

Guided Tour of Igor Pro

Overview	13
Terminology.....	13
About the Tour	13
Guided Tour 1 - General Tour.....	14
Creating an Igor64 Alias or Shortcut	14
Launching Igor Pro.....	14
Entering Data.....	14
Making a Graph	16
Touching up a Graph	16
Adding a Legend	17
Adding a Tag.....	18
Using Preferences	20
Making a Page Layout	20
Saving Your Work - Tour 1A	23
Loading Data	24
Appending to a Graph	25
Offsetting a Trace.....	26
Unoffsetting a Trace	26
Drawing in a Graph.....	27
Making a Window Recreation Macro	28
Recreating the Graph	29
Using the Data Browser	29
Saving Your Work - Tour 1B	29
Using Igor Documentation.....	30
Graphically Editing Data.....	31
Making a Category Plot (Optional).....	32
Category Plot Options (Optional)	33
The Command Window	35
Synthesizing Data	37
Zooming and Panning	37
Making a Graph with Multiple Axes	39
Saving Your Work - Tour 1C.....	40
Using Cursors.....	40
Removing a Trace and Axis	41
Creating a Graph with Stacked Axes	41
Creating a Page Layout.....	43
Saving Your Work - Tour 1D	43
Creating Controls.....	43
Creating a Dependency.....	46
Saving Your Work - Tour 1E	46
End of the General Tour	47
Guided Tour 2 - Data Analysis	48
Starting Guided Tour 2	48

Chapter I-2 — Guided Tour of Igor Pro

Creating Synthetic Data	48
Quick Curve Fit to a Gaussian	49
More Curve Fitting to a Gaussian	50
Sorting	50
Fitting to a Subrange	51
Extrapolating a Fit After the Fit is Done.....	52
Appending a Fit	53
Guided Tour 3 - Histograms and Curve Fitting.....	55
Starting Guided Tour 3	55
Creating Synthetic Data	55
Histogram of White Noise.....	55
Saving Your Work - Tour 3A	56
Histogram of Gaussian Noise	56
Saving Your Work - Tour 3B.....	57
Curve Fit of Histogram Data.....	57
Saving Your Work - Tour 3C.....	59
Curve Fit Residuals (Optional)	59
Saving Your Work - Tour 3D	61
Writing a Procedure (Optional).....	61
Saving a Procedure File (Optional)	64
Including a Procedure File (Optional).....	65
For Further Exploration	66

Overview

In this chapter we take a look at the main functions of Igor Pro by stepping through some typical operations. Our goal is to orient you so that you can comfortably read the rest of the manual or explore the program on your own. You will benefit most from this tour if you actually do the instructed operations on your computer as you read this chapter. Screen shots are provided to keep you synchronized with the tour.

Terminology

If you have read Chapter I-1, **Introduction to Igor Pro**, you already know these terms.

Experiment	The entire collection of data, graphs and other windows, program text and other data that make up the current Igor environment or workspace.
Wave	Short for waveform, this is basically a named array of data with optional extra information.
Name	Because Igor contains a built-in programming and data transformation language, each object must have a unique name.
Command	This is a line of text that performs some task. Igor is command-driven so that it can be easily programmed.

About the Tour

This tour consists of three sections: **Guided Tour 1 - General Tour** on page I-14, **Guided Tour 2 - Data Analysis** on page I-48, and **Guided Tour 3 - Histograms and Curve Fitting** on page I-55.

The General Tour is a rambling exploration intended to introduce you to the way things work in Igor and give you a general orientation.

The second and third tours guide you through Igor's data analysis facilities including simple curve fitting. When you've completed the first tour you may prefer to explore freely on your own before starting the second tour.

Guided Tour 1 - General Tour

In this exercise, we will generate data in three ways (typing, loading, and synthesizing) and we will generate graph, table, and page layout windows. We will jazz up a graph and a page layout with a little drawing and some text annotation. At the end, we will explore some of the more advanced features of Igor Pro.

Creating an Igor64 Alias or Shortcut

The 64-bit Igor Pro application is typically located at:

`/Applications/Igor Pro 7 Folder/Igor64.app` (*Macintosh*)

`C:\Program Files\WaveMetrics\Igor Pro 7 Folder\IgorBinaries_x64\Igor.exe` (*Windows*)

The “.app” and “.exe” extensions may be hidden on your system.

1. **Make an alias (Macintosh) or shortcut (Windows) for your Igor64 application file and put the alias or shortcut on your desktop. Name it Igor64.**

Launching Igor Pro

1. **Double-click your Igor64 alias or shortcut.**
Igor starts up.
On Windows, you can also launch Igor64 using the Start menu.
2. **Choose Misc→Preferences Off.**
Turning preferences off ensures that the tour works the same for everyone.

Entering Data

1. **If a table window is showing, click it to bring it to the front.**
When Igor starts up, it creates a new blank table unless this feature is turned off in the Miscellaneous Settings dialog. If the table is not showing, perform the following two steps:
 - 1a. **Choose the Windows→New Table menu item.**
The New Table dialog appears.
 - 1b. **Click the Do It button.**
A new blank table is created.
2. **Type “0.1” (without the quotes) and then press Return or Enter on your keyboard.**
This creates a wave named “wave0” with 0.1 for the first point. Entering a value in the first row (point 0) of the first blank column automatically creates a new wave.

3. Type the following numbers, pressing Return or Enter after each one:

1.2
1.9
2.6
4.5
5.1
5.8
7.8
8.3
9.7

The table should look like this:

Point	wave0
0	0.1
1	1.2
2	1.9
3	2.6
4	4.5
5	5.1
6	5.8
7	7.8
8	8.3
9	9.7
10	

4. Click in the first cell of the first blank column.
5. Enter the following numbers in the same way:

-0.12
-0.08
1.3
1
0.54
0.47
0.44
0.2
0.24
0.13

6. Choose Data→Rename.
7. Click “wave0” in the list and then click the arrow icon.
8. Replace “wave0” with “time”.
Notice that you can’t use the name “time” because it is the name of a built-in string function. We apologize for usurping such a common name.
9. Change the name to “timeval”.
10. Click “wave1” in the list, click the arrow icon, and replace “wave1” with “yval”.
11. Click Do It.

The column headers in the table change to reflect the name changes.

Making a Graph

1. Choose the **Windows**→**New Graph** menu item.

The New Graph dialog appears. This dialog comes in a simple form that most people will use and a more complex form that you can use to create complex multi-axis graphs in one step.

2. If you see a button labeled **Fewer Choices**, click it.
3. In the **Y Waves** list, click “yval”.
4. In the **X Wave** list, click “timeval”.
5. Click **Do It**.

A simple graph is created.

Touching up a Graph

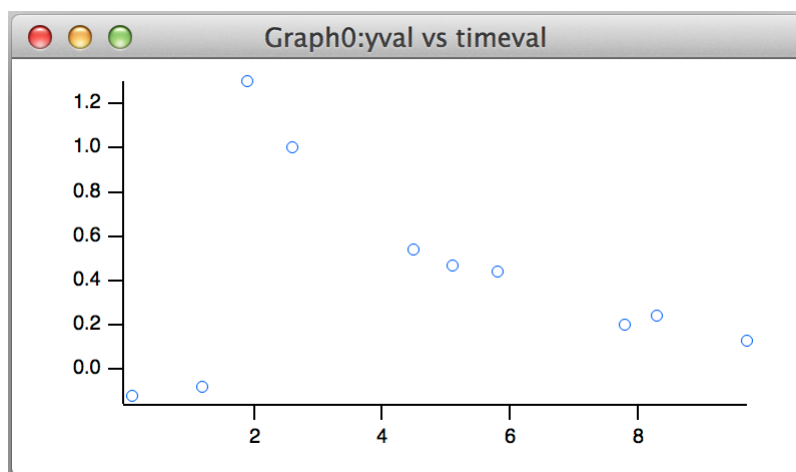
1. Position the cursor directly over the trace in the graph and double-click.

The Modify Trace Appearance dialog appears. You could also have chosen the corresponding menu item from the Graph menu.


Note: The Graph menu appears only when a graph is the target window. The *target* window is the window that menus and dialogs act on by default.

2. Choose **Markers** from the **Mode** pop-up menu.
3. Select the open circle from the **Marker pop-up** menu.
4. Set the marker color to blue.
5. Click **Do It**.


The graph should now look like this:



6. Position the cursor over the bottom axis line.

The cursor changes to this shape: . This indicates the cursor is over the axis and also that you can offset the axis, and the corresponding plot area edge, to a new position.

7. Double-click directly on the axis.

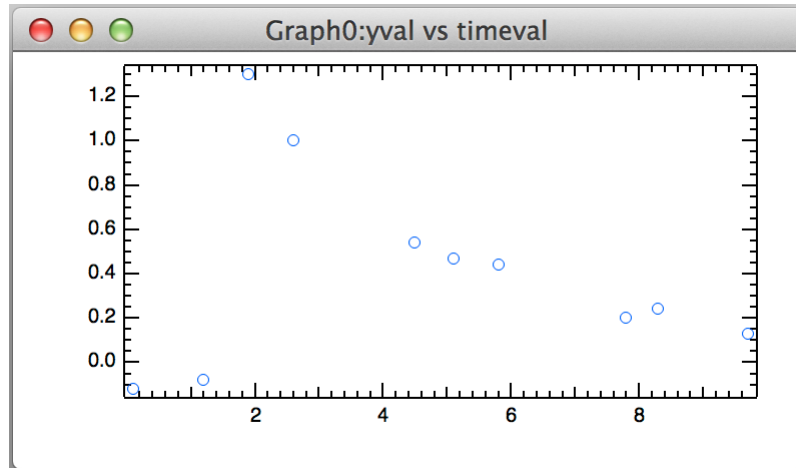
The Modify Axis dialog appears. If another dialog appears, click **Cancel** and try again, making sure the  cursor is showing when you double-click.

Note the **Live Update** checkbox in the top/right corner of the Modify Axis dialog. When it is checked, changes that you make in the dialog are immediately reflected in the graph. When it is unchecked, the changes appear only when you click **Do It**.

8. If it is not already showing, click the **Axis** tab.
9. Choose **On** from the **Mirror Axis** pop-up.
10. Click the **Auto/Man Ticks** tab.

11. Click the **Minor Ticks** checkbox so it is checked.
12. Click the **Ticks and Grids** tab.
13. Choose **Inside** from the **Location** pop-up.
14. Choose the left axis from the **Axis** pop-up menu in the top-left corner of the dialog and then repeat steps 8 through 13.
15. Click **Do It**.

The graph should now look like this:



16. **Again double-click the bottom axis.**
The **Modify Axis** dialog appears again.
17. **Click the Axis tab.**
18. **Uncheck the Standoff checkbox.**
19. **Choose the left axis from the Axis pop-up menu and repeat step 18.**
20. **Click Do It.**

Notice that some of the markers now overlap the axes. The axis standoff setting offsets the axis so that markers and traces do not overlap it. You can use Igor's preferences to ensure that this and other settings default to your liking, as explained below.

21. **Double-click one of the tick mark labels (such as "6") on the bottom axis.**
The **Modify Axis** dialog reappears, this time with the **Axis Range** tab showing. If another dialog or tab appears, cancel and try again, making sure to double click one of the tick mark labels on the bottom axis.
22. **Choose "Round to nice values" from the pop-up menu that initially reads "Use data limits".**
23. **Choose the left axis from the Axis pop-up menu and repeat step 22.**
24. **Click Do It.**

Notice that the limits of the axes now fall on "nice" values.

Adding a Legend

1. **Choose the Graph→Add Annotation menu item.**
The **Add Annotation** dialog appears.
2. **Click the Text tab if it is not already selected.**
3. **Choose Legend from the Annotation pop-up menu in the top-left corner of the dialog.**

Igor inserts text to create a legend in the **Annotation** text entry area. The **Preview** area shows what the annotation will look like. The text `\s (yval)` generates the symbol for the yval wave. This is an "escape sequence" which creates special effects such as this.

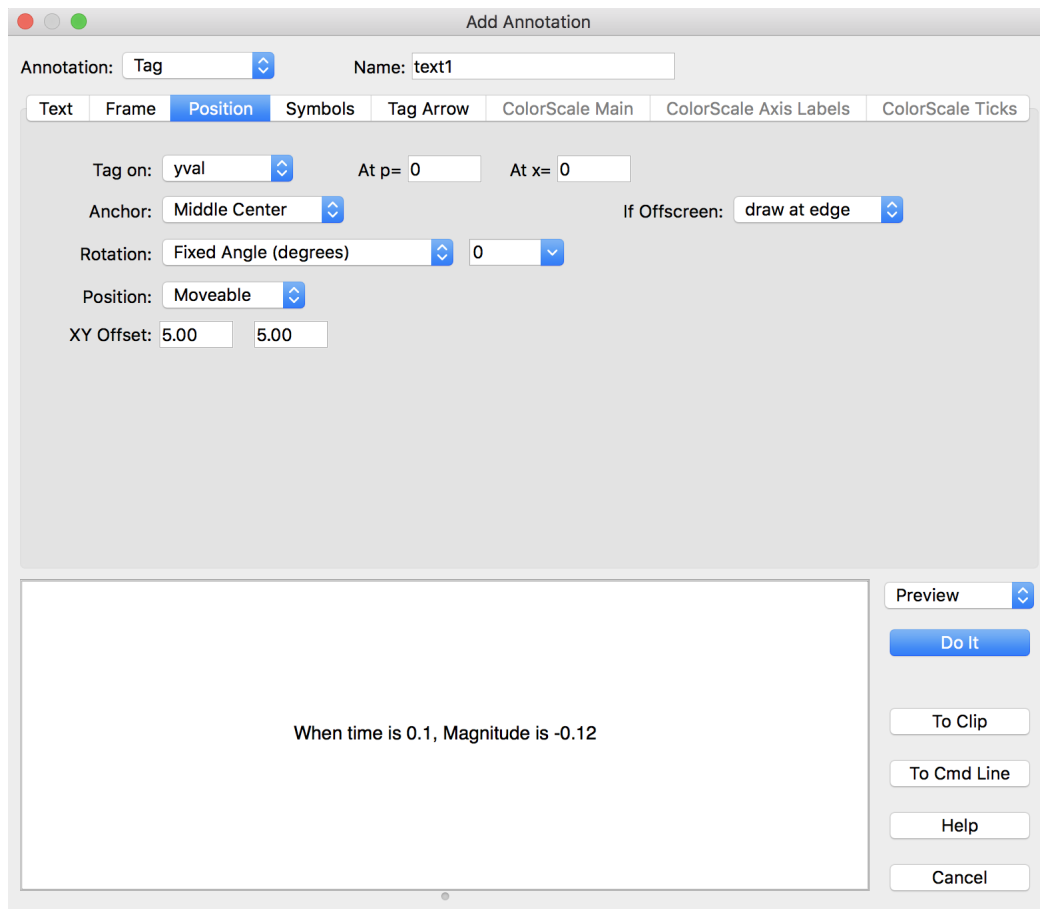
4. **Change the second instance of “yval” to “Magnitude”.**
The annotation text should now be “\s(yval) Magnitude”.
5. **Click the Frame tab and choose Box from the Annotation Frame pop-up menu.**
6. **Choose Shadow from the Border pop-up menu.**
7. **Click the Position tab and choose Right Top from the Anchor pop-up menu.**
Specifying an anchor point helps Igor keep the annotation in the best location as you make the graph window bigger or smaller.
8. **Click Do It.**

Adding a Tag

1. **Choose the Graph→Add Annotation menu item.**
2. **Choose Tag from the Annotation pop-up menu in the top-left corner of the dialog.**
3. **Click the Text tab, and in the annotation text entry area of the Text tab, type “When time is ”.**
4. **Choose Attach Point X Value from the Dynamic pop-up menu in the Insert area of the dialog.**
Igor inserts the \0X escape code into the annotation text entry area.
5. **In the annotation text entry area, add “, Magnitude is ”.**
6. **Choose Attach Point Y Value from the Dynamic pop-up menu.**
Igor inserts the \0Y escape code into the annotation text entry area.
7. **Click the Frame tab and choose None from the Annotation Frame pop-up menu.**
8. **Click the Tag Arrow tab and choose Arrow from the Connect Tag to Wave With pop-up menu.**

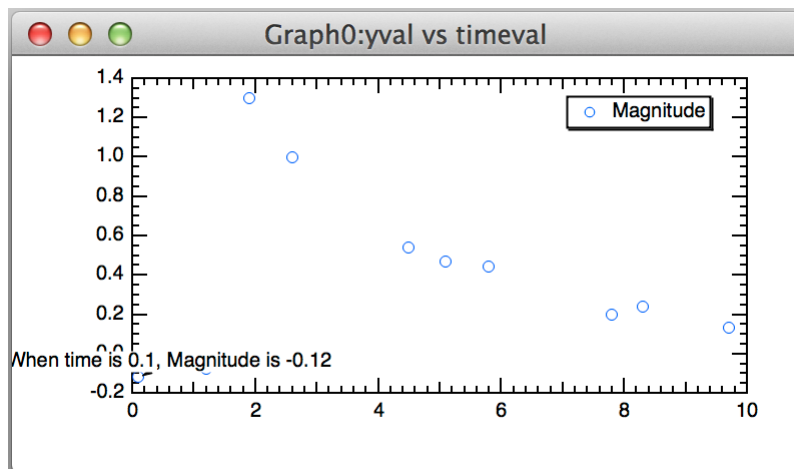
9. Click the **Position** tab and choose “Middle center” from the Anchor pop-up menu.

