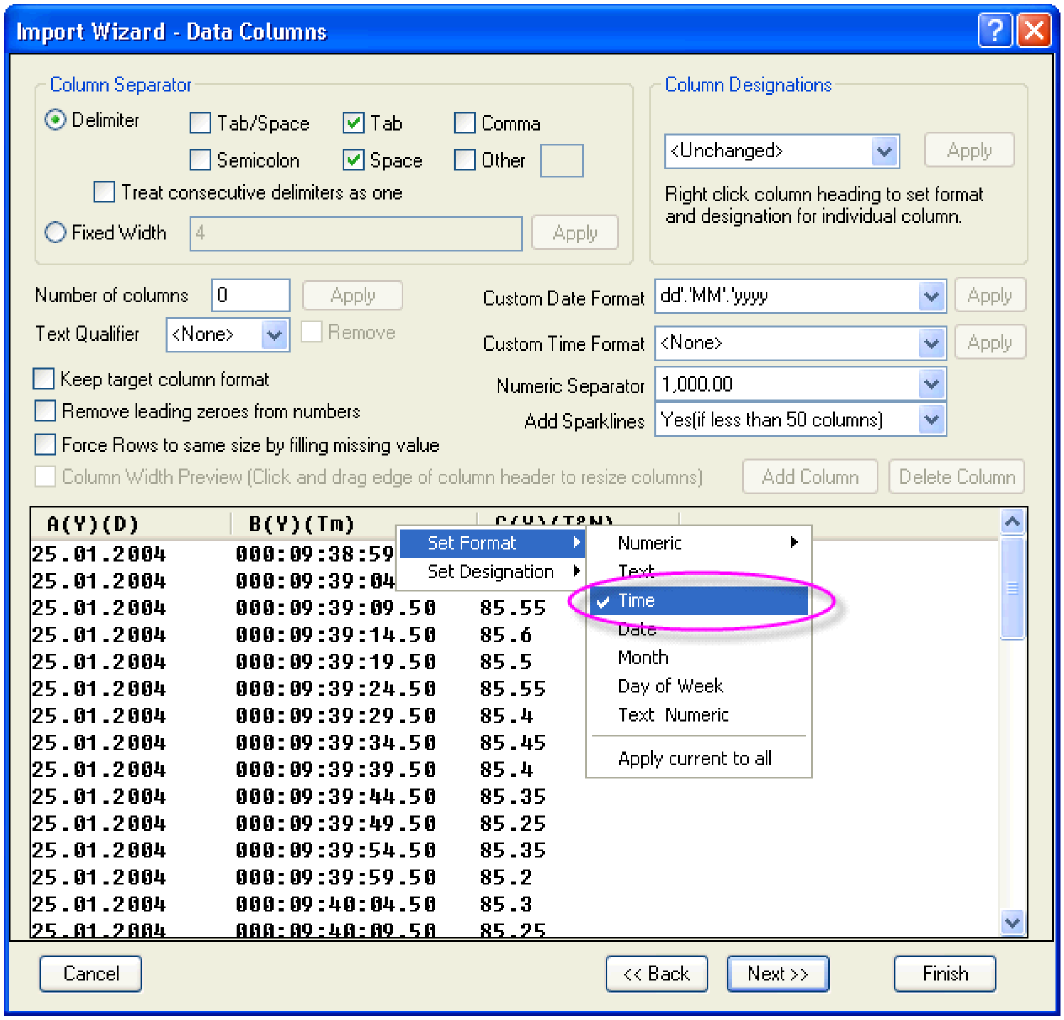
```
dd'.'MM'.'yyyy
```

where *dd*, *MM* and *yyyy* mean the days, months, and year respectively. Since the "."is used as a separator, we need to put single quotation marks around it in the format specification. After entering the custom format, press the **Apply** button next to the edit box. Then right-click on the header of the first column in the preview and select *Date* from the context menu:



Then the column title will turn into **A(Y)(D)** which means this is now a *Date* data column.

4. Similarly, right-click on the header of the second column and select *Time* to set that column as a time column:

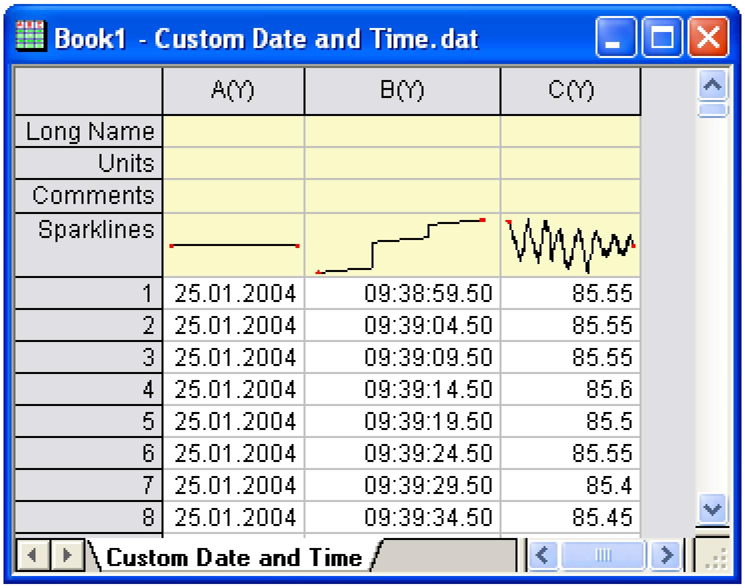


You can see the time data display in long format. We can change the display setting after imported.

5. Click **Finish** button to import data. Then double-click the second column title to open the **Column Properties** dialog, and set the Time display as:

HH:mm:ss.##

The final worksheet data after imported will looks like:



	A(Y)	B(Y)	C(Y)
Long Name			
Units			
Comments			
Sparklines			
1	25.01.2004	09:38:59.50	85.55
2	25.01.2004	09:39:04.50	85.55
3	25.01.2004	09:39:09.50	85.55
4	25.01.2004	09:39:14.50	85.6
5	25.01.2004	09:39:19.50	85.5
6	25.01.2004	09:39:24.50	85.55
7	25.01.2004	09:39:29.50	85.4
8	25.01.2004	09:39:34.50	85.45

Note: In the case of this particular data file, the first column simply has the exact same date in every row. So at this point you may want to set this first column as Disregard by right-clicking on the column header and selecting *Disregard* from the context menu. Then you can set the 2nd column as type X, and plot the data in the third column against the time data in the 2nd column.

7.1.4 Post Processing with Import Filter

Summary

The Import Wizard allows defining a custom filter to import ASCII and simple binary files. The filter can then be reused with similar data files once created. The filter mechanism also allows including LabTalk script lines that will then be run at the end of the import. This capability allows user to add post-processing script code to the filter.

Minimum Origin Version Required: Origin 8.0 SR6

What you will learn

- How to add post processing script to existing import wizard filter

Steps

1. Start a new workbook by clicking the **New Workbook** button on the Standard toolbar.
2. Click the **Import Wizard** button on the Standard toolbar to open the wizard.
3. In the **Data Source** group, click the button to the right of **File** and navigate to and add the file **Samples\Import and Export\S15-125-03.dat**.
4. A suitable filter for importing this file already exists in the data folder. Click the **Next** button to proceed thru all the pages of the wizard till you get to the **Save Filters** page.
5. On this page, check the **Save Filter** check box and also the **Specify advanced filter options** check box and then click **Next**. This will bring you to the **Advanced Options** page.

6. On this page, copy and paste the following lines in the edit box:

```
nlbegin iy:=(1,4) func:=gauss;
nlfit;
nlend output:=1 autoupdate:=au_auto;
```

7. Press **Finish**. This will save the filter along with these added lines of script, and the file will be imported and the script will run. The workbook will then have three sheets, which will include the custom report sheet and the fitted curve sheet, which are results of gaussian function fit to column 4 of the imported data.
8. Start a new project and open import wizard again and add all three files **S15_125_03.dat, S21-235-07, S32-014-04** in the file open dialog.
9. Check to see that the **Import mode** drop-down is set to **Start New Books** and click Finish. Your modified filter will be used and after each file is imported, the 4th column will be fit with the gaussian function.

7.2 Importing Data

7.2.1 Summary

Origin provides flexible ways of importing data, including simply dragging and dropping data files, using the ASCII import dialog to customize settings, and using the Import Wizard for advanced customization, extracting variables from header lines, and supporting custom file formats for many third-party files. This tutorial will highlight some of these features.

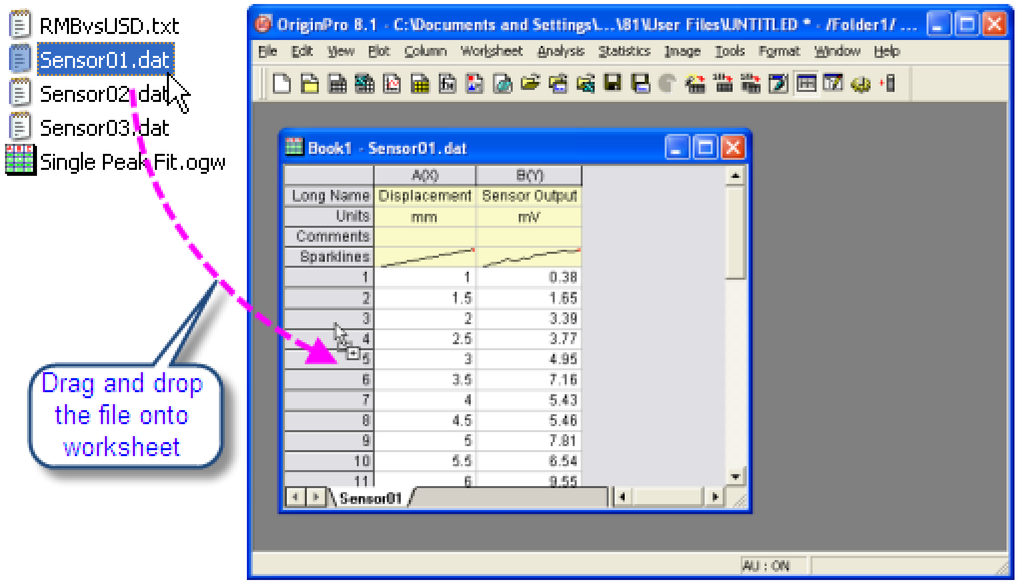
This tutorial will show you how to:

- Import files by drag-and-drop
- Import multiple ASCII files by customizing settings
- Save settings for future use
- Use the Import Wizard and import filters

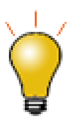
7.2.2 Drag-and-Drop Importing of ASCII Files

1. Start with an empty worksheet. Open Windows Explorer to the ***\Samples\Curve Fitting*** subfolder of your Origin system folder. Drag and drop the file **sensor01.dat** from the Windows

Explorer window into the empty Origin worksheet.



2. Data appears in the sheet. Now select two files: **sensor02.dat** and **sensor03.dat** and drag-and-drop them into the same worksheet. You will see that the first file replaces the data already in the worksheet, and the other creates a new workbook, as the default setting is to create new books, starting with the second file.



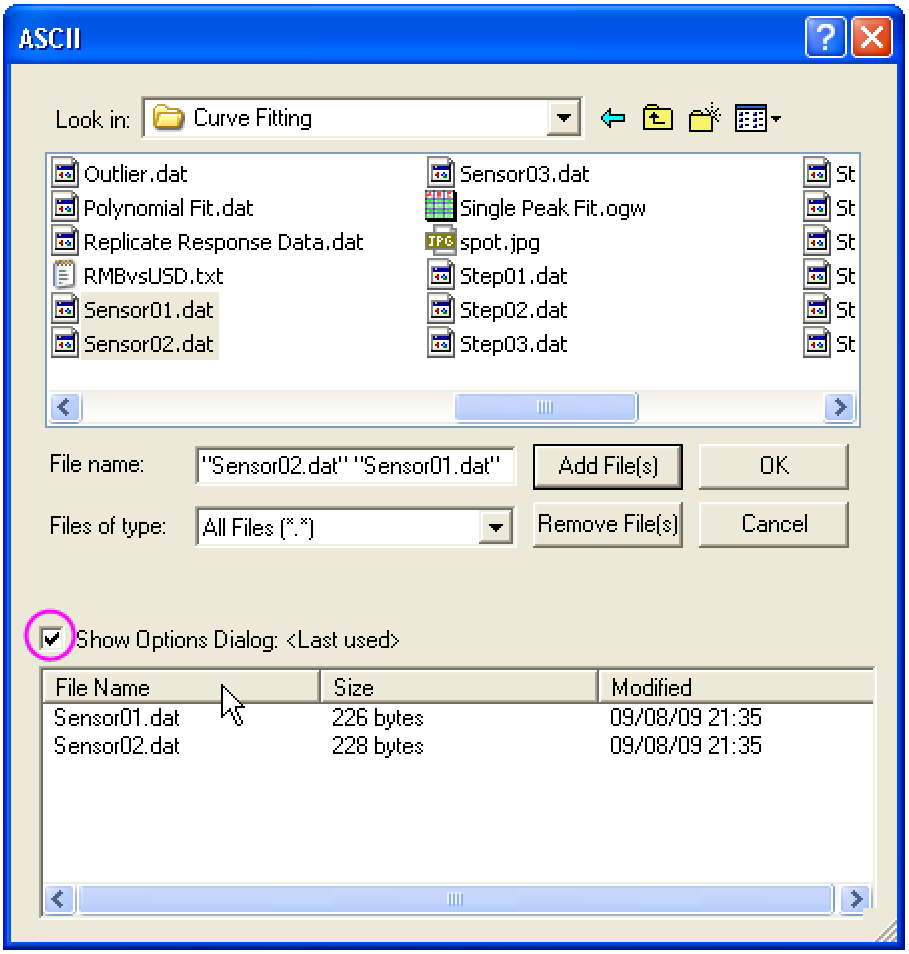
The default setting when dragging and dropping is to replace existing data. If you have some other data already in the sheet, you can drop the file onto the gray area outside of any window, or into a graph window, and Origin will create a new book and import the data into it.

7.2.3 Customizing ASCII Import Dialog Settings and Saving a Theme

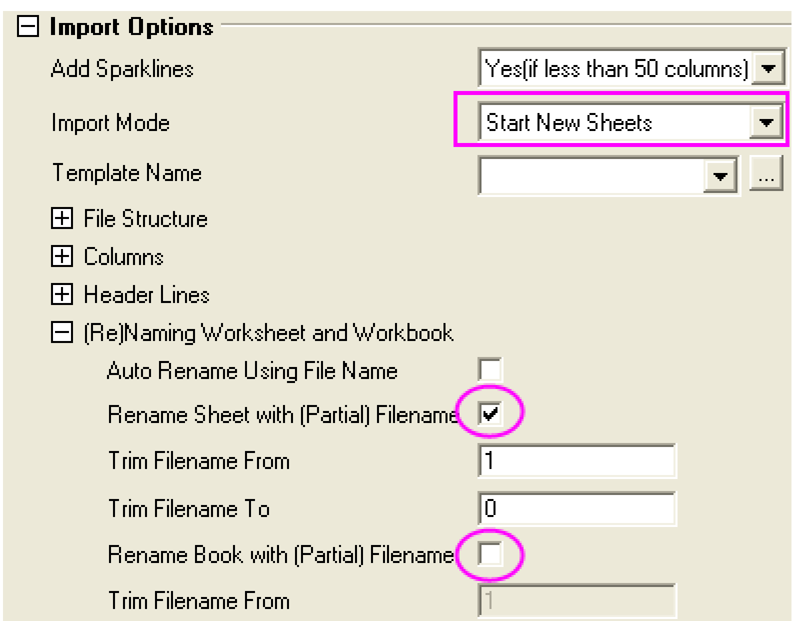
ASCII import and custom-file-format import both provide an *options* dialog, wherein a user can customize import settings and then save settings to use later on similar files.

1. Start with a new book and click the **Import Multiple ASCII** button  on the standard toolbar.
2. Select the files **sensor01.dat**, and **sensor02.dat** from ***\Samples\Curve Fitting*** and add them to the lower panel of the file dialog. Click the file name column header in the lower panel to sort the files by name. Keep the **Show Options Dialog** box checked and click **OK**. This will open a

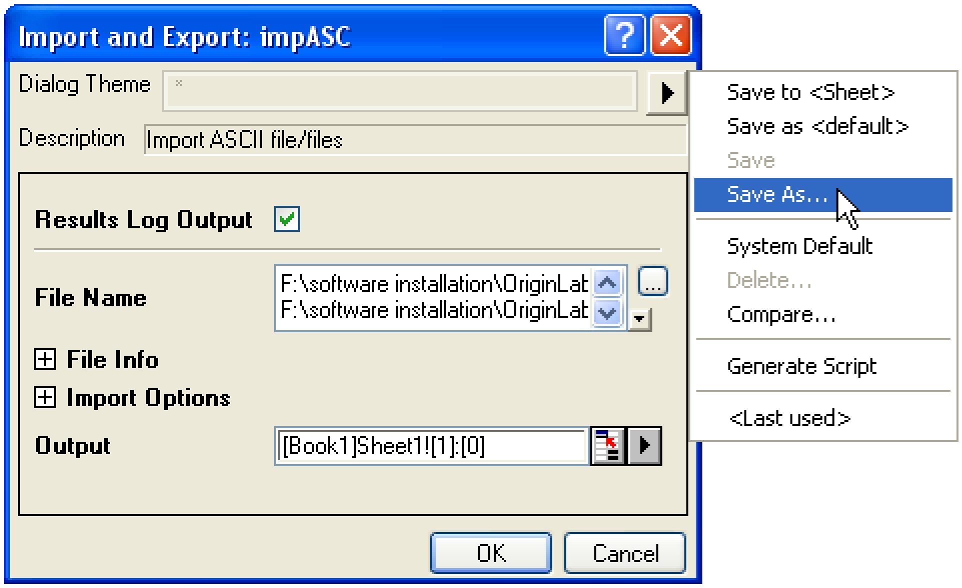
dialog for import settings.



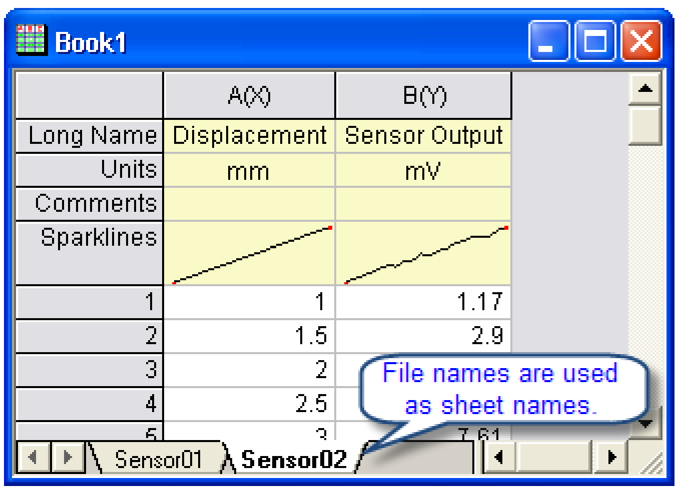
3. Change import mode to **Start New Sheets**. Expand the **(Re)Naming Worksheet and Workbook** node and change the settings so that only the sheet gets renamed.



4. Click on the right arrow button at the top of the dialog and select **Save As**, then give it a name like **My Multifile Import** and click **OK**. This saves your settings to a theme file on the disk.



5. Click **OK** and the first file gets imported into the current sheet, and a new sheet is created for the second file. File names are used as sheet names.



6. Now start a new book and invoke the menu **File: Recent Imports: impASC: My Multifile Import**. In the file dialog pick the three files: **step01.dat**, **step02.dat**, and **step03.dat** from ***\Samples\Curve Fitting***. Click the file name column header in the lower panel to sort the files by name. Click **OK**. The settings from the theme you saved and selected here are used to perform the importing.

7.2.4 Saving ASCII Import Settings to Worksheet

Custom ASCII import settings can be saved either to the disk as a theme file, or into the worksheet itself.

1. Start with a new book and invoke **File: Import: Single ASCII** from the menu, then select the file **sensor01.dat** from the ***\Samples*Curve Fitting** subfolder. Keep **Show Options Dialog** checked and click **OK**.
2. In the **Import Options** dialog, select **No** from the **Add Sparklines** drop-down list. Expand the **(Re)Naming Worksheet and Workbook** node and change the settings so that only the sheet gets renamed by file name, and not the book.
3. Click on the Top Arrow icon, select **Save to <Sheet>** and click **OK**. Then your custom settings get saved to the sheet and data are imported.
4. Select **File: Save Template As** from the main menu. Enter **SensorImport** as the template name, and click **OK** to save the workbook template. This template contains both the import settings and the worksheet property settings.
5. Now we create a new workbook from this template. Click the **Open Template** button on the **Standard** toolbar. Select SensorImport.otw under your User Files Folder and then click **Open**. A workbook is created from the template.
6. With this book active, drag-and-drop the file **sensor02.dat**. The data gets imported and only the sheet name changes to the new file name. The sparklines were not turned on.

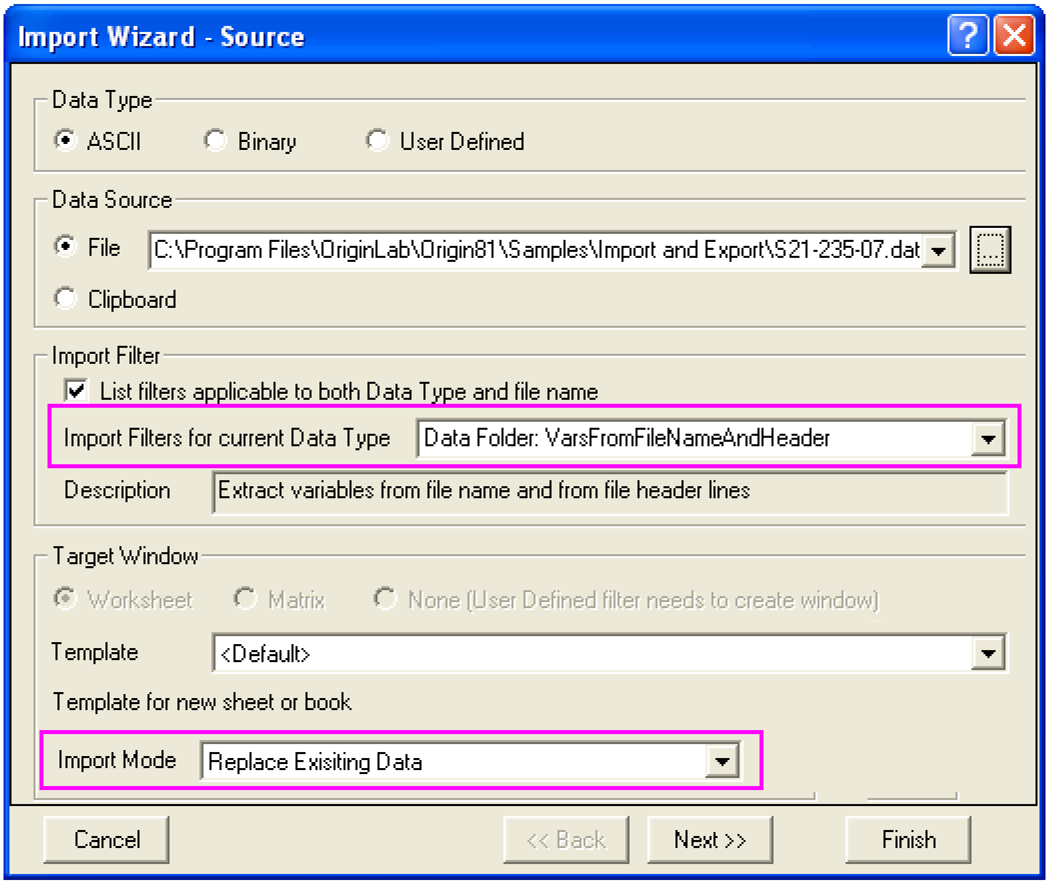
When saving custom settings to a worksheet, it is useful to then save the sheet, along with other desired analysis operations, as an **Analysis Template** for repeat analysis of similar data from multiple files. See the Batch Processing tutorial for details.

7.2.5 Import Wizard and Import Filters

The Import Wizard allows you to step through your data file, customize settings, including how to parse header lines to create variables, and then save all of your custom settings as an import filter (.OIF) file for repeat use. The filter file can reside in the data folder, in the ***\Filters*** subfolder of your **User Files Folder**, or can even be saved to the worksheet itself for use with **Analysis Templates**. The Wizard is typically useful when the file has header lines that need to be parsed, or the file needs custom settings such as fixed width, or for executing LabTalk script at the end of the import for post processing.

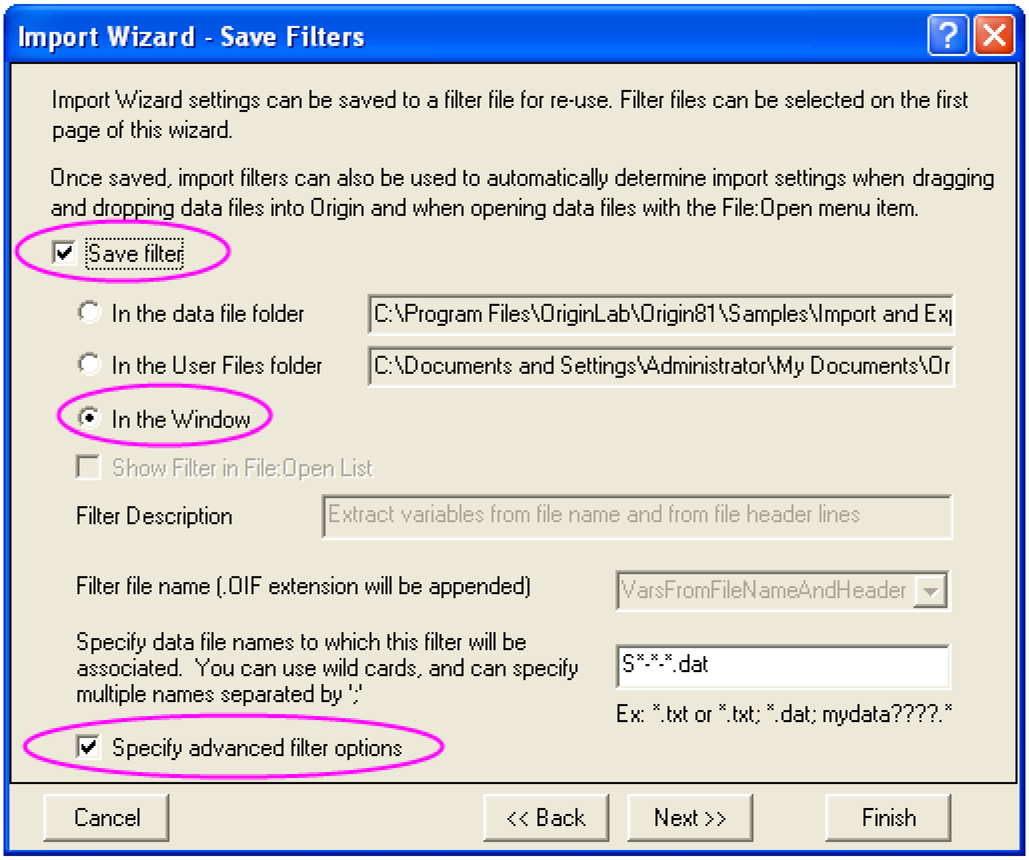
1. Start with a new book. Click on the Import Wizard button in the standard toolbar to launch the wizard.
2. Select the file ***\Samples\Import and Export\S15-125-03.dat***.
3. Note that the **Import Filter for Current Data Type** drop-down changes to show ***Data Folder: VarsFromFileNameAndHeader***. This is a filter already created for this file and shipped with Origin, and it is automatically picked up from the same folder as the data file you chose. Next

change **Import Mode** to **Replace Existing Data**.



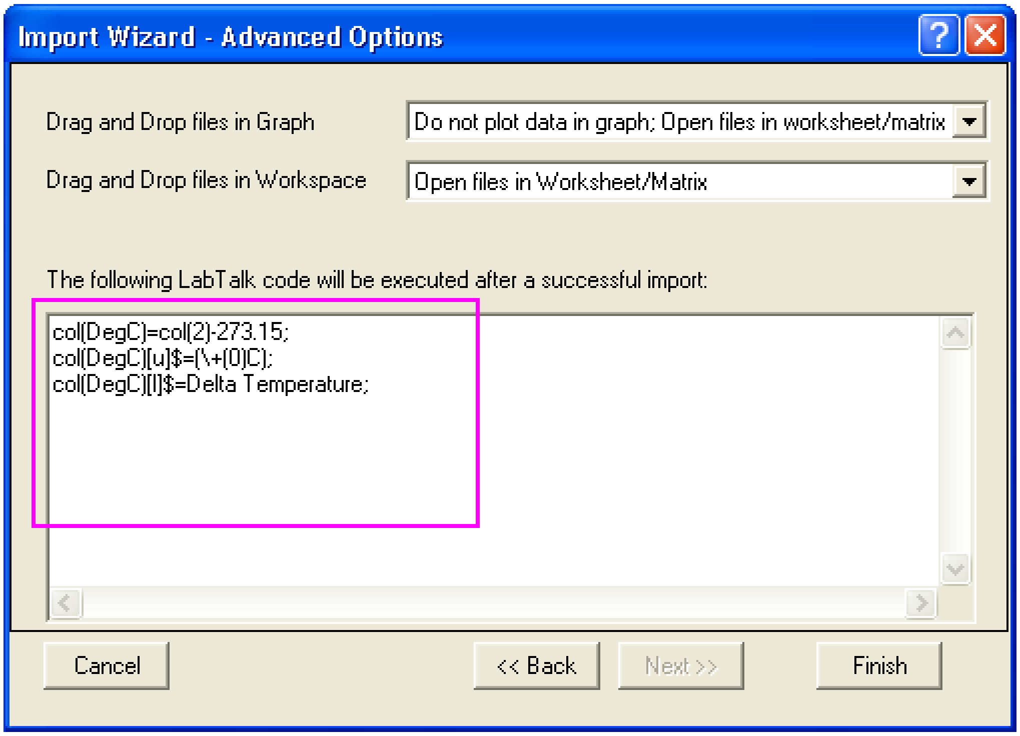
4. Click **Next** and walk through the pages. Notice controls on the **Header Lines** pages that allow flexible definitions of where the header lines end, where sub header lines are located, and what gets assigned to long name, units, etc.
5. For this file the **Variables Extraction** and **Variables Extraction by Delimiter** pages define how to parse the header lines to extract values from them.

6. Click **Next** until you get to the **Save Filters** page. Check the **Save filter** box and change the radio button to **In the Window**. This will save the filter in the active worksheet.



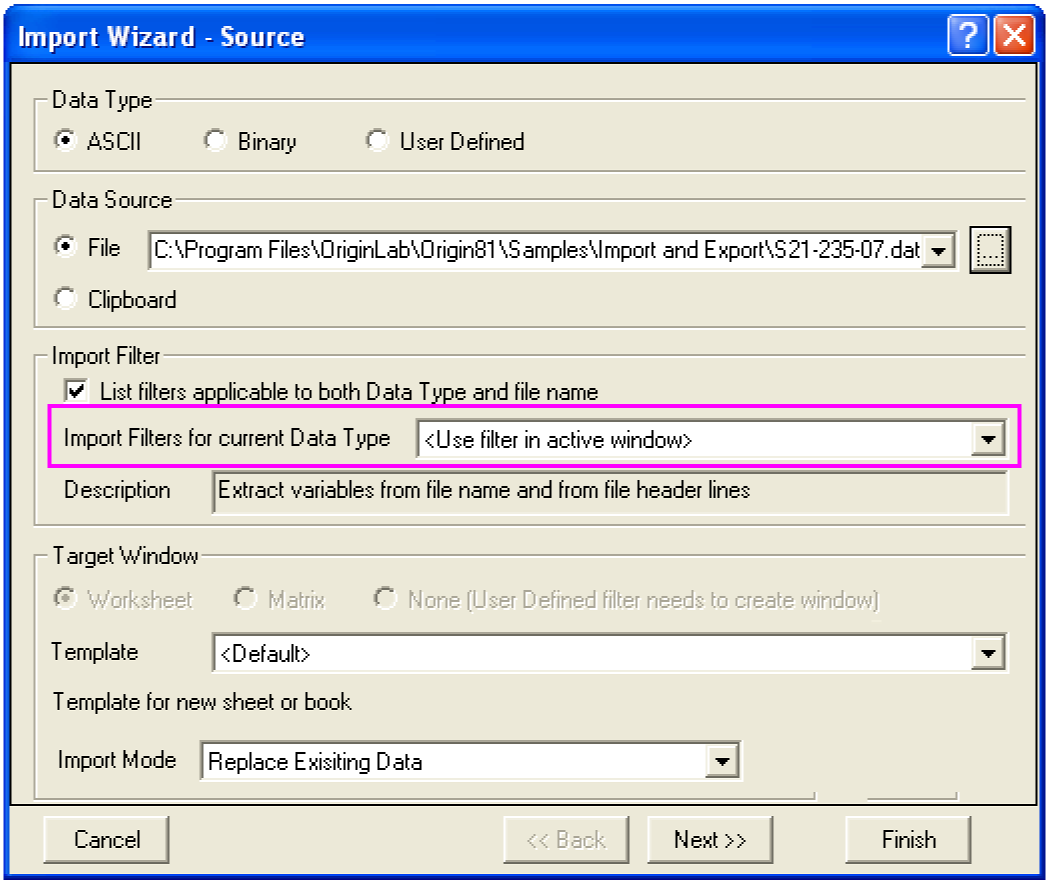
7. Now check **Specify advanced filter options**. This brings you to the last page where script (to run at the end of the import) can be specified. In the edit box enter:

```
col(DegC)=col(2)-273.15;
col(DegC)[u]$=(\+(0)C);
col(DegC)[l]$=Delta Temperature;
```

8. 