The dialog should now look like this:



10. Click **Do It**.


The graph should now look like this:



The tag is attached to the first point. An arrow is drawn from the center of the tag to the data point but you can't see it because it is hidden by the tag text.

11. **Position the cursor over the text of the tag.**

The cursor changes to a hand. This indicates you can reposition the tag relative to the data point it is attached to.

12. **Drag the tag up and to the right about 1 cm.**
You can now see the arrow.
13. **With the cursor over the text of the tag, press Option (Macintosh) or Alt (Windows).**
The cursor changes to this shape: . (You may need to nudge the cursor slightly to make it change.)
14. **While pressing Option (Macintosh) or Alt (Windows), drag the box cursor to a different data point.**
The tag jumps to the new data point and the text is updated to show the new X and Y values. Option-drag (Macintosh) or Alt-drag (Windows) the tag to different data points to see their X and Y values.
Notice that the tip of the arrow touches the marker. This doesn't look good, so let's change it.
15. **Double-click the text part of the tag.**
The Modify Annotation dialog appears.
16. **Click the Tag Arrow tab and change the Line/Arrow Standoff from "Auto" to "10".**
17. **Click the Change button.**
The tip of the arrow now stops 10 points from the marker.

Using Preferences

If you have already set preferences to your liking and do not want to disturb them, you can skip this section.

1. **Choose Misc→Preferences On.**
2. **Click the graph window if it is not already active.**
3. **Choose the Graph→Capture Graph Prefs menu item.**
The Capture Graph Preferences dialog appears.
4. **Click the checkboxes for XY plot axes and for XY plot wave styles.**
5. **Click Capture Preferences.**
6. **Choose Windows→New Graph.**
7. **Choose "yval" as the Y wave and "timeval" as the X wave.**
8. **Click Do It.**
The new graph is created with a style similar to the model graph.
9. **Press Option (Macintosh) or Alt (Windows) while clicking the close button of the new graph.**
The new graph is killed without presenting a dialog.
10. **Choose Graph→Capture Graph Prefs.**
11. **Click the checkboxes for XY plot axes and for XY plot wave styles.**
12. **Click Revert to Defaults.**
13. **Choose Misc→Preferences Off.**
We turn preferences off during the guided tour to ensure that the tour works the same for everyone. This is not something you would do during normal work.

Making a Page Layout

1. **Choose the Windows→New Layout menu item.**
The New Page Layout dialog appears. The names of all tables and graphs are shown in the list.
2. **In the Objects to Lay Out list, select Graph0.**
3. **Command-click (Macintosh) or Ctrl-click (Windows) on Table0.**

4. Click Do It.

A page layout window appears with a Table0 object on top of a Graph0 object.

The layout initially shows the page at 50% magnification but you may prefer to work at 100%. You can use the pop-up menu in the lower left corner of the window to change magnification.

5. Click the Table0 object in the layout window.

The table object becomes selected, resize handles are drawn around the edges, and the cursor changes to a hand when over the table.

6. Click in the middle of the table and drag it so you can see the right edge of the table.

7. Position the cursor over the handle (small black square) in the middle of the right edge of the table.

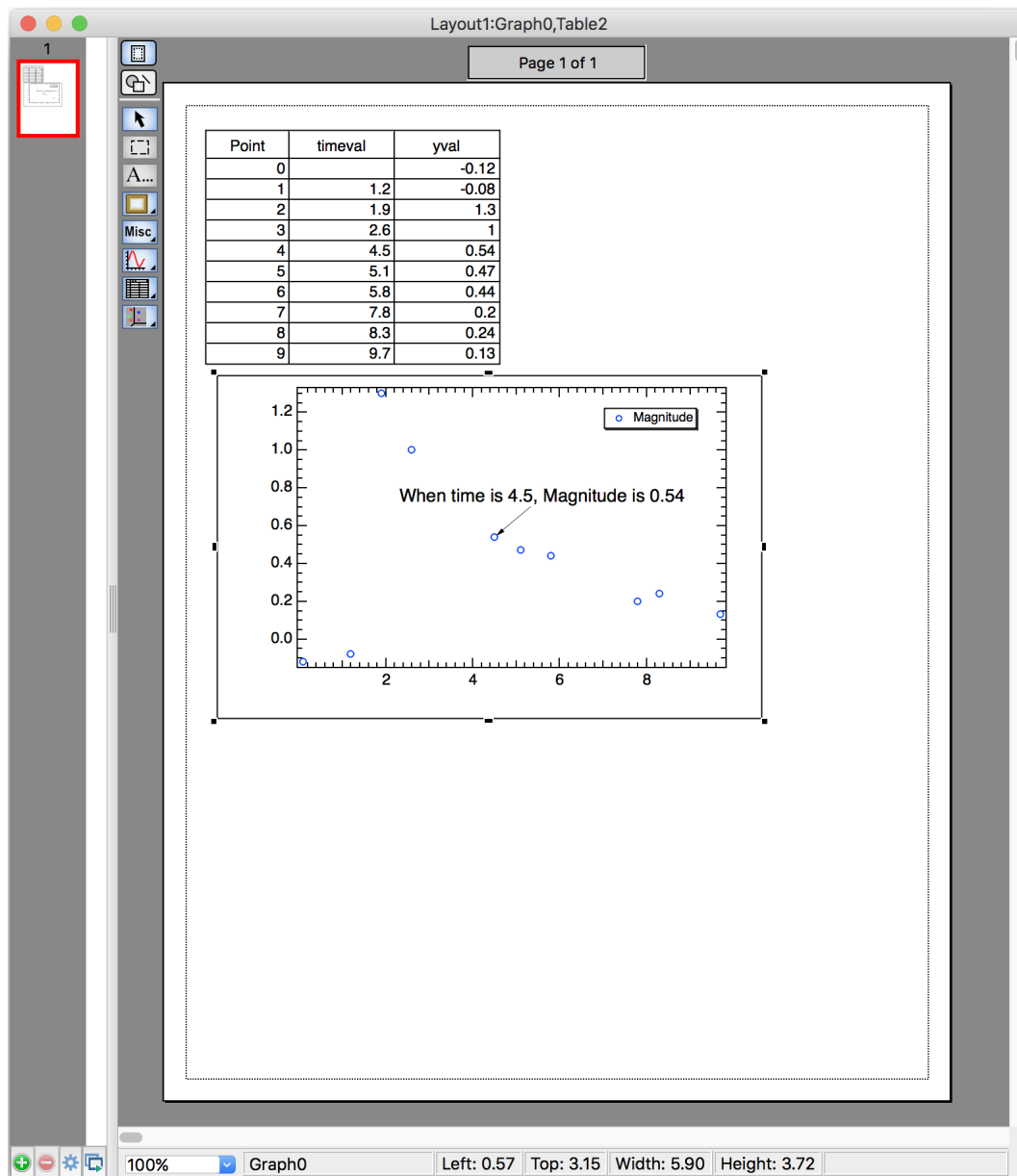
The cursor changes to a two headed arrow indicating you can drag in the direction of the arrows.



8. Drag the edge of the table to the left until it is close to the edge of the third column of numbers.

You need only get close — Igor snaps to the nearest grid line.

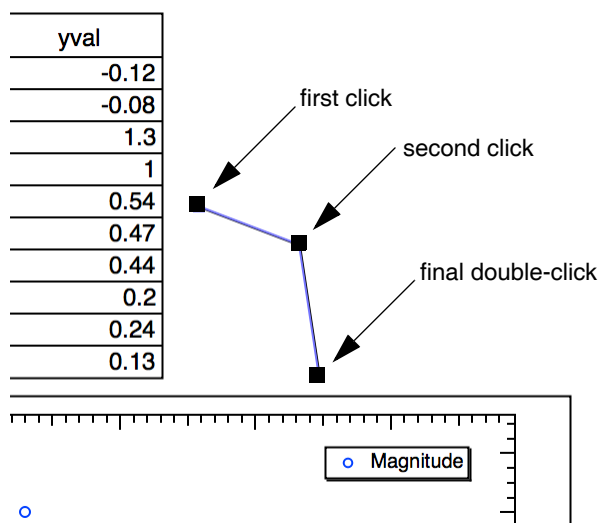
9. In a similar fashion, adjust the bottom of the table to show all the data but without any blank rows.

10. Arrange the table and graph objects in the layout to approximately match this picture:





11. Click this icon in the tool palette: . This activates the drawing tools.
12. Click this icon in the drawing tool palette: . This is the polygon tool.

13. Click once just to the right of the table, click again about 2 cm right and 1 cm down and finally double-click a bit to the right of the last click and just above the graph, as illustrated:



The double-click exits the “draw polygon” mode and enters “edit polygon mode”. If you wish to touch up the defining vertices of the polygon, do so now by dragging the handles (the square boxes at the vertices).

14. Click the Arrow tool in the palette.
This exits polygon edit mode.
15. Click the polygon to select it.
16. Click the drawing environment icon () and choose At End from the Line Arrow submenu.
17. Click this icon in the tool palette:  .
This is the operate icon. The drawing tools are replaced by the normal layout tools.
We are finished with the page layout for now.
18. Choose Windows→Send To Back.

Saving Your Work - Tour 1A

In this section, we will create a “Guided Tours” folder in which to save our work so far. Then we will save the current experiment in an Igor experiment file.

You can save Igor files anywhere. Here we will create the “Guided Tours” folder in the “Igor Pro User Files” folder. Igor creates “Igor Pro User Files” when it first starts. It is a good place to save Igor files.

1. Choose Help→Show Igor Pro User Files.

Igor opens the "Igor Pro User Files" folder on the desktop.

By default, this folder has the Igor Pro major version number in its name, for example, "Igor Pro 7 User Files", but it is generically called the "Igor Pro User Files" folder.

Note where in the file system hierarchy this folder is located as you will need to know this in a subsequent step. The default locations are:

Macintosh:

`/Users/<user>/Documents/WaveMetrics/Igor Pro 7 User Files`

Windows:

`C:\Users\<user>\Documents\WaveMetrics\Igor Pro 7 User Files`

2. Create a folder named “Guided Tours” in the “Igor Pro User Files” folder.
3. Activate Igor .

4. **Choose File→Save Experiment As.**

The save file dialog appears.

5. **Make sure that Packed Experiment File is selected as the file format.**

6. **Navigate to the “Guided Tours” folder.**

7. **Type “Tour 1A.pxp” in the name box.**

8. **Click Save.**

The “Tour 1A.pxp” file contains all of your work in the current experiment, including waves that you created, graphs, tables and page layout windows.

If you want to take a break, you can quit Igor Pro now.

Loading Data

Before loading data we will use a notebook window to look at the data file.

0. **If you are returning from a break, launch Igor and open your “Tour 1A.pxp” experiment file. Then turn preferences off using the Misc menu.**

Opening the “Tour 1A.pxp” experiment file restores the Igor workspace to the state it was in when you saved the file. You can open the experiment file by using the Open Experiment item in the File menu, by double-clicking the experiment file, or by choosing File→Recent Experiments→Tour #1a.pxp.

1. **Choose the File→Open File→Notebook menu item.**

2. **Navigate to the folder “Igor Pro 7 Folder:Learning Aids:Sample Data” folder and open “Tutorial Data #1.txt.”**

A notebook window showing the contents of the file appears. If desired, we could edit the data and then save it. For now we just observe that the file appears to be tab-delimited (tabs separate the columns) and contains names for the columns. Note that the name of the first column will conflict with the data we just entered and the other names have spaces in them.

3. **Click the close button or press Command-W (Macintosh) or Ctrl+W (Windows).**

A dialog appears asking what you want to do with the notebook window.

4. **Click the Kill button.**

The term “kill” means to “completely remove from the experiment”. The file will not be affected. Now we will actually load the data.

5. **Choose Data→Load Waves→Load Delimited Text.**

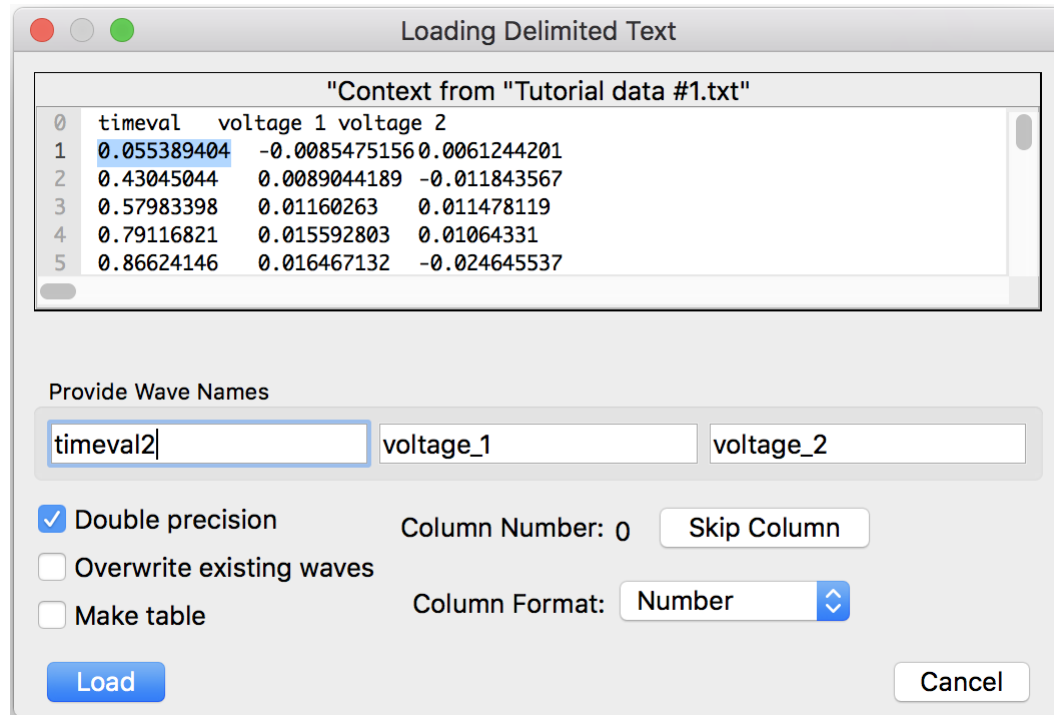
An Open File dialog appears.

6. **Again choose “Tutorial Data #1.txt” and click Open.**

The Loading Delimited Text dialog appears. The name “timeval” is highlighted and an error message is shown. Observe that the names of the other two columns were fixed by replacing the spaces with underscore characters.

7. Change “timeval” to “timeval2”.

The dialog should now look like this:



8. Click the Make Table box to check it and then click Load.

The data is loaded and a new table is created to show the data.

9. Click the close button of the new table window.

A dialog is presented asking if you want to create a recreation macro.

10. Click the No Save button.

The data we just loaded is still available in Igor. A table is just a way of viewing data and is not necessary for the data to exist.

The Load Delimited Text menu item that you used is a shortcut that uses default settings for loading delimited text. Later, when you load your own data files, choose Data→Load Waves→Load Waves so you can see all of the options.

Appending to a Graph

1. If necessary, click in Graph0 to bring it to the front.

The Graph menu is available only when the target window is a graph.

2. Choose the Graph→Append Traces to Graph menu item.

The Append Traces dialog appears. It is very similar to the New Graph dialog that you used to create the graph.

3. In the Y Waves list, select voltage_1 and voltage_2.

4. In the X Wave list, select timeval2.

5. Click Do It.

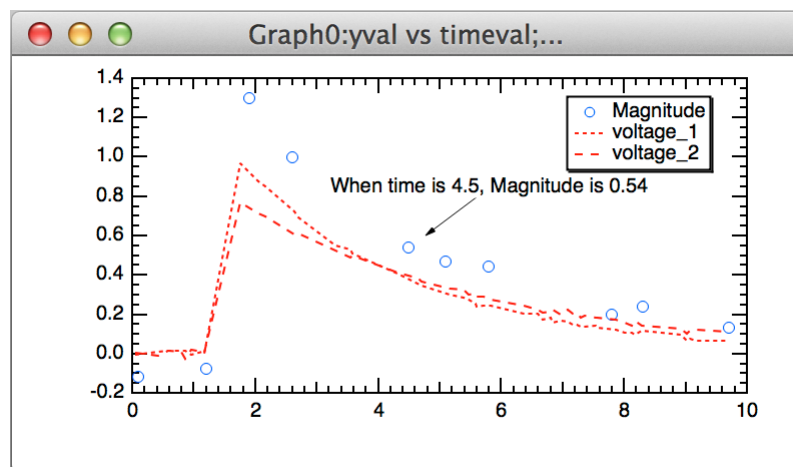
Two additional traces are appended to the graph. Notice that they are also appended to the Legend.

6. Position the cursor over one of the traces in the graph and double-click.

The Modify Trace Appearance dialog appears with the trace you double-clicked already selected.

7. If necessary, select `voltage_1` in the list of traces.
8. Choose dashed line #2 from the Line Style pop-up menu.
9. Select `voltage_2` in the list of traces.
10. Choose dashed line #3 from the Line Style pop-up menu.
11. Click Do It.

The graph should now look like this:



Offsetting a Trace


1. Position the cursor directly over the `voltage_2` trace.
The `voltage_2` trace has the longer dash pattern.
2. Click and hold the mouse button for about 1 second.
An XY readout appears in the lower-left corner of the graph and the trace will now move with the mouse.
3. With the mouse button still down, press Shift while dragging the trace up about 1 cm and release.
The Shift key constrains movement to vertical or horizontal.
You have added an offset to the trace. If desired, you could add a tag to the trace indicating that it has been offset and by how much. This trace offset does not affect the underlying wave data.


Unoffsetting a Trace

1. Choose the Edit→Undo Trace Drag menu item.
You can undo many of the interactive operations on Igor windows.
2. Choose Edit→Redo Trace Drag.
The following steps show how to remove an offset after it is no longer undoable.
3. Double-click the `voltage_2` trace.
The Modify Trace Appearance dialog appears with `voltage_2` selected. (If `voltage_2` is not selected, select it.) The Offset checkbox is checked.
4. Click the Offset checkbox.
This turns offset off for the selected trace and the offset controls in the dialog are hidden.
5. Click Do It.
The `voltage_2` trace is returned to its original position.


Drawing in a Graph

1. If necessary, click Graph0 to bring it to the front.
2. Choose the Graph→Show Tools menu item or press Command-T (Macintosh) or Ctrl+T (Windows).

A tool palette is added to the graph. The second icon from the top () is selected indicating that the graph is in drawing mode as opposed to normal (or “operate”) mode.

3. Click the top icon () to go into normal mode.

Normal mode is for interacting with graph objects such as traces, axes and annotations. Drawing mode is for drawing lines, rectangles, polygons and so on.

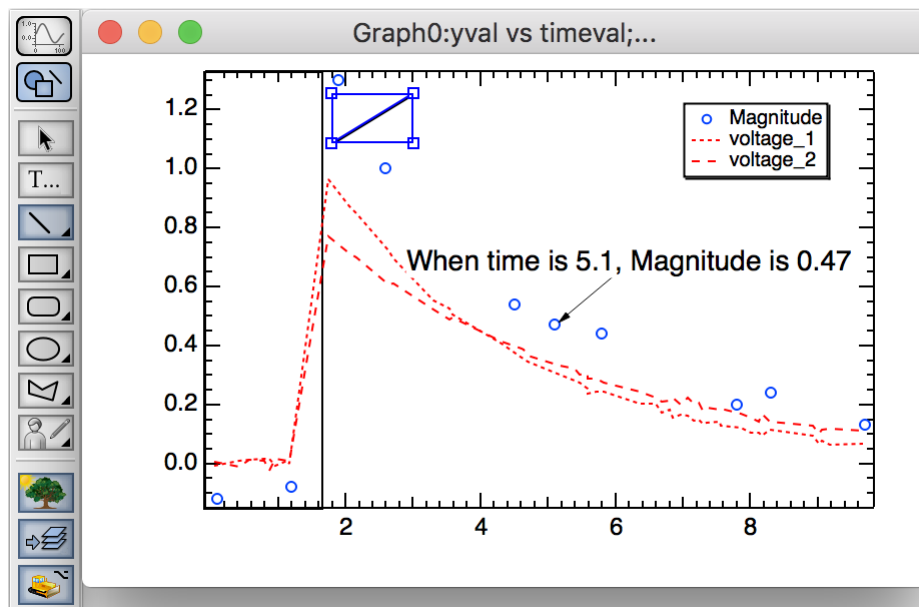
4. Click the second icon to return to drawing mode.
5. Click the drawing layer icon () .



A pop-up menu showing the available drawing layers and their relationship to the graph layers appears. The items in the menu are listed in back-to-front order.

6. Choose UserBack from the menu.

We will be drawing behind the axes, traces and all other graph elements.

7. Click the rectangle tool and drag out a rectangle starting at the upper-left corner of the plot area ($y=1.4$, $x=0$ on the axes) and ending at the bottom of the plot area and about 1.5 cm in width ($y=-0.2$, $x=1.6$).
8. Click the line tool and draw a diagonal line as shown, starting at the left, near the peak of the top trace, and ending at the right:



9. Click the drawing environment icon () and choose At Start from the Line Arrow item.
10. Click the Text tool icon () .
11. Click just to the right of the line you just drew.
The Create Text dialog appears.
12. Type “Precharge”.
13. From the Anchor pop-up menu, choose Left Center.
14. Click Do It.

15. **Click the graph's zoom button (*Macintosh*) or maximize button (*Windows*).**

Notice how the rectangle and line expand with the graph. Their coordinates are measured relative to the plot area (rectangle enclosed by the axes).
16. **Click the graph's zoom button (*Macintosh*) or restore button (*Windows*).**
17. **Click the Arrow tool and then double-click the rectangle.**

The Modify Rectangle dialog appears showing the properties of the rectangle.
18. **Enter 0 in the Thickness box in the Line Properties section.**

This turns off the frame of the rectangle.
19. **Choose Solid from the Fill Mode pop-up menu.**
20. **Choose a light gray color from the Fore Color pop-up menu under the Fill Mode pop-up menu.**
21. **Click Do It.**


Observe that the rectangle forms a gray area behind the traces and axes.
22. **Again, double-click the rectangle.**

The Modify Rectangle dialog appears.
23. **From the X Coordinate pop-up menu, choose Axis Bottom.**

The X coordinates of the rectangle will be measured in terms of the bottom axis — as if they were data values.
24. **Press Tab until the X0 box is selected and type "0".**
25. **Tab to the X1 box and type "1.6".**
26. **Tab to the Y0 box and type "0".**
27. **Tab to Y1 and type "1".**

The X coordinates of the rectangle are now measured in terms of the bottom axis and the left side will be at zero while the right side will be at 1.6.

The Y coordinates are still measured relative to the plot area. Since we entered zero and one for the Y coordinates, the rectangle will span the entire height of the plot area.
28. **Click Do It.**

Notice the rectangle is nicely aligned with the axis and the plot area.
29. **Click the operate icon () to exit drawing mode.**
30. **Press Option (*Macintosh*) or Alt (*Windows*), click in the middle of the plot area and drag about 2 cm to the right.**

The X axis range changes. Notice that the rectangle moved to align itself with the bottom axis.
31. **Choose Edit→Undo Modify.**

Making a Window Recreation Macro

1. **Click the graph's close button.**

Igor presents a dialog which asks if you want to save a window recreation macro. The graph's name is "Graph0" so Igor suggests "Graph0" as the macro name.
2. **Click Save.**

Igor generates a window recreation macro in the currently hidden procedure window. A window recreation macro contains the commands necessary to recreate a graph, table, page layout, control panel or 3D plot. You can invoke this macro to recreate the graph you just closed.

3. **Choose the Windows→Procedure Windows→Procedure Window menu item.**

The procedure window is always present but is usually hidden to keep it out of the way. The window now contains the recreation macro for Graph0. You may need to scroll up to see the start of the macro. Because of the way it is declared:

```
Window Graph0 () : Graph
```

this macro will be available from the Graph Macros submenu of the Windows main menu.

4. **Click the procedure window's close button.**

This hides the procedure window. Most other windows display a dialog asking if you want to kill or hide the window, but the built-in procedure window and the help windows simply hide themselves.

Recreating the Graph

1. **Choose the Windows→Graph Macros→Graph0 menu item.**

Igor executes the Graph0 macro which recreates a graph of the same name.

2. **Repeat step 1.**

The Graph0 macro is executed again but this time Igor gave the new graph a slightly different name, Graph0_1, because a graph named Graph0 already existed.

3. **While pressing Option (Macintosh) or Alt (Windows), click the close button of Graph0_1.**

The window is killed without presenting a dialog.

Using the Data Browser

The Data Browser lets you navigate through the data folder hierarchy and examine properties of waves and values of numeric and string variables.


1. **Choose the Data→Data Browser menu item.**

The Data Browser appears.

2. **Make sure all of the checkboxes in the top-left corner of the Data Browser are checked.**

3. **Click the timeval wave icon to select it.**

Note that the wave is displayed in the plot pane at the bottom of the Data Browser and the wave's properties are displayed just above in the info pane.

If you don't see this, click the info icon () half-way down the left side of the Data Browser window.

4. **Control-click (Macintosh) or right-click (Windows) on the timeval wave icon.**

A contextual menu appears with a number of actions that you can perform on the selection.

5. **Press Escape to dismiss the contextual menu.**

You can explore that and other Data Browser features later on your own.

6. **Click the Data Browser's close box to close it.**

Saving Your Work - Tour 1B

1. **Choose the File→Save Experiment As menu item.**

2. **Navigate to your "Guided Tours" folder.**

This is the folder that you created under **Saving Your Work - Tour 1A** on page I-23.

3. **Change the name to "Tour 1B.pxp" and click Save.**

If you want to take a break, you can quit from Igor now.

Using Igor Documentation

Now we will take a quick look at how to find information about Igor.

In addition to guided tours such as this one, Igor includes tool tips, general usage information, and reference information. The main guided tours, as well as the general and reference information, are available in both the online help files and in the Igor Pro PDF manual.

1. **Choose Misc→Miscellaneous Settings, click the Help icon on the left side, and verify that the Show Tool Tips checkbox is checked.**

If it is unchecked, check it..

2. **Click Save Settings to close the Miscellaneous Settings dialog.**

3. **Choose Data→Load Waves→Load Waves.**

Igor displays the Load Waves dialog. This dialog provides an interface to the LoadWave operation which is how you load data into Igor from text data files.

4. **On Macintosh only, move the cursor over the Load Columns Into Matrix checkbox.**

A tool tip appears in a yellow textbox. You can get a tip for most dialog items and icons this way.

5. **Click the Cancel button to quit the dialog.**

Now let's see how to get reference help for a particular operation.

6. **Choose Help→Command Help.**

The Igor Help Browser appears with the Command Help tab displayed.

The information displayed in this tab comes from the Igor Reference help file - one of many help files that Igor automatically opens at launch. Open help files are directly accessible through Help→Help Windows but we will use the Igor Help Browser right now.

7. **Click Show All above the left-hand list.**

8. **Click any item in the list and then type "Loa".**

Igor displays help for the LoadData operation. We want the LoadWave operation.

9. **Press the down-arrow key a few times until LoadWave is selected in the list.**

Igor displays help for the LoadWave operation in the help area on the right.

Another way to get reference help is to Control-click (Macintosh) or right-click (Windows) the name of an operation or function and choose the "Help For" menu item. This works in the command window and in procedure, notebook and help windows.

10. **In the Filter edit box just below the left-hand list, type "Matrix".**

The list now shows only operations, functions, and keywords whose names include "matrix".

11. **Click Show All above the left-hand list.**

The list shows all operations, functions, and keywords again.

12. **Click the Advanced Filtering control to reveal additional checkboxes and pop-up menus.**

These controls provide other ways to filter what appears in the left-hand list.

While we're in the Igor Help Browser, let's see what the other tabs are for.

13. **Click each of the Help Browser tabs and note their contents.**

You can explore these tabs in more detail later.

Next we will take a quick trip to the Igor Pro PDF manual. If you are doing this guided tour using the PDF manual, you may want to just read the following steps rather than do them to avoid losing your place.

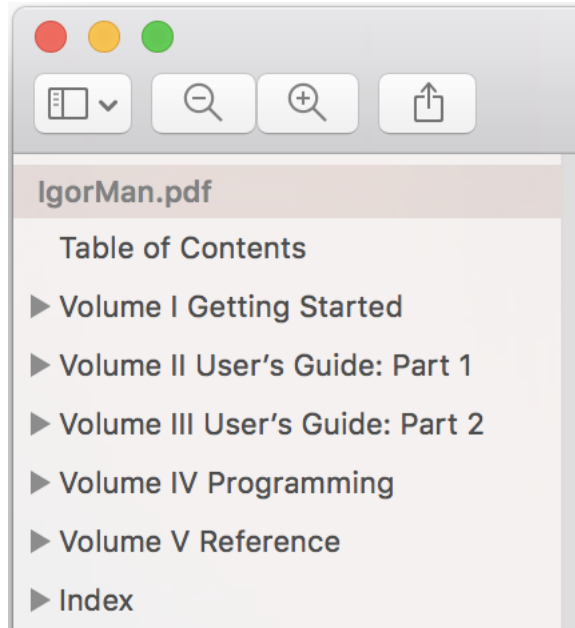
14. Click the Manual tab and then click the Open Online Manual button.

Igor opens the PDF manual in your PDF viewer - typically Adobe Reader or Apple's Preview program.

If you use Adobe Reader for viewing PDF files, you should have a Bookmarks pane on the left side of the PDF manual window. If not, choose View→Show/Hide→Navigation Panes→Bookmarks in Reader.

If you use Apple's Preview for PDF files, you should have a sidebar displaying the table of contents on one side of the main page. If not, choose View→Table of Contents in Preview.

Note in the Reader Bookmarks pane or the Preview table of contents that the PDF manual is organized into five volumes plus an index.

**15. Use the bookmarks or table of contents to get a sense of what's in the manual.**

Expand the volume bookmarks to see the chapter names.

You may notice that the Igor PDF manual is rather large - over 2,000 pages at last count. You'll be happy to know that we don't expect you to read it cover-to-cover. Instead, read chapters as the need arises.

The information in the manual is also in the online help files. The manual, being in book format, is better organized for linear reading while the online help is usually preferred for accessing reference information.

In case you ever want to open it directly, you can find the PDF manual in "Igor Pro 7 Folder/Manual".

That should give you an idea of where to look for information about Igor. Now let's get back to our hands-on exploration of Igor.

Graphically Editing Data



0. If you quit Igor after the last save, open your "Tour 1B.pxp" experiment and turn preferences off.

1. Adjust the positions of the graph and table so you can see both.

Make sure you can see the columns of data in the table when the graph is the front window.

2. If necessary, click the Graph0 window to bring it to the front.

3. Click the drawing mode icon () to activate the drawing tools.

4. **Move the cursor over the polygon icon () and click and hold the mouse button.**
A pop-up menu appears.
5. **Choose the Edit Wave menu item.**
6. **Click one of the open circles of the yval trace.**
The trace is redrawn using lines and squares to show the location of the data points.
7. **Click the second square from the left and drag it 1 cm up and to the right.**
Notice point 1 of yval and timeval changes in the table.
8. **Press Command-Z (Macintosh) or Ctrl+Z (Windows) or choose Edit→Undo.**
9. **Click midway between the first and second point and drag up 1 cm.**
A new data point is added to the yval and timeval waves.
10. **Press Option (Macintosh) or Alt (Windows) and click the new data point.**
The new data point is deleted.
You could also have pressed Command-Z (Macintosh) or Ctrl+Z (Windows) to undo the insertion.
11. **Press Command (Macintosh) or Ctrl (Windows), click the line segment between the second and third point and drag a few cm to the right.**
The line segment is moved and two points of yval and timeval are changed in the table.
12. **Press Command-Z (Macintosh) or Ctrl+Z (Windows) or choose Edit→Undo.**
13. **Click in the operate icon () to exit drawing mode.**
14. **Choose File→Revert Experiment and answer Yes in the dialog.**
This returns the experiment to the state it was in before we started editing the data.
15. **Choose File→New Experiment.**
This clears the windows and data from the previously open experiment, creating a new, empty experiment.

Making a Category Plot (Optional)

Category plots show continuous numeric data plotted against non-numeric text categories.

1. **Choose the Windows→New Table menu item.**
2. **Click in the Do It button.**
A new blank table is created.
3. **Type “Monday” (without the quotes) and then press Return or Enter.**
A wave named “textWave0” was created with the text Monday as the value of the first point. Entering a non-numeric value in the first row of the first blank column automatically creates a new text wave.
4. **Type the following lines, pressing Enter after each one:**
Tuesday
Wednesday
Thursday
5. **Click in the first cell of the next column and enter the following values:**
10
25
3
16

6. Click in the first cell of the next column and enter the following values:

0
12
30
17

7. Choose **Windows**→**New**→**Category Plot**.

A dialog similar to the New Graph dialog appears. This dialog shows only text waves in the right-hand list.

8. In the **Y Waves** list, select **wave0** and **wave1**.

9. In the **X Wave list** select **textWave0**.

10. Click **Do It**.

A category plot is created.

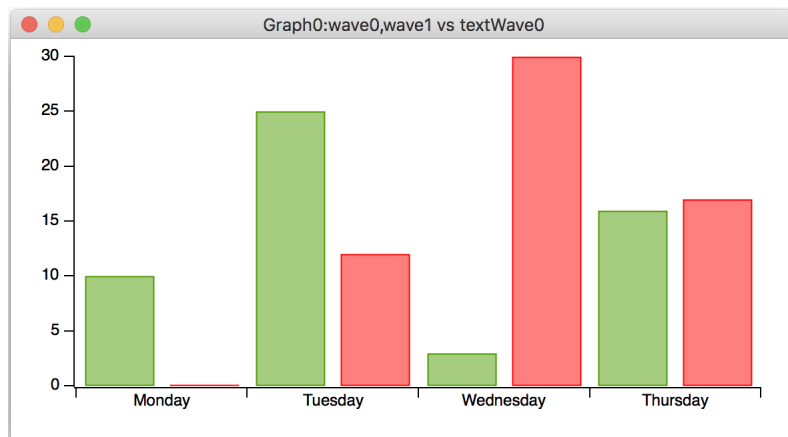
11. Double-click one of the bars.