9. Click **Finish**. The file gets imported and the import filter is now saved in your worksheet. The fifth column is a column added by the script. It is the Delta Temperature data in degrees Celsius.
10. With the worksheet active, click the **Import Wizard** button again and pick the file ***\S21-235-07.dat***. Note that the **Import Filter for Current Data Type** drop-down shows ***<use filter in***

active window>, so Origin picks up the filter settings that were saved in the worksheet.



11. Click **Finish**, and the file gets imported and the script gets executed (the values in column 5 are updated).

You can save the import settings to the worksheet, perform analysis on the imported data, and save the workbook as an **Analysis Template** for repeated processing of similar data files. See the tutorial on batch processing for more information.

8 Exporting and Presentation

Topics covered in this section:

1. Exporting (Tutorials)
2. Presentation (Tutorials)

8.1 Exporting

8.1.1 Exporting Graphs

Summary

When you have completed your graph for publication, exporting your final results is very easy with Origin. The graph export is highly customizable. You can specify the size of the image, the file format, and other advanced settings such as color format.

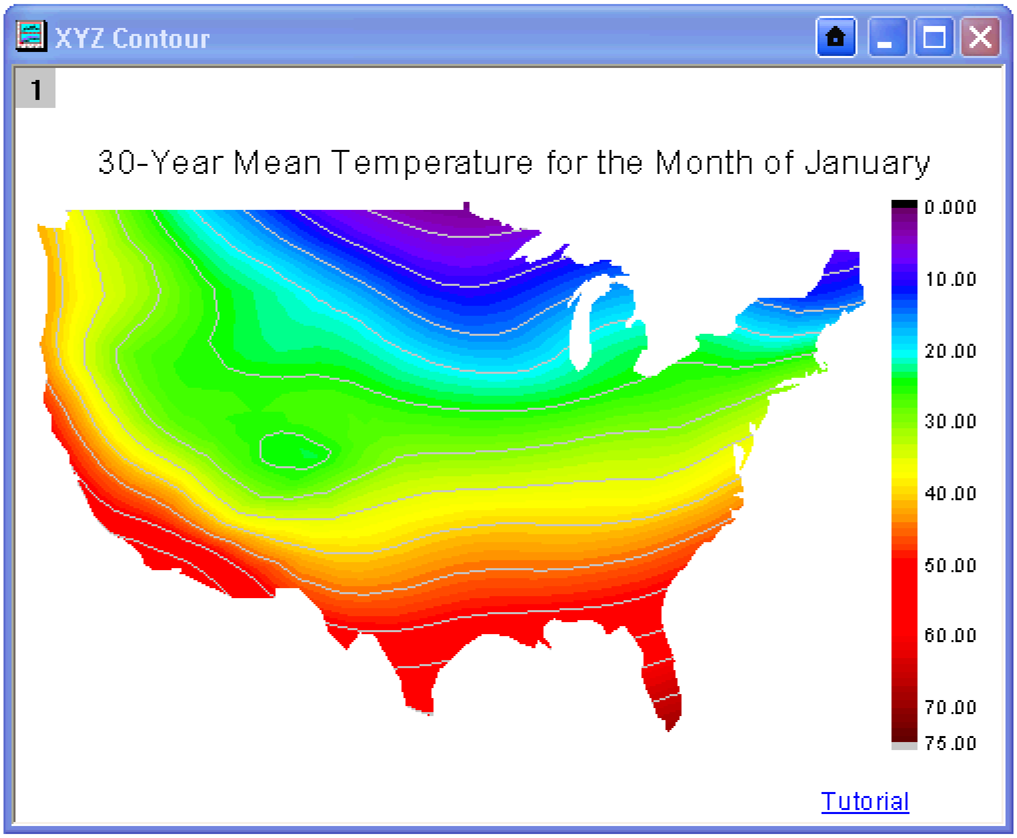
What you will learn

- How to export a graph and specify the settings.
- How to export specified graphs in the project.

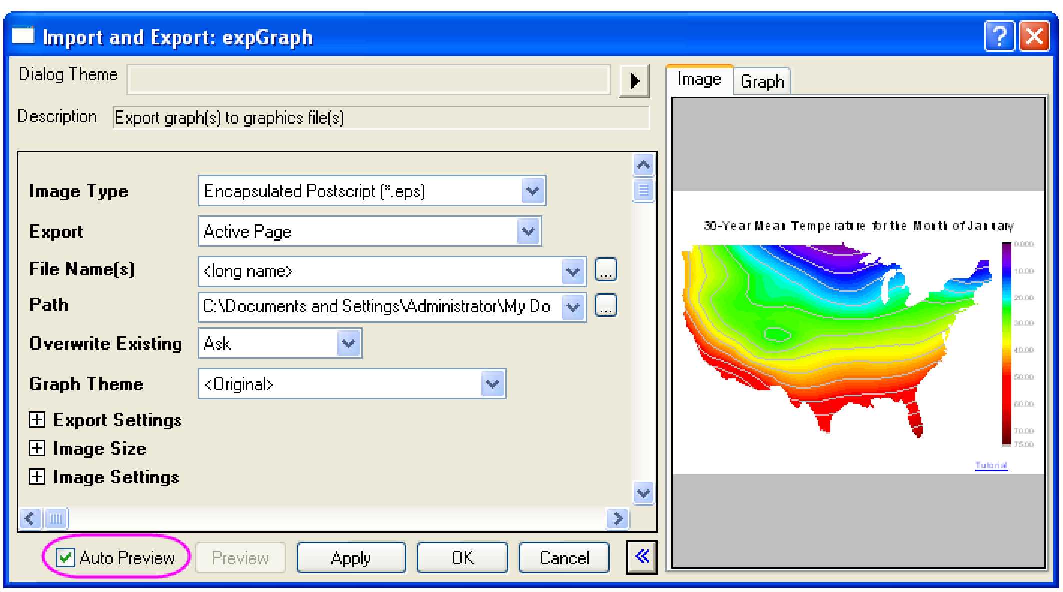
Export a graph as an eps image

This tutorial is associated with the Sample project (\Samples\2D and Contour Graphs.opj) which can be opened by selecting **File: Open Sample Projects: 2D and Contour Graphs** from the main menu.

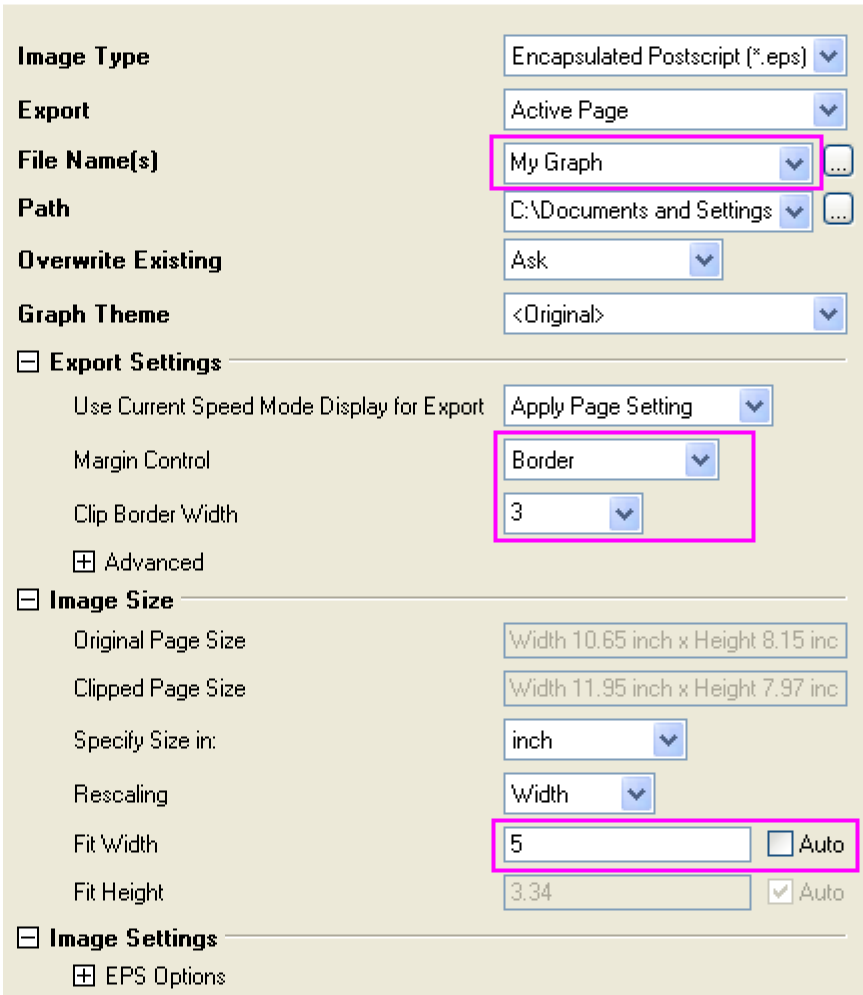
1. In the Project Explorer, browse to *\2D and Contour Graphs\Contour\XYZ Contour*, and make the graph window active.



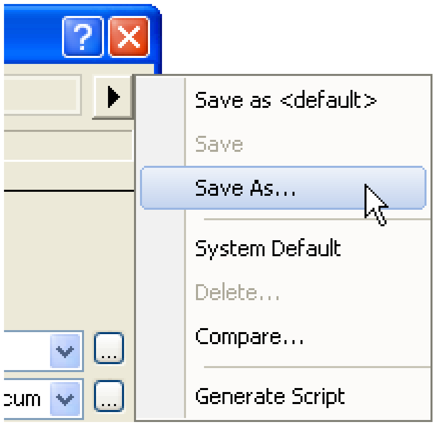
2. With the graph window active, select **File: Export Graphs** from the Origin menu to open the **expGraph** dialog.
3. Click the **Auto Preview** check box. Then the graph will auto preview the temporary copy in the right panel of the dialog.



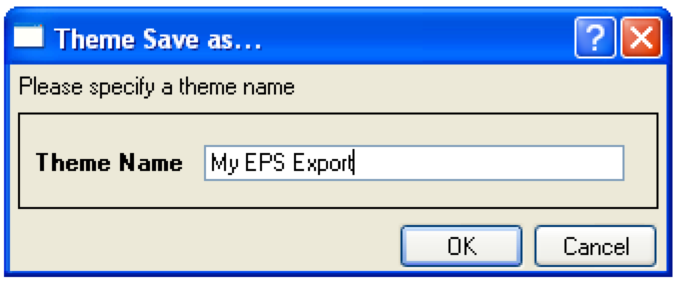
4. Specify the following settings to export the graph:
   - Customize the **File Name** to **My Graph**.
   - Expanding the **Export Settings** node, select **Border** from the **Margin Control** drop-down list and type **3** in the **Clip Border Width** box.
   - Expanding the **Image Size** node, uncheck **Auto** check box for **Fit Width** and set width to **5**.



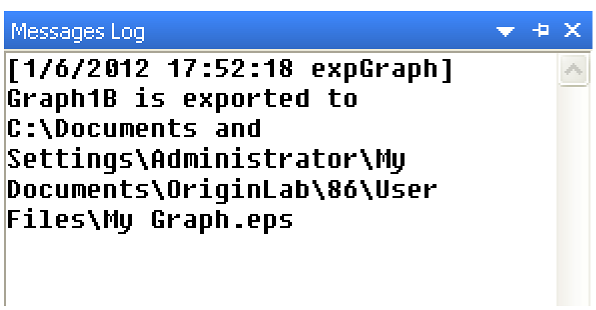
5. Click the triangle button to the right of the **Dialog Theme**. Select **Save As** from the shortcut menu. The **Theme Save as** dialog opens.



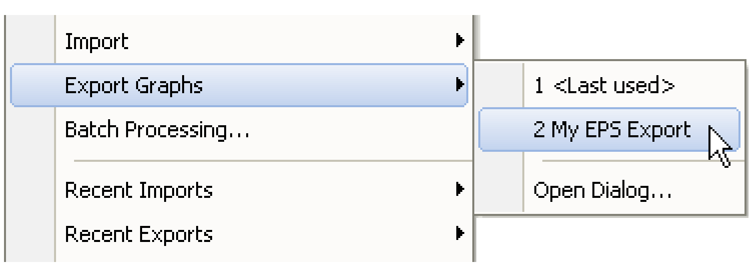
6. In the **Theme Save as** dialog, enter **My EPS Export** as **Theme Name**. And click **OK**.



7. Click the **OK** button in the **expGraph** dialog, then *My Graph.eps* is created in the **User File Folder**. And the graph path will display in the **Message Log**.

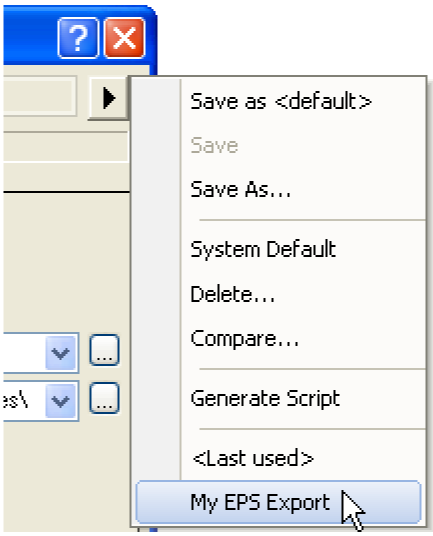


8. Repeat export a graph using the setting from above, select the theme **My EPS Export** in the fly-out menu of the export graph tool directly.



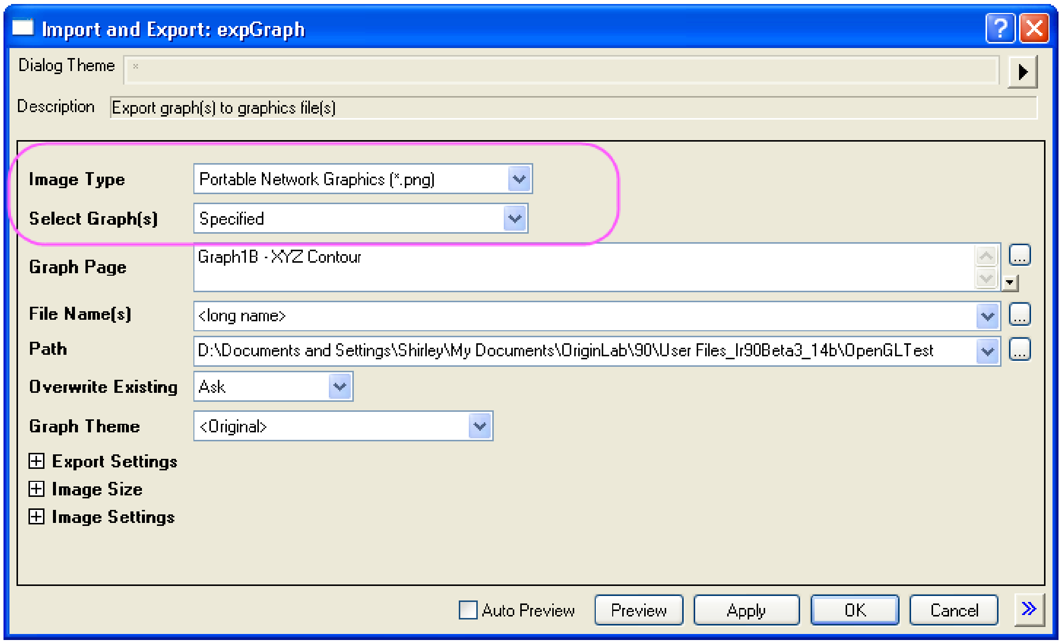
Or

Click the right-sided triangle button to the right of the Dialog Theme in the **expGraph** dialog. From the shortcut menu, pick **My EPS Export**. The settings from that theme will then be loaded in the dialog.



<u>Export the specified graphs</u>

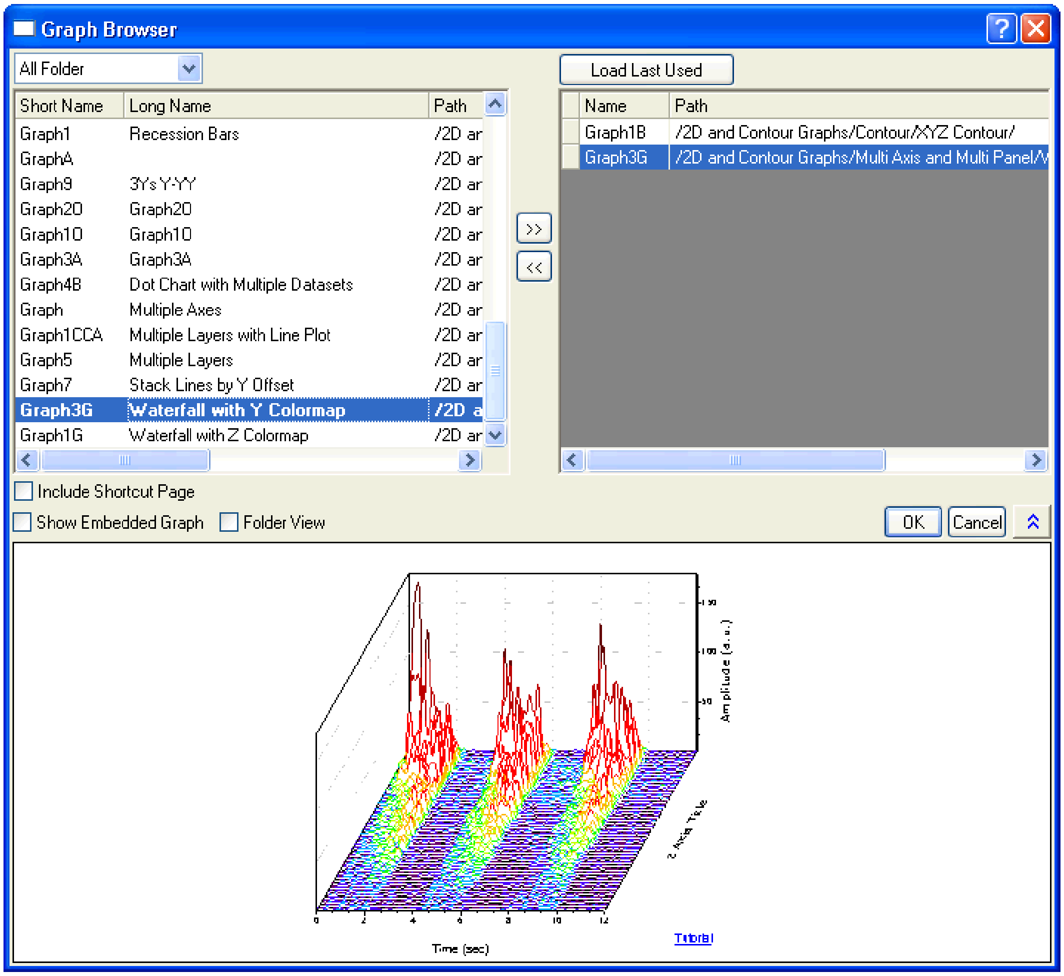
1. Redo steps 1 and 2 from above.
2. In the **expGraph** dialog, select **Portable Network Graphics(*.png)** for **Image Type**.
3. Select **Specified** from the **Select Graph(s)** drop-down list. Click the button to the right of the **Graph Page** edit box to open the **Graph Browse** dialog.



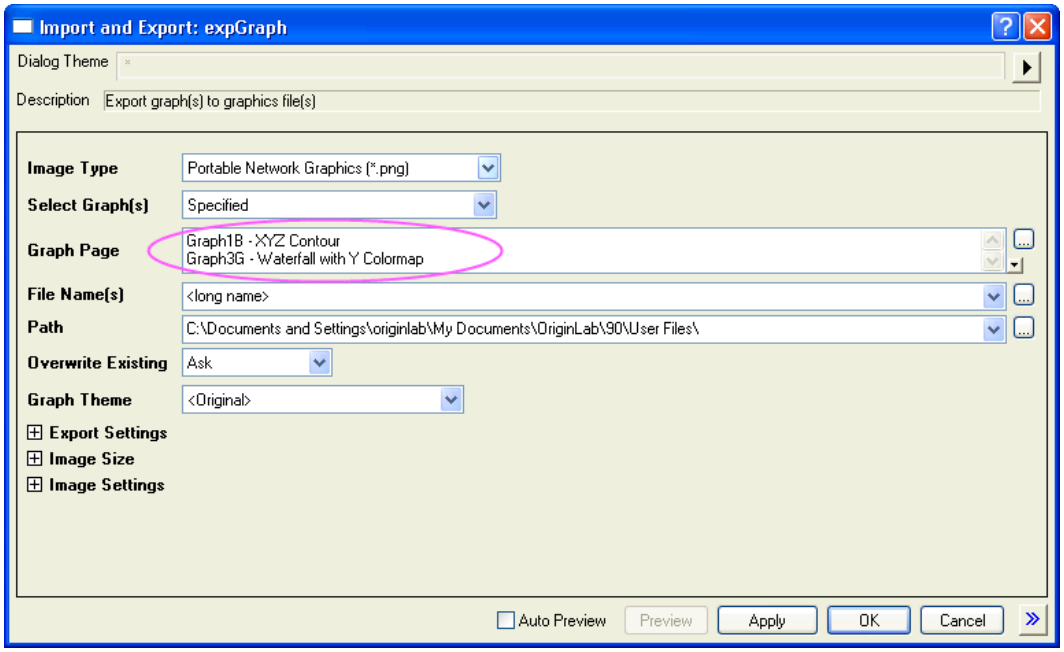
4. In the **Graph Browse** dialog, all the graphs in this project will list in the left panel of the dialog. Select **Graph3G** in the left panel, then the preview panel will show a preview of the graph.

5. Click the 

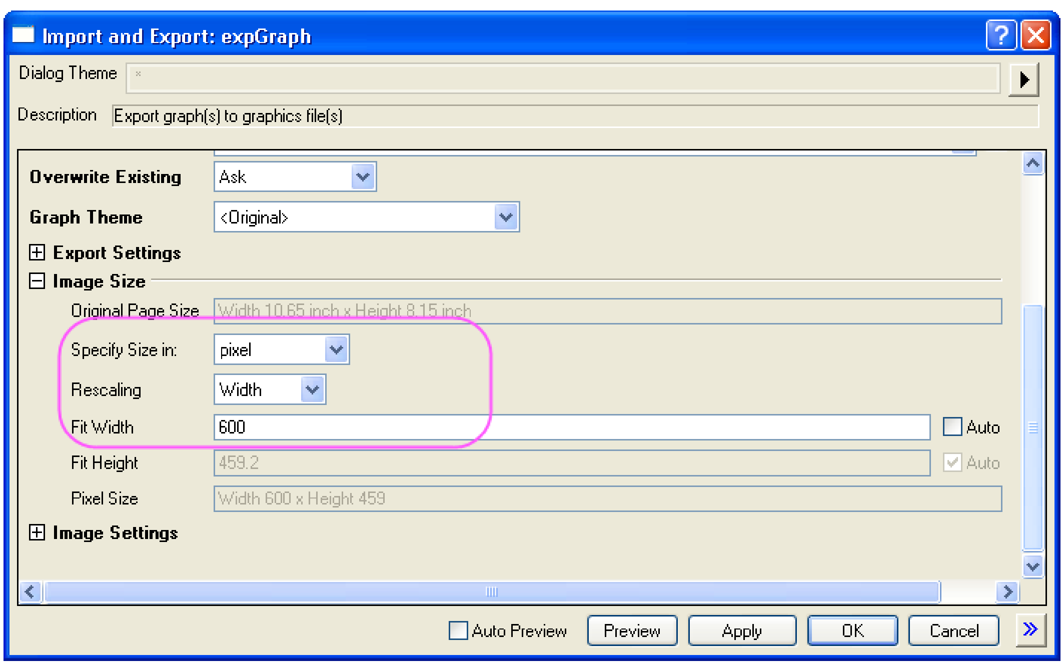
button to add the selected graph to the right panel, or double-click selected graph in left panel to add to the right panel.


Graph Browser
All Folder
Load Last Used
Short Name
Long Name
Path
Graph1 Recession Bars /2D ar
GraphA /2D ar
Graph9 3Ys Y-YY /2D ar
Graph20 Graph20 /2D ar
Graph10 Graph10 /2D ar
Graph3A Graph3A /2D ar
Graph4B Dot Chart with Multiple Datasets /2D ar
Graph Multiple Axes /2D ar
Graph1CCA Multiple Layers with Line Plot /2D ar
Graph5 Multiple Layers /2D ar
Graph7 Stack Lines by Y Offset /2D ar
Graph3G Waterfall with Y Colormap /2D a
Graph1G Waterfall with Z Colormap /2D ar
Name
Path
Graph1B /2D and Contour Graphs/Contour/XYZ Contour/
Graph3G /2D and Contour Graphs/Multi Axis and Multi Panel/V
>>
<<
Include Shortcut Page
Show Embedded Graph
Folder View
OK
Cancel
Time (sec)
Amplitude (a.u.)

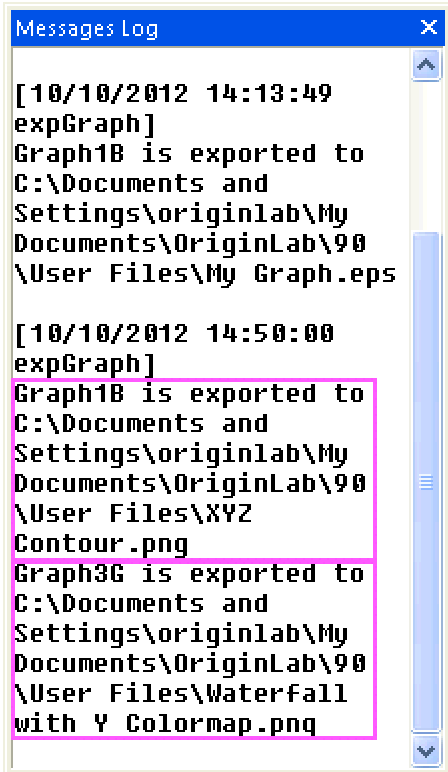
6. Then click **OK** button, the specified graphs will show in the **Graph Page** box.



7. Expanding the **Image Size** node, select **pixel** from the **Specify Size in** drop-down list. And uncheck the **Auto** check box for **Fit Width** and set width to **600**.



8. Click **OK** button in the **expGraph** dialog, then two images are created. And the graph path will display in the **Message Log**.



8.2 Presentation

8.2.1 Pasting or Embedding Graphs in Other Applications

Summary

It is possible to insert Origin graphs to other applications(e.g.Microsoft® PowerPoint/Excel/Word),either by object linking or embedding. Both linked and embedded graphs can be later edited with Origin.

The difference is that embedded graph is contained in the destination file while the linked graph is not. A linked graph can be dynamically updated if the source graph is changed.

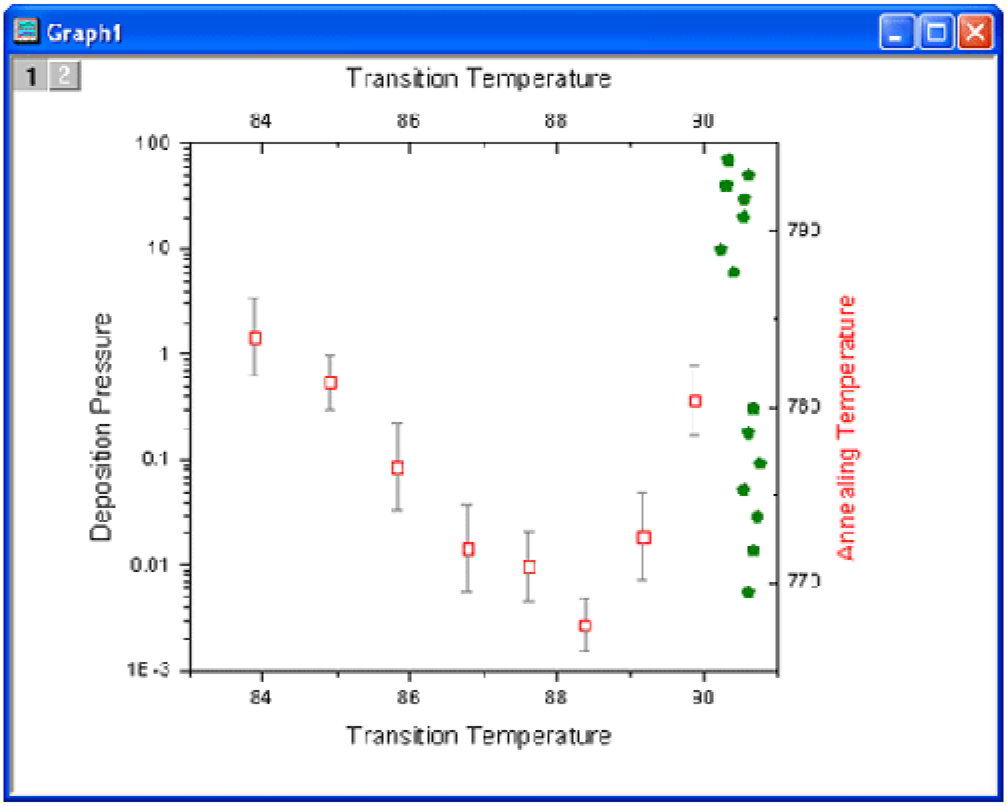
What you will learn

This tutorial will show you:

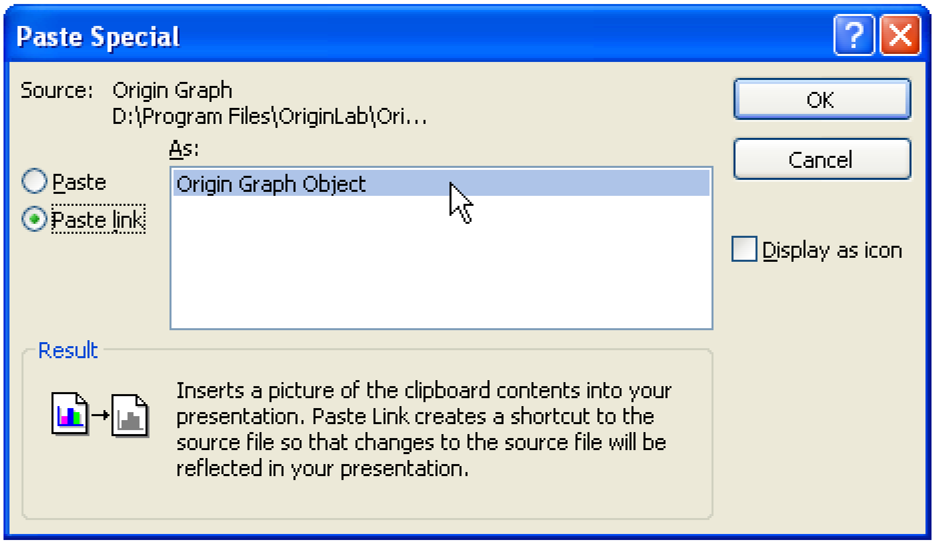
- How to insert an Origin graph to other applications as linked/embedded graphs.
- How to edit linked/embedded graphs after inserting to other applications.

Steps

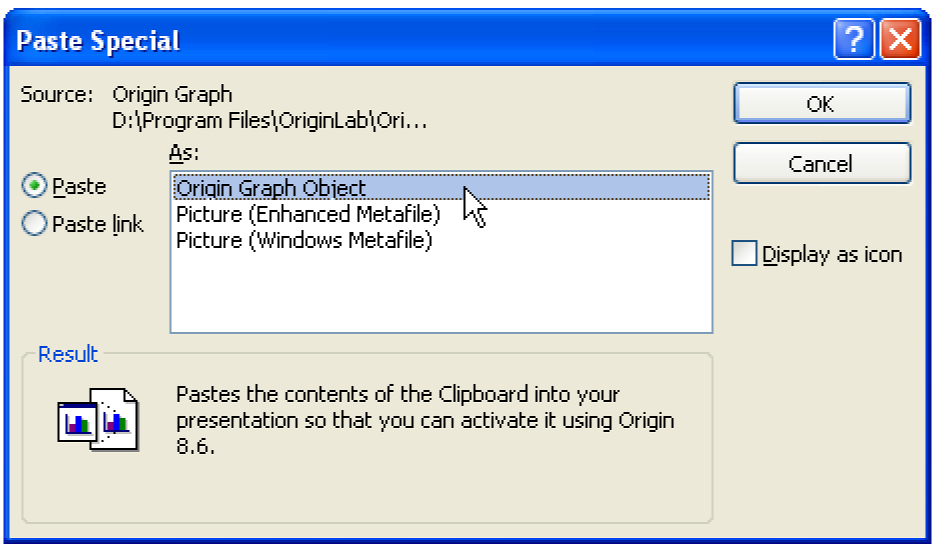
1. Launch Origin, open the OffsetY.opj project from *<Origin program folder>\Samples\Graphing*. This project contains a graph (Graph 1).