The Modify Trace Appearance dialog appears.

12. Using the **Color** pop-up menu, set the color of the **wave0** trace to green.

13. Click **Do It**.

The graph should now look like this:



14. Choose **File**→**Save Experiment As** and save the current experiment as “**Category Plots.pxp**”.

Category Plot Options (Optional)

This section explores various category-plot options. If you are not particularly interested in category plots, you can stop now, or at any point in the following steps, by skipping to the next section.

1. Double-click one of the bars and, if necessary, select the **wave0** in the list.

2. From the **Grouping** pop-up menu, choose **Stack on Next**.

3. Click **Do It**.

The left bar in each group is now stacked on top of the right bar.

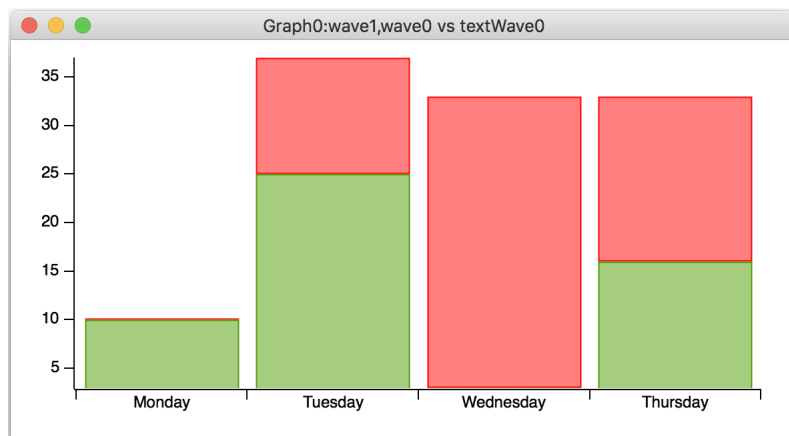
4. Choose the **Graph**→**Reorder Traces** menu item.

The Reorder Traces dialog appears.

5. Reverse the order of the items in the list by dragging the top item down and click **Do It**.

The bars are no longer stacked and the bars that used to be on the left are now on the right. The reason the bars are not stacked is that the trace that we set to Stack on Next mode is now last and there is no next trace.

6. Using the **Modify Trace Appearance** dialog, set the **wave1** trace to **Stack on next**. Click **Do It**. The category plot graph should now look like this:



7. Enter the following values in the next blank column in the table:

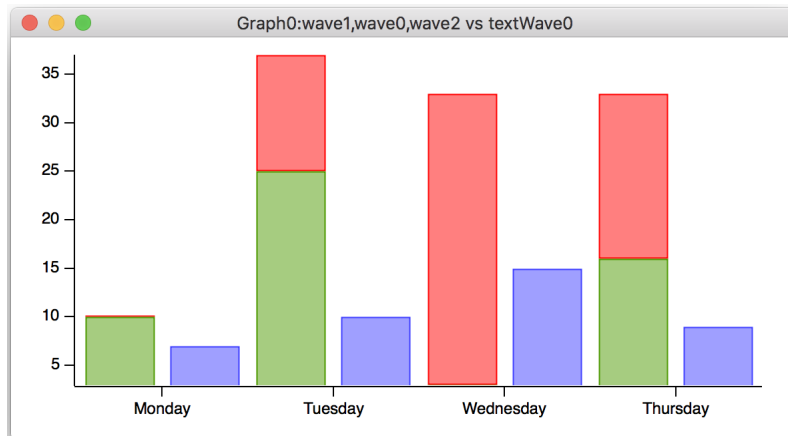
7
10
15
9

This creates a new wave named **wave2**.

8. Click the graph to bring it to the front.
9. Choose **Graph**→**Append to Graph**→**Category Plot**. The **Append Category Traces** dialog appears.
10. In the **Y Waves** list, select **wave2** and click **Do It**. This adds a red bar underneath each green bar.
11. Control-click (Macintosh) or right-click (Windows) one of the new red bars (underneath a green bar) to display the contextual pop-up menu and choose **blue** from the **Color** submenu. The new bars are now blue.
12. Using the **Modify Trace Appearance** dialog, change the grouping mode of the middle trace, **wave0**, to **none**. We now have **wave1** (red) stacked on **wave0** (green) and **wave0** not stacked on anything. Now the new **wave2** bars (blue) are to the right of a group of two stacked bars. You can create any combination of stacked and side-by-side bars.
13. Double-click the bottom axis. The **Modify Axis** dialog appears with the bottom axis selected.
14. Click the **Auto/Man Ticks** tab.

15. Click the Tick In Center checkbox and then click Do It.

Notice the new positions of the tick marks.



16. Double-click the bottom axis again.
17. Click the Axis tab.
18. Change the value of Bar Gap to zero and then click Do It.
Notice that the bars within a group are now touching.
19. Use the Modify Axis dialog to set the Category Gap to 50%.
The widths of the bars shrink to 50% of the category width.
20. Choose Graph→Modify Graph.
21. Click the “Swap X & Y Axes” checkbox and then click Do It.
This is how you create a horizontal bar plot.
22. Choose File→Save Experiment.

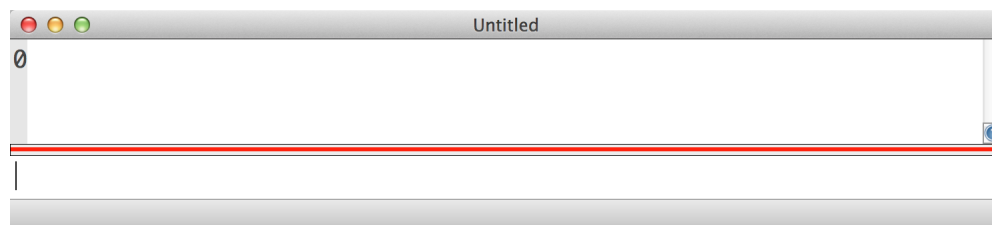
The Command Window

Parts of this tour make use of Igor’s command line to enter mathematical formulae. Let’s get some practice now.

1. Choose File→New Experiment.

This clears any windows and data left over from previous experimentation.

Your command window should look something like this:



If you don’t see the command window, choose Windows→Command Window.

The command line is the space below the red separator whereas the space above the separator is called the history area.

2. Click in the command line, type the following line and press Return or Enter.

```
Print 2+2
```

The Print command as well as the result are placed in the history area.

3. Press the Up Arrow key.

The line containing the print command is selected, skipping over the result printout line.

4. **Press Return or Enter.**

The selected line in the history is copied to the command line.
5. **Edit the command line so it matches the following and press Return or Enter.**

```
Print "The result is ", 2+2
```

The Print command takes a list of numeric or string expressions, evaluates them and prints the results into the history.
6. **Choose the Help→Igor Help Browser menu item.**

The Igor Help Browser appears.

You can also display the help browser by clicking the question-mark icon near the right edge of the command window and, on Windows, by pressing F1.
7. **Click the Command Help tab in the Igor Help Browser.**
8. **Click Advanced Filtering if necessary to reveal the advanced options.**
9. **Click Show All above the left-hand list.**
10. **Uncheck the Functions and Programming checkboxes and check the Operations checkbox.**

A list of operations appears.
11. **In the pop-up menu next to the Operations checkbox, choose About Waves.**
12. **Select PlaySound in the list.**

Tip: Click in the list to activate it and then type “p” to jump to PlaySound.

The reference help for the PlaySound operation appears in the help area on the right.
13. **Click the help area on the right, scroll down to the Examples section, and select the first four lines of example text (starting with “Make”, ending with “PlaySound sineSound”).**
14. **Choose the Edit→Copy menu to copy the selection.**
15. **Close the Igor Help Browser.**
16. **Choose Edit→Paste to paste the command into the command line.**

All four lines are pasted into the command line area.
17. **Make the command window taller and then drag the red divider line up so you can see the commands in the command line.**
18. **Press Return or Enter to execute the commands.**

The four lines are executed and a short tone plays.
19. **Click once on the last line in the history area, on “PlaySound sineSound”.**

The entire command is selected just as if you pressed the arrow key.
20. **Press Return or Enter once to transfer the command to the command line and a second time to execute it.**

The tone plays again as the line executes.

We are finished with the “sineSound” wave that was created in this exercise so let’s kill the wave to prevent it from cluttering up our wave lists.
21. **Choose Data→Kill Waves.**

The Kill Waves dialog appears.
22. **Select “sineSound” and click Do It.**

The sineSound wave is removed from memory.
23. **Again click once on the history line “PlaySound sineSound”.**
24. **Press Return or Enter twice to re-execute the command.**

An error dialog is presented because the sineSound wave no longer exists.
25. **Click OK to close the error dialog.**

26. **Choose Edit→Clear Cmd Buffer or press Command-K (Macintosh) or Ctrl+K (Windows).**

When a command generates an error, it is left in the command line so you can edit and re-execute it. In this case we just wanted to clear the command line.

Synthesizing Data

In this section we will make waves and fill them with data using arithmetic expressions.

0. **Choose File→New Experiment.**

This clears any windows and data left over from previous experimentation.

1. **Choose the Data→Make Waves menu item.**

The Make Waves dialog appears.

2. **Type “spiralY” in the first box, press the tab key, and type “spiralX” in the second box.**

3. **Change Rows to 1000.**

4. **Click Do It.**

Two 1000 point waves have been created. They are now part of the experiment but are not visible because we haven't displayed them in a table or graph.

5. **Choose Data→Change Wave Scaling.**

The Change Wave Scaling dialog appears. We will use it to set the X scaling of the waves.

6. **If a button labeled More Options is showing, click it.**

7. **In the Waves list, click spiralY and then Command-click (Macintosh) or Ctrl-click (Windows) spiralX.**

8. **Choose Start and Right in the SetScale Mode pop-up menu.**

9. **Enter “0” for Start and “50” for Right.**

10. **Click Do It.**

This executes a **SetScale** command specifying the X scaling of the spiralX and spiralY waves. X scaling is a property of a wave that maps a point number to an X value. In this case we are mapping point numbers 0 through 999 to X values 0 through 50.

11. **Type the following on the command line and then press Return or Enter:**

```
spiralY = x*sin(x)
```

This is a waveform assignment statement. It assigns a value to each point of the destination wave (spiralY). The value stored for a given point is the value of the right-hand expression at that point. The meaning of x in a waveform assignment statement is determined by the X scaling of the destination wave. In this case, x takes on values from 0 to 50 as Igor evaluates the right-hand expression for points 0 through 999.

12. **Execute this in the command line:**

```
spiralX = x*cos(x)
```

Now both spiralX and spiralY have their data values set.

Zooming and Panning

1. **Choose the Windows→New Graph menu item.**
2. **If necessary, uncheck the From Target checkbox.**
3. **In the Y Waves list, select “spiralY”.**
4. **In the X Wave list, select “_calculated_”.**

5. **Click Do It.**

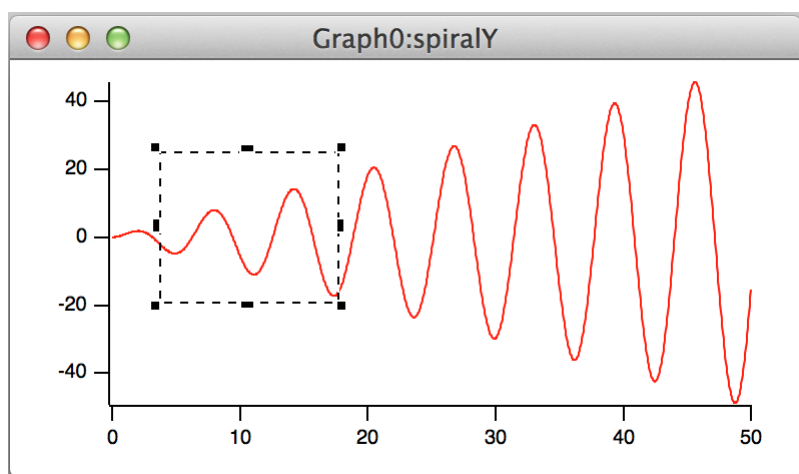
Igor creates a graph of spiralY's data values versus its X values.

Note that the X axis goes from 0 to 50. This is because the **SetScale** command we executed earlier set the X scaling property of spiralY which tells Igor how to compute an X value from a point number. Choosing `_calculated_` from the X Wave list graphs the spiralY data values versus these calculated X values.

6. **Position the cursor in the interior of the graph.**

The cursor changes to a cross-hair shape.

7. **Click and drag down and to the right to create a marquee as shown:**



You can resize the marquee with the handles (black squares). You can move the marquee by dragging the dashed edge of the marquee.

8. **Position the cursor inside the marquee.**

The mouse pointer changes to this shape: , indicating that a pop-up menu is available.

9. **Click and choose Expand from the pop-up menu.**

The axes are rescaled so that the area enclosed by the marquee fills the graph.

10. **Choose Edit→Undo Scale Change or press Command-Z (Macintosh) or Ctrl+Z (Windows).**

11. **Choose Edit→Redo Scale Change or press Command-Shift-Z (Macintosh) or Ctrl+Shift+Z (Windows).**

12. **Press Option (Macintosh) or Alt (Windows) and position the cursor in the middle of the graph.**

The cursor changes to a hand shape. You may need to move the cursor slightly before it changes shape.

13. **With the hand cursor showing, drag about 2 cm to the left.**

14. **While pressing Option (Macintosh) or Alt (Windows), click the middle of the graph and gently fling it to the right.**

The graph continues to pan until you click again to stop it.

15. **Click the plot area of the graph to stop panning.**

16. **Choose Graph→Autoscale Axes or press Command-A (Macintosh) or Ctrl+A (Windows).**

Continue experimenting with zooming and panning as desired.

17. **Press Command-Option-W (Macintosh) or Ctrl+Alt+W (Windows).**

The graph is killed. Option (Macintosh) or Alt (Windows) avoided the normal dialog asking whether to save the graph.

Making a Graph with Multiple Axes

1. Choose the **Windows**→**New Graph** menu item.
2. If you see a button labeled **More Choices**, click it.

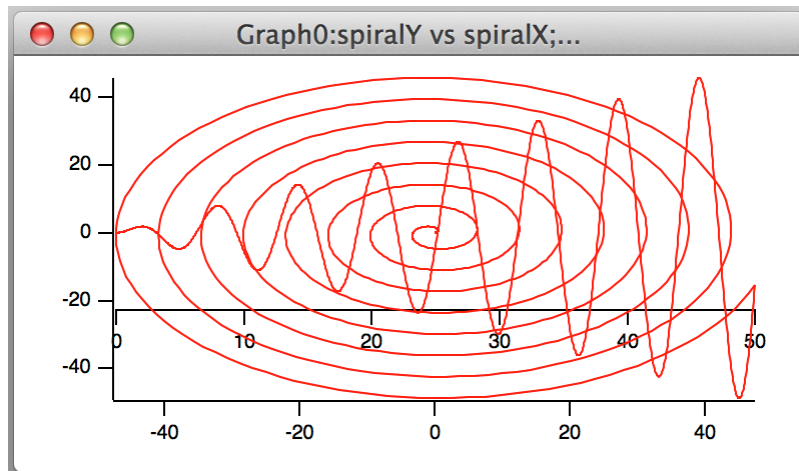
We will use the more complex form of the dialog to create a multiple-axis graph in one step.

3. In the **Y Waves** list, select **"spiralY"**.
 4. In the **X Wave** list, select **"spiralX"**.
 5. Click **Add**.
- The selections are inserted into the lower list in the center of the dialog.
6. In the **Y Waves** list, again select **"spiralY"**.
 7. In the **X Wave** list, select **"_calculated_"**.
 8. Choose **New** from the **Axis pop-up** menu under the **X Waves** list.
 9. Enter **"B2"** in the name box.
 10. Click **OK**.


Note the command box at the bottom of the dialog. It contains two commands: a **Display** command corresponding to the initial selections that you added to the lower list and an **AppendToGraph** command corresponding to the current selections in the **Y Waves** and **X Wave** lists.

11. Click **Do It**.

The following graph is created:



The interior axis is called a "free" axis because it can be moved relative to the plot rectangle. We will be moving it outside of the plot area but first we must make room by adjusting the plot area margins.

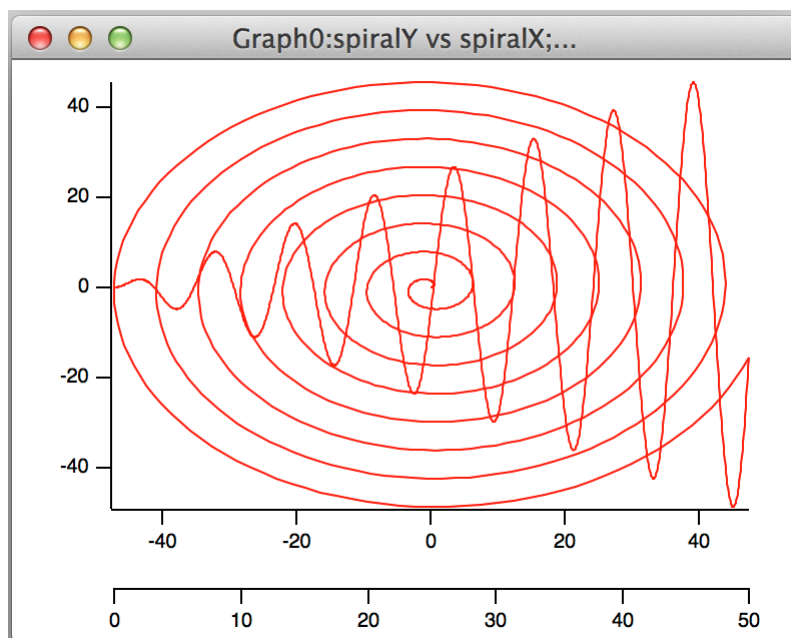
12. Press **Option (Macintosh)** or **Alt (Windows)** and position the cursor over the bottom axis until the cursor changes to this shape: .

This shape indicates you are over an edge of the plot area rectangle and that you can drag that edge to adjust the margin.

13. Drag the margin up about 2 cm. Release the **Option (Macintosh)** or **Alt (Windows)**.
14. Drag the interior axis down into the margin space you just created.

15. **Resize the graph so the spiral is nearly circular.**

The graph should now look like this:



Saving Your Work - Tour 1C

1. **Choose the File→Save Experiment As menu item.**
2. **Navigate to your “Guided Tours” folder.**

This is the folder that you created under **Saving Your Work - Tour 1A** on page I-23.

3. **Type “Tour 1C.pxp” in the name box and click Save.**

If you want to take a break, you can quit from Igor now.

Using Cursors

0. **If you are returning from a break, open your “Tour 1C.pxp” experiment and turn preferences off.**
1. **Click in the graph and choose the Graph→Show Info menu item.**


A cursor info panel appears below the graph.

2. **Control-click (Macintosh) or right-click (Windows) in the name area for graph cursor A (the round one), where it says “A:”.**
3. **Choose “spiralY” from the pop-up menu.**

The A cursor is placed on point zero of spiralY.

4. **Repeat for cursor B but choose “spiralY#1” from the pop-up menu.**

The wave spiralY is graphed twice. The #1 suffix is used to distinguish the second instance from the first. It is #1 rather than #2 because in Igor indices start from zero.

5. **Position the mouse pointer over the center of the cursor position control .**
6. **Click the blue slider and gently drag it to the right.**

Both cursors move to increasing point numbers. They stop when one or both get to the end.

You can also move the cursors using the left and right arrow keys on the keyboard or by clicking to the left or right of the blue slider.

7. **Practice moving the slider to the left and right.**
Notice that the cursors move with increasing speed as the slider is displaced farther from the center.
8. **Click once on the dock for cursor A (the round black circle).**
The circle turns white, indicating that cursor A is deselected.
9. **Move the slider to the left and right.**
Notice that only cursor B moves.
10. **Click cursor B in the graph and drag it to another position on either trace.**
You can also drag cursors from their docks to the graph.
11. **Click cursor A in the graph and drag it completely outside the graph.**
The cursor is removed from the graph and returns to its dock.
12. **Choose Graph→Hide Info.**
13. **Click in the command window, type the following and press Return or Enter.**
`Print vcsr(B)`
The Y value at cursor B is printed into the history area. There are many functions available for obtaining information about cursors.
14. **Click in the graph and then drag cursor B off of the graph.**

Removing a Trace and Axis

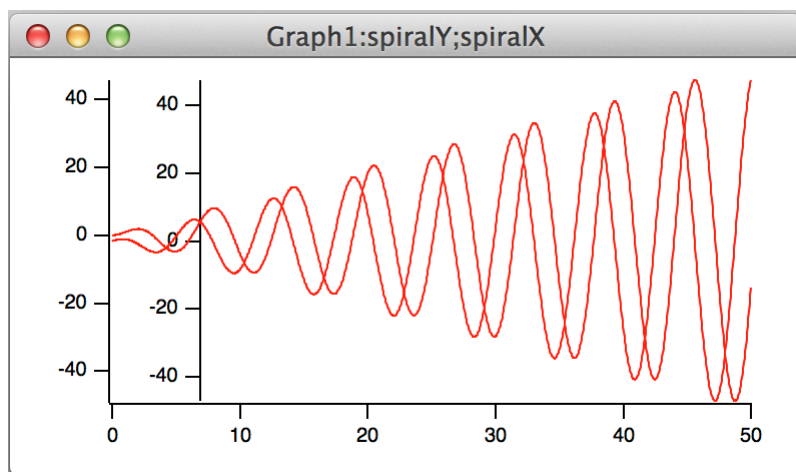
1. **Choose the Graph→Remove from Graph menu item.**
The Remove From Graph dialog appears with spiralY listed twice. When we created the graph we used spiralY twice, first versus spiralX to create the spiral and second versus calculated X values to show the sine wave.
2. **Click the second instance of spiralY (spiralY#1) and click Do It.**
The sine wave and the lower axis are removed. An axis is removed when its last trace is removed.
3. **Drag the horizontal axis off the bottom of the window.**
This returns the margin setting to auto. We had set it to a fixed position when we option-dragged (*Macintosh*) or Alt-dragged (*Windows*) the margin in a previous step.

Creating a Graph with Stacked Axes

1. **Choose the Windows→New Graph menu item.**
2. **If you see a button labeled More Choices, click it.**
3. **In the Y Waves list, select “spiralY”.**
4. **In the X Wave list, select “_calculated_”.**
5. **Click Add.**
6. **In the Y Waves list, select “spiralX”.**
7. **In the X Wave list, select “_calculated_”.**
8. **Choose New from the Axis pop-up menu under the Y Waves list.**
9. **Enter “L2” in the name box.**
10. **Click OK.**

11. **Click Do It.**

The following graph is created.



In the following steps we will stack the L2 axis on top of the left axis.

12. **Double-click the far left axis.**

The Modify Axis dialog appears. If any other dialog appears, cancel and try again making sure the cursor is over the axis.

The Left axis should be selected in the Axis pop-up menu in the upper-left corner of the dialog.

13. **Click the Axis tab.**

14. **Set the Left axis to draw between 0 and 45% of normal.**

15. **Choose L2 from the Axis pop-up menu.**

16. **Set the L2 axis to draw between 55 and 100% of normal.**

17. **In the Free Axis Position box, pop up the menu reading Distance from Margin and select Fraction of Plot Area.**

18. **Verify that the box labeled “% of Plot Area” is set to zero.**

Steps 17 and 18 move the L2 axis so it is in line with the Left axis.

Why don't we make this the default? Good question — positioning as percent of plot area was added in Igor Pro 6; the default behavior maintains backward compatibility.

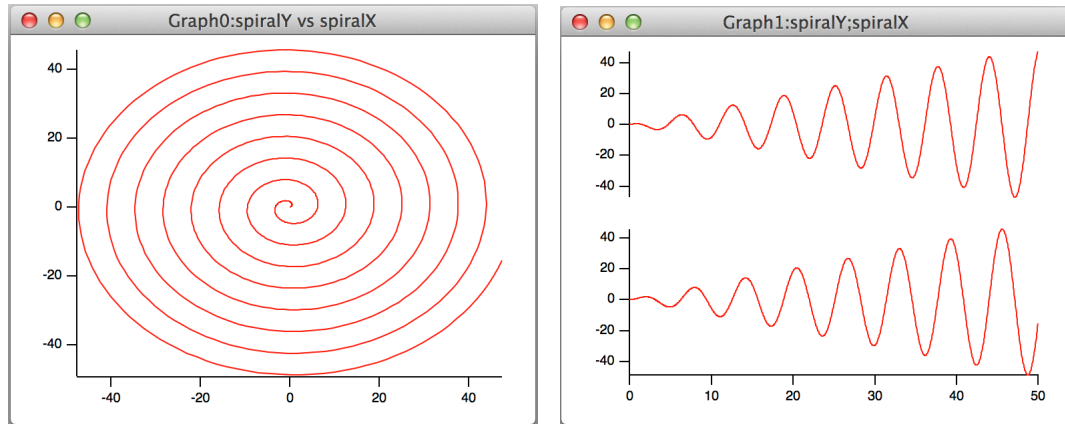
19. **Choose Bottom from the Axis pop-up menu.**

20. **Click the Axis Standoff checkbox to turn standoff off.**



21. **Click Do It.**

22. **Resize and reposition the Graph0 and Graph1 windows so they are side-by-side and roughly square.**

The graphs should look like this:



Creating a Page Layout

1. **Choose the Windows→New Layout menu item and click Do It.**
A new blank page layout window is created.
2. **Click in the graph icon () and choose “Graph0”.**
Graph0 is added to the layout.
3. **Again, click in the graph icon and choose “Graph1”.**
Graph1 is added to the layout.
4. **Click the marquee icon .**
5. **Drag out a marquee that approximately fills the bottom half of the page.**
6. **Choose Layout→Arrange Objects.**
The Arrange Objects dialog appears.
7. **Select both Graph0 and Graph1 and leave the Use Marquee checkbox checked.**
8. **Click Do It.**
The two graphs are tiled inside the area defined by the marquee.
9. **Click in the page area outside the marquee to dismiss it.**
10. **Choose Windows→Send to Back.**
This sends the page layout window behind all other windows.

Saving Your Work - Tour 1D

1. **Choose the File→Save Experiment As menu item.**
2. **Navigate to your “Guided Tours” folder.**
This is the folder that you created under **Saving Your Work - Tour 1A** on page I-23.
3. **Type “Tour 1D.pxp” in the name box and click Save.**
If you want to take a break, you can quit Igor Pro now.

Creating Controls

This section illustrates adding controls to an Igor graph — the type of thing a programmer might want to do. If you are not interested in programming, you can skip to the **End of the General Tour** on page I-47.

0. **If you are returning from a break, open your “Tour 1D.pxp” experiment and turn off preferences.**

1. **Click the graph with the spiral (Graph0) to bring it to the front.**
2. **Choose the Graph→Show Tools menu item or press Command-T (Macintosh) or Ctrl+T (Windows).**

A tool palette is displayed to the left of the graph. The second icon is selected indicating that the graph is in the drawing as opposed to normal mode.

The selector tool (arrow) is active. It is used to create, select, move and resize controls.
3. **Choose Graph→Add Control→Control Bar.**

The Control Bar dialog appears.
4. **Enter a height of 30 points and click Do It.**

This reserves a space at the top of the graph for controls.
5. **Click in the command line, type the following and press Return or Enter.**

```
Variable ymult=1, xmult=1
```

This creates two numeric variables and sets both to 1.0.
6. **Click Graph0 and then choose Graph→Add Control→Add Set Variable.**

The SetVariable Control dialog appears.

A SetVariable control provides a way to display and change the value of a variable.
7. **Choose ymult from the Value pop-up menu.**
8. **Enter 100 in the Width edit box.**

This setting is back near the top of the scrolling list.
9. **Set the High Limit, Low Limit, and Increment values to 10, 0.1, and 0.1 respectively.**

You may need to scroll down to find these settings.
10. **From the Font Size pop-up menu, choose 12.**

You may need to scroll down to find this pop-up menu.
11. **Click Do It.**

A SetVariable control attached to the variable ymult appears in the upper-left of the control bar.
12. **Double-click the ymult control.**

The SetVariable Control dialog appears.
13. **Click the Duplicate button at the bottom of the dialog.**
14. **Choose xmult as the value.**
15. **Click Do It.**

A second SetVariable control appears in the control bar. This one is attached to the xMult variable.
16. **Choose Graph→Add Control→Add Button.**

The Button Control dialog appears.
17. **Tab to the Title edit box and enter "Update".**
18. **Click the New button adjacent to Procedure.**

The Control Procedure dialog appears in which you can create or edit the procedure to be called when control-related events occur. Such procedures are called "control action procedures".
19. **Make sure the "Prefer structure-based procedures" checkbox is checked.**
20. **Edit the procedure text so it looks like this:**

```
Function ButtonProc (ba) : ButtonControl
    STRUCT WMBUTTONACTION& ba

    switch (ba.eventCode)
        case 2: // Mouse up
            WAVE spiralX
            NVAR xmult
```

```

    spiralX = x*cos(xmult*x)

    WAVE spiralY
    NVAR ymult
    spiralY = x*sin(ymult*x)

    break
endswitch

return 0
End

```

Proofread the function to make sure you entered it as shown above.

21. Click the Save Procedure Now button.

The Control Procedure dialog disappears and the text you edited is inserted into the (currently hidden) procedure window.

22. Click Do It.

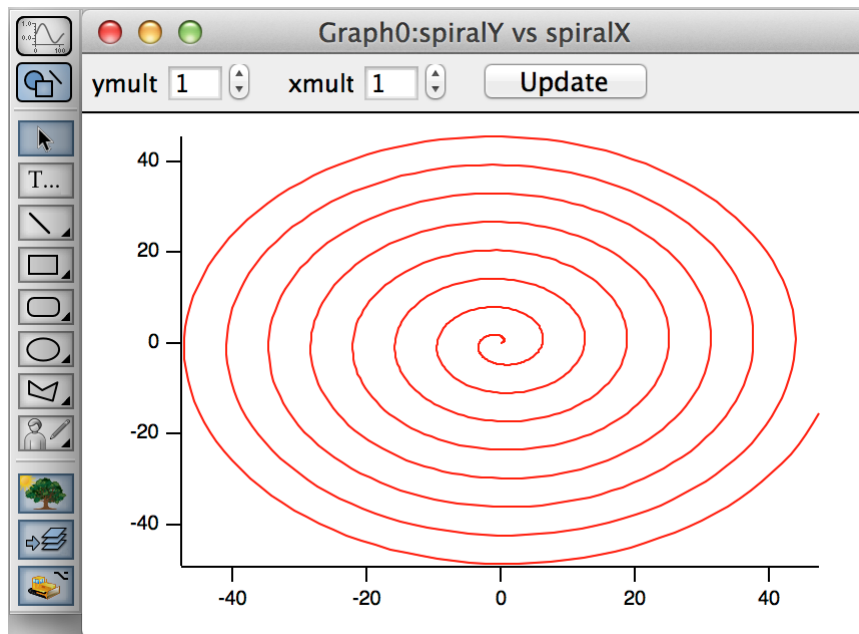
A Button control is added to the control bar.

The three controls are now functional but are not esthetically arranged.

23. Use the Arrow tool to rearrange the three controls into a more pleasing arrangement. Expand the button so it doesn't crowd the text by dragging its handles.

After selecting a control with the arrow tool, you can drag it or use the arrow keys on the keyboard to finetune its position.

The graph now looks like this:



24. Click the top icon in the tool palette to enter "operate mode".

25. Choose Graph→Hide Tools or press Command-T (Macintosh) or Ctrl+T (Windows).

26. Click the up arrow in the ymult control.

The value changes to 1.1.

27. Click the Update button.

The ButtonProc procedure that you created executes. The spiralY and spiralX waves are recalculated according to the expressions you entered in the procedure and the graphs are updated.

28. Experiment with different ymult and xmult settings as desired.

29. Set both values back to 1 and click the Update button.

You can edit a value by typing in the SetVariable control and enter it by pressing Return or Enter.

Creating a Dependency

A dependency is a rule that relates the value of an Igor wave or variable to the values of other waves or variables. By setting up a dependency you can cause Igor to automatically update a wave when another wave or variable changes.

1. **Click the command window to bring it to the front.**

2. **Execute the following commands in the command line:**

```
spiralY := x*sin(ymult*x)
spiralX := x*cos(xmult*x)
```

This is exactly what you entered before except here := is used in place of =. The := operator creates a dependency formula. In the first expression, the wave spiralY is made dependent on the variable ymult. If a new value is stored in ymult then the values in spiralY are automatically recalculated from the expression.

3. **Click Graph0 to bring it to the front.**

4. **Adjust the ymult and xmult controls but do not click the Update button.**

When you change the value of ymult or xmult using the SetVariable control, Igor automatically executes the dependency formula. The spiralY or spiralX waves are recalculated and both graphs are updated.

5. **On the command line, execute this:**

```
ymult := 3*xmult
```

The ymult SetVariable control as well as the graphs are updated.

6. **Adjust the xmult value.**

Again notice that ymult as well as the graphs are updated.

7. **Choose the Misc→Object Status menu item.**

The Object Status dialog appears. You can use this dialog to examine dependencies.

8. **Click the the Current Object pop-up and choose spiralY from the Data Objects list.**

The list on the right indicates that spiralY depends on the variable ymult.

9. **Double-click the ymult entry in the right-hand list.**

ymult becomes the current object. The list on the right now indicates that ymult depends on xmult.

10. **Click the Delete Formula button.**

Now ymult no longer depends on xmult.

11. **Click Done.**

12. **Adjust the xmult setting.**

The ymult value is no longer automatically recalculated but the spiralY and spiralX waves still are.

13. **Click the Update button.**

14. **Adjust the xmult and ymult settings.**

The spiralY and spiralX waves are no longer automatically recalculated. This is because the ButtonProc function called by the Update button does a normal assignment using = rather than := and that action removes the dependency formulae.

In real work, you should avoid using dependencies because they are hard to keep track of and debug. If a button can do the job then use it rather than the dependency.

Saving Your Work - Tour 1E

1. **Choose the File→Save Experiment As menu item.**

2. **Navigate to your “Guided Tours” folder.**

This is the folder that you created under **Saving Your Work - Tour 1A** on page I-23.