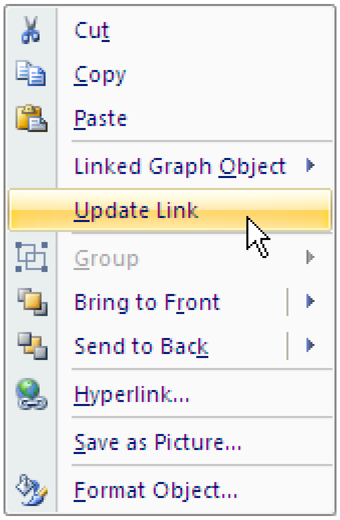
2. With the graph window active, select **Edit: Copy Page**.
3. Switch to Microsoft® PowerPoint 2007, create two new slides.
4. Go to the first slide, click the **Paste** button, in the fly-out menu choose **Paste Special**.
5. In the **Paste Special** dialog, select the **Paste Link** radio button and then choose **Origin Graph Object** in the **As** list. Click OK to close the dialog. In the first slide the graph is pasted as a linked object.



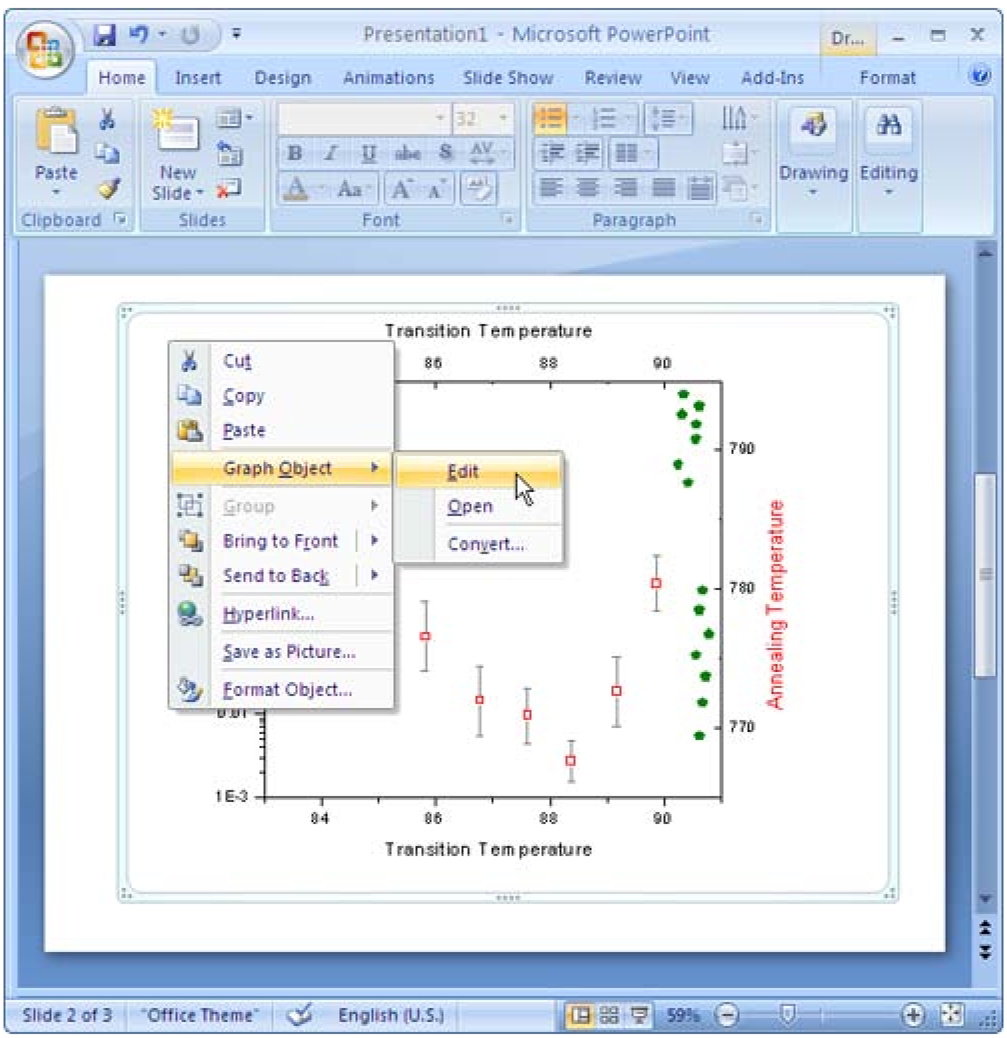
6. Go to the second slide, open the **Paste Special** dialog again similarly as the previous step.
7. This time, select the **Paste** radio button then choose **Origin Graph Object** in the **As** list. Click OK to close the dialog. In the second slide the graph is pasted as an embedded object.



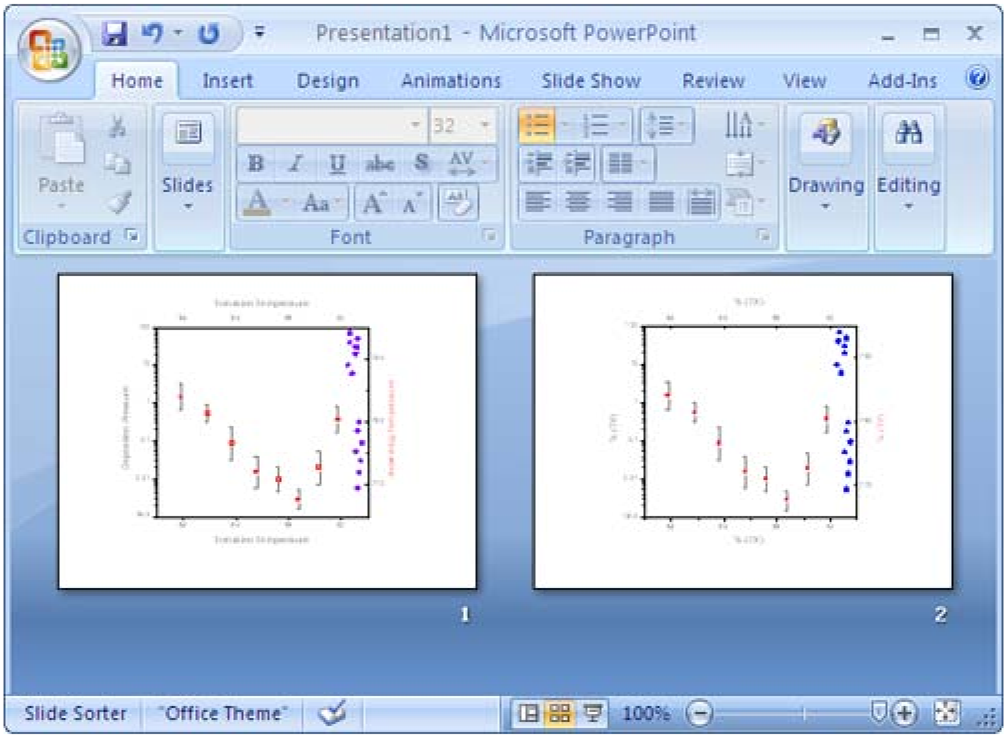
8. Go back to the OffsetY.opj file in Origin, double click on the olive dot plot in Graph 1, bring up the **Plot Details** dialog, change the symbol color to **Violet** and click OK to close the dialog.
9. Now switch to PowerPoint, in the first slide, right click on the linked graph, and choose **Update Link** from the fly-out menu. The color of the dot plot in this graph will be changed to violet.



10. Go to the second slide, right click on the embedded graph, and choose **Graph Object:Edit** to reopen the graph in Origin.Note that the graph name is **Graph in Presentation 1**, indicating that the graph is contained in the destination file.



11. In the **Graph in Presentation 1**, change the color of olive dot plot to blue similar as previously, and close Origin.Note that the linked graph in slide 1 and the embedded graph in slide 2 looks different now.



9 Collaboration and Connectivity

Topics covered in this section:

1. Connectivity (Tutorials)

9.1 Connectivity

9.1.1 Excel

- Working with Excel

Working with Excel

Summary

Origin provides flexible ways to interact with Excel. You can either import Excel data into an Origin workbook, or open an Excel book inside Origin. If you require full access to all of Origin's graphing and analysis features, you will probably want to import your Excel data files into Origin. If it is important to maintain a separate Excel workbook file—perhaps so that other colleagues who do not work with Origin have access to that file—you will probably want to open your Excel data files directly. We give a brief introduction to working with Excel in this tutorial.

This tutorial will show you how to:

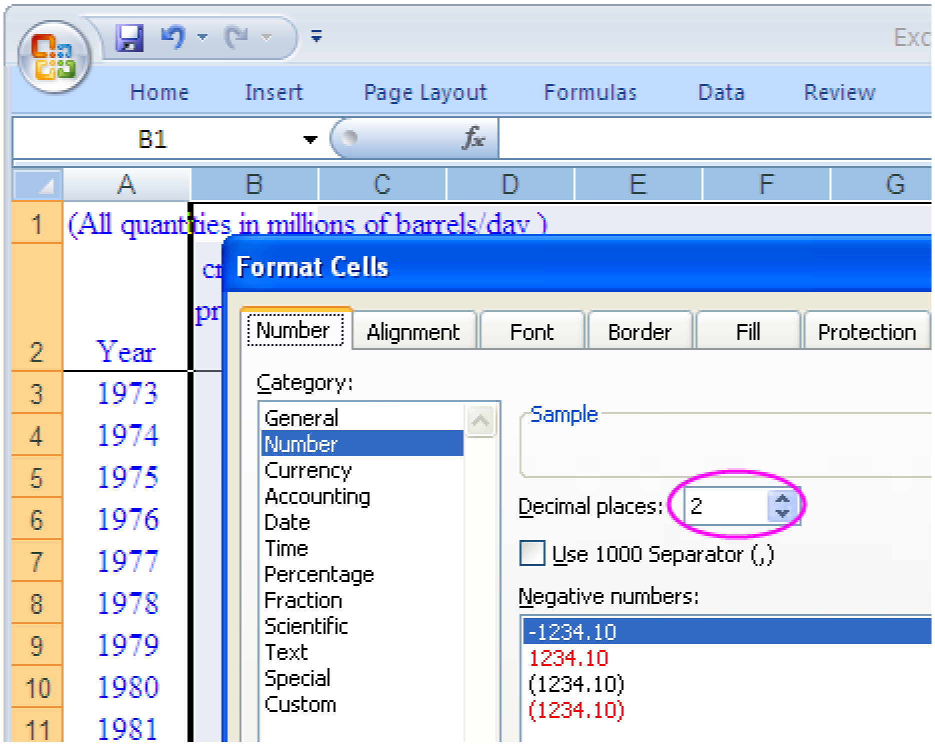
1. Copy and paste data from Excel with full precision
2. Import an Excel file into an Origin workbook
3. Open an Excel file in Origin
4. Save an Excel file with path relative to the Origin Project file

Copying and Pasting Data from Excel

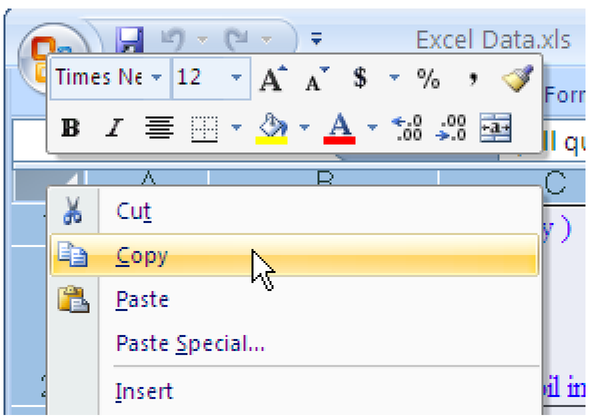
It may be desirable at times to simply copy and paste data from Excel to Origin instead of importing or opening Excel. These steps show that such a copy/paste operation can bring in data with full precision.

1. Launch Excel and Origin separately.
2. Open the file **<Origin Installation Folder>\Samples\Graphing\ExcelData.XLS** in Excel.

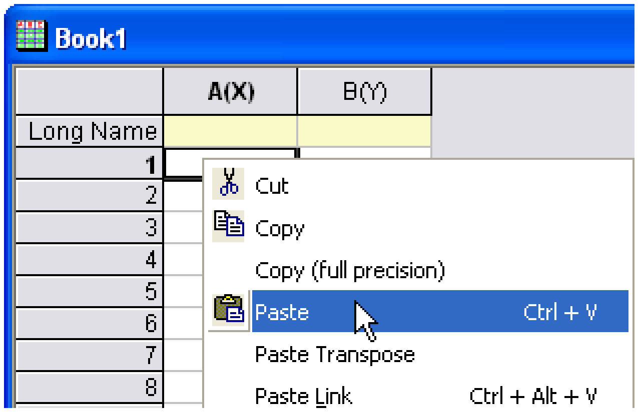
3. Select columns B through L, then right-click and bring up the **Format Cells** dialog, and set the number of decimal places to 2. So now Excel shows fewer decimal places.



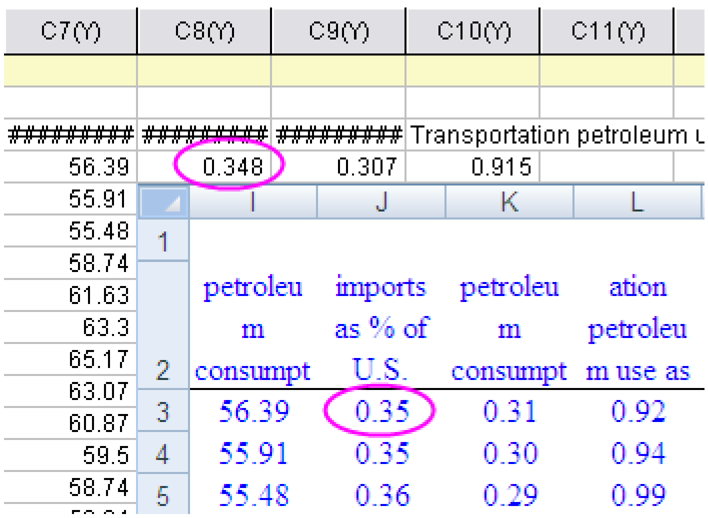
4. Click on the top left cell in the Excel sheet to select the entire sheet and right-click and select **Copy**, or use the keyboard shortcut **Ctrl+C** to copy.



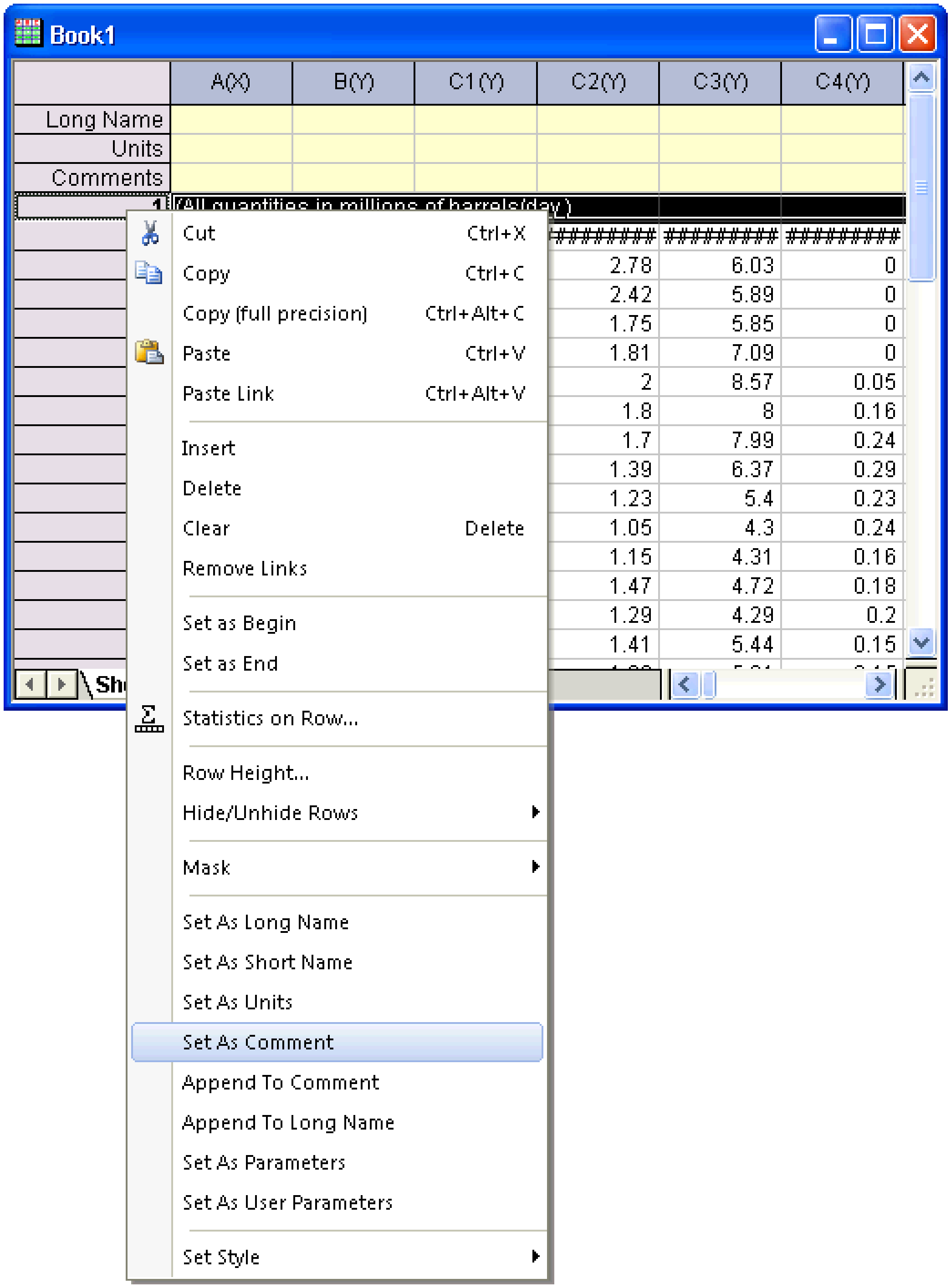
5. Go to a new book in Origin, place the cursor in row 1 of column 1 and do **Ctrl+V** or right-click and **Paste**.



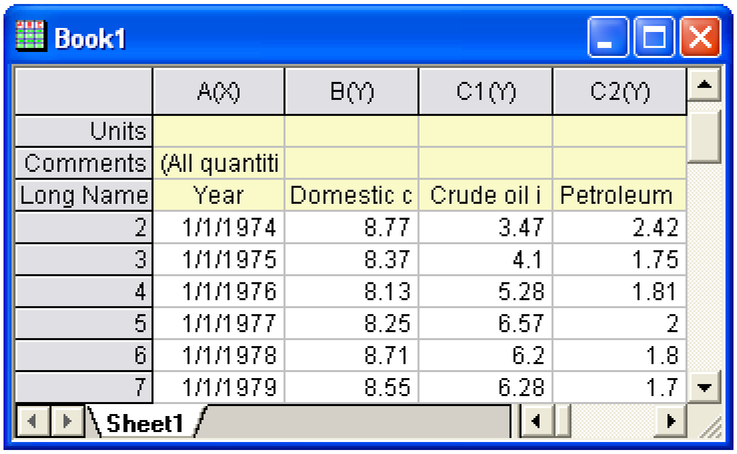
6. Note that the numbers come into Origin with full precision, not the number of displayed digits (2) in Excel.



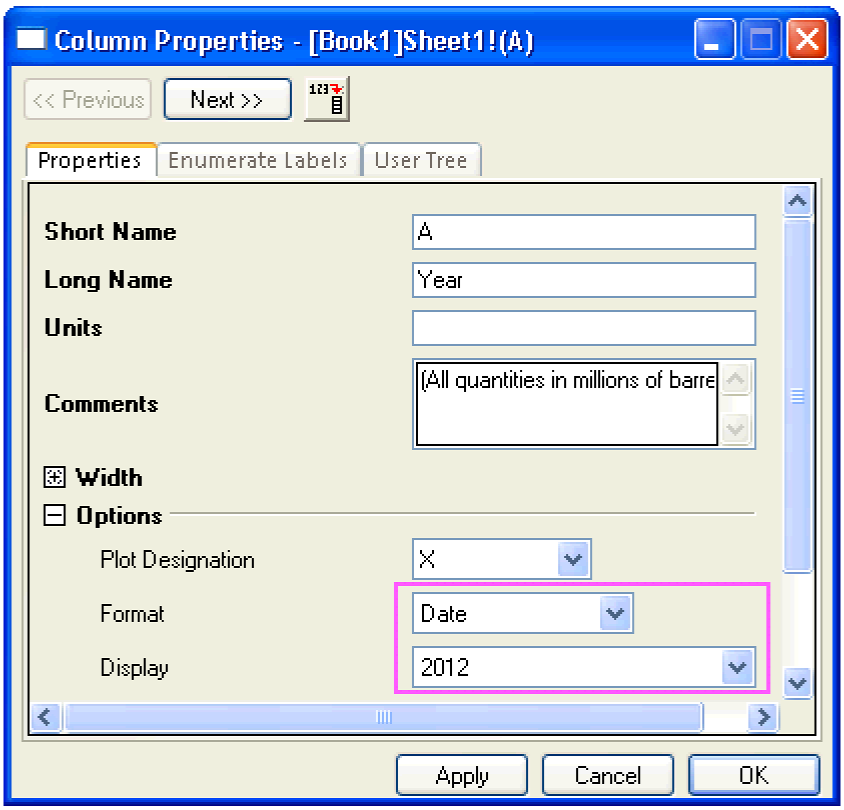
7. In Origin, right-click on the row 1 header and select **Set as Comment** to make this row an Origin column comment.



8. Right-click on row 1 again and select **Set as Long Name**. Then rows 1 and 2 in Excel become the worksheet header in Origin:



9. You can now double-click column 1 and set it as **Date** and then plot the data.

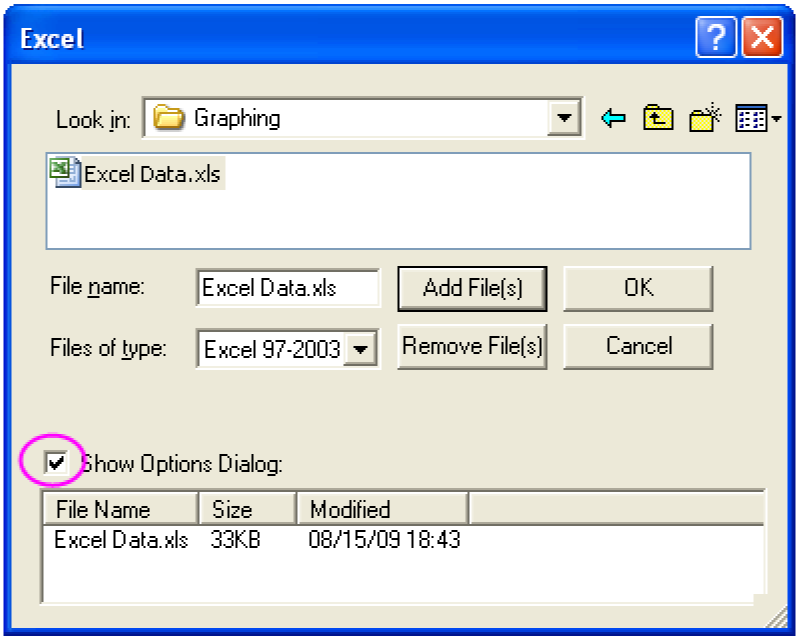


Importing Excel Files

Origin supports importing Excel files directly into Origin workbooks. Multiple sheets are supported and controls are available for setting specific rows in the Excel sheet to be brought into an Origin worksheet as header information, including Long Name and Comments. If you want to perform analysis or data manipulation operations on your Excel data, we recommend importing your data into Origin.

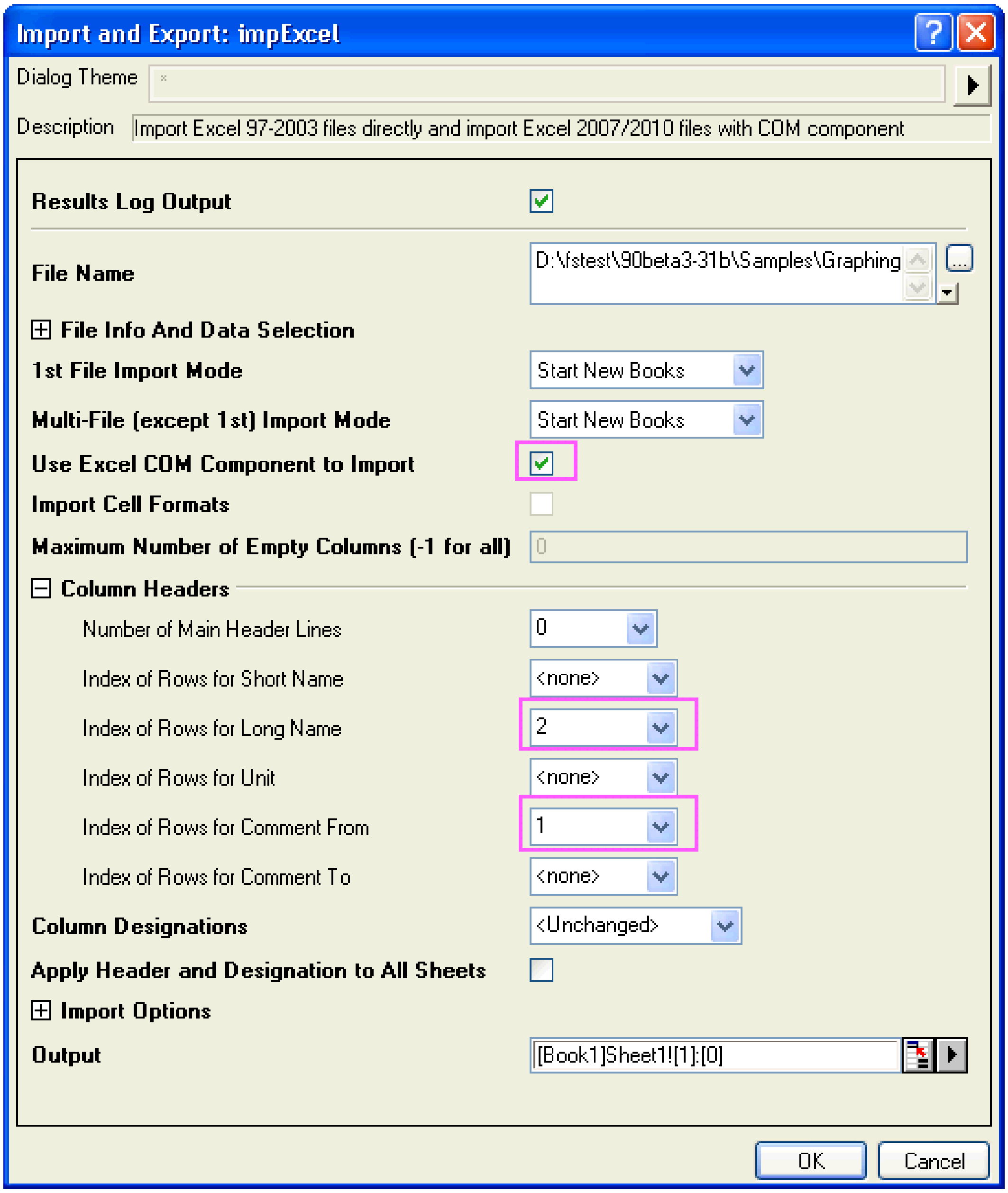
1. With a new book active in Origin, use the menu to select **File: Import: Excel (XLS, XLSX)**.

2. Select the file **\samples\graphing\Excel Data.xls**, and make sure **Show Options Dialog** is checked.

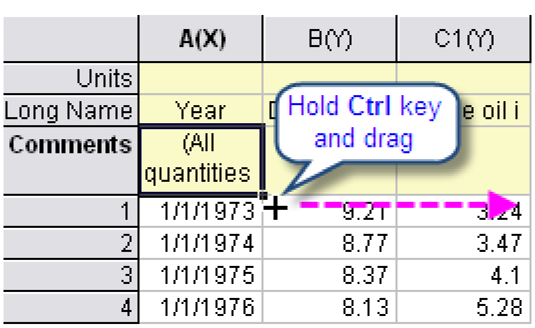


3. In the dialog that comes up, leave the **Use Excel COM Component to Import** check box checked.
4. Set the **Index of Rows for Comments** to 1.

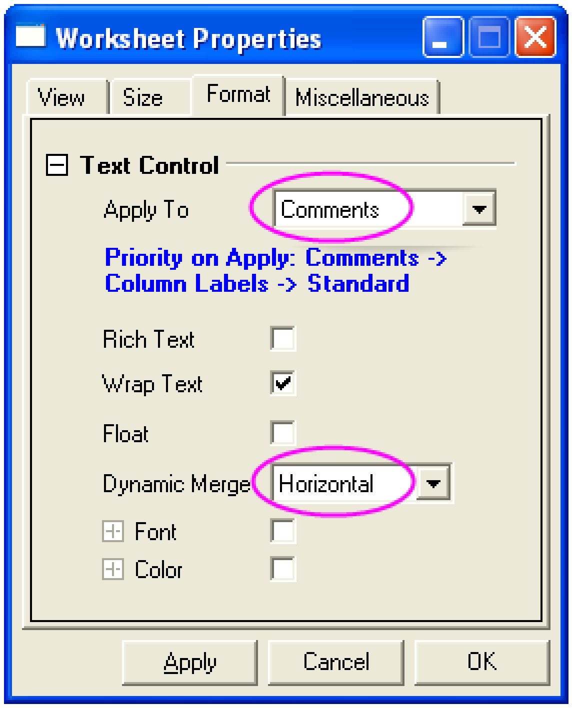
5. Set the **Index of Rows for Long Name** drop-down to 2 and click **OK** to Import.



6. Click and select the **Comments** cell under column 1, hold down the **Ctrl** key and drag the bottom right point of the selected cell to stretch across all columns with data, copying the same comments to all columns.



7. Press **F4** to bring up the format dialog, switch to the **Format** tab, change the **Apply To** drop-down to **Comments** and set **Dynamic Merge** to **Horizontal**, then click **OK**.



This sets the comments cells to be merged and to show in the center of all data columns.

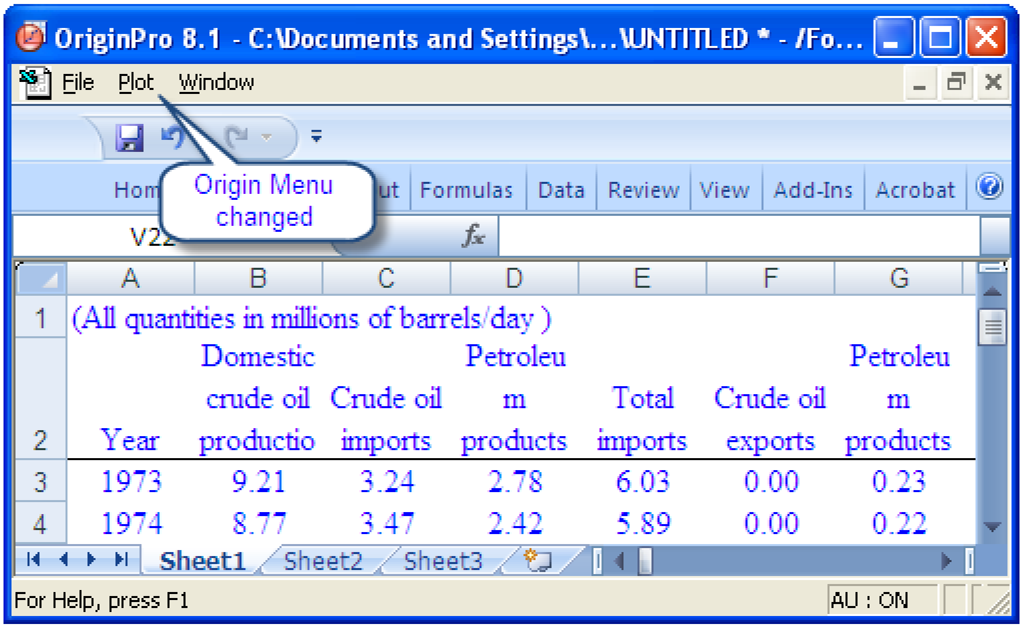
C3(Y)	C4(Y)	C5(Y)	C6(Y)
Total impor	Crude oil e	Petroleum	U.S. petrole
(All quantities in millions of barrels/day)			
6.03	0	0.23	17.31
5.89	0	0.22	16.65
5.85	0	0.2	16.32
7.09	0	0.22	17.46
8.57	0.05	0.19	18.43

Open Excel File in Origin

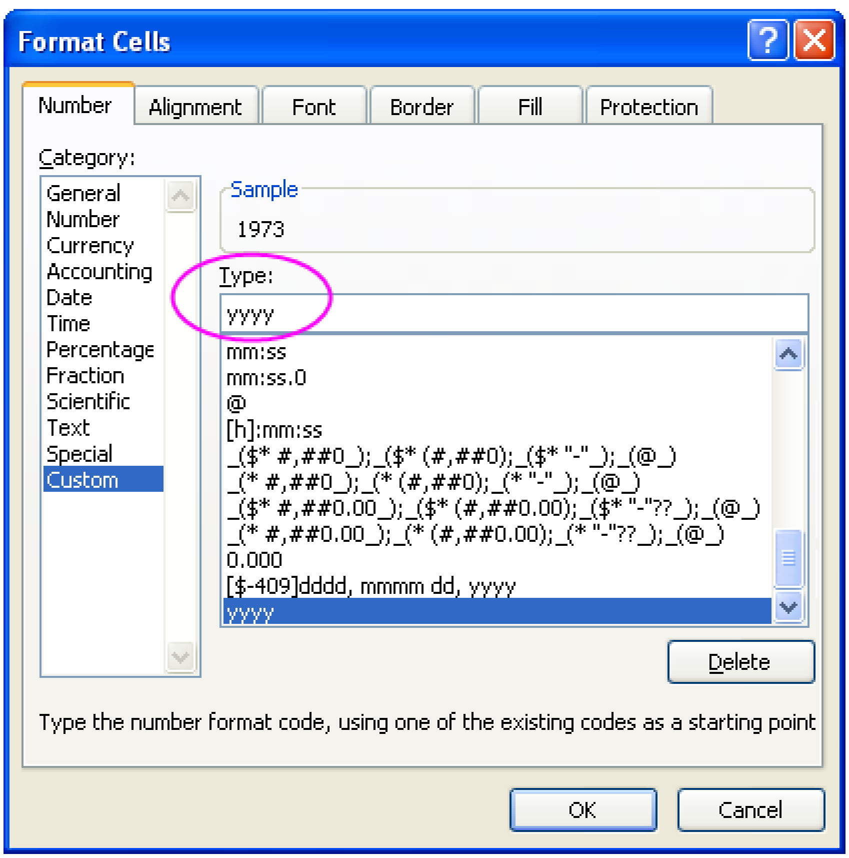
At times it may be desirable to keep the data in an external XLS file and simply open the file inside Origin as an Excel window, and then work with the data. When you open Excel (.XLS or .XLSX) files as Excel workbooks in Origin, an OLE instance of Microsoft Excel is launched. You can plot directly using Excel workbook data, but many analysis features, as well as 3D plotting, will be inaccessible.