3. Type "Tour 1E.pxp" in the name box and click Save.

End of the General Tour

This is the end of the general tour of Igor Pro.

If you want to take a break, you can quit from Igor Pro now.

Guided Tour 2 - Data Analysis

In this tour we will concentrate on the data analysis features of Igor Pro. We will generate synthetic data and then manipulate it using sorting and curve fitting.

Starting Guided Tour 2

0. **If Igor is already running, activate it and choose File→New Experiment.**
In this case, skip to step 2.
1. **Double-click your Igor64 alias or shortcut.**
Instructions for creating this alias or shortcut can be found under **Creating an Igor64 Alias or Shortcut** on page I-14.
On Windows, you can also launch Igor64 using the Start menu.
2. **Choose Misc→Preferences Off.**
Turning preferences off ensures that the tour works the same for everyone.

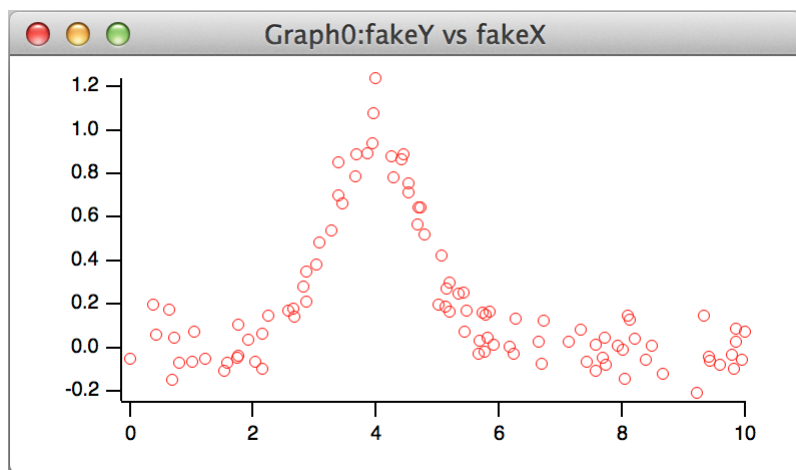
Creating Synthetic Data

We need something to analyze, so we generate some random X values and create some Y data using a math function.

1. **Type the following in the command line and then press Return or Enter:**
`SetRandomSeed 0.1`
This initializes the random number generator so you will get the same results as this guided tour.
2. **Type the following in the command line and then press Return or Enter:**
`Make/N=100 fakeX = enoise(5)+5, fakeY`
This generates two 100 point waves and fills fakeX with evenly distributed random values ranging from 0 to 10.
3. **Execute this in the same way:**
`fakeY = exp(-(fakeX-4)^2)+gnoise(0.1)`
This generates a Gaussian peak centered at 4.
4. **Choose the Windows→New Graph menu item.**
5. **If you see a button labeled Fewer Choices, click it.**
6. **In the Y Waves list, select “fakeY”.**
7. **In the X Wave list, select “fakeX”.**
8. **Click Do It.**
The graph is a rat’s nest of lines because the X values are not sorted.
9. **Double-click the red trace.**
The Modify Trace Appearance dialog appears.
10. **From the Mode pop-up choose Markers.**
11. **From the Markers pop-up menu choose the open circle.**

12. Click Do It.

Now the graph makes sense.



Quick Curve Fit to a Gaussian

Our synthetic data was generated using a Gaussian function so let's try to extract the original parameters by fitting to a Gaussian of the form:

$$y = y_0 + A \cdot \exp\left(-\left(\frac{x-x_0}{\text{width}}\right)^2\right)$$

Here y_0 , A , x_0 and width are the parameters of the fit.

1. **Choose the Analysis→Quick Fit→gauss menu item.**

Igor generated and executed a CurveFit command which you can see if you scroll up a bit in the history area of the command window. The CurveFit command performed the fit, appended a fit result trace to the graph, and reported results in the history area.

At the bottom of the reported results we see the values found for the fit parameters. The amplitude parameter (A) should be 1.0 and the position parameter (x_0) should be 4.0. We got 0.99222 ± 0.0299 for the amplitude and 3.9997 ± 0.023 for the position.

Let's add this information to the graph.

2. **Choose Analysis→Quick Fit→Textbox Preferences.**

The Curve Fit Textbox Preferences dialog appears.

You can add a textbox containing curve fit results to your graph. The Curve Fit Textbox Preference dialog has a checkbox for each component of information that can be included in the textbox.

3. **Click the Display Curve Fit Info Textbox checkbox to check it and then click OK.**

You have specified that you want an info textbox. This will affect future Quick Fit operations.

4. **Choose Analysis→Quick Fit→gauss again.**

This time, Igor displays a textbox with the curve fit results. Once the textbox is made, it is just a textbox and you can double-click it and change it. But if you redo the fit, your changes will be lost unless you rename the textbox.

That textbox is nice, but it's too big. Let's get rid of it.

You could just double-click the textbox and click Delete in the Modify Annotation dialog. The next time you do a Quick Fit you would still get the textbox unless you turn the textbox feature off.

5. **Choose Analysis→Quick Fit→Textbox Preferences again. Click the Display Curve Fit Info Textbox checkbox to uncheck it. Click OK.**

6. **Choose Analysis→Quick Fit→gauss again.**

The textbox is removed from the graph.

More Curve Fitting to a Gaussian

The Quick Fit menu provides easy access to curve fitting using the built-in fit functions, with a limited set of options, to fit data displayed in a graph. You may want more options. For that you must use the Curve Fitting dialog.

1. **Choose the Analysis→Curve Fitting menu item.**
The Curve Fitting dialog appears.
2. **Click the Function and Data tab.**
3. **From the Function pop-up menu, choose gauss.**
4. **From the Y Data pop-up menu, choose fakeY.**
5. **From the X Data pop-up menu, choose fakeX.**
6. **Click the Data Options tab.**
The Weighting and Data Mask pop-up menus should read “_none_”.
7. **Click the Output Options tab.**
The Destination pop-up menu should read “_auto_” and Residual should read “_none_”.
8. **Click Do It.**
During the fit a Curve Fit progress window appears. After a few passes the fit is finished and Igor waits for you to click OK in the progress window.
9. **Click OK.**
The curve fit results are printed in the history. They are the same as in the previous section.

Sorting

In the next section we will do a curve fit to a subrange of the data. For this to work, the data must be sorted by X values.

1. **Double-click one of the open circle markers in the graph.**
The Modify Traces Appearance dialog appears with fakeY selected. If fakeY is not selected, click it.
2. **From the Mode pop-up choose Lines between points and click Do It.**
The fakeY trace reverts to a rat’s nest of lines.
3. **Choose the Analysis→Sort menu item.**
The Sorting dialog appears.
4. **If necessary choose Sort from the Operation pop-up menu.**
5. **Select “fakeX” in the Key Wave list and both “fakeX” and “fakeY” in the Waves to Sort list.**
This will sort both fakeX and fakeY using fakeX as the sort key.
6. **Uncheck any checkboxes in the dialog that are checked, including the Display Output In checkbox.**
7. **Click Do It.**
The rat’s nest is untangled. Since we were using the lines between points mode just to show the results of the sort, we now switch back to open circles.
8. **Press Control and click (Macintosh) or right-click (Windows) on the fakeY trace (the jagged one).**
A pop-up menu appears with the name of the trace at the top. If it is not “Browse fakeY” try again.
9. **Choose Markers from the Mode submenu.**

Fitting to a Subrange

Here we will again fit our data to a Gaussian but using a subset of the data. We will then extrapolate the fit outside of the initial range.

1. **Choose the Graph→Show Info menu item.**

A cursor info panel is appended to the bottom of the graph.

Two cursors are "docked" in the info panel, Cursor A and Cursor B.

2. **Place cursor A (the round one) on the fakeY trace.**

One way to place the cursor is to drag it to the trace. Another way is to control-click (*Macintosh*) or right-click (*Windows*) on the name area which is just to the right of the cursor icon in the cursor info panel.

Note that the cursor A icon in the dock is now black. This indicates that cursor A is selected, meaning that it will move if you use the arrow keys on the keyboard or the slider in the cursor position control.

3. **Move cursor A to point #14.**

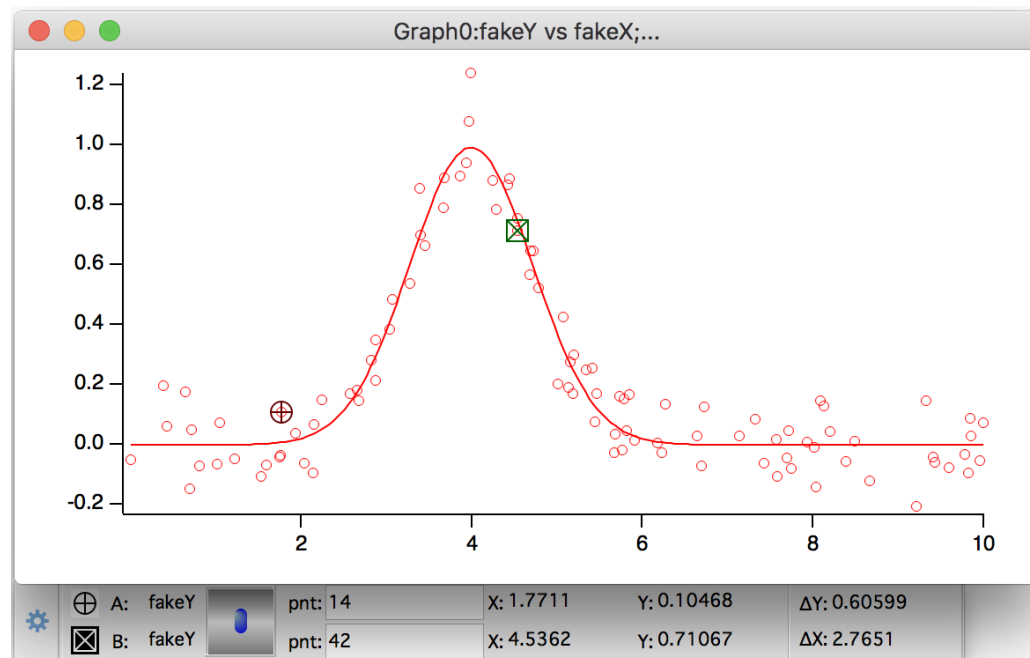
To move the cursor one point at a time, use the arrow keys on the keyboard or click on either side of the slider in the cursor position control.

4. **Click the dock for cursor A in the cursor info panel to deselect it.**

This is so you can adjust cursor B without affecting the position of cursor A.

5. **Place cursor B (the square one) on the fakeY trace and move it to point #42.**

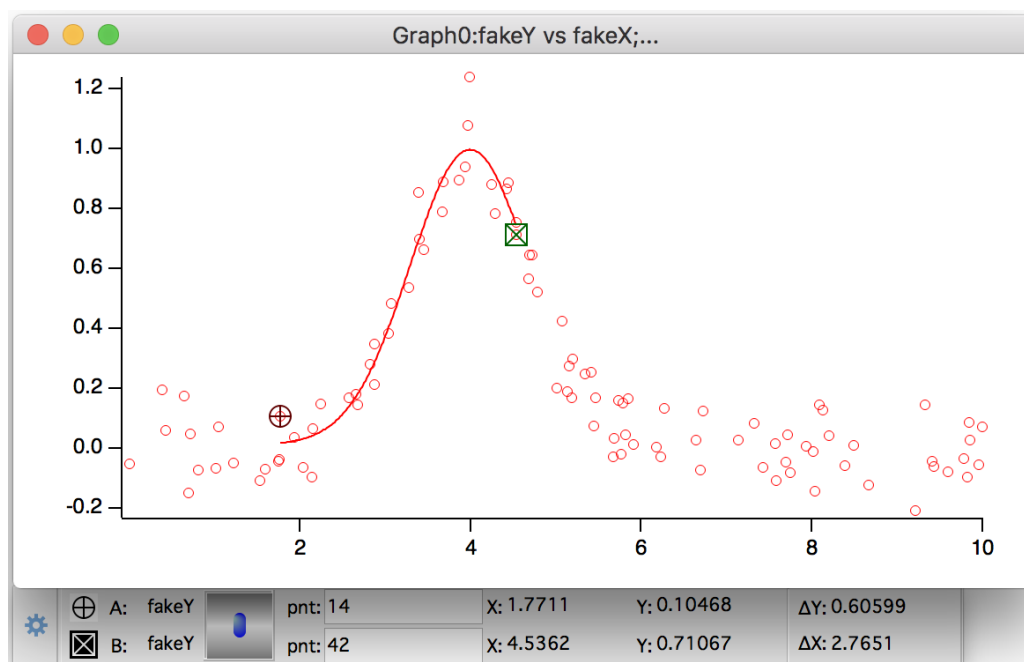
The graph should look like this:



6. **In the Analysis→Quick Fit menu make sure the Fit Between Cursors item is checked. If it is not, select it to check it.**

7. **Choose Analysis→Quick Fit→gauss.**

Note that the fit curve is evaluated only over the subrange identified by the cursors.



We would like the fit trace to extend over the entire X range, while fitting only to the data between the cursors. This is one of the options available only in the Curve Fitting dialog.

8. **Choose Analysis→Curve Fitting and then click the Function and Data tab.**

The curve fitting dialog appears and the settings should be as you left them. Check that the function type is gauss, the X data is fakeY, the X data is fakeX.

9. **Click the Data Options tab.**

10. **Click the Cursors button in the Range area.**

This puts the text “pcsr(A)” and “pcsr(B)” in the range entry boxes.

pcsr is a function that returns the wave point number at the cursor position.

11. **Select the Output Options tab and click the X Range Full Width of Graph checkbox to check it.**

12. **Click Do It.**

The curve fit starts, does a few passes and waits for you to click OK.

13. **Click OK.**

The fit was done using only the data between the cursors, but the fit trace extends over the entire X range.

In the next section, we need the short version of the fit curve, so we will simply do the fit again:

14. **Choose Analysis→Quick Fit→gauss.**

Extrapolating a Fit After the Fit is Done

When you used the Quick Fit menu, and when you chose “_auto_” from the Destination pop-up menu in the Curve Fitting dialog, Igor created a wave named fit_fakeY to show the fit results. This is called the “fit destination wave.” It is just an ordinary wave whose X scaling is set to the extent of the X values used in the fit.

In the preceding sections you learned how to make the curve fit operation extrapolate the fit curve beyond the subrange. Here we show you how to do this manually to illustrate some important wave concepts.

To extrapolate, we simply change the X scaling of fit_fakeY and re-execute the fit destination wave assignment statement which the CurveFit operation put in the history area.

1. Choose the **Data→Change Wave Scaling** menu item.
2. If you see a button labeled **More Options**, click it.
3. From the **SetScale Mode pop-up menu**, choose **Start and End**.
4. Double-click **“fit_fakeY”** in the list.

This reads in the current X scaling values of fit_fakeY. The starting X value will be about 1.77 and the ending X will be about 4.53.

5. Press **Tab** until the **Start box** is selected and enter **1.0**.
6. **Tab** to the **End box** and type **“8.0”**.
7. Click **Do It**

The fit_fakeY trace is stretched out and now runs between 1 and 8.

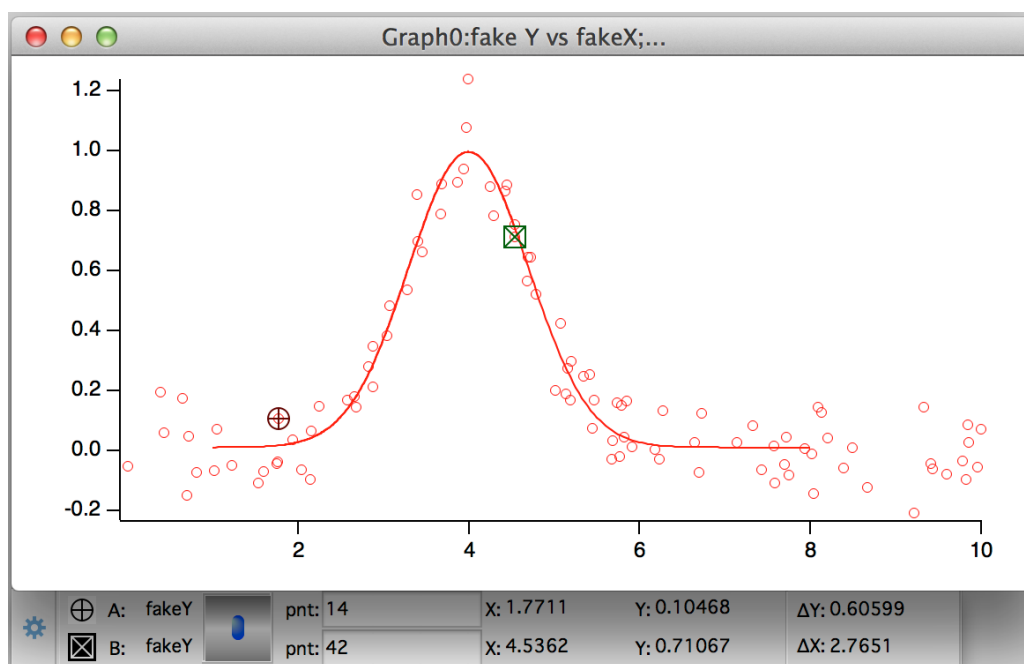
Now we need to calculate new Y values for fit_fakeY using its new X values.

8. In the history, find the line that starts **“fit_fakeY=”** and click it.

The entire line is selected. (The line in question is near the top of the curve fit report printed in the history.)

9. Press **Return** or **Enter** once to copy the selection from the history to the command line and a second time to execute it.

The fit_fakeY wave now contains valid data between 1 and 8.



Appending a Fit

The fit trace added automatically when Igor does a curve fit uses a wave named by adding “fit_” to the start of the Y data wave’s name. If you do another fit to the same Y data, that fit curve will be overwritten. If you want to show the results of several fits to the same data, you will have to somehow protect the fit destination wave from being overwritten. This is done by simply renaming it.

1. Choose the **Data→Rename** menu item.
2. Double-click the wave named **fit_fakeY** to move it into the list on the right.
3. Edit the name in the **New Name box** to change the name to **“gaussFit_fakeY”** and click **Do It**.

Chapter I-2 — Guided Tour of Igor Pro

4. **Position the A and B cursors to point numbers 35 and 61, respectively.**

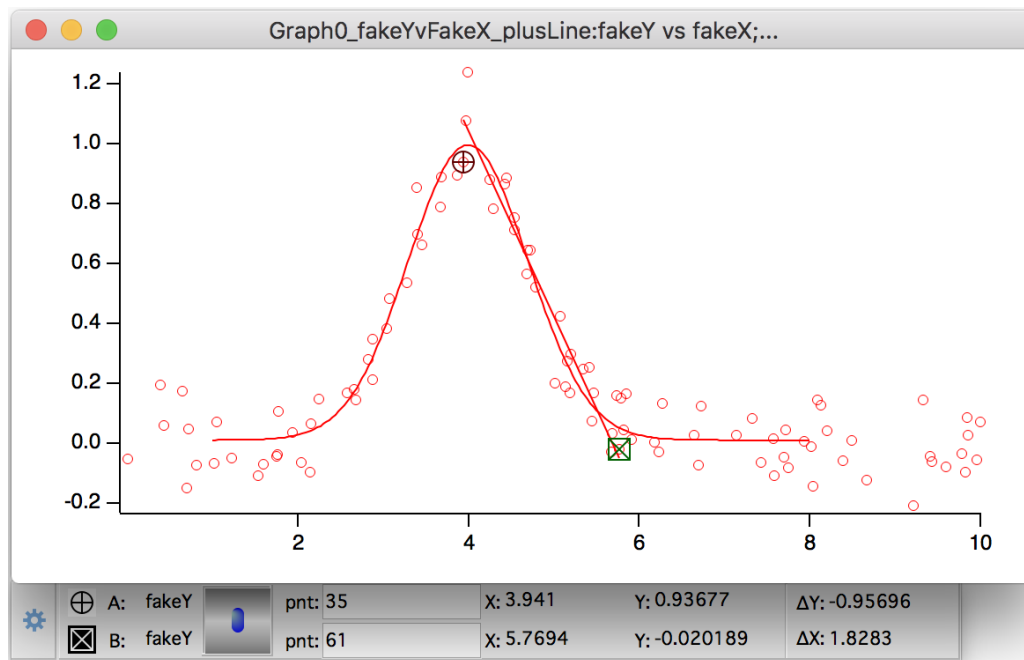
A quick way to do this is to enter the numbers 35 and 61 in the edit boxes to the right of the cursor position control.

5. **Choose Analysis→Quick Fit→line.**

Because there are two traces on the graph, Quick Fit doesn't know which one to fit and puts up the Which Data Set to Quick Fit dialog.

6. **Select fakeY from the list and click OK.**

The line fit is appended to the graph:



This concludes Guided Tour 2.

Guided Tour 3 - Histograms and Curve Fitting

In this tour we will explore the Histogram operation and will perform a curve fit using weighting. The optional last portion creates a residuals plot and shows you how to create a useful procedure from commands in the history.

Starting Guided Tour 3

0. **If Igor is already running, activate it and choose File→New Experiment.**
In this case, skip to step 2.
1. **Double-click your Igor64 alias or shortcut.**
Instructions for creating this alias or shortcut can be found under **Creating an Igor64 Alias or Shortcut** on page I-14.
On Windows, you can also launch Igor64 using the Start menu.
2. **Choose Misc→Preferences Off.**
Turning preferences off ensures that the tour works the same for everyone.

Creating Synthetic Data

We need something to analyze, so let's generate some random values.

1. **Type the following in the command line and then press Return or Enter:**
`SetRandomSeed 0.1`
This initializes the random number generator so you will get the same results as this guided tour.
2. **Type the following in the command line and then press Return or Enter:**
`Make/N=10000 fakeY = enoise(1)`
This generates a 10,000 point wave filled with evenly distributed random values from -1 to 1.

Histogram of White Noise

Here we will generate a histogram of the evenly distributed "white" noise.

1. **Choose the Analysis→Histogram menu item.**
The Histogram dialog appears.
2. **Select fakeY from the Source Wave list.**
3. **Verify that Auto is selected in the Output Wave menu.**
4. **Uncheck any checkboxes in the dialog that are checked, including the Display Output Wave checkbox.**
5. **Click the Auto-set Bin Range radio button.**
6. **Set the Number of Bins box to 100.**
Note in the command box at the bottom of the dialog there are two commands:
`Make/N=100/O fakeY_Hist;DelayUpdate`
`Histogram/B=1 fakeY, fakeY_Hist`
The first command makes a wave to receive the results, the second performs the analysis. The Histogram operation in the "Auto-set bin range" mode takes the number of bins from the output wave.
7. **Click the Do It button.**
The histogram operation is performed.
Now we will display the results.
8. **Choose Windows→New Graph.**

9. Select `fakeY_Hist` in the Y Waves list and “_calculated_” in the X list.
10. Click the Do It button.

A graph is created showing the histogram results. Next we will touch it up a bit.

11. Double-click the trace in the graph.

The Modify Trace Appearance dialog appears.

“Left” is selected in the Axis pop-up menu in the top/left corner of the dialog indicating that changes made in the dialog will affect the left axis.

12. Choose “Sticks to zero” from the Mode pop-up menu and click Do It.

The graph is redrawn using the new display mode.

13. Double-click one of the tick mark labels (e.g., “100”) of the left axis.

The Modify Axis dialog appears, showing the Axis Range tab.

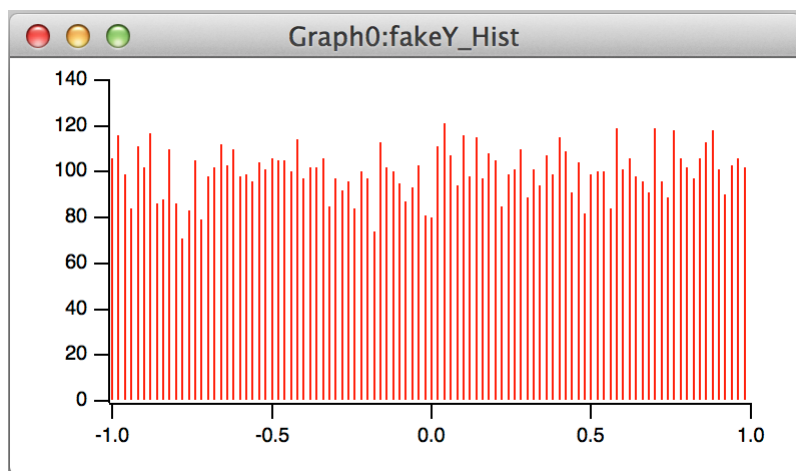
14. From the two pop-up menus in the Autoscale Settings area, choose “Round to nice values” and “Autoscale from zero”.

15. Choose Bottom from the Axis pop-up menu.

16. From the two pop-up menus in the Autoscale Settings area, choose “Round to nice values” and “Symmetric about zero”.

17. Click the Do It button.

The graph should now look like this:



Saving Your Work - Tour 3A

1. Choose the File→Save Experiment As menu item.
2. Navigate to your “Guided Tours” folder.

This is the folder that you created under **Saving Your Work - Tour 1A** on page I-23.

3. Type “Tour 3A.pxp” in the name box and click Save.

Histogram of Gaussian Noise

Now we'll do another histogram, this time with Gaussian noise.

1. Type the following in the command line and then press Return or Enter:

```
fakeY = gnoise(1)
```

2. Choose the Analysis→Histogram menu item.

The dialog should still be correctly set up from the last time.

3. **Click the radio button labeled “Auto-set bins: $3.49 * S_{dev} * N^{-1/3}$ ”.**
The information text at the bottom of the Destination Bins box tells you that the histogram will have 48 bins.
This is a method by Sturges for selecting a “good” number of bins for a histogram. See the **Histogram** operation on page V-297 for a reference.
4. **Click the Bin-Centered X Values checkbox to check it.**
By default, the Histogram operation sets the X scaling of the output wave such that the X values are at the left edge of each bin, and the right edge is given by the next X value. This makes a nice bar plot. In the next section you will do a curve fit to the histogram. For curve fitting you need X values that represent the center of each bin.
5. **Click the Create Square Root(N) Wave checkbox to check it.**
Counting data, such as a histogram, usually has Poisson-distributed values. The estimated mean of the Poisson distribution is simply the number of counts (N) and the estimated standard deviation is the square root of N.
The curve fit will be biased if this is not taken into account. You will use this extra wave for weighting when you do the curve fit.
6. **Click the Do It button.**
Note that the histogram output as shown in Graph0 has a Gaussian shape, as you would expect since the histogram input was noise with a Gaussian distribution.
7. **Choose Data→Data Browser.**
The Data Browser shows you the waves and variables in your experiment. You should see three waves now: fakeY, fakeY_Hist, and W_SqrtN. FakeY_Hist contains the output of the Histogram operation and W_SqrtN is the wave created by the Histogram operation to receive the square root of N data.
8. **Close the Data Browser.**
9. **Click Graph0 and then double-click the trace to invoke the Modify Trace Appearance dialog.**
10. **Select Markers from the Mode menu, then select the open circle marker.**
11. **Click the Error Bars checkbox.**
The Error Bars dialog appears.
12. **Select “+/- wave” from the Y Error Bars menu.**
13. **Pop up the Y+ menu and select W_SqrtN.**
Note that the Y- menu is set to Same as Y+. You could now select another wave from the Y- menu if you needed asymmetric error bars.
14. **Click OK, then Do It.**

Saving Your Work - Tour 3B

1. **Choose the File→Save Experiment As menu item.**
2. **Navigate to your “Guided Tours” folder.**
This is the folder that you created under **Saving Your Work - Tour 1A** on page I-23.
3. **Type “Tour 3B.pxp” in the name box and click Save.**

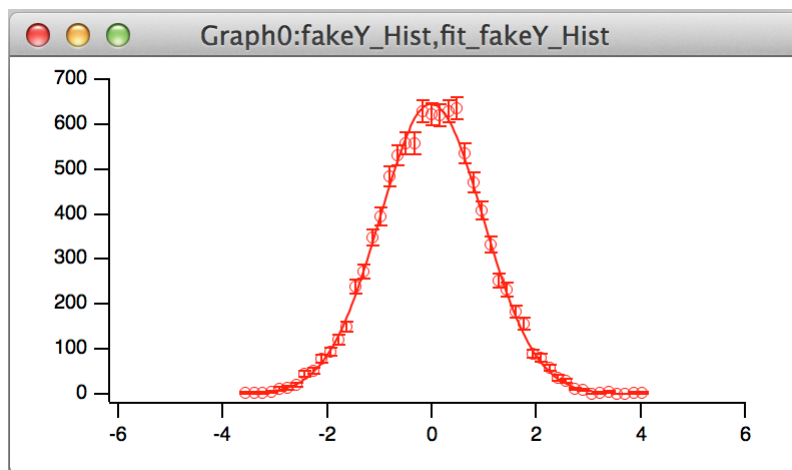
Curve Fit of Histogram Data

The previous section produces all the pieces required to fit a Gaussian to the histogram data, with proper weighting to account for the variance of Poisson-distributed data.

1. **Click the graph to make sure it is the target window.**

2. In the Analysis→Quick Fit menu make sure the Weight from Error Bar Wave item is checked. If it is not, select it to check it.
3. Choose Analysis→Quick Fit→gauss.

The graph now looks like this:



As shown in the history area, the fit results are:

```
Coefficient values ± one standard deviation
y0  =-0.35284 ± 0.513
A    =644.85 ± 7.99
x0   =-0.0014111 ± 0.00997
width=1.406 ± 0.0118
```

The original data was made with a standard deviation of 1. Why is the width 1.406? The way Igor defines its gauss fit function, width is $\sigma \cdot 2^{1/2}$.

4. Execute this command in the command line:

```
Print 1.406/sqrt(2)
```

The result, 0.994192, is pretty close to 1.0.
It is often useful to plot the residuals from a fit to check for various kinds of problems. For that you need to use the Curve Fitting dialog.
5. Choose Analysis→Curve Fitting.
6. Click the Function and Data tab and choose gauss from the Function menu.
7. Choose fakeY_Hist (not fakeY) from the Y Data menu.
8. Leave the X Data pop-up menu set to “_calculated_”.
9. Click the Data Options tab. If there is text in the Start or End Range boxes, click the Clear button in the Range section.
10. Choose W_SqrtN from the Weighting pop-up menu.
11. Just under the Weighting pop-up menu there are two radio buttons. Click the top one which is labeled “Standard Dev”.
12. Click the Output Options tab and choose “_auto_” from the Destination pop-up menu.
13. Set the Residual pop-up menu to “_auto trace_”.

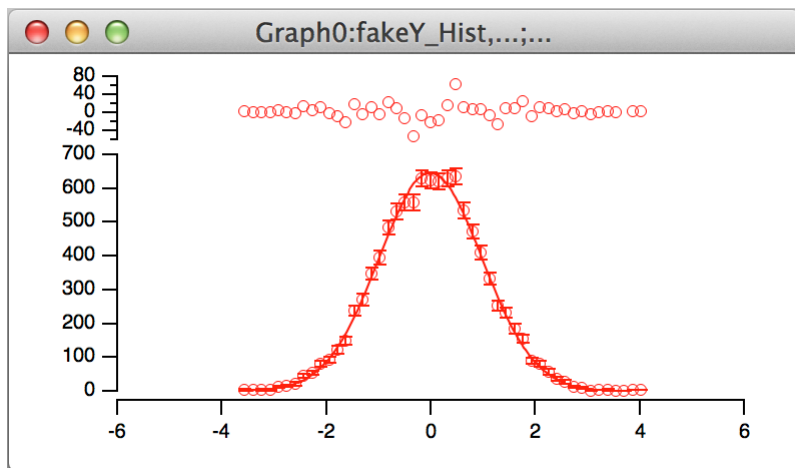
Residuals will be calculated automatically and added to the curve fit in our graph.

14. Uncheck the X Range Full Width of Graph checkbox.

This is appropriate only when we are fitting to a subset of the source wave.

15. Click Do It.

The curve fit starts, does a few passes, and waits for you to click OK.



There is one small issue not addressed above. One of the bins contains zero; the square root of zero is, of course, zero. So the weighting wave contains a zero, which causes the curve fit to ignore that data point. It's not clear what is the best approach for fixing that problem. Some replace the zero with a one. These commands replace any zeroes in the weighting wave and re-do the fit:

```
W_SqrtN = W_SqrtN[p] == 0 ? 1 : W_SqrtN[p]
CurveFit/NTHR=0 gauss fakeY_Hist /W=W_SqrtN /I=1 /D /R
```

This doesn't change the result very much, since there was just one zero in the histogram:

```
Coefficient values ± one standard deviation
y0      =-0.40357 ± 0.464
A       =644.76 ± 7.98
x0      =-0.0014186 ± 0.00996
width   =1.4065 ± 0.0115
```

Saving Your Work - Tour 3C

1. Choose the File→Save Experiment As menu item.
2. Navigate to your "Guided Tours" folder.

This is the folder that you created under **Saving Your Work - Tour 1A** on page I-23.

3. Type "Tour 3C.pxp" in the name box and click Save.

Curve Fit Residuals (Optional)

The remaining guided tour sections are primarily of interest to people who want to use Igor programming to automate tasks. You may want to skip them and come back later. If so, jump ahead to **For Further Exploration** on page I-66.

In the next section, as an illustration of how the history area can be used as a source of commands to generate procedures, we will create a procedure that appends residuals to a graph. The preceding section illustrated that Igor is able to automatically display residuals from a curve fit, so the procedure that we write in the next section is not needed. Still, it demonstrates the process of creating a procedure. In preparation for writing the procedure, in this section we append the residuals manually.

If the curve fit to a Gaussian function went well and if the gnoise function truly produces noise with a Gaussian distribution, then a plot of the difference between the histogram data and the fitted function should not reveal any curvature.

0. To remove the automatically generated residual from the Gaussian fit in the previous section, Control-click (*Macintosh*) or right-click (*Windows*) directly on the residual trace at the top of the graph and select Remove Res_fakeY_Hist from the pop-up menu.

1. **Choose the Data→Duplicate Waves menu item.**
2. **Choose fakeY_Hist from the Template pop-up menu.**
3. **In the first Names box, enter “histResids”.**
4. **Click Do It.**

You now have a wave suitable for containing residuals.