1. Select **File: Open Excel** and select the file **\Samples\Graphing\Excel Data.xls**.
2. A new Excel window opens inside the Origin workspace. When this window is active, the Origin main menu has different entries, some of which are specific to Excel, and the Excel toolbars are

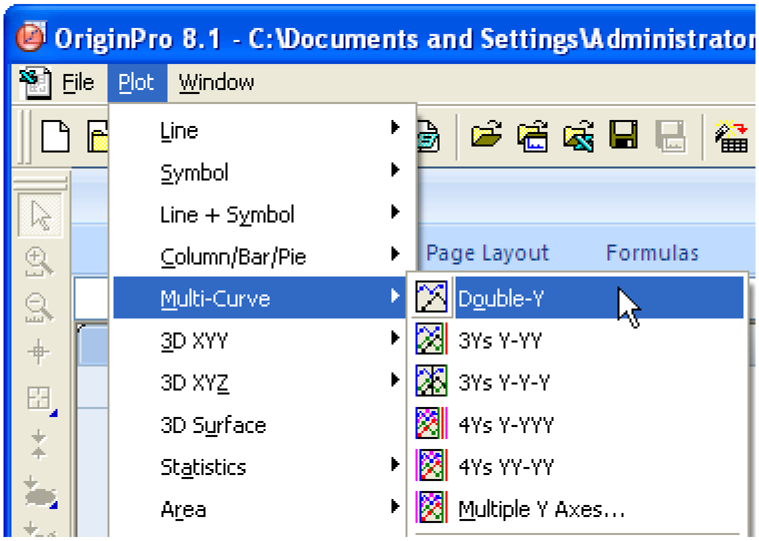
available.



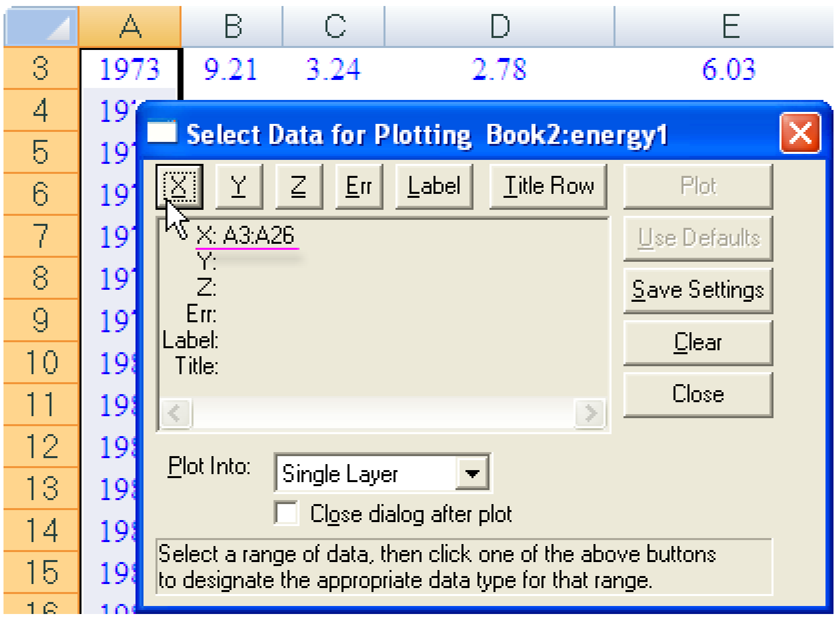
3. Highlight the Excel data range **A3:A26** and right-click, then select **Format Cells** to make sure that the data is in **Date** format.



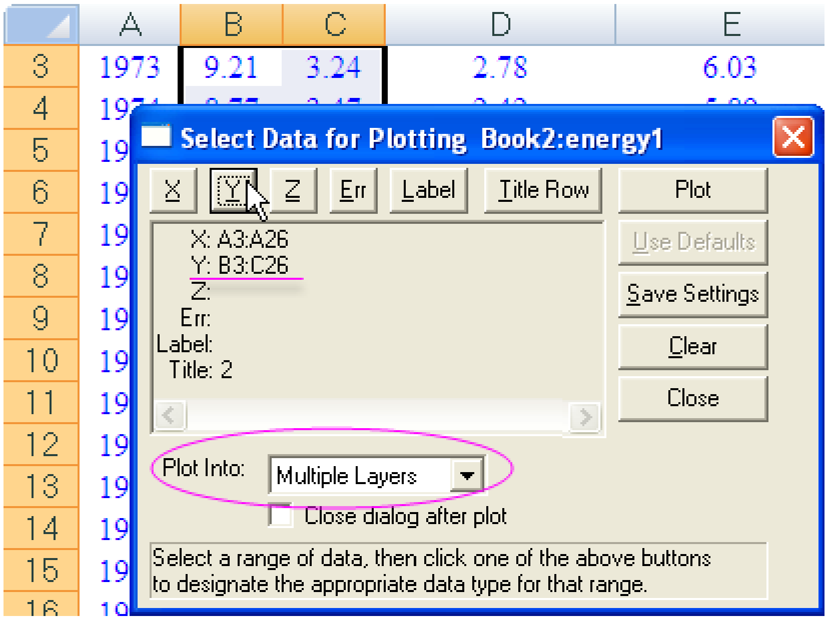
4. Now select the **Plot** menu in Origin and select the **Multi-Curve: Double-Y** plot type.



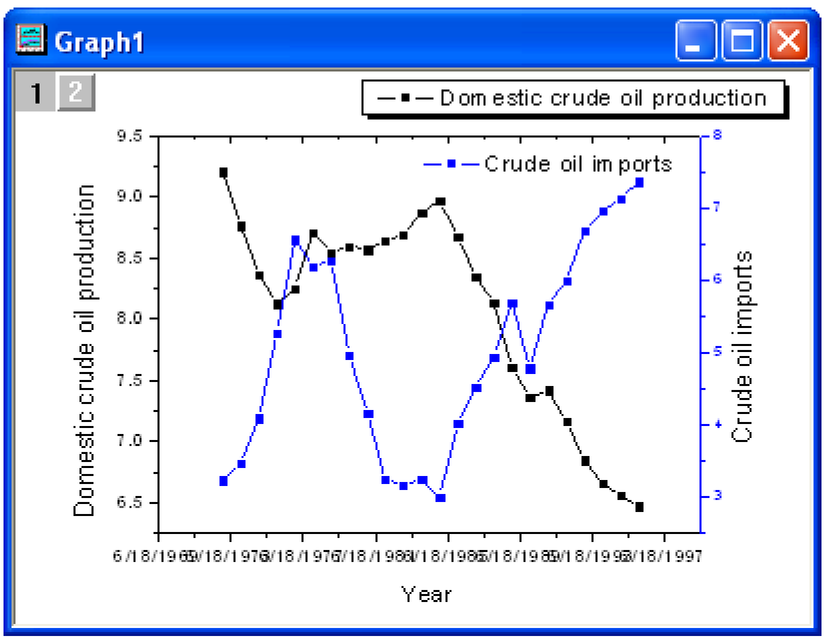
5. Select **A3:A26** in the Excel sheet and click **X** in the **Select Data for Plotting** dialog to assign the X data.



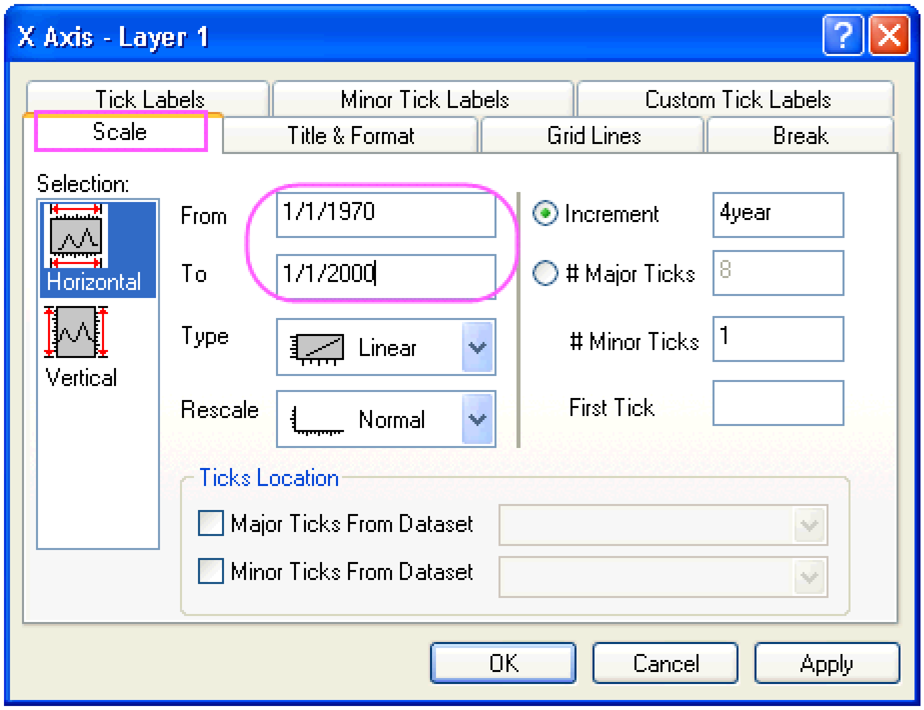
6. Select **B3:C26** in the Excel sheet and click **Y** in the **Select Data for Plotting** dialog to assign the Y data, and then change the **Plot Into** drop-down to be **Multiple Layers**.



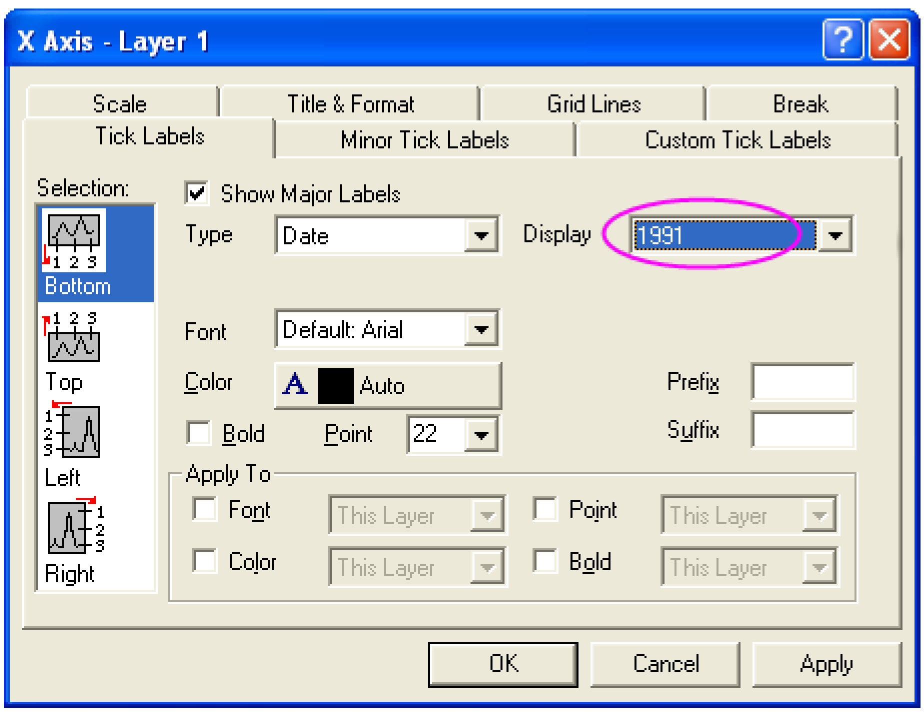
7. Click **Plot** and a double-y plot is created.



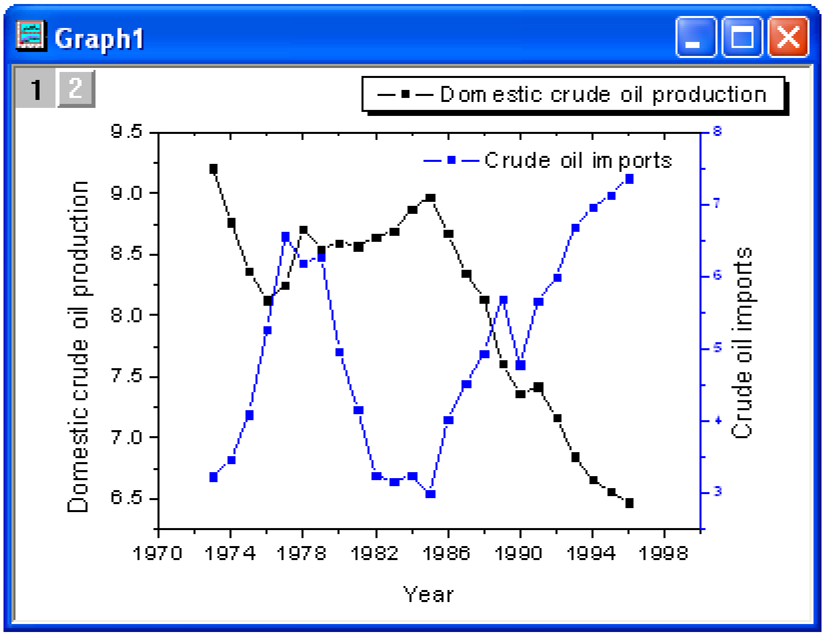
8. By default, Origin displays the tick labels for time on the X-axis in MM/DD/YYYY format. Double-click the X-axis to open an X-Axis Properties dialog box. On the **Scale** tab, change the scale from **1/1/1970** to **1/1/2000**.



On the **Tick Labels** tab, change the **Display** to year only.



Then we get:



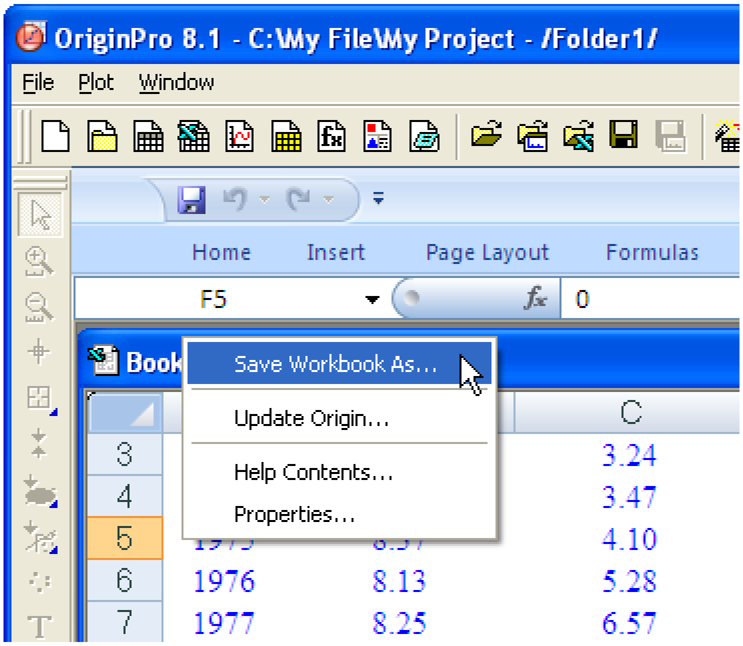
After opening Excel in Origin, if you switch to another Origin window, a toolbar spacer is visible where the Excel menu used to be. Right-click and you can select **Hide Toolbar Spacer** or **Hide Toolbar Spacer Always**.

Setting External Excel File Path Relative to OPJ Path

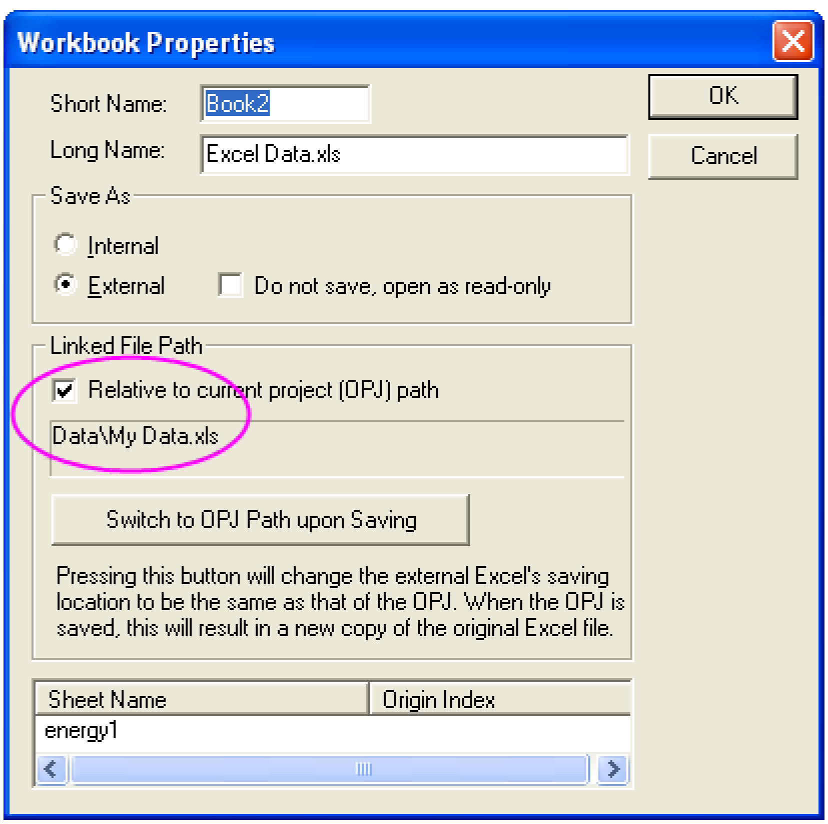
The Origin Project can contain an Excel window which is linked to an external Excel file. It may then be beneficial to save the Excel file in the same folder as the Origin project, or in a subfolder under the Origin project folder, which will then make the two files more portable, as seen in the following steps:

1. Close Excel if it is running.
2. Perform the steps under the **Open Excel File in Origin** section above, and (optionally) create a plot.
3. First save the OPJ to some folder location such as "C:\My Files\My Project.opj".

4. Now right-click on the Excel window and select **Save Workbook As** and save it in a (new) subfolder under the OPJ save location, such as "C:\My Files\Data\My Data.xls".



5. Right-click again on the Excel window title and select **Properties**, and then check the box that says **Relative to current project (opj) path**. Note that the Excel file path in the box below changes to the relative path "Data\My Data.xls".



6. Save the OPJ again. Now you can copy the entire subfolder structure, starting from where the OPJ is saved, and put it on an external memory device (i.e., a memory stick or similar), or zip the entire folder structure. When taken to another computer and opened, Origin will look relative to the OPJ path to find the Excel file.

If your Excel file is in a different location and you want to save it to the same path as the OPJ, you can open the Excel file in Origin, then right-click the title, select **Properties** and click the **Switch to OPJ path upon Saving** button. On saving the OPJ, the Excel file will be copied from its current location to the same path where the OPJ is saved.

10 Programming

Topics covered in this section:

1. LabTalk (Tutorials)
2. Origin C (Tutorials)
3. X-Functions (Tutorials)

10.1 LabTalk

Topics covered in this section:

1. Advanced Scripting using LabTalk

10.1.1 Advanced Scripting using LabTalk

Summary

This tutorial demonstrates how to use some advanced LabTalk scripting commands and methods to organize your script files. To learn more about all the commands and methods supported in LabTalk, please refer to **Help: Programming: LabTalk**.

Minimum Origin Version Required: Origin 8.1 SR1

What you will learn

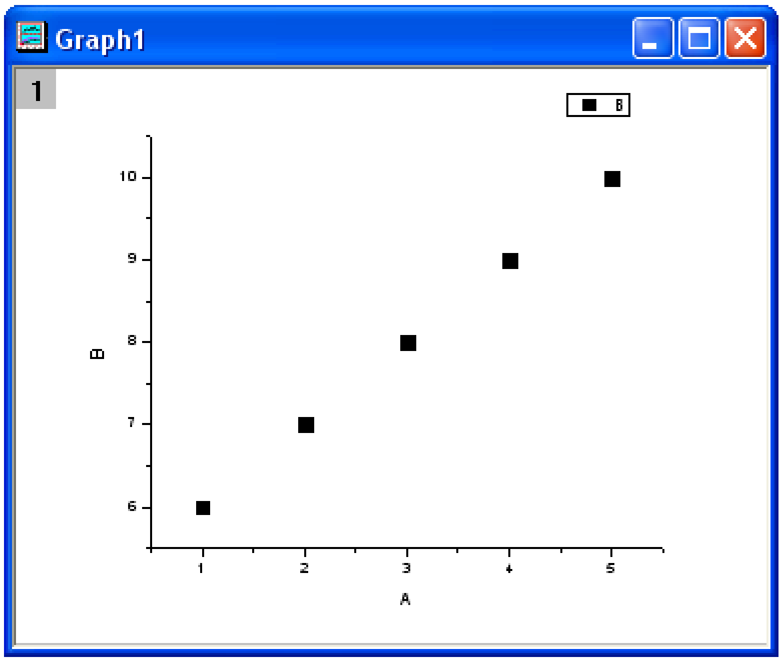
This tutorial will show you how to:

- Modify Plot Attributes via Script
- Define a LabTalk Script Macro Command
- Define a Macro
- See Origin?s Predefined System Macros
- Load and Compile your Origin C Function using LabTalk script
- Use .OGS Files to Store Script.

Modifying Plot Attributes via Script

This section demonstrates how to use script commands to change the attributes of a data plot.

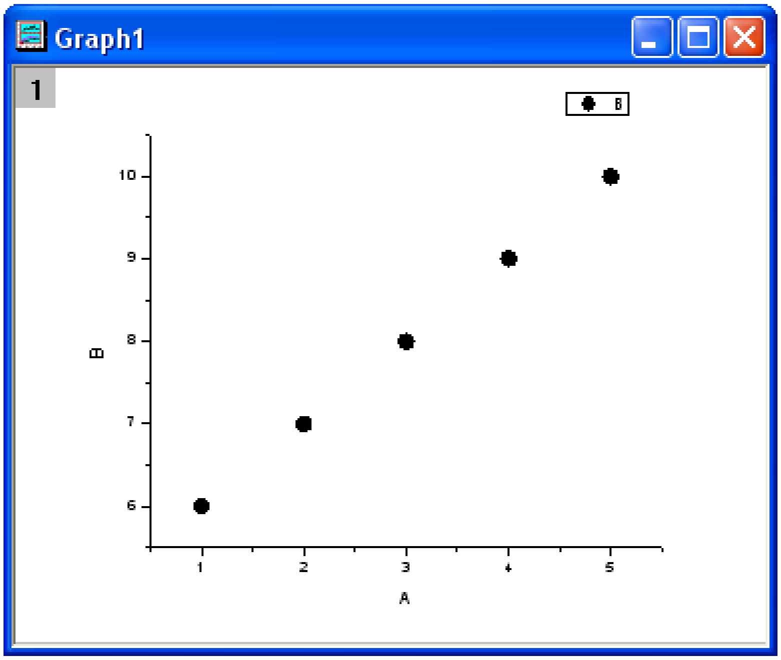
1. Start a new project, enter the numbers 1 through 5 in column A of the Data1 worksheet and numbers 6 through 10 in column B. If you have not already done so, open the Script Window by selecting **Window: Script Window** from the Origin program menu.
2. Using the worksheet data from the previous exercise, create a scatter plot. Note that the scatter plot symbol is a black filled square (symbol size is increased for clarity).



3. To change the symbol shape, type the following:

```
set %C -k 2
```

4. Press **ENTER**.

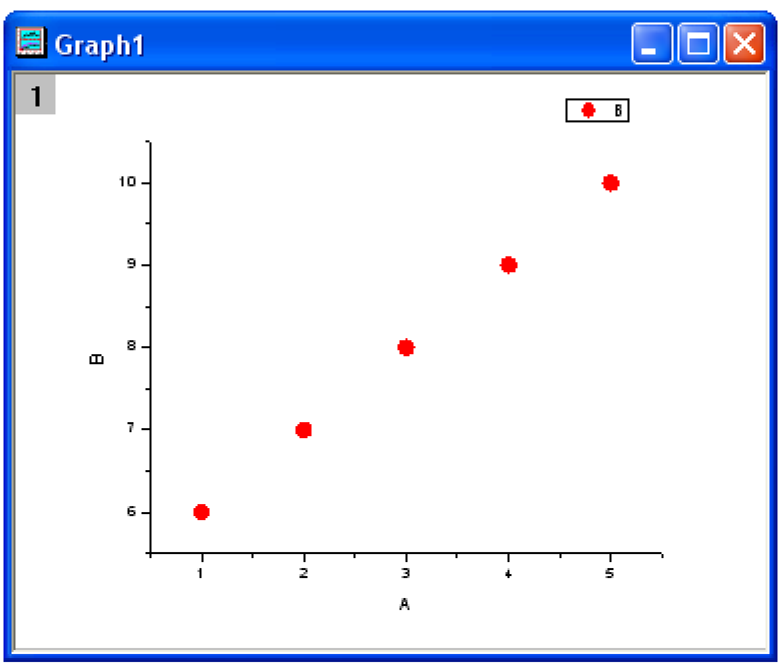


The data plot symbol changes from a filled square to a filled circle (the numbers correspond to Origin's symbol list; 1 = square, 2 = circle, etc.).

5. To change the symbol color, type the following:

```
set %C -c 2
```

6. Press **ENTER**.



The data plot symbol color changes from black to red.

To modify the axis scale values

7. Type the following:

```
X1=0;X2=20;Y1=0;Y2=10
```

8. Press **ENTER**.

Your X-axis scale now reads from 0-20, and the Y-axis reads from 0-10.

> **Notes:** As this example illustrates, you can type multiple lines of script in a single line by separating commands with a semi-colon.

You can also use the set command to specify the data display range.

9. Type the following:

```
Set %C -b 2
```

10. Press **ENTER**.

The graph?s display range now begins with the second data point in the data set.

11. Type the following:

```
Set %C -e 4
```

12. Press **ENTER**.

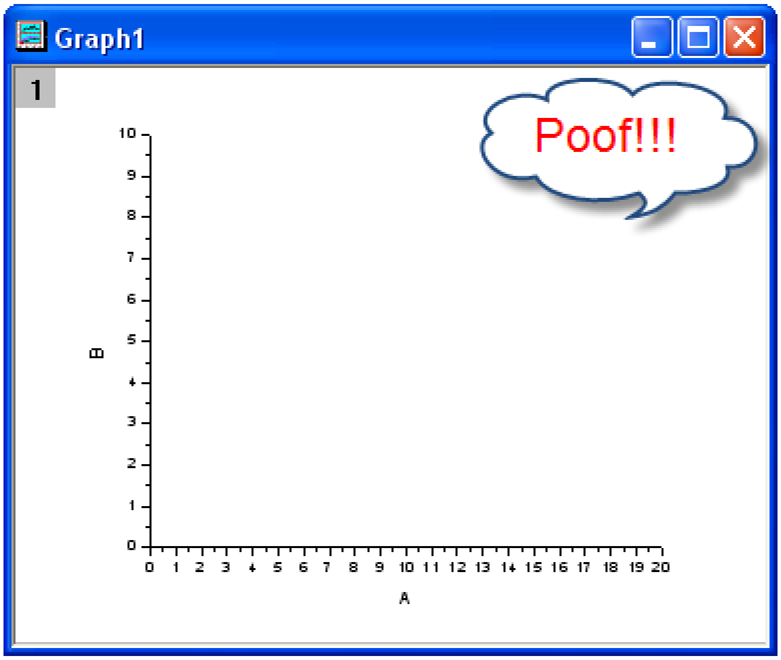
The graph?s display range now ends with the fourth data point in the data set.

You can also hide or show a data plot using the set command?s -s switch.

13. Type the following:

```
Set %C -s 0
```

14. Press **ENTER**.



The active data set is now hidden.

15. To show the hidden data set, type:

```
Set %C -s 1
```

16. Press **ENTER**.

Defining a LabTalk Script Macro Command

A macro is a convenient method of aliasing a LabTalk script. When you define a macro you are associating an entire script with a specific name. This name becomes a command that invokes the associated script.

When developing scripts, macros can provide several advantages.

- Modular code can streamline a script by replacing repetitive or similar blocks of code with multiple calls of the same macro.
- Modifications to your code become easier to implement because you only have to redefine your macro as opposed to modifying repeated blocks of code that are scattered throughout your application.
- There is a limit to the number of tokens that can be included between a set of curly braces that enclose script. Macros provide a means to shorten the code between braces by calling on a pre-defined macro.
- You can modify the behavior of a LabTalk command by creating a macro of the same name. The functionality of the LabTalk command is restored when the macro is deleted.

Defining a Macro

A macro is defined using the define command. The general syntax is:

```
define macroName {
        script
}
```

where *macroName* and *script* are the name of the macro and the body of the macro, respectively.

To define a macro using LabTalk?s define command:

1. From the Origin menu, select **Window: Script Window**.
2. Type the following:

```
def hello {
        type -b "Hello World!!!";
}
```

This script defines a macro named hello that will type "Hello World!!!".

> **Notes:** The define command can be abbreviated as def

We will now use the Script Window to call our hello macro.

3. Type the following into the Script window:

```
hello
```

4. Highlight all codes in the Script Window and press **ENTER**.

An attention dialog opens to say "Hello World!!!".

Origin's Predefined System Macros

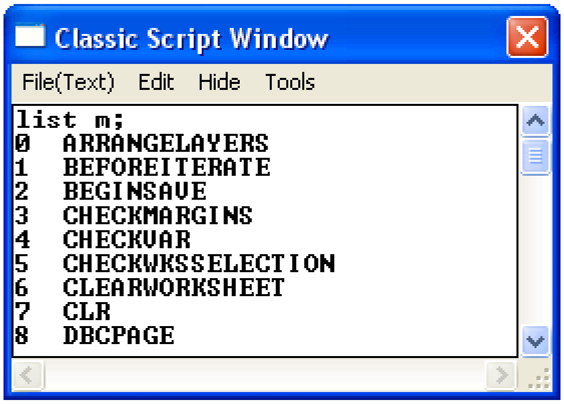
Let's look at Origin's predefined system macros, some of which take in arguments. Macros can take up to five arguments. Use the %1, %2,? %5 notation within the script definition to indicate that the macro expects one or more arguments (%1= 1st argument, %2= 2nd argument,?%5 = 5th argument).

1. In the Script Window, type the following:

```
list m
```

2. Press **ENTER**.

Origin responds by typing the names of predefined macros into the Script Window.



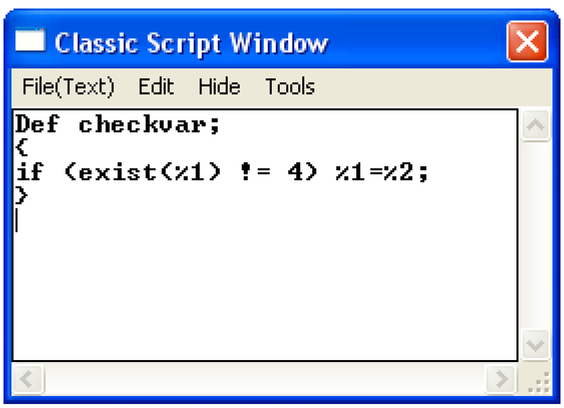
To see the definition of any system macro:

3. To see how a system macro is defined, type **def *macroname***. For example, type the following in the Script Window:

```
def checkvar
```

4. Press **ENTER**.

Origin responds to the Script Window as:



The %1 notation in the macro definition indicates that this macro takes one argument.

To define a new macro as a system macro:

5. Type the following in the Script Window:

```
def graph {
    set %1 -s 1;layer -i %1
}
```

6. Press **ENTER**.

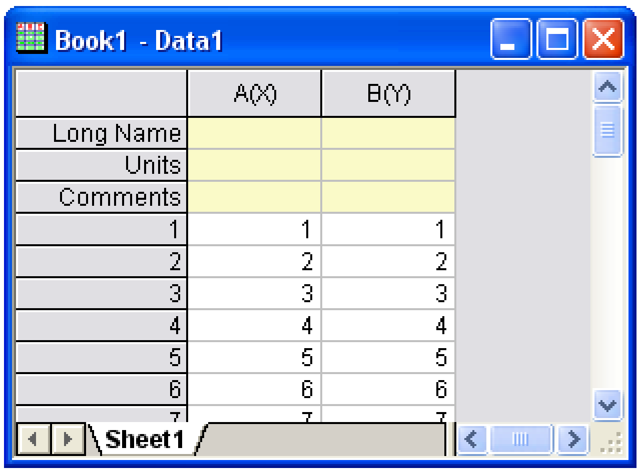
Notes: We used the set data set ?s 1 command to show a data plot. The layer -i data set command

adds (plots) the named data set onto the active layer.

To call a macro:

7. Click the **New Worksheet** button 

8. Create a worksheet named *Book1* and type in the following data:



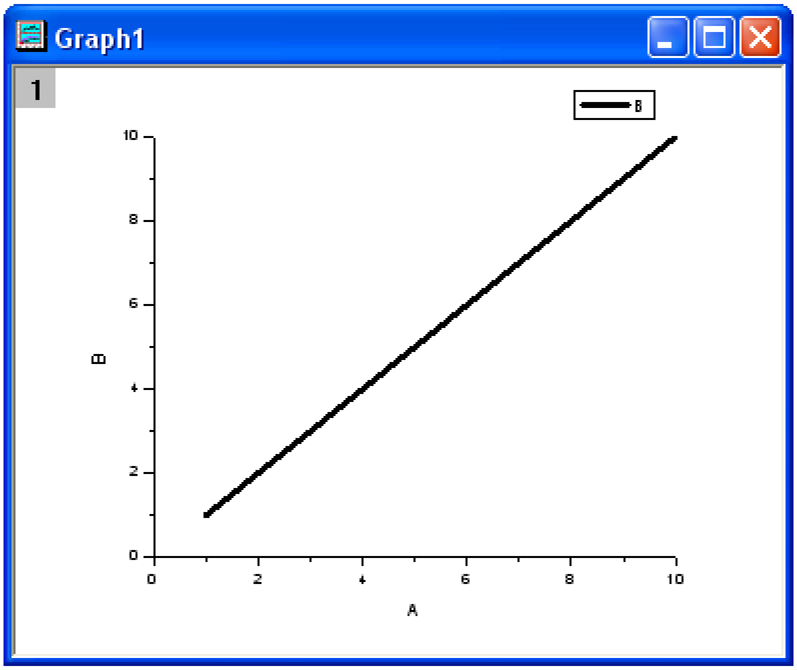
9. Click the **New Graph** button 

10. In the Script Window, type the following:

```
graph book1_b
```

11. Press **ENTER**.

A line plot of the data set book1_b is included in the graph window.



We will modify the macro definition so that it creates a scatter plot using a red, ?up triangle? as the symbol.

You could hard code the appropriate values for scatter plot, red, and up triangle in the macro definition, but it is more efficient to pass the value of a variable as an argument. This way, the macro may be used in other instances when you want to set the color and symbol shape to something other than red and upward-pointing triangle .

> **Notes:** LabTalk often uses integer values to specify plot details. If you look at the color palette, for instance (from the menu, **Format: Color Palette**), you will see (assuming that you have not modified the default color palette) that black =1, red = 2, green = 3, blue = 4, etc. For more information, see documentation on the **Set** command in the LabTalk Language Reference section of the Programming Help file.

12. Type the following in the Script Window to redefine the graph macro:

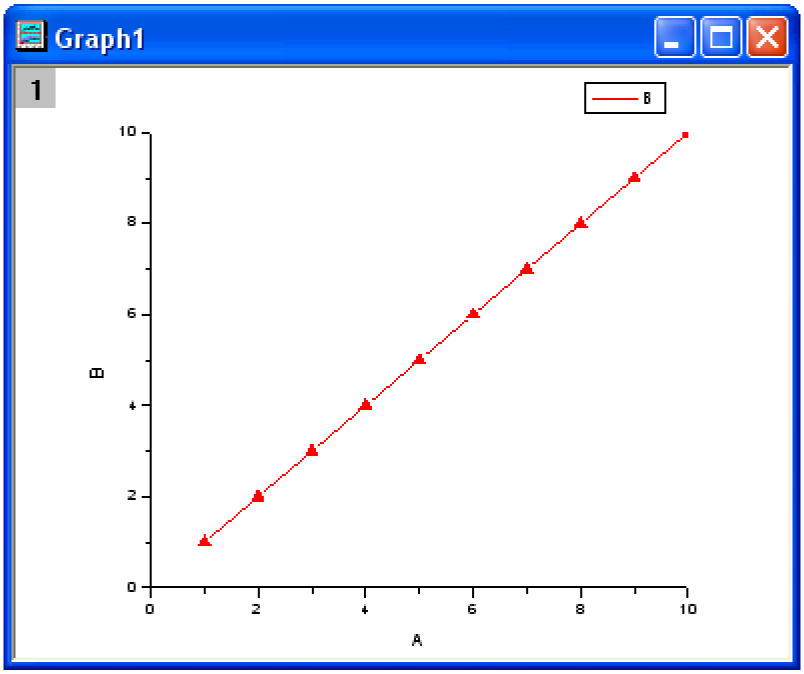
```
def graph {
    set %1 -s 1;
    layer -i %1;
    set %1 -c %2;
    set %1 -k %3;
}
```

13. Type the following:

```
graph book1_b 2 3
```

14. Press **ENTER**.

This plots the data set book1_b as red, upward-pointing triangles.



To better understand what we actually did, let?s examine our macro line by line.

First, we used the **def** command to tell Origin that we are defining a macro.

```
def graph
```

Secondly, we used the LabTalk **set** command with the **? s** option.

The **set ?s** command syntax is:

```
set dataset -s value
```

where *dataset* is the name of a data set, and *value* is either 1 (show plot) or 0 (hide plot),

```
set %1 ?s 1;
```

The layer **?i#** command syntax is:

```
layer ?igraphType dataset;
```

Note that the data set name has been assigned to %1.

The **set ?c** command syntax is:

```
set ?c color#
```

This is used to specify the plot symbol color. Note that color will be assigned to %2.

The **set ?k** command syntax is:

```
set ?k shape#
```

This is used to specify symbol shape. Note that symbol shape is assigned to %3.

When we execute our macro by typing

```
graph book1_b 2 3
```

we are passing three arguments to the macro:

- book1_b, which is substituted for %1.
- 2, which is substituted for %2.
- 3, which is substituted for %3.

Note that it is merely coincidence that we chose to substitute a value of 2 for %2, and a value of 3 for %3. We could have chosen any allowed value for symbol color or shape.

Remember that any macro that you define is only available for the duration of your Origin session. If you restart Origin, you cannot execute your macro until you define it again. If you want your macro to be defined automatically when you start Origin, you can save your macro definition to Origin's MACROS.CNF file. Each time Origin starts, it reads MACROS.CNF, and your macro is defined.

> **Notes:** MACROS.CNF is located in the Origin software folder. Because of a turf battle over the .CNF file extension, it will probably only be listed as MACROS and will display a terminal icon. In reality, this file is a text file and can be opened in any text editor, such as Notepad and Origin Code Builder.

For more information on macros, see **Help: Programming: Labtalk** in the Help menu.

Loading and Compiling your Origin C Function using LabTalk script

Before an Origin C function can be used, it must be compiled and linked in the current Origin session. Origin provides the following method to programmatically compile and link a source file, or to programmatically build a workspace, from LabTalk.

```
err = run.LoadOC("myFile",[option]);
```

> **Notes:** For more on the LabTalk run object in **Help: Programming:Labtalk** Help.

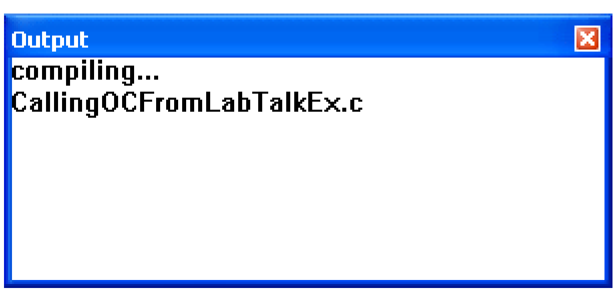
The following example demonstrates how to programmatically load and compile an Origin C source file.

To begin this tutorial:

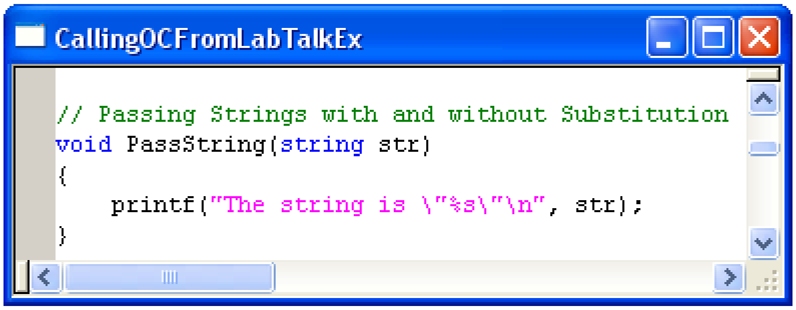
1. On the **Standard** Toolbar, click the **Code Builder** button .
2. Return to the Origin workspace and open a New Project (**File: New?Project**).
3. Open the Script Window (**Window: Script Window**) and type the following:

```
string fld$="Samples\Origin C Examples\Programming Guide\Calling
Functions\";
string fname$=System.path.program$ + fld$ + "CallingOCFromLabTalkEx.c";
run.LoadOC(%(fname$), 1);
```

The CallingOCFromLabTalkEx.c function is compiled now and can be found under the User folder in Code Builder's Workspace view. The Output Window reports as follows in the Code Builder workspace:



The Origin C functions in CallingOCFromLabTalkEx.c are now accessible. You can call the following section from the C file. Note the comments (the green text with leading //).



To execute the PassString function:

4. Return to the Origin workspace, open the Script Window (**Window: Script Window**) and type in the following:

```
PassString abc
```

5. Press **ENTER**. Origin returns:

```
The string is "abc"
```

Using .OGS Files to Store Script

As an alternative to associating your LabTalk script or Origin C function with a button, you could save your script to .OGS files. The advantage is that these .OGS files are self-contained and can be called from many buttons.

These .OGS files are organized by sections. Sections are identified by a name surrounded by square brackets, as in this example:

```
[Main]
```

To execute the code in a portion of a .OGS file, you need only identify the .OGS file and refer to the section containing the code by name, as in this example:

```
run.section(test.ogs, Main)
```

Most of Origin's menu and toolbar commands run LabTalk script in a .OGS file. These files can be opened and edited in Code Builder.

In this tutorial, we will create a new .OGS file, and associate the .OGS file with a new toolbar button.

> **Notes:** This tutorial assumes that you have already created and saved an Origin C file called test.c, as prescribed under **Tutorial: Organizing and Accessing Origin C Functions**.

To create a new .OGS file:

1. From the **Code Builder** menu, select **File: New**.
2. In the **New File** dialog box, select **LabTalk Script File**.
3. In the **File Name** text box, type:

```
Test
```

4. Click **OK**.

You now have an empty document called **test.ogs**.

We will use the run.LoadOC script command to programmatically compile and link the test.C Origin C source file. The advantage of this method is that it allows you to program your buttons or other user-created visual objects to make behind-the-scenes calls to your Origin C functions.

To make a call to an ?uncompiled? Origin C function from Origin:

5. In the blank **test.ogs** window, type the following:

```
[CreateGraph]
run.LoadOC("test.c");
Plot_Data("scatter","book1_b");
```

6. From the Code Builder menu, select **File: Save As** and save the file to your main Origin software folder.

7. Return to the Origin workspace.

8. From the Origin menu, select **View: Toolbars**. This opens the **Customize Toolbar** dialog.

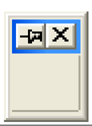
9. On the **Toolbars** tab, click the **New** button to open the **New Toolbar** dialog.

10. Type in the following name for your new toolbar:

```
My Toolbar
```

11. Click **OK**.

My Toolbar is added to the **Toolbars** list. A new toolbar is added to the Origin workspace.

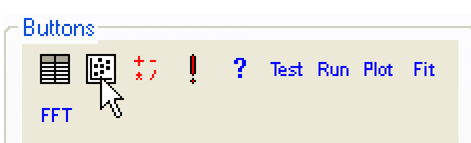


Notes: The toolbar may be a bit hard to spot because it does not contain any buttons.

12. Return to the **Customize Toolbars** dialog and select the **Button Groups** tab.

13. Scroll to the bottom of the **Groups** list and select **User Defined**.

14. Select the second button in this group.



15. Click the **Settings** button. This opens the **Button Settings** dialog box

16. In the File Name text box, type **test.ogs**.

17. In the Section Name text box, type **CreateGraph**. We are not passing arguments to the section, so we can skip the **Argument List** text box.

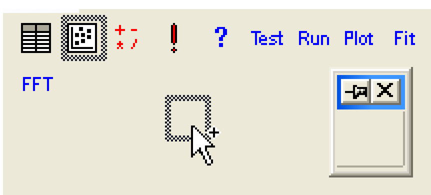
18. In the **Tool Tip Text** box, type **CreateGraph**. A Tool Tip is the message that displays when you mouse over a toolbar button.

19. In the **Status Bar** text box, type **Example, plotting data from Origin C** as Status Bar Text. When you mouse over a toolbar button, the Status Bar message displays in the lower left corner of your Origin workspace.

20. In the **Context** group, verify that the **Windows** radio button is selected and clear the **Graph**, **Matrix**, **Layout**, and **Excel** check boxes. Leave only **Worksheet** selected. This limits toolbar availability to active worksheets.

21. Click **OK**.

22. Point to the toolbar button, hold down the left mouse button and drag the button to the floating toolbar.



23. Click **Close** to close the **Customize Toolbar** dialog.

To test this method, close and restart Origin. Remember that the second argument to our function is book1_b, so the Origin workspace will need to have a worksheet named book1, a B(Y) column and some data in both the X and Y columns. Note, too, that our toolbar button is grayed out when a graph is the active window.

This concludes the tutorial on **Advanced Scripting Using LabTalk**.

10.2 Origin C

Topics covered in this section:

1. Introduction to Origin C and Code Builder
2. The Code Builder Workspace
3. Adding New Origin C Functions to Origin
4. Organizing and Accessing Origin C Functions

5. Calling NAG Functions From Origin C
6. Accessing Internal Origin Objects by Origin C

10.2.1 Introduction to Origin C and Code Builder

Summary

Origin C supports a nearly-complete ANSI C language syntax as well as a subset of C++ features including internal and DLL-extended classes. In addition, Origin C is "Origin aware". This means that Origin objects such as worksheets and graphs are mapped to classes in Origin C, allowing direct manipulation of these objects and their properties from Origin C.

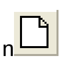
Origin C's integrated development environment (IDE) is called Code Builder. Code Builder provides standard tools for writing, compiling, and debugging your Origin C programs. Once an Origin C function is compiled, the function can be called in various ways from the Origin or Code Builder workspaces.

Minimum Origin Version Required: Origin 8.0 SR0

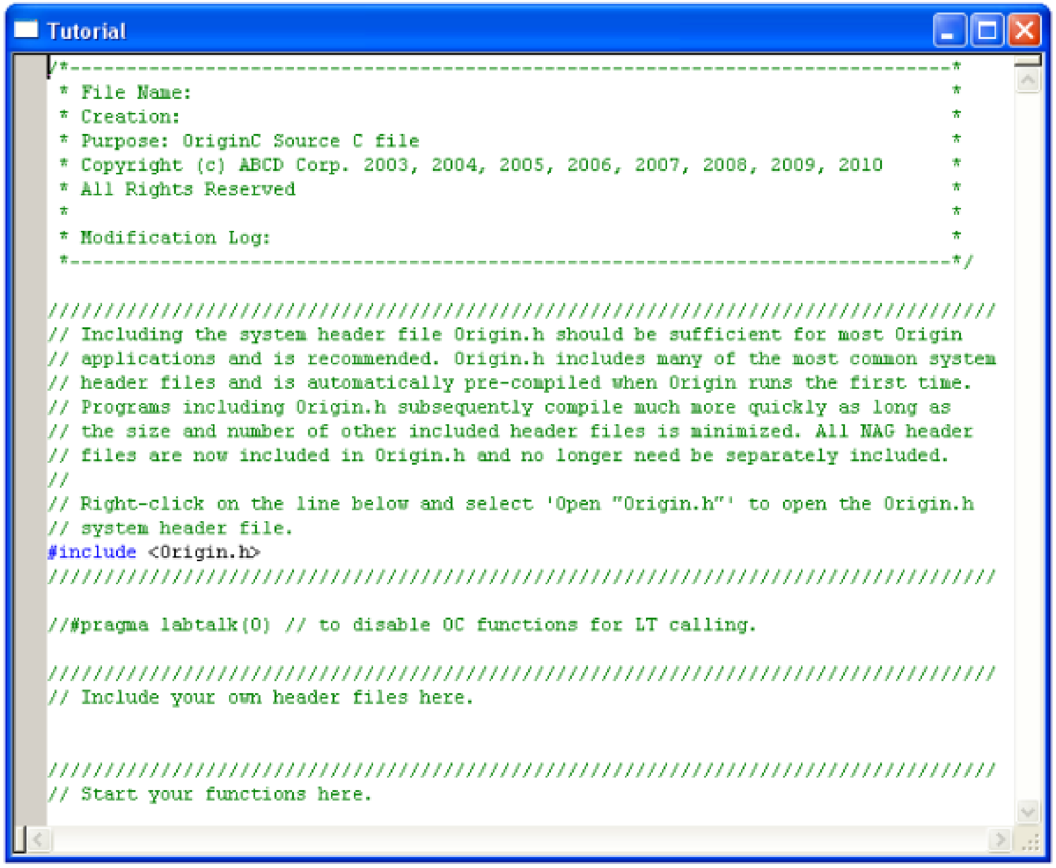
What you will learn

This tutorial introduces you to Origin C and Code Builder by showing you how to write, compile and call a function that types the message "Hello World!!!".

Steps

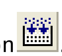
1. On the Origin Standard toolbar, click the **Code Builder** button
2. On the Code Builder toolbar, click the **New** button

   In the **New File** dialog, select **C File**.
   In the **File Name** text box, type **Tutorial**.
   In the **Location** text box, specify where to save your source file, the default path is Origin C subfolder under User File Folder(Note: The Browse button looks like this:).
   Click **OK**. A file named Tutorial.c opens in the Code Builder workspace.



3. Type the following beneath the line that reads // start your functions here:

```
void test ()
{
   printf("Hello World!!!\n");
}
```

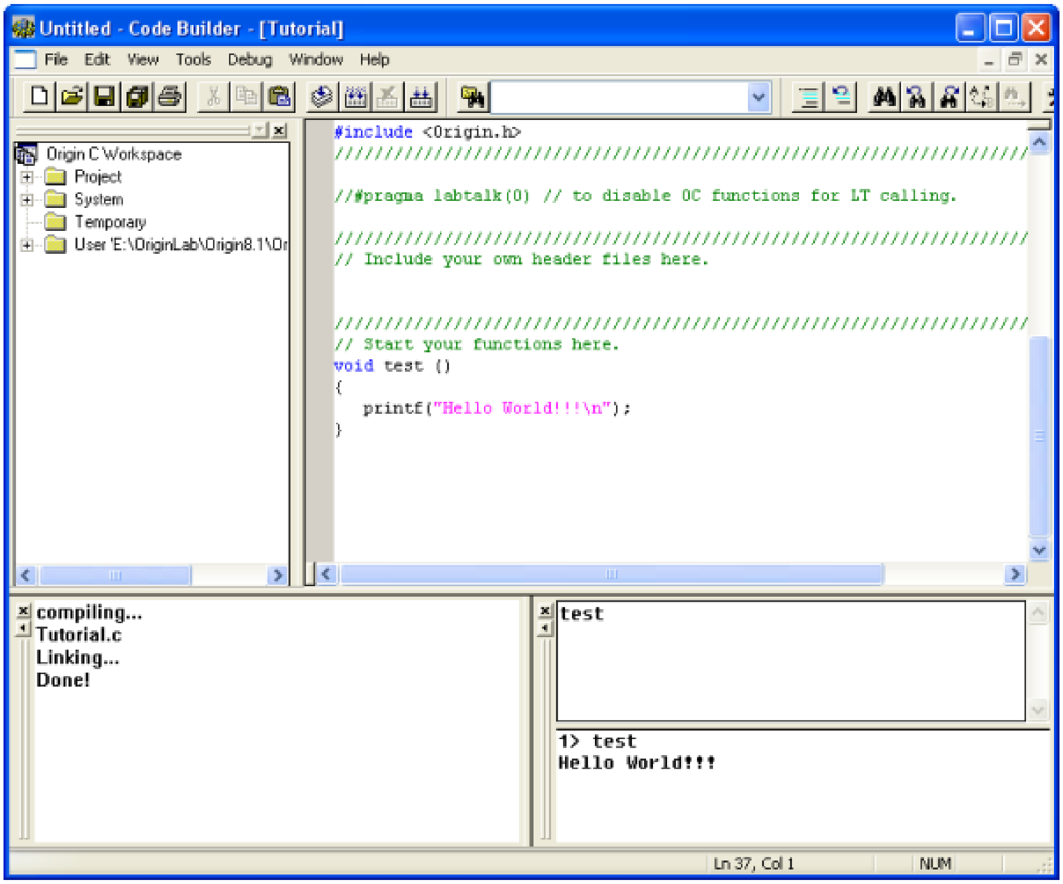
4. On the Code Builder workspace Standard toolbar, click the **Build** button . This compiles the test function.

5. To call this function, click in the upper pane of the **LabTalk Console**. This is located in the lower right corner of the **Code Builder** workspace (This is the default location. If the LabTalk console isn't visible, select **View: LabTalk Console** from the Code Builder menu and make sure that the menu item is checked).

6. Type the following LabTalk function call in the **LabTalk Console**:

```
test
```

7. Press **ENTER**.



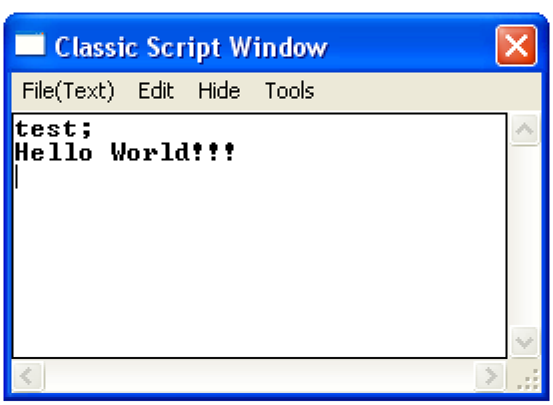
To test this function in the Origin Script Window:

8. Return to the Origin workspace, and select **Window: Script Window**.

9. In the Script window, type the following:

```
test
```

10. Press **ENTER**.



"**Hello World!!!**" displays in the Script Window.

This concludes the **Introduction to Origin C and Code Builder** tutorial.

10.2.2 The Code Builder Workspace

Summary

In this exercise, we will create a workspace, add a source file with a new function, then build, test and save the workspace file.

Minimum Origin Version Required: Origin 8.0 SR0

What you will learn

This tutorial will show you how to:

- Build a Workspace File
- Build Workspace Folders
- Build on Startup

The Workspace File

A workspace is a collection of files that can be opened by a single menu option (**File: Open Workspace?**) in Origin's Code Builder. Any text file can be a part of the collection. They do not necessarily have to be source code files; they could be notes, for example.

All files opened in the Multiple Document Window by a workspace can be edited and saved individually. In addition to files being opened in the Multiple Document Window, source code files can be added to the Workspace Window with the **File: Add to Workspace** menu option.

By including source code files in the Workspace Window, you can build individual or multiple files with the appropriate menu option or toolbar button. Header files can be referenced within source files and do not need to be loaded in the Workspace Window or even open in the Multiple Document Window.

Since you can save a workspace with a new name, you can have multiple workspace files. However, only one workspace file can be open at a time.

To create a workspace:

1. On the **Standard** toolbar, click the **Code Builder** button .
2. From the Code Builder menu, select **File: New Workspace**. This creates a new workspace with the default name of "Untitled.ocw".
3. From the Code Builder menu, select **File: New**. This opens the **New** File dialog.
4. Choose **C File**, and type **foo** in the **File Name** text box. The **Add to Workspace** and **Fill with Default Contents** check boxes should be selected. You may accept the default Location. Click **OK**.
5. In FOO.C, starting below the line that says ?\\start your functions here", type the following:

```
void bar()
{
```

```
        printf("Hello World!\n");
}
```

6. Click the **Build** button . Origin automatically saves the source file and compiles and links the function.

7. From the Origin menu choose **Window: Script Window**.

8. To test our new function, type:

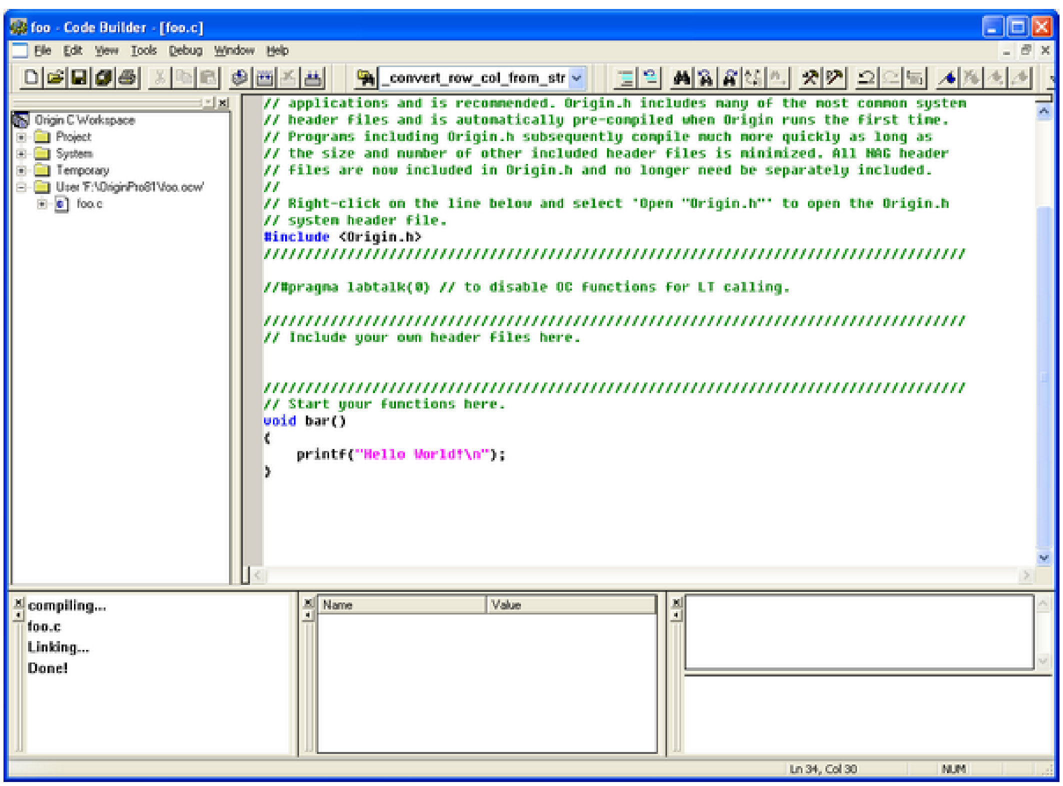
```
bar
```

9. Press **ENTER**

Origin responds by typing **Hello World!**

10. From the Code Builder menu, choose **File: Save Workspace As?.**

The figure shows the foo.ocw workspace file containing a single source file, foo.c, in the Multiple Document Window. The file has been added to the Workspace Window. The Output Window shows that the file has been compiled. The source file contains a single function ? bar( ) ? which is listed in the tree structure of the workspace.



Workspace Folders

The Code Builder workspace has four subfolders named Project, System, Temporary, and User. Files added by user, such as foo.c that you just added, are placed in the User subfolder. Origin, itself, uses Origin C for many analysis routines. When these routines are accessed, the Origin C source files are loaded into the Workspace into either the System subfolder or the Temporary subfolder. The Project subfolder is reserved for files that are saved and loaded with the Origin project. This aspect of attaching a file to the project is described in a separate tutorial.

Build on Startup

If you right click on "Origin C Workspace", the shortcut menu has a **Build on Startup** option. When this option is checked, the last workspace you saved will be loaded when you restart Origin. All source files in the Workspace will be built and all functions in the source files will be available for immediate use.

For information on building individual source files on startup by including information in the ORIGIN.INI file, please view Build on Startup

This concludes the tutorial on the **Code Builder Workspace**.

10.2.3 Adding New Origin C Functions to Origin

Summary

Functions written in Origin C are accessible from various locations within the Origin interface, such as the Script Window, provided they meet the following criteria: the function should return either void (as in the previous tutorial), double, string, or vectors of type double or string. Variables passed to the function from Origin should be of type double or string, or vectors of these types. Functions that do not meet these criteria are not callable from the Origin interface, but can be called within other Origin C functions. Note that although an Origin C function that accepts and returns type int can be called from the Origin interface, the data may be truncated since the interface only supports type double.

In this tutorial, you will be introduced to writing a math function that returns computed values. We will first create a function that returns type double to Origin, and then we will create a function that returns vectors of type double.

Minimum Origin Version Required: Origin 8.0 SR0

What you will learn

How to add a new function and how to run this function in the Script Window.

Steps

1. Start a new Origin C file in **Code Builder**.
2. Enter the following code:

```
double myfunc1(double x, double a)
{
        return sin( a * x );
}
```

3. Click the **Build** button to compile the function.

This function can now be called from the Origin interface, in places such as the Script Window.

4. Go to the Script Window, and type in the following lines, pressing **ENTER** at the end of each line:

```
y = myfunc1(2, 3)
y =
```

You can also use worksheet cells instead of absolute numbers:

5. Make a worksheet active, enter a number in the first row of column A. Then type the following into the Script Window and press **ENTER**:

```
col(B)[1] = myfunc1(col(A)[1], 3)
```

Note that a function such as myfunc1, that accepts and returns type double, can also be used to perform vector operations.

6. Fill rows 1 through 10 of Column A with numbers, and type the following into the Script Window:

```
col(B) = myfunc1(col(A), 3)
```

In the above example, Origin calls the myfunc1 function for each row of column A. For performing vector operations as above, it is more efficient to write functions that accept and return vectors.

7. Go back to Code Builder and add the following function to the same file, and compile the file by clicking the **Build** button 

```
vector<double> myfunc2(vector<double> vecIn, double a)
{
        vector<double> vecOut;
        vecOut = sin( a * vecIn );
        return vecOut;
}
```

8. Go back to the Origin interface, fill Column A with some new numbers, and type the following into the Script Window:

```
col(B) = myfunc2(col(A), 3)
```

The function myfunc2 is called only once for computing the entire column.

Note that you can use such functions in other places such as the "Set Column Values" dialog. The Auto Update feature of "Set Column Values" can be enabled by checking the appropriate check box in this dialog. As long as the Origin C function is compiled and ready to be called from Origin, any changes to the source column will result in an update of the destination column.

10.2.4 Organizing and Accessing Origin C Functions

Summary

Techniques for using your Origin C functions.

Minimum Origin Version Required: Origin 8.0 SR0

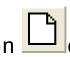
What you will learn

This tutorial will show you how to:

- Save your Origin C Function with your Project
- Associate your Programs with Visual Objects
- Load and Compile your Origin C Function from script

Saving your Origin C Function with your Project

One way to load and compile your Origin C function is to save the Origin C file as an attachment to your Origin Project (*.OPJ) file. When a project file is opened, all files attached to it are separated out and stored in a temporary folder. In addition, any Origin C file that was attached is also automatically loaded into the Code Builder workspace, and compiled. The function is then ready to be called upon opening the Origin Project file.