5. **In the history area of the command window, find the line that reads:**

```
fit_fakeY_Hist= W_coef[0]+W_coef[1]*exp(-(x-W_coef[2])/W_coef[3])^2)
```

W_coef is a wave created by the CurveFit operation to contain the fit parameters. W_coef[0] is the y0 parameter, W_coef[1] is the A parameter, W_coef[2] is the x0 parameter and W_coef[3] is the width parameter.

This line shows conceptually what the CurveFit operation did to set the data values of the fit destination wave.

6. **Click once on the line to select it and then press Return or Enter once.**

The line is transferred to the command line.

7. **Edit the line to match the following:**

```
histResids = fakeY_Hist - (W_coef[0]+W_coef[1]*exp(-(x-W_coef[2])/W_coef[3])^2)
```

In other words, change fit_fakeY_Hist to histResids, click after the equals and type fakeY_Hist - (and then add a) to the end of the line.

The expression inside the parentheses that you added represents the model value using the parameters determined by the fit. This command computes residuals by subtracting the model values from the data values on which the fit was performed.

If the fit had used an X wave rather than calculated X values then it would have been necessary to substitute the name of the X wave for the “x” in the expression.

8. **Press Return or Enter.**

This wave assignment statement calculates the difference between the measured data (the output of the Histogram operation) and the theoretical Gaussian (as determined by the CurveFit operation).

Now we will append the residuals to the graph stacked above the current contents.

9. **Choose Graph→Append Traces to Graph.**
10. **Select histResids from the Y waves list and “_calculated_” from the X wave list.**
11. **Choose New from the Axis pop-up menu under the Y Waves list.**
12. **Enter “Lresid” in the Name box and click OK.**
13. **Click Do It.**

The new trace and axis is added.

Now we need to arrange the axes. We will do this by partitioning the available space between the Left and Lresid axes.

14. **Double-click the far-left axis.**

The Modify Axis dialog appears. If any other dialog appears, cancel and try again making sure the cursor is over the axis.

If you have enough screen space you will be able to see the graph change as you change settings in the dialog. Make sure that the Live Update checkbox in the top/right corner of the dialog is selected.

15. **Click the Axis tab.**

The Left axis should already be selected in the pop-up menu in the top-left corner of the dialog.

16. **Set the Left axis to draw between 0 and 70% of normal.**
17. **Choose Lresid from the Axis pop-up menu.**
18. **Set the Lresid axis to draw between 80 and 100% of normal.**

19. Choose Fraction of Plot Area in the “Free axis position” menu.

The Lresid axis is a “free” axis. This moves it horizontally so it is in line with the Left axis.

20. Choose Bottom from the Axis pop-up menu.

21. Click the Axis Standoff checkbox to turn standoff off.

Just a couple more touch-ups and we will be done. The ticking of the Lresid axis can be improved. The residual data should be in dots mode.

22. Choose Lresid from the Axis pop-up menu again.

23. Click the Auto/Man Ticks tab.

24. Change the Approximately value to 2.

25. Click the Axis Range tab.

26. In the Autoscale Settings area, choose “Symmetric about zero” from the menu currently reading “Zero isn’t special”.

27. Click the Do It button.

28. Double-click the histResids trace.

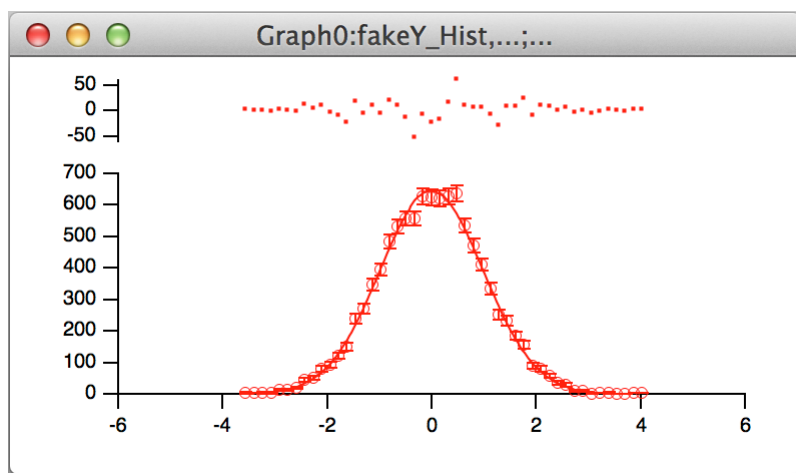
The Modify Trace Appearance dialog appears with histResids already selected in the list.

29. Choose Dots from the Mode pop-up menu

30. Set the line size to 2.00.

31. Click Do It.

The graph should now look like this:



Saving Your Work - Tour 3D

1. Choose the File→Save Experiment As menu item.

2. Navigate to your “Guided Tours” folder.

This is the folder that you created under **Saving Your Work - Tour 1A** on page I-23.

3. Type “Tour 3D.pxp” in the name box and click Save.

Writing a Procedure (Optional)

In this section we will collect commands that were created as we appended the residuals to the graph. We will now use them to create a procedure that will append a plot of residuals to a graph.

1. Make the command window tall enough to see the last 10 lines by dragging the top edge of the window upward.

2. Find the fifth line from the bottom that reads:
 - AppendToGraph/L=Lresid histResids
3. Select this line and all the lines below it and copy them to the clipboard.
4. Choose the Windows→New→Procedure menu item.
5. Type “Append Residuals” (without the quotes) in the Document Name box and click OK.
A new procedure window appears. We could have used the always-present built-in procedure window, but we will save this new procedure window as a standalone file so it can be used in other experiments.
6. Add a blank line to the window, type “Function AppendResiduals()”, and press Return or Enter.
7. Paste the commands from the clipboard into the new window.
8. Type “End” and press Return or Enter.
9. Select the five lines that you pasted into the procedure window and then choose Edit→Adjust Indentation.

This removes the bullet characters copied from the history and prepends tabs to apply the normal indentation for procedures.

Your procedure should now look like this:

```
Function AppendResiduals()
  AppendToGraph/L=Lresid histResids
  SetAxis/A/E=2 Lresid
  ModifyGraph nticks(Lresid)=2,standoff(bottom)=0,axisEnab(left)={0,0.7};DelayUpdate
  ModifyGraph axisEnab(Lresid)={0.8,1}, freePos(Lresid)=0;DelayUpdate
  ModifyGraph mode(histResids)=2,lsize(histResids)=2
End
```

10. Delete the “;DelayUpdate” at the end of the two ModifyGraph commands.

DelayUpdate has no effect in a function.

We now have a nearly functional procedure but with a major limitation — it only works if the residuals wave is named “histResids”. In the following steps, we will change the function so that it can be used with any wave and also with an XY pair, rather than just with equally-spaced waveform data.

11. Replace the first two lines of the function with the following:

```
Function AppendResiduals(yWave, xWave)
  Wave yWave
  Wave xWave // X wave or null if there is no X wave

  if (WaveExists(xWave))
    AppendToGraph/L=Lresid yWave vs xWave
  else
    AppendToGraph/L=Lresid yWave
  endif
  String traceName = NameOfWave(yWave)
```

12. In the last ModifyGraph command in the function, change both “histResids” to “\$traceName” (without the quotes).

The “\$” operator converts the string expression that follows it into the name of an Igor object.

Here is the completed procedure.

```
Function AppendResiduals(yWave,xWave)
  Wave yWave
  Wave/Z xWave // X wave or null if there is no X wave

  if (WaveExists(xWave))
    AppendToGraph/L=Lresid yWave vs xWave
  else
    AppendToGraph/L=Lresid yWave
  endif
  String traceName = NameOfWave(yWave)
  SetAxis/A/E=2 Lresid
  ModifyGraph nticks(Lresid)=2,standoff(bottom)=0,axisEnab(left)={0,0.7}
```

```

ModifyGraph axisEnab(Lresid)={0.8,1},freePos(Lresid)={0,kwFraction}
ModifyGraph mode($traceName)=2,lsize($traceName)=2
End

```

Let's try it out.

13. **Click the Compile button at the bottom of the procedure window to compile the function.**

If you get an error, edit the function text to match the listing above.

14. **Click the close button in the Append Residuals procedure window. A dialog asks if you want to kill or hide the window. Click Hide.**

If you press Shift while clicking the close button, the window will be hidden without a dialog.

15. **Choose Windows→New Graph and click the Fewer Choices button if it is available.**
 16. **Choose fakeY_Hist from the Y Waves list and _calculated_ from the X Wave list and click Do It.**
- A graph without residuals is created.

17. **In the command line, execute the following command:**

```
AppendResiduals(histResids, "$")
```

Because we have no X wave, we use "\$" to pass NULL to AppendResiduals for the xWave parameter.

The AppendResiduals function runs and displays the residuals in the graph, above the original histogram data.

Next we will add a function that displays a dialog so we don't have to type wave names into the command line.

18. **Choose Windows→Procedure Windows→Append Residuals to show the Append Residuals procedure window.**

19. **Enter the following function below the AppendResiduals function.**

```

Function AppendResidualsDialog()
String yWaveNameStr, xWaveNameStr

Prompt yWaveNameStr,"Residuals Data",popup WaveList("",";","")
Prompt xWaveNameStr,"X Data", popup "_calculated_"+WaveList("",";","")
DoPrompt "Append Residuals", yWaveNameStr, xWaveNameStr
if (V_flag != 0)
    return -1 // User canceled
endif

AppendResiduals($yWaveNameStr, $xWaveNameStr)

return 0 // Success
End

```

This function displays a dialog to get parameters from the user and then calls the AppendResiduals function.

Let's try it out.

20. **Click the Compile button at the bottom of the procedure window to compile procedures.**

If you get an error, edit the function text to match the listing above.

21. **Shift-click the close button to hide the procedure window. Then activate Graph1.**
22. **Control-click (Macintosh) or right-click (Windows) the residual trace at the top of the graph and select Remove histResids from the pop-up menu.**

The Lresid axis is removed because the last wave graphed against it was removed.

23. **On the command line, execute this command:**

```
AppendResidualsDialog()
```

The AppendResidualsDialog function displays a dialog to let you choose parameters.

24. **Choose histResids from the Residuals Data pop-up menu.**

25. **Leave the X Wave pop-up set to “_calculated_”.**
26. **Click Continue.**

The graph should once again contain the residuals plotted on an axis above the main data.
Next we will add a menu item to the Macros menu.
27. **Choose Windows→Procedure Windows→Append Residuals to show the Append Residuals procedure window.**
28. **Enter the following code before the AppendResiduals function:**

```
Menu "Macros"  
    "Append Residuals...", AppendResidualsDialog()  
End
```
29. **Click the Compile button.**

Igor compiles procedures and adds the menu item to the Macros menu.
30. **Shift-click the close button to hide the procedure window. Then activate Graph1.**
31. **Control-click (Macintosh) or right-click (Windows) the residual trace at the top of the graph and select “Remove histResids” from the pop-up menu.**
32. **Click the Macros menu and choose the “Append Residuals” item**

The procedure displays a dialog to let you choose parameters.
33. **Choose histResids from the Residuals Data pop-up menu.**
34. **Leave the X Wave pop-up menu set to “_calculated_”.**
35. **Click the Continue button.**

The graph should once again contain the residuals plotted on a new axis above the main data.

Saving a Procedure File (Optional)

Note: *If you are using the demo version of Igor Pro beyond the 30-day trial period, you cannot save a procedure file.*

Now that we have a working procedure, let's save it so it can be used in the future. We will save the file in the "Igor Pro User Files" folder - a folder created by Igor for you to store your Igor files.

More precisely, we will save the procedure file in the "User Procedures" subfolder of the Igor Pro User Files folder. You could save the file anywhere on your hard disk, but saving in the User Procedures subfolder makes it easier to access the file as we will see in the next section.

1. **Choose Help→Show Igor Pro User Files.**

Igor opens the "Igor Pro User Files" folder on the desktop.

By default, this folder has the Igor Pro major version number in its name, for example, "Igor Pro 7 User Files", but it is generically called the "Igor Pro User Files" folder.

Note where in the file system hierarchy this folder is located as you will need to know this in a subsequent step. The default locations are:

Macintosh:

```
/Users/<user>/Documents/WaveMetrics/Igor Pro 7 User Files
```

Windows:

```
C:\Users\<user>\Documents\WaveMetrics\Igor Pro 7 User Files
```

Note the "User Procedures" subfolder of the Igor Pro User Files folder. This is where we will save the procedure file.

2. **Back in Igor, choose Windows→Procedure Windows→Append Residuals to show the Append Residuals procedure window.**
3. **Choose File→Save Procedure As.**
4. **Enter the file name “Append Residuals.ipf”.**

5. **Navigate to your User Procedures folder inside your Igor Pro User Files folder and click Save.**
The Append Residuals procedure file is now saved in a standalone file.
6. **Click the close button on the Append Residuals procedure window.**
Igor asks if you want to kill or hide the file.
7. **Click the Kill button.**
This removes the file from the current experiment, but it still exists on disk and you can open it as needed.

There are several ways to open the procedure file to use it in the future. One is to double-click it. Another is to choose the File→Open File→Procedure menu item. A third is to put a #include statement in the built-in procedure window, which is how we will open it in the next section.

Including a Procedure File (Optional)

The preferred way to open a procedure window that you intend to use from several different experiments is to use a #include statement. This section demonstrates how to do that.

Note: *If you are using the demo version of Igor Pro beyond the 30-day trial period, you did not create the Append Residuals.ipf file in the preceding section so you can't do this section.*

1. **In Igor, use the Windows→Procedure Windows→Procedure Window to open the built-in procedure window.**
2. **Near the top of the built-in procedure window, notice the line that says:**

```
#pragma rtGlobals = 3
```


This is technical stuff that you can ignore.
3. **Under the rtGlobals line, leave a blank line and then enter:**

```
#include "Append Residuals"
```
4. **Click the Compile button at the bottom of the built-in procedure window.**
Igor compiles the procedure window. When it sees the #include statement, it looks for the "Append Residuals.ipf" procedure file in the User Procedures folder and opens it. You don't see it because it was opened hidden.
5. **Use the Windows→Procedure Windows menu to verify that the Append Residuals procedure file is in fact open.**

To remove the procedure file from the experiment, you would remove the #include statement from the built-in procedure window.

#include is powerful because it allows procedure files to include other procedure files in a chain. Each procedure file automatically opens any other procedure files it needs.

User Procedures is special because Igor searches it to satisfy #include statements.

Another special folder is Igor Procedures. Any procedure file in Igor Procedures is automatically opened by Igor at launch time and left open till Igor quits. This is the place to put procedure files that you want to be open all of the time. It is the easiest way to make a procedure file available and is recommended for frequently-used files.

This concludes Guided Tour 3.

For Further Exploration

We developed the guided tours in this chapter to provide an overview of the basics of using Igor Pro and to give you some experience using features that you will likely need for your day-to-day work. Beyond these fundamentals, Igor includes a wide variety of features to facilitate much more advanced graphing and analysis of your data.

As you become more familiar with using Igor, you will want to further explore some of the additional learning and informational aids that we have included with Igor Pro.

- The Igor Pro manual is installed on your hard disk in PDF format. You can access it through the Igor Help Browser or open it directly from the Manual folder in the Igor Pro 7 Folder.

The material in the manual is the same as the material in the online help files but is organized in book format and is therefore better suited for linear reading. Unlike the help files, the PDF manual includes an index.

The most important chapters at this point in your Igor learning curve are Chapter II-3, **Experiments, Files and Folders** and Chapter II-5, **Waves**. If you want to learn Igor programming, read Chapter IV-1, **Working with Commands**, Chapter IV-2, **Programming Overview**, and Chapter IV-3, **User-Defined Functions**.

- The Igor Help Browser provides online help, including reference material for all built-in operations, functions, and keywords, an extensive list of shortcuts, and the ability to search Igor help files, procedure files, examples and technical notes for key phrases. See **The Igor Help Browser** on page II-2 for more information.
- The Examples folder contains a wide variety of sample experiments illustrating many of Igor's advanced graphing and programming facilities. You can access these most easily through the File→Example Experiments submenus.
- The Learning Aids folder contains additional guided tours and tutorials including a tutorial on image processing. You can access these through the File→Example Experiments→Tutorials submenu.
- A tutorial on 3D visualization can be found in the "3D Graphics" help window, accessible through the Help→Help Windows menu.
- The WaveMetrics Procedures folder contains a number of utility procedures that you may find useful for writing your own procedures and for your more advanced graphing requirements. For an overview of the WaveMetrics procedures and easy loading of the procedure files, choose Help→Help Windows→WM Procedures Index.
- The "More Extensions (64-bit)" folder contains a number of External Operations (XOPs), which add functionality not built into the Igor Pro application. Read the included help files to find out more about the individual XOPs and how to install them, or consult the External Operations Index in the XOP Index help file, which has brief description of each XOP.
- The Technical Notes folder contains additional information. Tech Note #000 contains an index to all of the other notes.
- The Igor Pro mailing list is an Internet mailing list where Igor users share ideas and help each other. See **Igor Mailing List** on page II-4 or select the Help→Support menu and then select Igor Mailing List from the Support Options list for information about the mailing list.
- **IgorExchange** is a user-to-user support web page and a repository for user-created Igor Pro projects. Choose Help→IgorExchange to visit it.