1. Start a new Origin project file by clicking on the **New** Project button  on the **Standard** Toolbar.
2. On the **Standard** Toolbar, click the **Code Builder** button .
3. From the Code Builder menu, select **File: New**. This opens the **New File** dialog box.
4. In the top list-box, select **C File**.
5. In the **File Name** text box, type: **Test**. Keep the **Add to Workspace** check box selected. Click **OK**. The file Test.c is added to the workspace.
6. Select and copy the following function, and paste it into the Test.c file. Be sure to paste the text below the line that reads "//start your functions here."

```
void Plot_Data(string strTemplate, string strData)
{
	// Create a graph window from a Template
	GraphPage gp;
	BOOL bOK = gp.Create(strTemplate, CREATE_VISIBLE);
	if( !bOK )
		return;

	// Attach the first layer (0) to a GraphLayer object
	GraphLayer gl = gp.Layers(0);

	//Attach a dataset to a Curve object
	Curve crv(strData);

	// Add the Curve to the graph layer
	int nPlot = gl.AddPlot(crv);
	if(nPlot >= 0)
	{
		// Set plot color to Green(2)
		gl.DataPlots(nPlot).SetColor(2, TRUE);

		// Rescale this graph layer
```

```
                gl.Rescale();
        }
}
```

The Plot_Data function takes two arguments: (1) the template name and (2) the name of a Y dataset to include (plot) in the layer.

7. Click the **Build** button to compile and link the file.

8. Drag-and-drop the file Test.c from the User subfolder branch of the Workspace tree, to the Project subfolder. (**Hint:** You may need to first expand the User subfolder branch to display the Test.c entry prior to dragging the file).

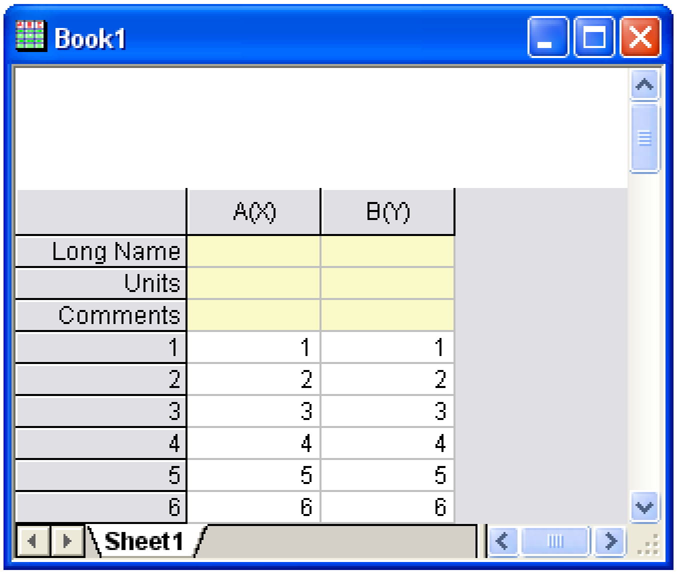
9. Go back to the Origin interface and save the project by clicking the **Save** button on the Standard Toolbar. Give the project the name Test.OPJ, and save it in a location of your choosing.

10. The Origin C file, Test.c, is now saved with the Project. To verify this, close the project, and go back to Code Builder. You will see that there are no entries under the Project subfolder of the Workspace tree. Now go back to Origin interface and reopen the project. Go to Code builder and verify that Test.c is now listed under the Project subfolder (**Hint:** you may need to expand the Project subfolder branch to see the Test.c entry).

Associating your Programs with Visual Objects

You will now learn how to create a button on a worksheet and program the button to call the Origin C function in the Test.c file that you saved with the project.

1. Open the Test.OPJ project that you saved under step 9 (previous section).
2. Highlight the A(X) and B(Y) columns, right-click and select **Fill Columns With: Row Numbers.**
3. From the menu, select **Format: Worksheet** to open the **Worksheet Properties** dialog box.
4. In the **Size** tab, **Worksheet Measurement** branch, set the **Gap from Top** to **40** and click **OK** to close the dialog.



The worksheet now has sufficient space above the column headings to add a text label.

Right-click in the area directly above the two columns and choose **Add Text**.

5. At the cursor, type: **Plot Data**.

6. Click once outside the text label to deselect it.

7. Right-click on the text label and choose **Programming Control** to open the **Programming Control** dialog box. (**Hint:** Please choose **Label Control** in Origin 7.5)

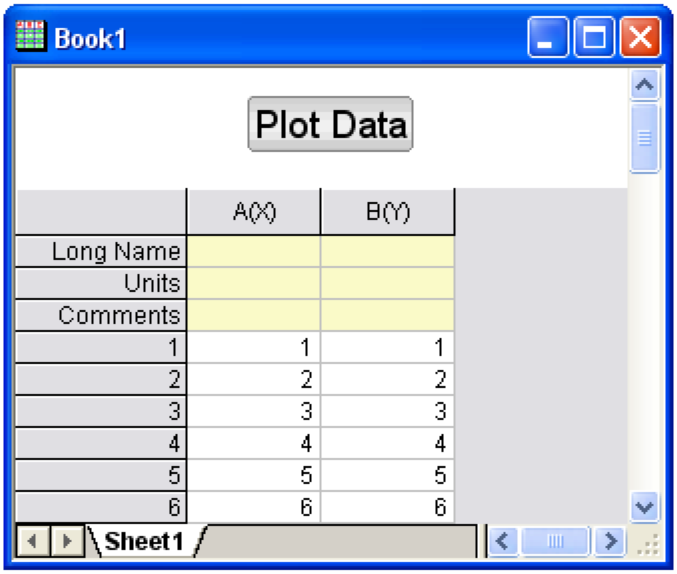
8. From the **Script, Run After** drop-down list, choose **Button Up**.

9. Type the following script in the text box at the bottom of the dialog box:

```
Plot_Data("scatter","book1_b");
```

10. Click **OK**.

Your text label will now look like a button.



11. Click the **Plot Data** button on your worksheet.

12. The Plot_Data function in your Test.c file is called, and a scatter plot is created.

> **Notes:** The script behind the button assumes that you have data in column B(Y) of the Data1 worksheet and that there is an associated X data set.

Loading and Compiling your Origin C Function from Script

In this tutorial we learned how to save an Origin C function along with the project file and then access the function from the Origin interface. Saving an Origin C file with a project limits access to functions within that file to only that project. When a new project is opened, the functions are not available any more.

To access functions in an Origin C file that is saved on disk, the file can be programmatically loaded and compiled using LabTalk script. The script command for performing the programmatic load and compile is run.LoadOC. Refer to the **LabTalk** Help files (**Help: Programming: Labtalk**) for more information on using this command.

This concludes the **Origin C Functions** tutorial.

10.2.5 Calling NAG Functions From Origin C

Summary

Calling a NAG function from an Origin C function is very much like calling any other Origin C function. You must familiarize yourself with the desired NAG function to gain an understanding of what parameters the function requires to be passed as arguments and what parameters the function returns. Once familiar with the function, you must develop code that follows the function's requirements.

The NAG header file containing the function's prototype must be included, required parameters must be correctly declared, sized, and initialized, and the function call must follow the function's prototype as described in header file. The objective of this tutorial is to demonstrate how to call a NAG function from an Origin C function.

Minimum Origin Version Required: Origin 8.1 SR1

What you will learn

This tutorial will show you how to:

- Understand NAG functions
- Get Ready to Debug Sample Code
- Include the NAG Header
- How to See the Declaration of NAG Function
- How to Get NAG Error Code
- How to Use Function Pointer

Understand NAG Functions

The primary resource for understanding NAG functions is NAG library. The library also can be found in Origin C Reference. For example, d01ajc NAG function:

1. From the Origin menu select Help:Programming:OriginC. In the Origin C Reference book, expand the Global Functions book, expand the NAG Functions book, and choose Accessing NAG Functions Category and Help.
2. Select Quadrature (d01) category and then select nag_1d_quad_gen (d01ajc) function.
3. The selected page is one PDF file. Study the nag_1d_quad_gen function as needed to understand the description of the function, the function's prototype, and the description of all arguments. Sample data and an example program calling the function are also often included.

Secondary resource for understanding the Origin C NAG functions is Examples book. From the Origin menu select Help:Programming:OriginC, expand Examples book, expand Analysis book, choose Accessing NAG Functions, there are some examples to show how to call NAG functions in Origin C.

Get Ready to Debug Sample Code

The best way to understand how to write an Origin C function that calls a NAG function is to step through an example function in Debug mode. Follow the steps below to set up Origin and Code Builder to execute such a sample Origin C function in Debug mode.

1. From the Code Builder menu, select **File: New**. This opens the **New File** dialog box.
2. In the **File Name** text box, type: **TestNAG**, Keep **Add to Workspace** checkbox is checked. Click **OK**. The file TestNAG.c is added to the workspace.
3. Select and copy the following function, and paste it into the TestNAG.c file. Be sure to paste the text below the line that reads "// Include your own header files here."

```
// Include your own header files here.
#include <OC_nag8.h>

////////////////////////////////////////////////////////////////
// Start your functions here.
```

```
//NAG_CALL denotes proper calling convention. You may treat it
//like a function pointer and define your own integrand
double NAG_CALL f(double x)
{
  return (x*sin(x*30.0)/sqrt(1.0-x*x/(PI*PI*4.0)));
}

void nag_d01ajc_ex()
{
        double a = 0.0;
        double b = PI * 2.0;  // integration interval

        double epsabs, abserr, epsrel, result;
        // you may use epsabs and epsrel and this quantity to enhance
        // your desired precision when not enough precision encountered
        epsabs = 0.0;
        epsrel = 0.0001;
        // The max number of sub-intervals needed to evaluate the
        // function in the integral. The more diffcult the integrand
        // the larger max_num_subint should be.
        // For most problems 200 to 500 is adequate and recommmended
        int max_num_subint = 200;

        Nag_QuadProgress qp;
        static NagError fail;
        d01ajc(f, a, b, epsabs, epsrel, max_num_subint,
                       &result, &abserr, &qp, &fail);

        // For the error other than the following three errors.
        // which are due to bad input parameters or allocation failure
        // NE_INT_ARG_LT NE_BAD_PARAM   NE_ALLOC_FAIL.
        // You will need to free the memory allocation before calling
        // the integration routine again to avoid memory leakage.
        if (fail.code != NE_INT_ARG_LT &&
                fail.code != NE_BAD_PARAM &&
                fail.code !=  NE_ALLOC_FAIL)
        {
                NAG_FREE(qp.sub_int_beg_pts);
                NAG_FREE(qp.sub_int_end_pts);
                NAG_FREE(qp.sub_int_result);
                NAG_FREE(qp.sub_int_error);
        }

        printf("%10.6f", result);
}
```

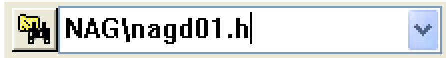
Include the NAG Header

```
#include <OC_nag8.h>
```

This header file containing all the header files of NAG functions, and all type define and error code define. So just include this one function should be enough.

How to See the Declaration of NAG Function

See the declaration of NAG functions from the header file:

1. Acivate TestNAG.c file that created in above section, move scoll bar to find out #include <OC_nag.h> line.
2. Right-click anywhere in the line and select **Open "OC_nag.h"**. This opens the header file containing all nag headers.
3. In the Search combo box 

type **NAG\nagd01.h** and press **ENTER** button to find out this line. Function **d01ajc** belongs to d01 category, so the header file name should be nagd01.h.
4. Right-click anywhere of this line to choose **Open "NAG\nagd01.h"**. This opens the header file including the prototype of the functions.
5. In the Search combo box, type **d01ajc** and press **ENTER** button to go to the declaration of this function.

To see the declaration of functions from NAG PDF:

- NAG PDF Online
- NAG PDF Files

How to Get NAG Error Code

1. Reactivate the TestNAG.c window in Code Builder. In this file, define a **NagError** variable **fail** and pass it as last argument to function **d01ajc**.
2. NAG function returns error code into NagError variable code item. In this example, can access NAG error code by **fail.code**.

How to know what error codes will be got:

3. Open Origin C Help from Origin menu **Help: Programming: OriginC**, expand Origin C Reference book, Global Functions book, NAG Functions book and choose Accessing NAG Functions Category and Help.

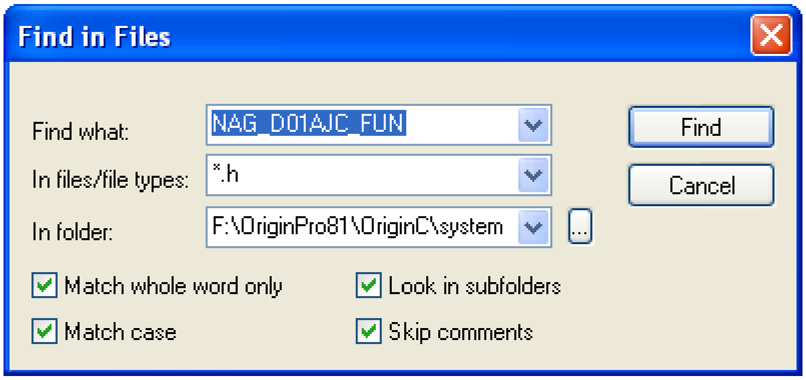
4. In **Chapters of NAG C Library** table, choose **d01** to enter Quadrature page, select d01ajc in the table of this category to open the PDF help of this NAG function.

5. Drag scall bar to page down to **5 Error Indicators and Warnings** part, there list all error codes for this function and the related description. Can directly use these error codes in Origin C if included correct header file (include <OC_nag8.h>, this header containing all NAG headers, or directly include <NAG8\nag_errlist.h>), for example NE_INT_ARG_LT, NE_BAD_PARAM, NE_ALLOC_FAIL used in TestNAG.c file.

How to Use Function Pointer

1. Open nagd01.h file from Origin program path \OriginC\system\NAG folder.
2. In this file, find out the declaration of **d01ajc** function. The first argument type of this function is **NAG_D01AJC_FUN**.

3. Double-click **NAG_D01AJC_FUN** to high-light it. From menu choose **Edit: Find in Files** to open **Find in Files** dialog. Set settings same as the following picture, click Find button.



4. Searching result display in Output window. Double click nag_types.h line to go to this file, **typedef NAG_D01_FUN NAG_D01AJC_FUN** line, you can find the define of NAG_D01_FUN nearby.
5. The define of NAG_D01_FUN is

```
typedef double (NAG_CALL * NAG_D01_FUN)(double);
```

6. User defined function should keep the same return type and argument list as this define. And **NAG_CALL** denotes proper calling convention and it should be used in your own function.
7. Activate TestNAG.c file. There is a function named **f** and it used as function pointer in **d01ajc** as the first argument.

```
double NAG_CALL f(double x)
{
  return (x*sin(x*30.0)/sqrt(1.0-x*x/(PI*PI*4.0)));
}
```

10.2.6 Accessing Internal Origin Objects by Origin C

<u>Summary</u>

Internal Origin objects (such as Project Explorer folders, Origin windows (pages), layers, plots, graphic objects, data sets, etc.) are accessed using Origin C classes. To access or programmatically control an internal Origin object, you must attach it to an Origin C object.

To attach something to an internal Origin object you must first "find" it using the properties, methods, and collections of a container class. Common container classes include the Project, Folder, Page, GraphPage, Layer, GraphLayer, Worksheet, MatrixLayer, and Collection classes. Once found, an internal Origin object can easily be attached to an Origin C object of the appropriate type.

The internal Origin object is then programmatically controlled by manipulating the class methods and properties of the attached Origin C object. The objective of this tutorial is to demonstrate how to find particular internal Origin objects, attach things to those objects, and access the objects by manipulating the methods and properties of the attached Origin C objects.

Minimum Origin Version Required: Origin 8.1 SR1

What you will learn

This tutorial will show you how to:

- Access Worksheet Related Objects
- Access Graph Related Objects

Accessing Worksheet Related Objects

Familiarity with the Origin C Project class (Project.h), the Collection class (Collection.h), and the Folder class (Folder.h), is valuable when attempting to understand how to find particular internal Origin Objects. Users may find it helpful to preview these classes in the **Origin C Reference: Classes** book of **Origin C Help** or in the above header files located in the ..\Origin\OriginC\system subfolder.

To begin this tutorial:

1. On the **Standard** toolbar, click the **New Project** button .
2. On the **Standard** toolbar, click the **Code Builder** button .
3. On the **Code Builder** menu, select **File: New Workspace**.
4. On the **Code Builder** menu, click the **Open** button .
5. Browse to the \Samples\Origin C Examples\Programming Guide\Introduction to Accessing Origin Objects folder in the Origin software directory, select AccessWorksheetObjectsTutorial.c, check the **Add to Workspace** check box and click **Open**.
6. On the **Code Builder** toolbar, click the **Rebuild All** button . This compiles and links the file.
7. On the Code Builder **View** menu, verify that the **LabTalk Console** (Command & Results) and the **Local Variables** windows are visible (the corresponding menu items should be checked).
8. From the **Code Builder** menu, select **Tools: Customize**. Select the Toolbars tab and make sure that the **Debug** toolbar check box is selected.
9. In Code Builder, activate AccessWorksheetObjectsTutorial.c.
10. Near the top of the file, locate and click on the line:

```
PageBase pb;
```

You can position the cursor anywhere on the line.

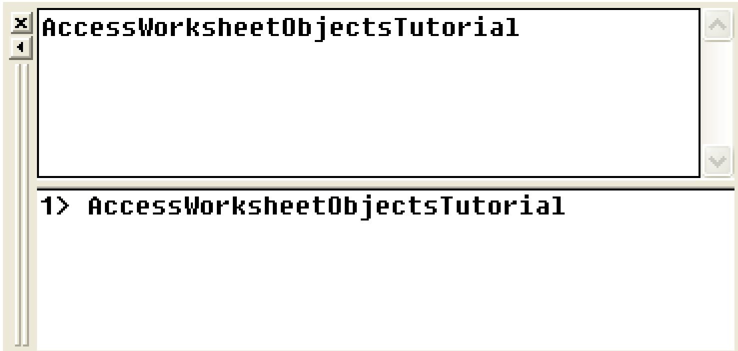
11. From the Code Builder menu, select **Debug: Toggle Breakpoints**. Alternately, press **F9** or click the **Toggle Breakpoint** button on the **Debug** toolbar.

A brown circle is displayed in the gray margin to the left of the above line indicating that a Debug breakpoint has been set for that line.

12. In the Code Builder workspace, activate the LabTalk Console (Command & Results window) and type in the following:

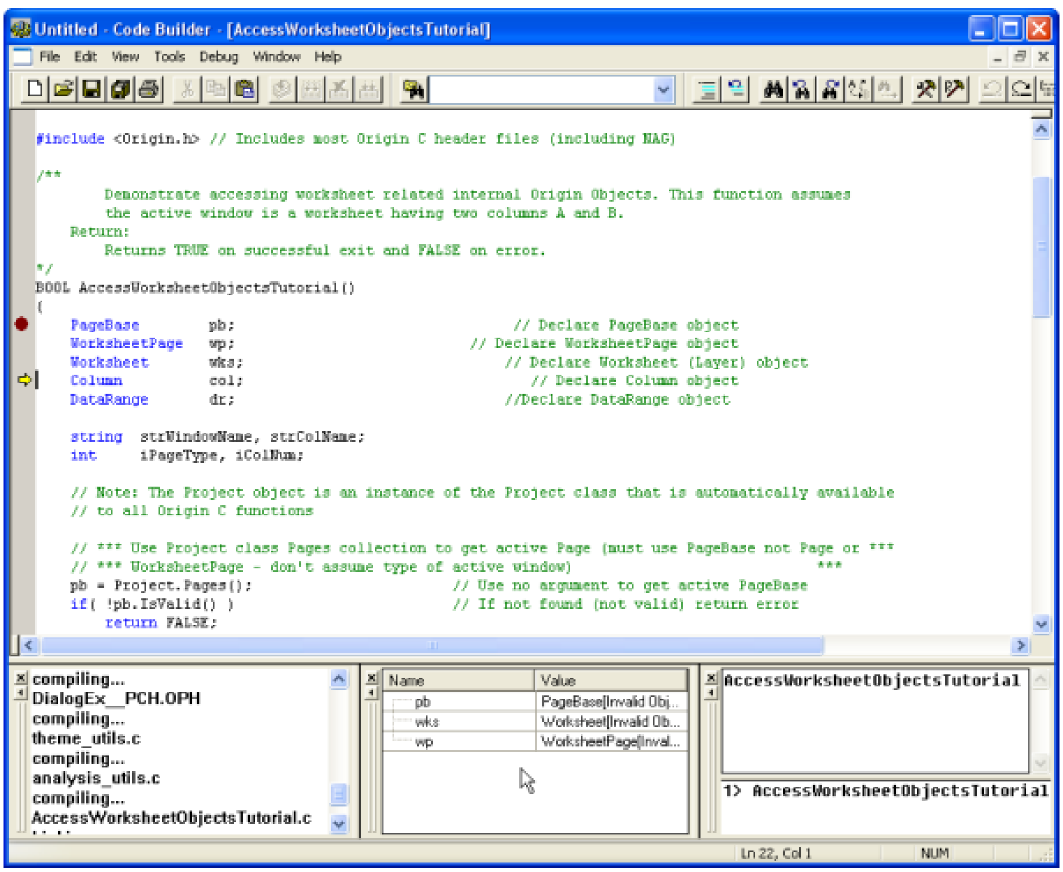
```
AccessWorksheetObjectsTutorial
```

13. Press **ENTER** to execute the function.



14. On the **Debug** toolbar, press the **Step Into** button 

15. Press the **Step Into** button repeatedly, stopping to read the comments for each statement. Periodically stop and resize and/or reposition the Local Variables window to view the current run-time value of each variable.



Accessing Graph Related Objects

1. Return to the *Origin* workspace and, on the **Standard** toolbar, click the **Open** button .

2. Browse to the \Samples\Origin C Examples\Programming Guide\Introduction to Accessing Origin Objects subfolder, select AccessGraphObjectsTutorial.OPJ, and click **Open**. You may be prompted to save changes to an untitled project. Click **No** and a worksheet and graph should open.
3. From the Code Builder menu, select **File: New Workspace**. Click **No** when prompted to save workspace changes.
4. In Code Builder, click the **Open** button .
5. Browse to the \Samples\Origin C Examples\Programming Guide\Introduction to Accessing Origin Objects subfolder, select AccessGraphObjectsTutorial.c, select the **Add to Workspace** check box, and click **Open**.
6. Click the **Rebuild All** button to compile and link the file.
7. On the Code Builder **View** menu, verify that the **LabTalk Console** (Command & Results) and the **Local Variables** windows are visible (the corresponding menu items should be checked).
8. From the Code Builder menu, select **Tools: Customize**. Select the Toolbars tab and make sure that the **Debug** toolbar check box is selected.
9. In the Code Builder workspace, activate AccessGraphObjectsTutorial.c.
10. Near the top of the file locate and click on the line:

```
GraphPage gp;
```

You can position the cursor anywhere on the line.

11. From the Code Builder menu, select **Debug: Toggle Breakpoints**. Alternately, press **F9** or click the **Toggle Breakpoint** button on the **Debug** toolbar.

A brown circle is displayed in the gray margin to the left of the above line indicating that a Debug breakpoint has been set for that line.

12. Activate the LabTalk Console (Command & Results window) in Code Builder and type the following:

```
AccessGraphObjectsTutorial
```

13. Press **ENTER** to execute the function.
14. On the **Debug** toolbar, press the **Step Into** button
15. Press the **Step Into** button repeatedly, stopping to read the comments for each statement. Periodically stop and resize and/or reposition the Local Variables window to view the current run-time value of each variable.

This concludes the **Internal Origin Objects** tutorial.

10.3 X-Functions

Topics covered in this section:

10.3.1 Command Window and X-Functions

Summary

Many of Origin's analysis tools and other data processing tools have been implemented using X-Functions. The Command Window provides a convenient way to run these functions.

Another important use for the Command Window is to send LabTalk script commands to Origin. Script commands can range from simple math and data operations, to user-created X-Functions or Origin C functions.

Minimum Origin Version Required: Origin 8.0 SR6

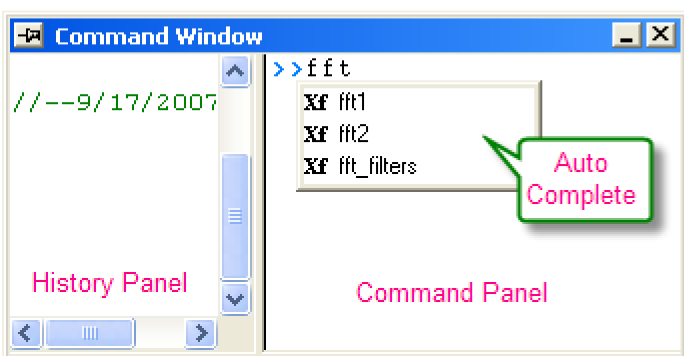
What you will learn

This tutorial will introduce you to the Command Window and show you how to:

- Perform simple calculation
- Access worksheet cells/columns
- Access X-Functions

Command Window

The Command Window consists of two panels: the **Command Panel** and **History Panel**:



The Command Window is normally located at the bottom right corner of the screen, but if it is not visible, you can access it by pressing **Alt+3** or by selecting **View: Command Window**.

When typing in the Command Panel, the Auto Complete support allows you to choose among X-Function script commands and OGS files in the current working folder. The command and OGS file name will be respectively preceded by Xf and LT. You can move up and down the list using the arrow keys; pressing Enter selects the item. After your selection, press the space bar and the Auto Complete now shows you the available options for the command.

Examples

Perform Calculations

The Command Window can be used as a calculator or to access any of Origin's mathematical functions. See the examples below.

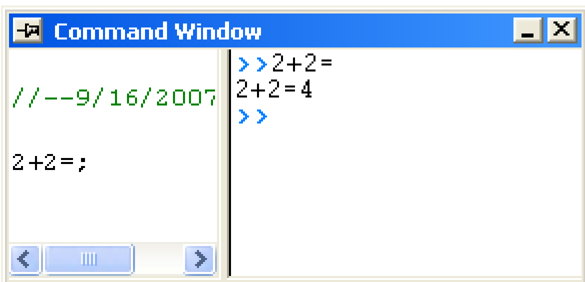
Single Line

One of the more basic uses for the Command Window is as an interface to perform simple calculations. For example, type the following:

```
2+2=
```

Press **ENTER**. Origin returns

```
2+2=4
```



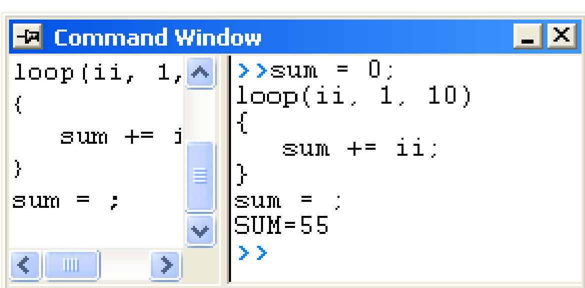
Multiple Lines

If you are typing multiple lines of scripts, first edit it in Code Builder (**View:Code Builder**) or any text editor, such as Windows Notepad, ending each line with a semi-colon, and then Copy + Paste the script in the Command Window, and press **ENTER** to execute. For example, paste the following script in the Command Window and **ENTER**:

```
sum = 0;
loop(ii, 1, 10)
{
   sum += ii;
}
sum = ;
```

Origin returns:

```
SUM=55
```

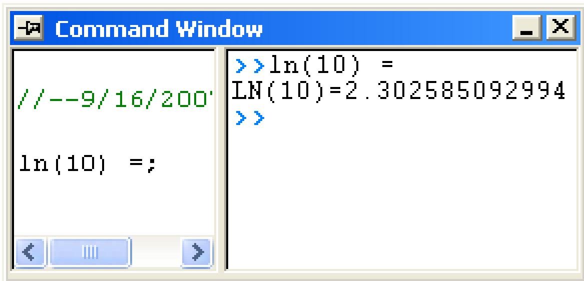


Functions

Any mathematical function, built-in as well as user-created, can be executed from the Command Window. For example, type:

```
ln(10) =
```

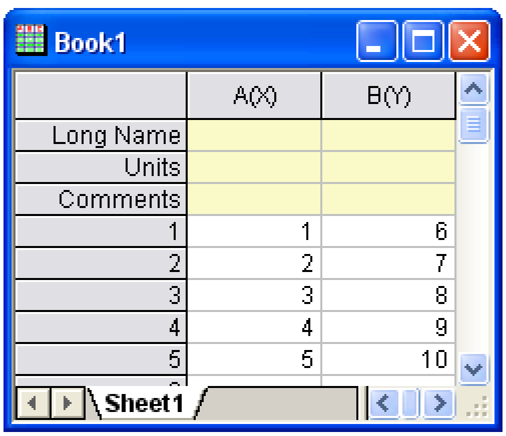
Origin returns natural logarithm value of 10.



Access Worksheet Values

You can also use the Script Window to read and write worksheet values, or to perform math operations on datasets.

1. Enter the following data into a fresh worksheet:



2. To return the value in the first cell of the second column, type the following:

```
cell(1,2)=
```

3. Press **ENTER**. Origin returns:

```
CELL(1,2)=6
```

You can also use the column name and row number to reference cell values.

4. Type the following:

```
col(B)[1]=
```

5. Press **ENTER**. Origin returns:

```
COL(B)[1]=6
```

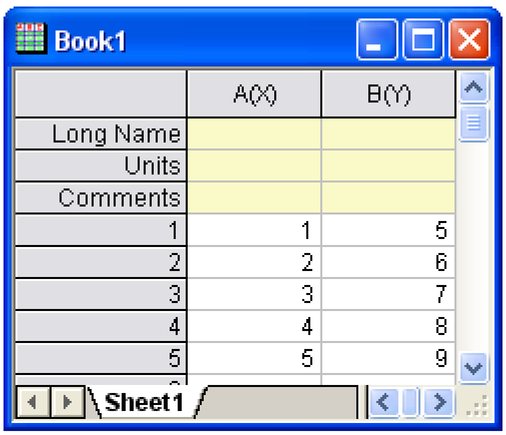
> **Notes:** In addition to using the column name, you can also use the dataset name. In LabTalk, the syntax for naming datasets is *worksheetName_columnName*. So, For example, *Book1_A[1]*= would return the first element of column A in worksheet Data1. Also, if the worksheet that you are referencing is the active window, you can use the LabTalk string variable %H, in place of the worksheet name. For example, %H_A[1].

To subtract the value in row 1 of column A, from all the values in column B?

6. Type the following:

```
col(B)=col(B)-col(A)[1]
```

7. Press **ENTER**. Your worksheet now reads:



Let's use what we have learned about executing multiple lines of script in the Script window. We will multiply every value in a column of data by some constant b.

8. Type the following:

```
b=3;
```

Press **CTRL+ENTER**. Recall that this gives us a carriage return without executing the command.

9. Now type:

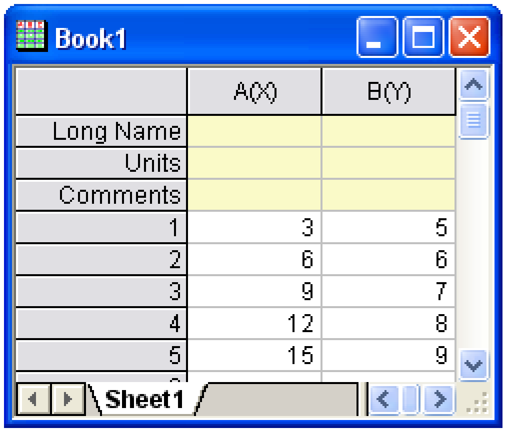
```
col(A)=col(A)*b;
```

Again, Press **CTRL+ENTER**.

10. Choose **Edit** from the Script window's menu bar. **Script Execution** should have a check mark next to it; if not, single-click on the menu item to place a check mark there.

11. Now, select the two lines of script that you just entered into the Script window and press **ENTER**.

Your worksheet now reads:



Notes: The following C notation is also supported:

b=3;

col(A)*=b;

You can also use linear interpolation or extrapolation on a specified *X dataset* to find the corresponding interpolated or extrapolated value in a *Y dataset*. This requires using a new notation with parentheses ( ) instead of brackets [ ].

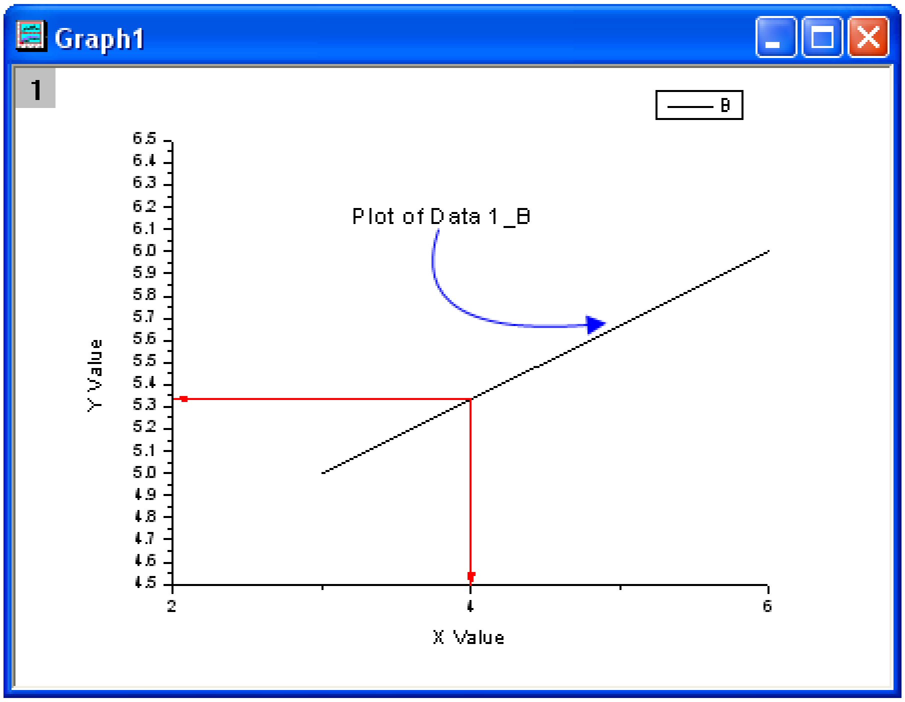
In this example, book1_b is a Y dataset and (4) is a value in an X dataset (book1_a) for which you want to find a corresponding, interpolated Y value.

12. Type the following:

```
book1_b(4) =
```

13. Press **ENTER**. Origin returns:

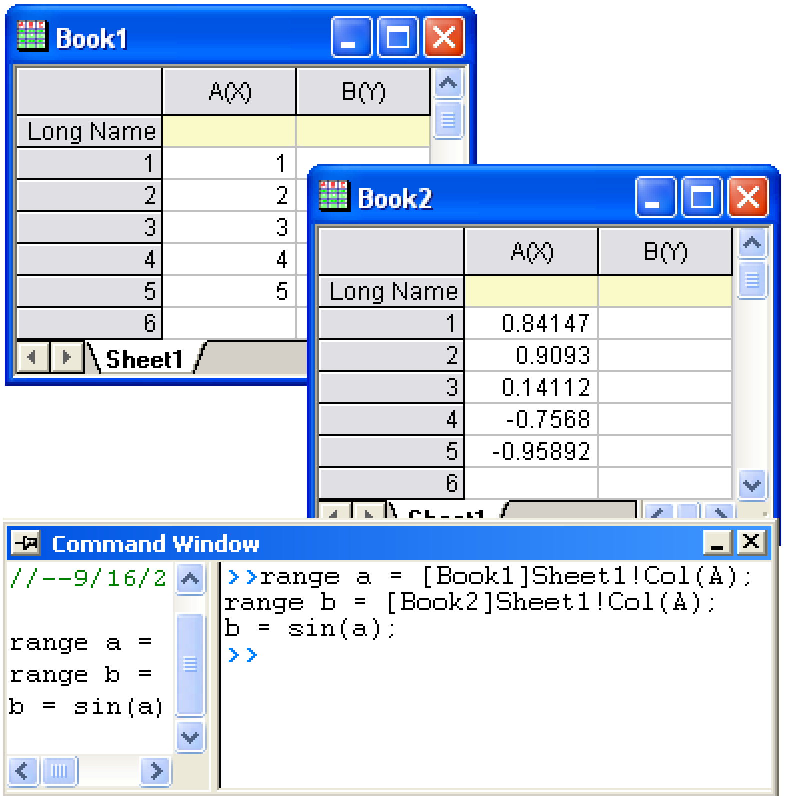
```
BOOK1_B(4)=5.333333
```



This is a line plot of our simple worksheet data. You can see that our interpolated Y value ? the one corresponding to X = 4 ? is 5.333333.

14. If the columns you work with are in different worksheet/workbook, you should use the range variables to represent the worksheet columns. For example, this script calculates the sine value on Book1 column A, and puts the result in Book2 column A (You must have Book2 before hitting Enter):

```
range a = [Book1]Sheet1!Col(A);
range b = [Book2]Sheet1!Col(A);
b = sin(a);
```



Access X-Functions

Origin 8 provides a large collection of X-Functions for performing a wide variety of data processing tasks. Of this collection, many of the X-Functions are accessible from LabTalk script. The functions accessible from script provide a powerful environment for users to create custom script code for their routine tasks.

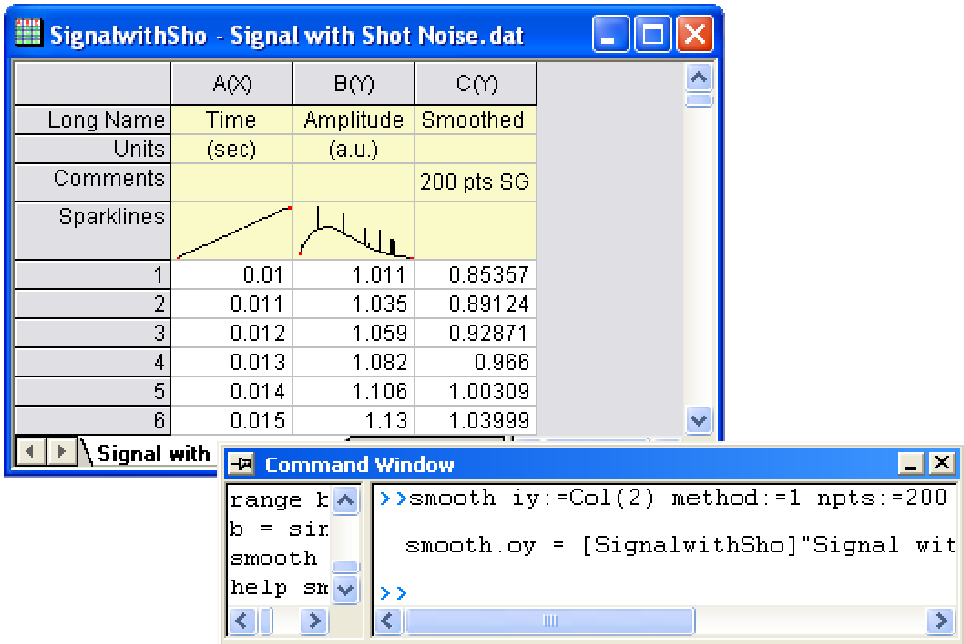
X-Functions that are accessible from script can be listed in the Command Window, and you can also obtain help on the command syntax as well as make use of auto completion of commands for such functions.

X-Functions accept data range string or range variable for specifying source and destination data for the operation. For example, the smooth X-Function under signal processing can be accessed from the Command Window as follows:

1. Import the file *\Samples\Single Processing\Signal with Shot Noise.dat*.
2. In the Command Window, type the following:

```
smooth iy:=Col(2) method:=1 npts:=200
```

3. When you press **ENTER**, the result will append to the source worksheet.



4. For help in using this smooth X-Function, you can type

```
help smooth
```

5. to open the corresponding Help.

10.3.2 Introduction to X-Functions

Summary

X-Functions provide a structured programming environment that offers a framework for building Origin tools. Different from the simple GetN box, creating tools by using X-Functions allows the user to focus on the actual data processing code and not have to worry about codes for the user interface.

Most of the dialogs/functions in Origin 8 are X-Functions, and many of them can be run from both menu and command line mode. The flexibility of running X-Functions makes them an attractive approach to customizing Origin

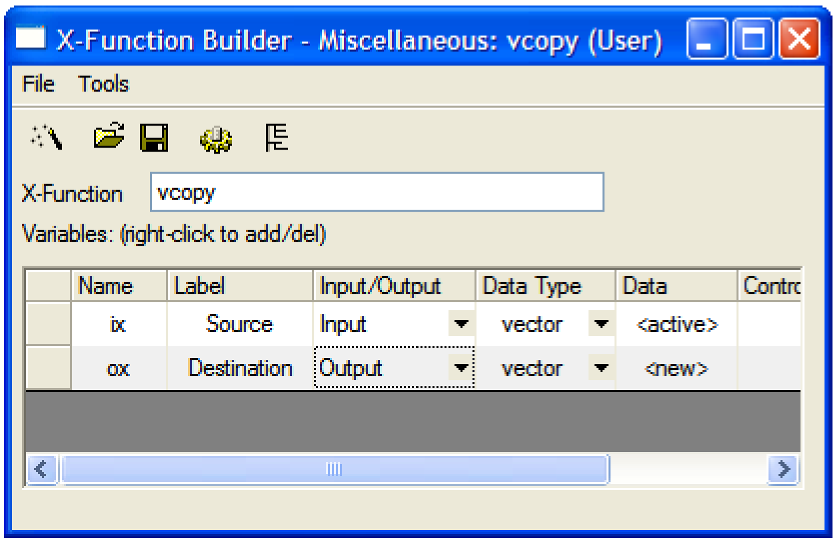
Minimum Origin Version Required: Origin 8.0 SR0

What you will learn

- How to create an X-Function
- How to make the X-Function script accessible
- How to use the X-Funciton in dialog mode

Create an X-Function

1. Select **Tools: X-Function Builder** or press **F10** to open the **X-Function Builder** dialog
2. Set **Name**, **Label** and **Data** of the 1st variable as **ix**, **Source** and **<active>**
3. Right click in the list panel and select **Add Variables** from the context menu.
4. Set **Name**, **Label**, **Input/Output** and **Data** of the 2nd variable as **ox**, **Destination**, **Output** and **<new>**
5. Select **File:Save** to save the x-function as vcopy

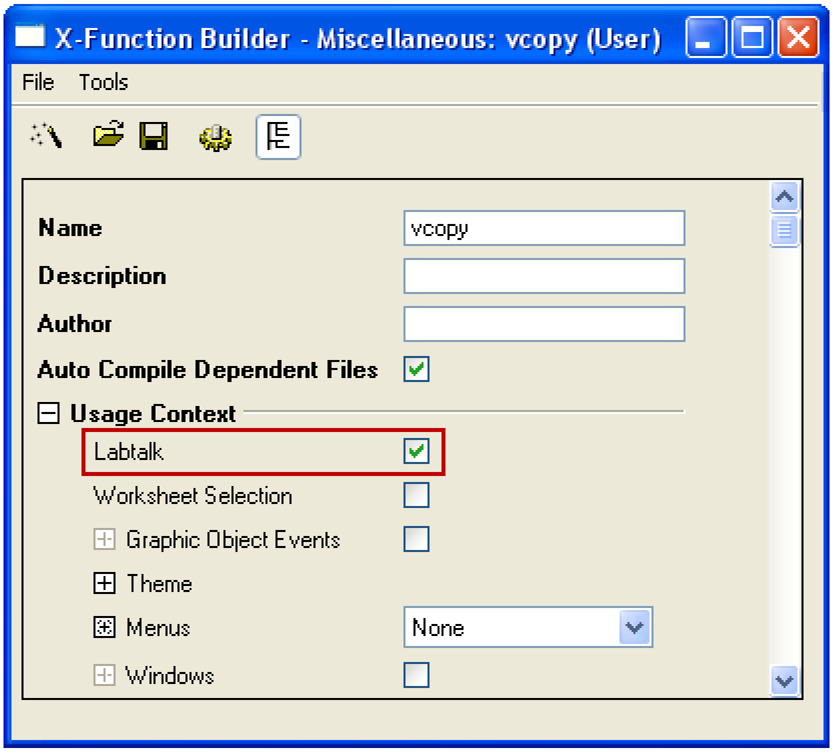


6. Click  to open **Code Builder**
7. Add the following codes in the vcopy function in code builder

```
void vcopy(const vector& ix, vector& ox)
{

        if (!ix || !ox)
            XF_THROW(CER_NO_DATA);

        ox = ix;

}
```

Making the X-Function Script Accessible

1. Click the **Return to Dialog** button in **Code Builder**
2. In the **X-Function Builder**, save your changes 
3. Open the X-Function in **Tree View** by clicking 
4. Open the **Usage Context** branch. Make sure the **Labtalk** check box is selected

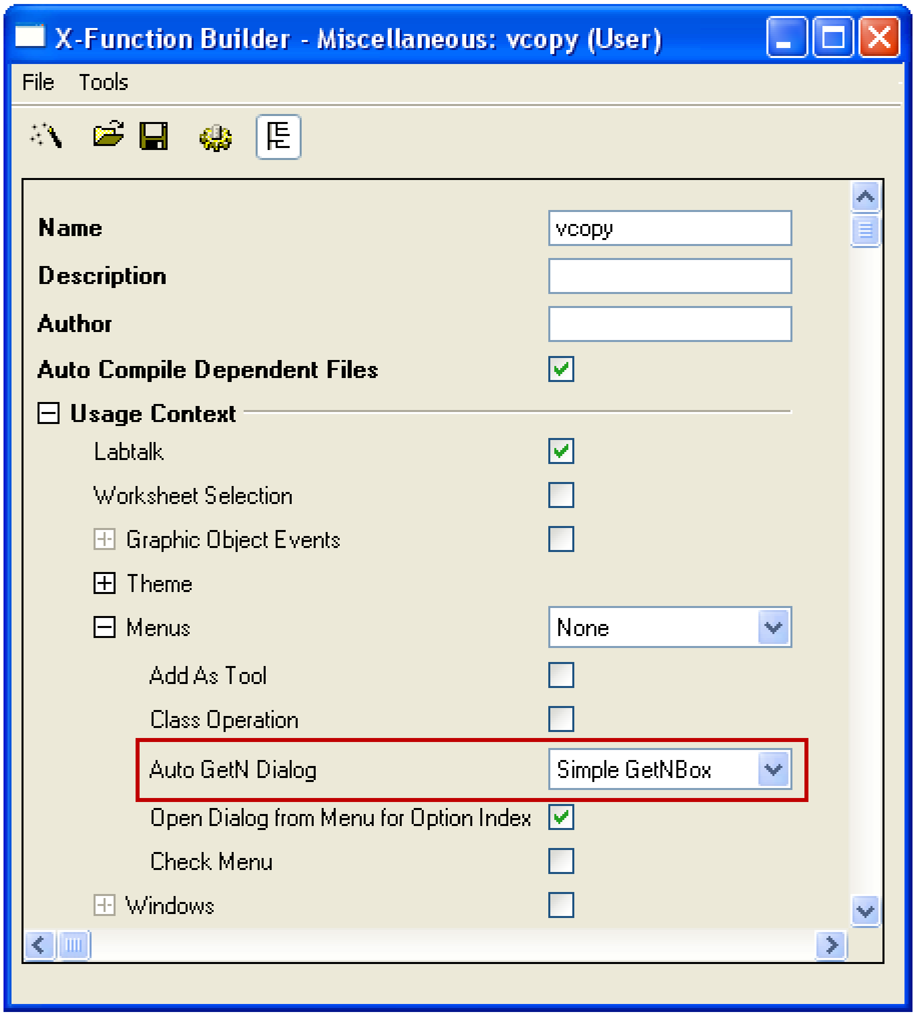


5. Save the x-function and close the **X-Function Dialog**
6. Fill column(A) with row numbersw in the active worksheet (Highlight column(A), right-click and select **Fill Column with: Row Numbers**)
7. Type the following script in the command window, Column(A) will be copied to Column(B)

```
vcopy col(a) col(b)
```

Using the X-Function in Dialog Mode

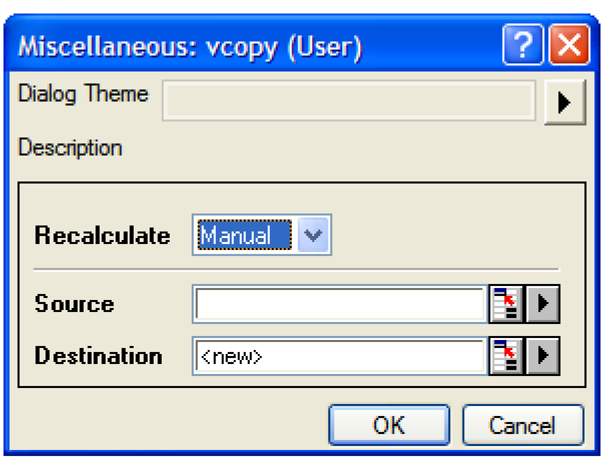
1. Open the **X-Function Dialog** and open VCOPY.OXF in **Tree View**
2. Open the **Usage Context** branch
3. Open the **Menus** branch, make sure **Simple GetNBox** is selected from the **Auto GetN Dialog** list box



4. Save the x-function and close the **X-Function Dialog**

5. Type following script in the command window, Dialog of VCOPY.OXF will be opened

```
vcopy -d
```



10.3.3 How to Create a Wizard

Summary

A wizard is a graphical user interface that includes a series of dialogs to direct a user to complete a mission step by step. A wizard makes a complex task easier to perform. Origin provides several classes in Origin C for users to develop a wizard. The dialog for each step in the wizard can be created using an X-Function.

In this example, the wizard will perform a normality test and then a one-sample t-test for data in a column. The normality test's result can be shared in one-sample t-test.

> **Note:** This tutorial requires the Develop Kit.

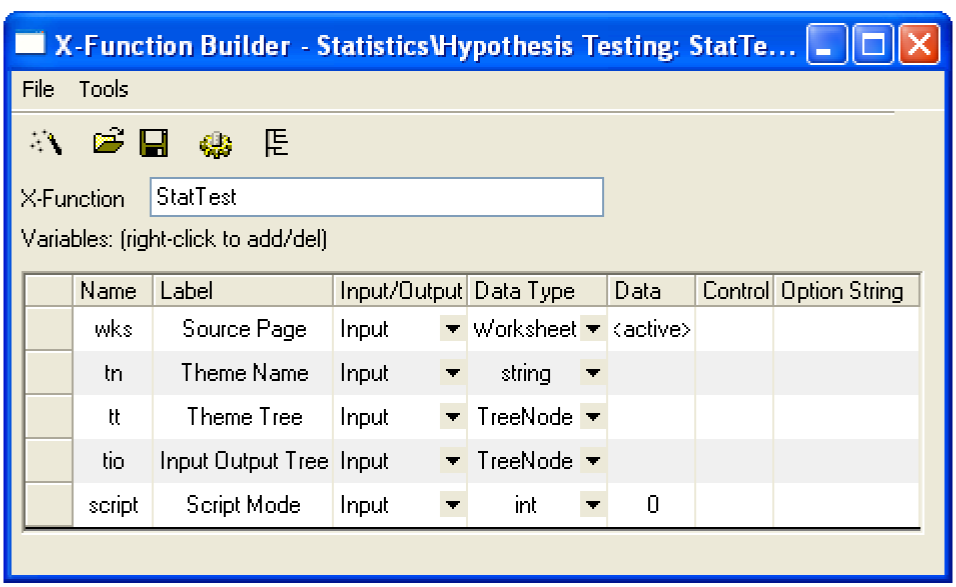
Minimum Origin Version Required: Origin 8.1SR0

What you will learn

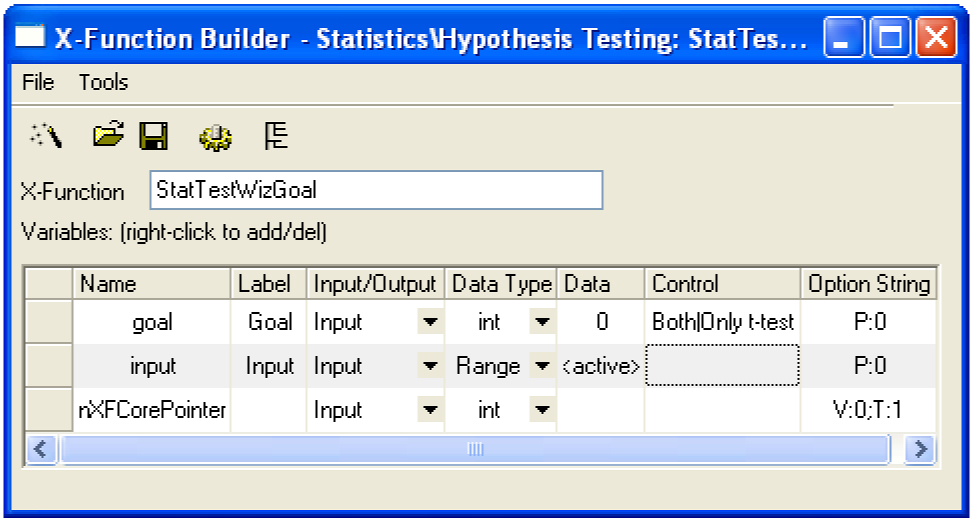
- How to create an X-Function.
- How to share a variable or a DataRange in different steps.
- How to call an X-Function in OriginC.
- How to create a wizard.

Create four X-Functions

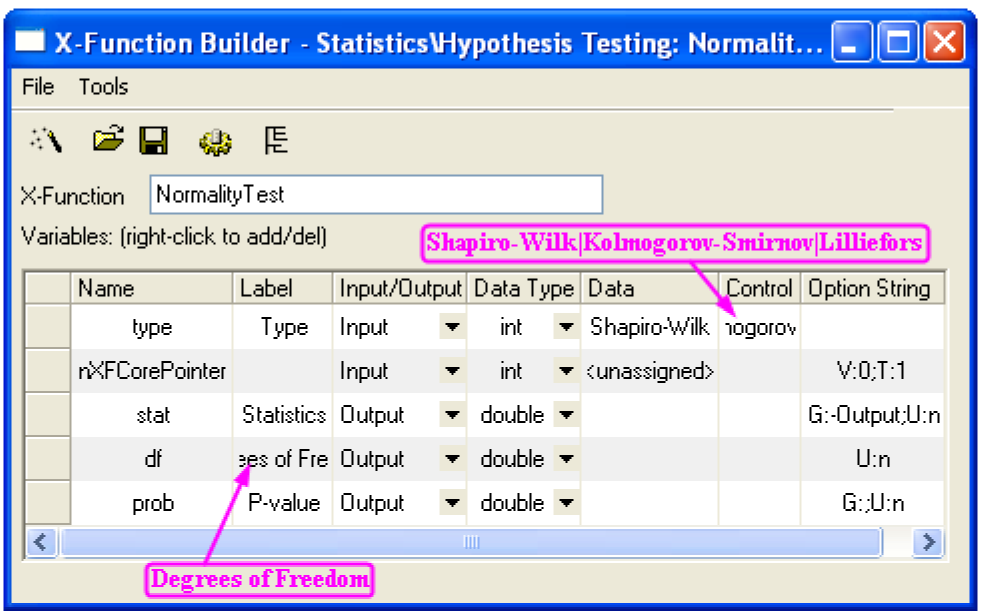
1. Select **Tools: X-Function Builder** or press **F10** to open **X-Function Builder dialog**
2. Add the variables as follows and save the X-Function as "StatTest" in the User Files folder, User Files\X-Functions\Statistics\Hypothesis Testing.



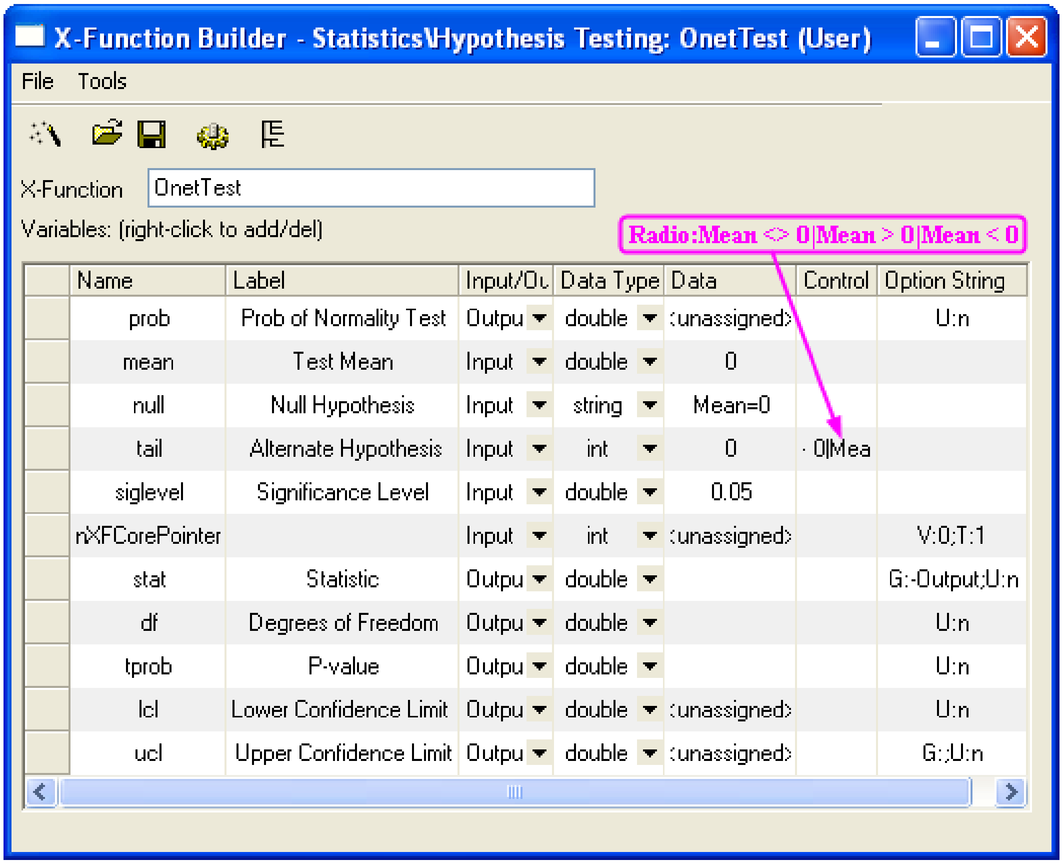
3. Click the **New X-Function Wizard** button. Add the variables as follows and save the X-Function as "StatTestWizGoal" in the User Files folder, User Files\X-Functions\Statistics\Hypothesis Testing.



4. Click the **New X-Function Wizard** button. Add the variables as follows and save the X-Function as "NormalityTest" in the User Files folder, User Files\X-Functions\Statistics\Hypothesis Testing.



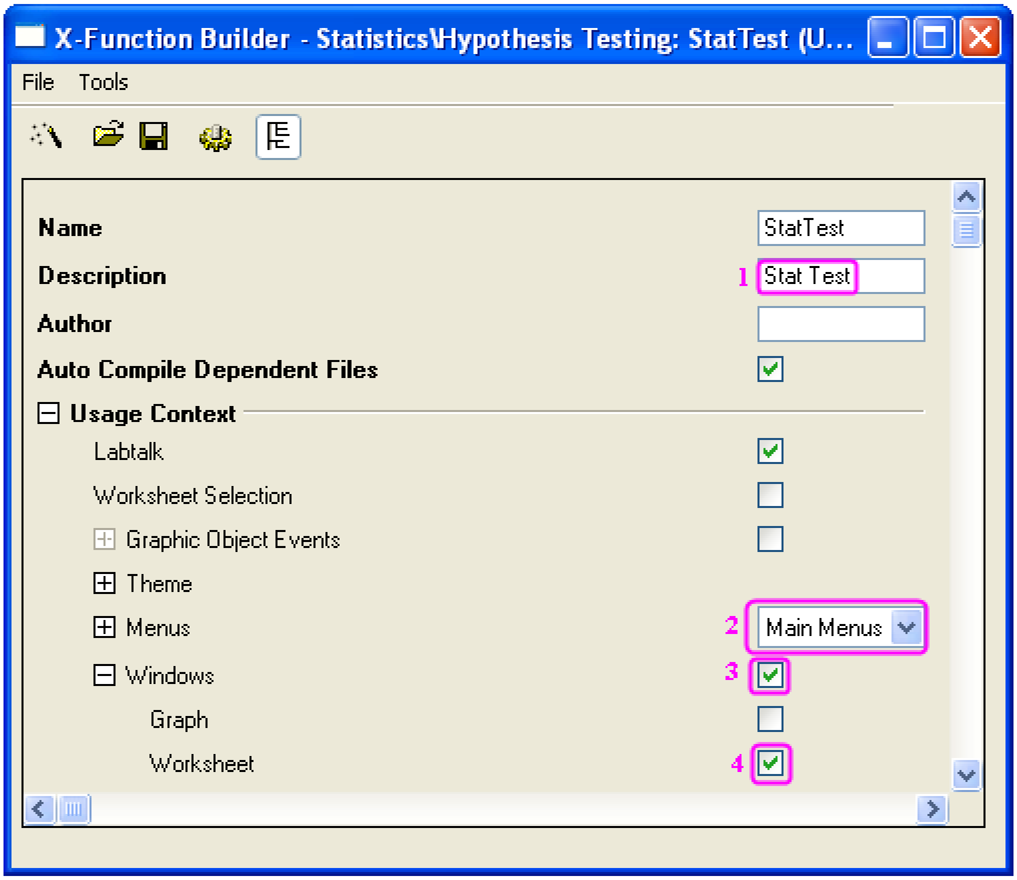
5. Click the **New X-Function Wizard** button. Add the variables as follows and save the X-Function as "OnetTest" in the User Files folder, User Files\X-Functions\Statistics\Hypothesis Testing.



Note that the X-Functions **NormalityTest** and **OnetTest** have the same variable "prob", which is a shared variable and will be declared in the source file.

Update X-Function's Property in Tree View

1. Open the X-Function **StatTest**. Click the **TreeView** button to open the Tree View. Make the following settings in the Tree View.



2. Click the **Save OXF file** button to save the X-Function.
3. Open the X-Function **StatTestWizGoal**, **NormalityTest** and **OnetTest** respectively in X-Function Builder. Click the **TreeView** button and type "Select Wizard Goal", "Normality Test" and "One-Sample t-test" in the **Description** edit box of each X-Function's Tree View, which will be shown in the dialogs.

Create Files for the Wizard

- Click the **Code Builder** button on the **Standard** toolbar. In Code Builder, click the **New** button. In the **New File** dialog, select **H File**, click the **Browse** button, and select the User Files folder, User Files\OriginC as the new header file's **Location**. Then type StatTestWiz in the **File Name** edit box. Click **OK** to close the dialog.

Add the following script to StatTestWiz.h file.

```
#ifndef __STAT_TEST_WIZ_H__
#define __STAT_TEST_WIZ_H__

#include <..\OriginLab\XFWiz.h>
#include <..\OriginLab\XFCore.h>
```

```
#include <..\OriginLab\XFWizard_utils.h>

class StatTestWizCore : public XFCore
{
public:
   StatTestWizCore();

public:
   void ChangeGoal(int nGoal);
   DataRange GetRange();

   int nStep;
protected:

};

int stat_test_run_wiz_nodlg(LPCSTR lpcszThemeName = NULL, const
XFWizTheme *pXFWizTheme
= NULL, const XFWizInputOutputRange *pXFWizIO = NULL, DWORD dwOPUID =
0);

int stat_test_open_wiz_dlg(LPCSTR lpcszThemeName = NULL, const
XFWizTheme *pXFWizTheme
= NULL, const XFWizInputOutputRange *pXFWizIO = NULL, DWORD dwOPUID =
0);

#endif  //__STAT_TEST_WIZ_H__
```

Click the **Save** button to save StatTestWiz.h file.

- Repeat the same operation to create a new **C File**, StatTestWiz.c .

Add the following script to StatTestWiz.c file.

```
//////////////////////////////////////////////////////////////////////
////////////
#include <..\OriginLab\XFWizManager.h>

#include <..\OriginLab\WizOperation.h>
#include <..\OriginLab\XFWizNavigation.h>

#include <..\OriginLab\XFWizScript.h>
#include <..\OriginLab\XFWizDlg.h>

//////////////////////////////////////////////////////////////////////
////////////
// Include your own header files here.
#include "StatTestWiz.h"

enum
{
        GOAL_ALL = 0,
        GOAL_SIMPLE,
};

//Names of three X-Functions
#define STR_STEP_GOAL  "StatTestWizGoal"
```

```
#define STR_STEP_Normal        "NormalityTest"
#define STR_STEP_TTest "OnetTest"

//Names of steps shown in the wizard.
#define STR_LABEL_STEP_GOAL    "Goal"
#define STR_LABEL_STEP_Normal  "Normality Test"
#define STR_LABEL_STEP_TTest   "One-Sample t-test"

////////////////////////////////////////////////////////////////////////
////////////
//Class StatTestWizTheme
class StatTestWizTheme : public XFWizTheme
{
public:
   StatTestWizTheme();
};

//Name of the variable prob shared by X-Functions NormalityTest and
OnetTest
#define STR_GETN_VAR_SHARED_NProb "prob"

StatTestWizTheme::StatTestWizTheme()
:XFWizTheme()
{
   m_saSharedList.Add(STR_GETN_VAR_SHARED_NProb); //Add the shared
variable
}

////////////////////////////////////////////////////////////////////////
////////////
class StatTestWizInputOutputRange : public XFWizInputOutputRange
{
};

////////////////////////////////////////////////////////////////////////
////////////
//Class StatTestWizManager

#define STR_CLASS_NAME_TEST    "StatTestWiz"
#define TEST_VERSION_NUMBER    1.0

class StatTestWizManager : public XFWizManager
{
public:
   StatTestWizManager(LPCSTR lpcszThemeName = NULL, const XFWizTheme
*pXFWizTheme
= NULL, const XFWizInputOutputRange *pXFWizIO = NULL, DWORD dwUIDOp =
0);

protected:
   virtual double GetVersion() { return TEST_VERSION_NUMBER; }
   virtual XFCore* CreateXFCore() { return new StatTestWizCore; }
   virtual XFWizTheme* CreateXFWizTheme() { return new StatTestWizTheme;
}
   virtual XFWizInputOutputRange* CreateXFWizInputOutputRange()
          { return new StatTestWizInputOutputRange; }
   virtual string GetClassName() { return STR_CLASS_NAME_TEST; }
```

```
};

StatTestWizManager::StatTestWizManager(LPCSTR lpcszThemeName, const
XFWizTheme
*pXFWizTheme, const XFWizInputOutputRange *pXFWizIO, DWORD dwUIDOp)
: XFWizManager(lpcszThemeName, pXFWizTheme, pXFWizIO, dwUIDOp)
{
   StringArray saMapXFNames = {STR_STEP_GOAL, STR_STEP_Normal,
STR_STEP_TTest};
   StringArray saMapXFLabels = {STR_LABEL_STEP_GOAL,
STR_LABEL_STEP_Normal,
                                 STR_LABEL_STEP_TTest};
   m_saMapXFNames = saMapXFNames;
   m_saMapXFLabels = saMapXFLabels;
   ASSERT( m_saMapXFNames.GetSize() == m_saMapXFLabels.GetSize() );

   StringArray saDefaultXFNames = {STR_STEP_GOAL, STR_STEP_Normal,
STR_STEP_TTest};
   m_saDefaultXFNames = saDefaultXFNames;

   m_strRunDlgName = _L("Stat Test");
}

///////////////////////////////////////////////////////////////////////
////////////
//Class StatTestWizCore

StatTestWizCore::StatTestWizCore()
:XFCore()
{
   StringArray vsXFsRecalculateShown = {STR_STEP_GOAL};
   m_vsXFsRecalculateShown = vsXFsRecalculateShown;
   nStep = GOAL_ALL;
}

//Select steps in the Goal Step
void StatTestWizCore::ChangeGoal(int nGoal)
{
   XFWizNavigation *pXFWizNavg = (XFWizNavigation
*)GetXFWizNavigation();
   ASSERT(pXFWizNavg);

   nStep = nGoal;

   if ( pXFWizNavg )
   {
      StringArray saXFNames;
      saXFNames.Add(STR_STEP_GOAL);
      switch (nGoal)
      {
      case GOAL_ALL:
         saXFNames.Add(STR_STEP_Normal);
         saXFNames.Add(STR_STEP_TTest);
         break;
      case GOAL_SIMPLE:
         saXFNames.Add(STR_STEP_TTest);
         break;
```

```
      }

      pXFWizNavg->SetSteps(saXFNames);

   }
}

//Get input DataRange in the Goal Step.
DataRange StatTestWizCore::GetRange()
{
   XFWizNavigation *pXFWizNavg =
(XFWizNavigation*)GetXFWizNavigation();
   XFWizInputOutputRange* pIORange = pXFWizNavg-
>GetXFWizInputOutputRange();

   DataRange drInput;
   if(!pIORange)
   {
      error_report("Fail to get io ranges!");
      return drInput;
   }

   Array<DataRange&> drs;
   //Get input DataRange.
   if(!pIORange->Get(&drs, STR_STEP_GOAL, true))
   {
      error_report("Fail to get range from WizCore!");
      return drInput;
   }

   drInput = drs.GetAt(0);

   return drInput;
}

///////////////////////////////////////////////////////////////////////
////////////

int stat_test_run_wiz_nodlg(LPCSTR lpcszThemeName, const XFWizTheme
*pXFWizTheme, const
XFWizInputOutputRange *pXFWizIO, DWORD dwOPUID)
{
   TEMPLATE_run_wiz_nodlg(StatTestWizManager, lpcszThemeName,
pXFWizTheme, pXFWizIO, dwOPUID)
}

int stat_test_open_wiz_dlg(LPCSTR lpcszThemeName, const XFWizTheme
*pXFWizTheme, const
XFWizInputOutputRange *pXFWizIO, DWORD dwOPUID)
{
   TEMPLATE_open_wiz_dlg(StatTestWizManager, lpcszThemeName,
pXFWizTheme, pXFWizIO, dwOPUID)
}

int stat_test_run_wiz(UINT msg, const XFWizTheme *pXFWizTheme, const
XFWizInputOutputRange *pXFWizIO, DWORD dwOPUID, int nExeMode)
{
```

```
    TEMPLATE_run_wiz(StatTestWizManager, msg, pXFWizTheme, pXFWizIO,
dwOPUID, nExeMode)
}
```

Click the **Save** button to save StatTestWiz.c file.

Note that StatTestWiz.c should be compiled after the X-Function **StatTest** is compiled, since the included files in StatTestWiz.c are not yet in the workspace until the X-Function **StatTest** is compiled. In fact StatTestWiz.h is included in X-Function **StatTest**, so StatTestWiz.c will be compiled automatically when X-Function **StatTest** is compiled.

Add Script for X-Functions

Script for X-Function StatTest

In the **X-Function Builder**, click the **Open** button and open the X-Function StatTest. Click the **Edit X-Function in Code Builder** and add the following script.

- Include header files

```
#include <..\OriginLab\XFWiz.h>
#include <..\OriginLab\WizOperation.h>
#include <..\OriginLab\XFCore.h>
#include <..\OriginLab\XFWizNavigation.h>
#include <..\OriginLab\XFWizManager.h>
#include <..\OriginLab\XFWizScript.h>
#include <..\OriginLab\XFWizDlg.h>

#include <..\OriginLab\XFWizard_utils.h>

#include <..\OriginLab\WksOperation.h>

#include <event_utils.h>

#include "StatTestWiz.h"
```

- StatTest()

Add the function body, which specifies the dialog mode.

```
if( script )
    stat_test_run_wiz_nodlg(tn);
else
    stat_test_open_wiz_dlg(tn);
```

- StatTest_before_execute()

Add the function body, which determines not to show this dialog before the wizard is opened.

```
nRet = XFEVT_PROCEED_NO_DLG;
```

Click **Compile** button to compile the file. Then click **Return to Dialog** button to return to **X-Function Builder**. In the **X-Function Builder**, click **Save OXF file** button to save the X-Function.

Script for X-Function StatTestWizGoal

Open the X-Function StatTestWizGoal. Click **Edit X-Function in Code Builder** button, add the following script.

- Include header files

```
#include "StatTestWiz.h"
```

- Add a static function _check_input()

This function is used to check whether the input DataRange is a single column.

```
static bool _check_input(const TreeNode trGetN, string& strErr)
{
   TreeNode trRange = trGetN.input;
   DataRange drInput;

   drInput.Create(trRange.strVal);

   if( drInput.GetNumRanges() == 0 )
   {
      strErr = "Input can't be empty, and it should be a valid
column.";
      return false;
   }
   else
   {
      if( drInput.GetNumRanges() == 1)
      {
         Worksheet wksInput;
         int nC1, nC2;
         drInput.GetRange(wksInput, nC1, nC2);
         if( nC1 == nC2 )
            return true;
      }

         strErr = "Please select one column.";
         return false;
   }
}
```

- StatTestWizGoal_event1()

Add the function body, which updates the dialog.

```
   StatTestWizCore* pstatwc =
(StatTestWizCore*)get_xf_core_handler(trGetN);
   ASSERT(pstatwc);

   //Update the Wizard page.
   if ( 0 == lstrcmp(lpcszNodeName, "goal") )
      pstatwc->ChangeGoal(trGetN.goal.nVal);


   //Error message is shown at the bottom of the dialog,
   //and OK button is disenabled for incorrect choice of DataRange.
```

```
  bOKEnable = _check_input(trGetN, strErrMsg);

  return false;
```

Click **Compile** button to compile the file. Then click **Return to Dialog** button to return to **X-Function Builder**, and click **Save OXF file** button to save the X-Function.

Script for X-Function NormalityTest

Open the X-Function **NormalityTest**. Click the **Edit X-Function in Code Builder** button and add the following script.

- Include header files

```
#include "StatTestWiz.h"

#include <XFbase.h>
```

- Add a static function _update_GUI()

This function is used to update the dialog's edit boxes for normality test result.

```
static void _update_GUI(TreeNode& trGetN)
{
   vector vRes;
   vRes = _norm_test(trGetN.nXFCorePointer.nVal, trGetN.type.nVal);

   trGetN.stat.dVal = vRes[0];
   trGetN.df.dVal = vRes[1];
   trGetN.prob.dVal = vRes[2];
}
```

- Add a static function _update_strErr()

This function is used to update the string shown at the bottom of the dialog.

```
static void _update_strErr(const TreeNode tr, string& strErr)
{
   if(tr.prob.dVal >= 0.05 && tr.prob.dVal <= 1)
      strErr = "At the 0.05 level, the data was significantly drawn
from a
            normally distributed population.";
   else if(tr.prob.dVal < 0.05 && tr.prob.dVal >= 0)
      strErr = "At the 0.05 level, the data was not significantly drawn
from a
            normally distributed population.";
   else
      strErr = "There is not enough information to draw a conclusion.";
}
```

Note that the string is divided into two lines shown in the page. It should be a command of one line in the script.

- Add a static function _norm_test()

This function is used to perform Normality Test using related X-Functions.

```
static vector _norm_test(const int nXFCorePointer, const int nType)
{
   StatTestWizCore* pstatwc =
(StatTestWizCore*)get_xf_core_handler(nXFCorePointer);
   ASSERT(pstatwc);

   vector vRes(3);
   vRes[2] = -1;
   DataRange drInput;
   drInput = pstatwc->GetRange();
   if( !drInput )
      return vRes;

   vector<string> vsXFName = {"swtest","kstest","lillietest"};
   XFBase xfNorm(vsXFName[nType]);
   if( !xfNorm.SetArg("irng", drInput) )
   {
      error_report("Failed to set argument image type");
      return vRes;
   }
   if( !xfNorm.SetArg("stat", vRes[0]) )
   {
      error_report("Failed to set argument image type");
      return vRes;
   }
   if( !xfNorm.SetArg("df", vRes[1]) )
   {
       error_report("Failed to set argument image type");
       return vRes;
   }
   if( !xfNorm.SetArg("prob", vRes[2]) )
   {
       error_report("Failed to set argument image type");
       return vRes;
   }

   if( !xfNorm.Evaluate() )
   {
      error_report("Failed to evaluate the stats X-Function.");
      return vRes;
   }

   return vRes;
}
```

- NormalityTest()

Update the function body, which exports the result into a worksheet when the **Next** button is pressed.

```
   DataRange drInput;
   StatTestWizCore* pstatwc =
(StatTestWizCore*)get_xf_core_handler(nXFCorePointer);
   ASSERT(pstatwc);
   drInput = pstatwc->GetRange();
```

```
    if( !drInput )
        return;
    string strBook, strSheet;
    if(!drInput.GetBookSheet(strBook, strSheet))
    {
        error_report("Workbook and worksheet names can't be obtained.");
        return;
    }
    WorksheetPage wpData(strBook);

    int nLayer = wpData.AddLayer("Normality Test");

    if(nLayer >= 0)
    {
        Worksheet wksRes = wpData.Layers(nLayer);
        vector<string> vsTypeName = {"Shapiro-Wilk","Kolmogorov-
Smirnov","Lilliefors"};
        vector<string> vsNProb = {"Prob<W", "Prob>D", "Prob>D"};
        vector<string> vsParaName = {"Statistic", "DF", ""};
        vsParaName[2] = vsNProb[type];

        vector vRes;
        vRes = _norm_test(nXFCorePointer, type);

        wksRes.Columns(1).SetLongName(vsTypeName[type]);
        for(int ii=0; ii<3; ii++)
        {
            wksRes.SetCell(ii, 0, vsParaName[ii], false);
            wksRes.SetCell(ii, 1, vRes[ii]);
        }
    }
    else
    {
        error_report("New worksheet can't be created.");
    }
```

- NormalityTest_event1()

Update the function body, which will update the results in the dialog as the method of normality test changes. Strings shown at the bottom of the dialog will also be updated.

```
    _update_GUI(trGetN);
    _update_strErr(trGetN, strErrMsg);

    return true;
```

- NormalityTest_before_execute()

Update the function body, which will make the edit boxes for results grayed out, and show the result in the dialog.

```
    trGetN.stat.Enable = false;
    trGetN.df.Enable = false;
    trGetN.prob.Enable = false;
```

Click the **Compile** button to compile the file. Then click the **Return to Dialog** button to return to **X-Function Builder**, and click the **Save OXF file** button to save the X-Function.

Script for X-Function OnetTest

Open the X-Function **OnetTest**. Click the **Edit X-Function in Code Builder** button and add the following script.

- Include header files

```
#include "StatTestWiz.h"

#include <XFbase.h>
```

- Define strings

```
const vector<string> vsNull = {"Mean = ","Mean <= ","Mean >= "};
const vector<string> vsAlter = {"Mean <> ","Mean > ","Mean < "};
const vector<string> vsAcceptNull = {"Not significantly different
from","Not
significantly greater than","Not significantly less than"};
const vector<string> vsRejectNull = {"significantly different
from","significantly
greater than","significantly less than"};
const vector<string> vsProb = {"Prob>|t|", "Prob>t", "Prob<t"};
```

- Add a static function _update_null()

This function is used to update the **Null** edit box.

```
static void _update_null(TreeNode& trGetN, bool bMean = false)
{
   string strNull;

   strNull = vsNull[trGetN.tail.nVal] + ftoa(trGetN.mean.dVal);
   trGetN.null.strVal = strNull;

   if(bMean)
   {
      string strAlter = vsAlter[0] + ftoa(trGetN.mean.dVal) + "|";
      strAlter = strAlter + vsAlter[1] + ftoa(trGetN.mean.dVal) + "|";
      strAlter = strAlter + vsAlter[1] + ftoa(trGetN.mean.dVal);

      trGetN.tail.SetAttribute(STR_COMBO_ATTRIB, strAlter);
   }
}
```

- Add a static function _check_sig_level()

This function is used to check the **Significance Level** edit box value.

```
static bool _check_sig_level(TreeNode& trGetN, string& strErr)
{
   if( trGetN.siglevel.dVal > 0 && trGetN.siglevel.dVal < 1 )
   {
      return true;
   }
```

```
    else
    {
        strErr = "Significance Level should be between 0 and 1.";
        return false;
    }
}
```

- Add a static function _update_strErr()

This function is used to define the string for the conclusion of t-test at the bottom based on P-value.

```
static void _update_strErr(const TreeNode tr, string& strErr)
{
    if(tr.tprob.dVal >= tr.siglevel.dVal && tr.tprob.dVal <= 1)
        strErr.Format("Null Hypothesis is %s%s.\r\nAlternative Hypothesis
is %s%s.
            At the %s level, the population mean is %s the test
mean(%s).",
            vsNull[tr.tail.nVal], ftoa(tr.mean.dVal),
vsAlter[tr.tail.nVal], ftoa(tr.mean.dVal),
            ftoa(tr.siglevel.dVal), vsAcceptNull[tr.tail.nVal],
ftoa(tr.mean.dVal) );
    else if(tr.tprob.dVal < tr.siglevel.dVal && tr.tprob.dVal >= 0)
        strErr.Format("Null Hypothesis is %s%s.\r\nAlternative Hypothesis
is %s%s.
            At the %s level, the population mean is %s the test
mean(%s).",
            vsNull[tr.tail.nVal], ftoa(tr.mean.dVal),
vsAlter[tr.tail.nVal], ftoa(tr.mean.dVal),
            ftoa(tr.siglevel.dVal), vsRejectNull[tr.tail.nVal],
ftoa(tr.mean.dVal) );
    else
        strErr = "There is not enough information to draw a conclusion.";
}
```

Note that the command is divided into several lines shown in the page. It should be a command of one line in the script.

- Add a static function _update_GUI()

This function is used to update edit boxes for results in the dialog.

```
static void _update_GUI(TreeNode& trGetN)
{
    vector vRes;
    vRes = _one_sample_t_test(trGetN.nXFCorePointer.nVal,
trGetN.mean.dVal, trGetN.tail.dVal, trGetN.siglevel.dVal);

    trGetN.stat.dVal = vRes[0];
    trGetN.df.dVal = vRes[1];
    trGetN.tprob.dVal = vRes[2];
    trGetN.lcl.dVal = vRes[4];
    trGetN.ucl.dVal = vRes[5];
}
```

- Add a static function _one_sample_t_test()

This function is used to perform One-Sample t-Test using an X-Function.

```
static vector _one_sample_t_test(const int nXFCorePointer, const double
dMean, const int nTail, const double dSiglevel)
{
   DataRange drInput;
   StatTestWizCore* pstatwc =
(StatTestWizCore*)get_xf_core_handler(nXFCorePointer);
   ASSERT(pstatwc);

   vector vRes(6);
   vRes[2] = -1;
   drInput = pstatwc->GetRange();
   if( !drInput )
      return vRes;

   vRes[3] = 100 - 100*dSiglevel;

   XFBase xfTTest("ttest1");

   if( !xfTTest.SetArg("irng", drInput) )
   {
      error_report("Failed to set argument irng");
      return vRes;
   }
   if( !xfTTest.SetArg("mean", dMean) )
   {
      error_report("Failed to set argument mean");
      return vRes;
   }
   if( !xfTTest.SetArg("tail", nTail) )
   {
      error_report("Failed to set argument tail");
      return vRes;
   }
   if( !xfTTest.SetArg("alpha", dSiglevel) )
   {
      error_report("Failed to set argument alpha");
      return vRes;
   }

   if( !xfTTest.SetArg("stat", vRes[0]) )
   {
      error_report("Failed to set argument stat");
      return vRes;
   }
   if( !xfTTest.SetArg("df", vRes[1]) )
   {
       error_report("Failed to set argument df");
       return vRes;
   }
   if( !xfTTest.SetArg("prob", vRes[2]) )
   {
       error_report("Failed to set argument prob");
       return vRes;
   }
   if( !xfTTest.SetArg("lcl", vRes[4]) )
```

```
    {
        error_report("Failed to set argument lcl");
        return vRes;
    }
    if( !xfTTest.SetArg("ucl", vRes[5]) )
    {
        error_report("Failed to set argument ucl");
        return vRes;
    }

    if( !xfTTest.Evaluate() )
    {
       error_report("Failed to evaluate the ttest1 X-Function.");
       return vRes;
    }

    return vRes;
}
```

- OnetTest()

Update the function body, which exports the result into a worksheet when the **Finish** button is pressed.

```
   DataRange drInput;
   StatTestWizCore* pstatwc =
(StatTestWizCore*)get_xf_core_handler(nXFCorePointer);
   ASSERT(pstatwc);

   drInput = pstatwc->GetRange();
   if( !drInput )
      return ;

   string strBook, strSheet;
   if(!drInput.GetBookSheet(strBook, strSheet))
   {
      error_report("Workbook and worksheet names can't be obtained.");
      return;
   }
   WorksheetPage wpData(strBook);

   int nLayer = wpData.AddLayer("One-Sample t-test");

   if(nLayer >= 0)
   {
      Worksheet wksRes = wpData.Layers(nLayer);

      vector<string> vsParaName = {"t Statistic", "DF","", "Conf.
Levels in %", "Lower Limits", "Lower Limits"};
      vsParaName[2] = vsProb[tail];

      vector vRes;
      vRes = _one_sample_t_test(nXFCorePointer, mean, tail, siglevel);

      wksRes.SetSize(-1, 4);
      wksRes.Columns(0).SetLongName("Test Statistics");
```

```
      string strNull = "Null Hypothesis is " + vsNull[tail] +
ftoa(mean);
      wksRes.Columns(1).SetLongName(strNull);
      wksRes.Columns(3).SetLongName("Confidence Intervals for Mean");
      for(int ii=0; ii<3; ii++)
      {
         wksRes.SetCell(ii, 0, vsParaName[ii], false);
         wksRes.SetCell(ii, 1, vRes[ii]);

         wksRes.SetCell(ii, 2, vsParaName[ii + 3], false);
         wksRes.SetCell(ii, 3, vRes[ii + 3]);
      }
   }
   else
   {
      error_report("New worksheet can't be created.");
   }
```

- OnetTest_event1()

Update the function body, which will update results and show a conclusion at the bottom of the dialog according to the result. As settings change in the dialog, the **Null** edit box will be updated as the mean and hypothesis change, and the **Significance Level** edit box's value is checked.

```
   if( 0 == lstrcmp(lpcszNodeName, "mean") )
      _update_null(trGetN, true);
   if( 0 == lstrcmp(lpcszNodeName, "tail") )
      _update_null(trGetN);
   if( 0 == lstrcmp(lpcszNodeName, "siglevel") )
      bOKEnable = _check_sig_level(trGetN, strErrMsg);

   _update_GUI(trGetN);
   _update_strErr(trGetN, strErrMsg);

   return false;
```

- OnetTest_before_execute()

Update the function body, to show/hide or disable the controls in the dialog.

```
   StatTestWizCore* pstatwc =
(StatTestWizCore*)get_xf_core_handler(trGetN.nXFCorePointer.nVal);
   ASSERT(pstatwc);
   trGetN.prob.Show = 1 - pstatwc->nStep;
   trGetN.prob.Enable = false;

   trGetN.null.Enable = false;
   trGetN.stat.Enable = false;
   trGetN.df.Enable = false;
   trGetN.tprob.Enable = false;
   trGetN.lcl.Enable = false;
   trGetN.ucl.Enable = false;
```

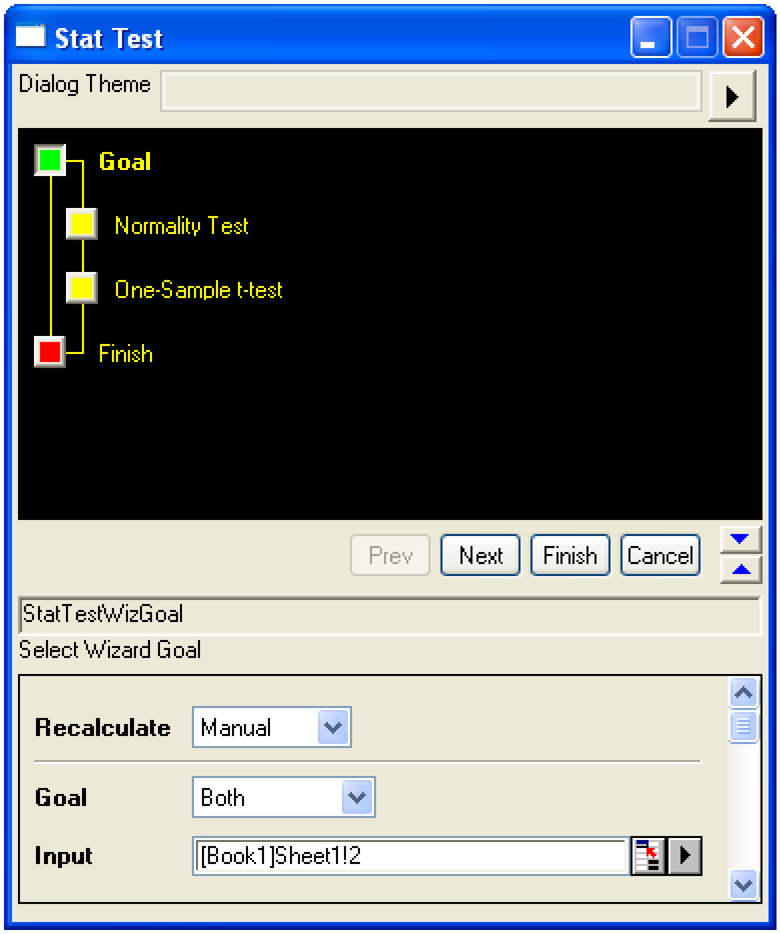
Click the **Compile** button to compile the file. Then click the **Return to Dialog** button to return to the X-Function Builder. Click the **Save OXF file** button to save the X-Function.

Close Origin. Then start Origin and you will notice that a new item **Stat Test** is added to Origin's menu **Statistics: Hypothesis Testing**.

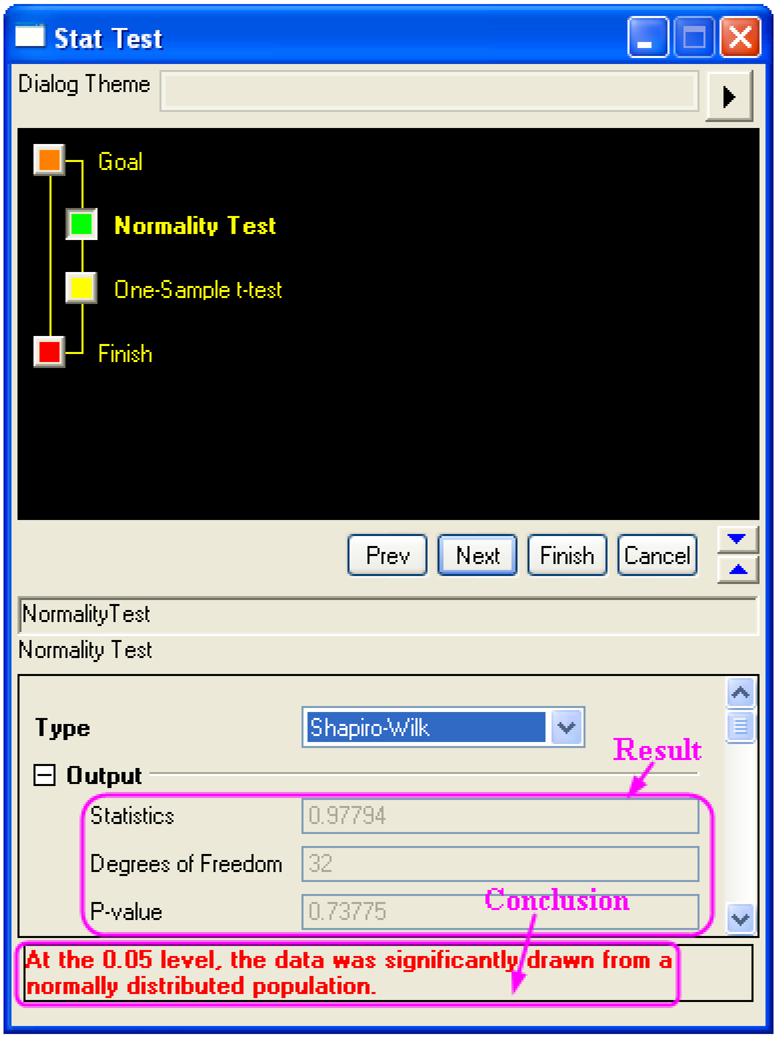
How to Use the Wizard

The following example shows how to use the wizard.

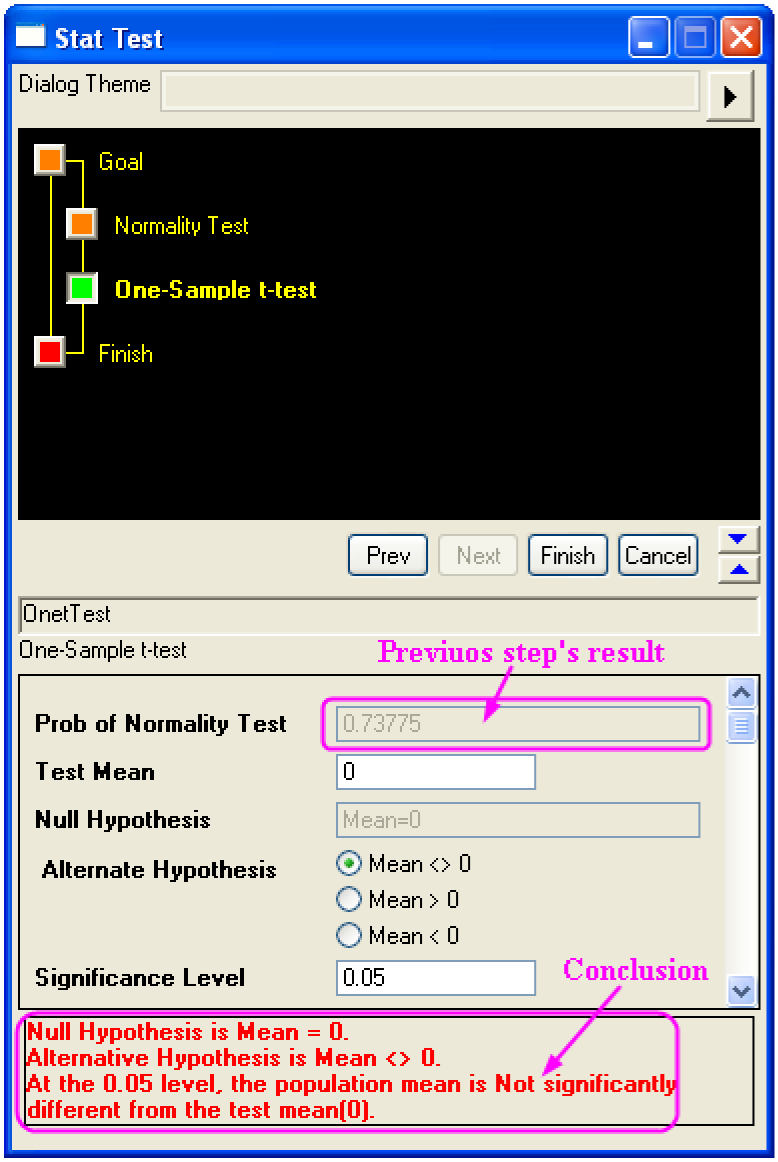
1. Select a column in the worksheet.
2. Select **Statistics: Hypothesis Testing: Stat Test** from the Origin menu or type the command "StatTest -d" in the command window. The **Stat Test** wizard dialog will open.

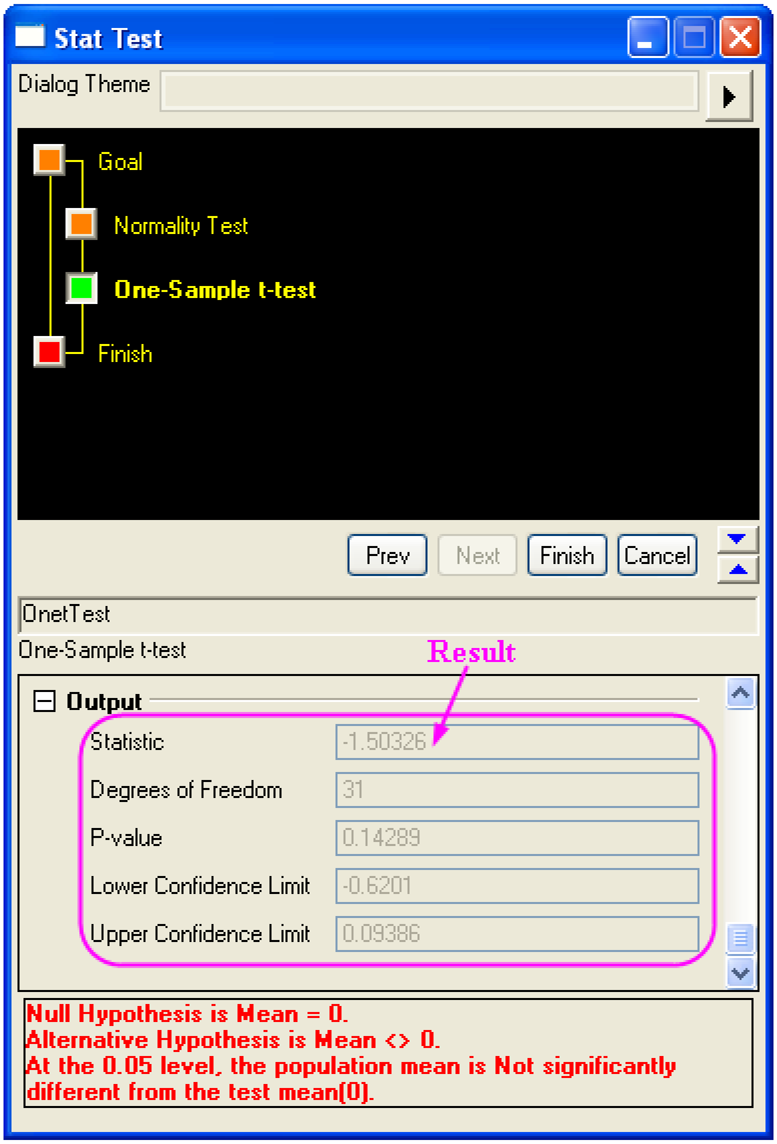


3. Click the **Next** button. The **Normality Test** dialog is opened. The result is shown in the **Output** branch. A conclusion is drawn at the bottom of the dialog.



4. Click the **Next** button. The **One-Sample t-test** dialog is opened. The result is shown in the **Output** branch. A conclusion is drawn at the bottom of the dialog. Previous step's result of normality test is shown at the top. You can also change the setting in the dialog, and notice the result changes.


Stat Test
Dialog Theme
Goal
Normality Test
One-Sample t-test
Finish
Prev
Next
Finish
Cancel
OnetTest
One-Sample t-test
Previuos step's result
Prob of Normality Test
0.73775
Test Mean
0
Null Hypothesis
Mean=0
Alternate Hypothesis
Mean <> 0
Mean > 0
Mean < 0
Significance Level
0.05
Conclusion
Null Hypothesis is Mean = 0.
Alternative Hypothesis is Mean <> 0.
At the 0.05 level, the population mean is Not significantly different from the test mean(0).


Stat Test
Dialog Theme
Goal
Normality Test
One-Sample t-test
Finish
Prev
Next
Finish
Cancel
OnetTest
One-Sample t-test
Result
Output
Statistic
-1.50326
Degrees of Freedom
31
P-value
0.14289
Lower Confidence Limit
-0.6201
Upper Confidence Limit
0.09386
Null Hypothesis is Mean = 0.
Alternative Hypothesis is Mean <> 0.
At the 0.05 level, the population mean is Not significantly different from the test mean(0).

5. Click the **Finish** button to end the wizard. Two worksheets for results are created.