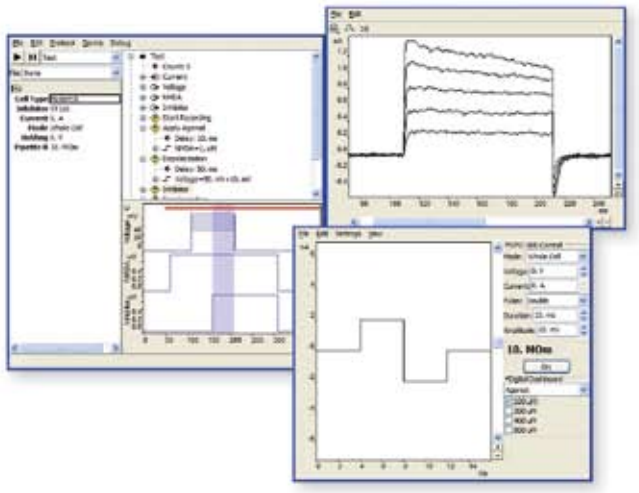
Data Acquisition

Easy to Use and Extremly Flexible Protocols & Instrument Configuration

- Unlimited number of Analog and Digital Inputs & Outputs
- Can be configured to work with any number of custom Instruments
- Custom Stimulus protocols and Data Analysis
- Perfusion, Liquid Handling and Temperature Control
- USB, Parallel and Serial Port Controls
- Ships installed and configured for your experimental protocols
- Installation includes on-site training and consulting

Features	Standard configurations:
ADC	12/16 bit, 8/16 channels
	up to 500 kSamples/sec
	range from -10 to 10V (other options are available)
DAC	12/16 bit, 2/4/8/16 channels
Digital I/O	8-32 digital I/O channels
RS232/USB	RS232, parallel and USB port control



Bioscience Tools, 4527 52nd Street San Diego CA 92115
Ph: 877-853-9755, Fax: 619-582-0919
www.biosciencetools.com

Contents

Introduction

This data acquisition system can simultaneously read data from multiple sources and control multiple output signals in a synchronous manner. The data acquisition process is defined by a protocol, where you can set a sequence of events, such as start and stop of data recording and output of analog signals. A single protocol can simultaneously control different devices, such as analog interfaces and amplifiers. The protocol can also specify the analysis of recorded data, which is performed during data acquisition. The system saves all its recorded data, analysis results, and protocol definitions in a single data file. You can also define custom data that will be associated with recordings to better describe the recorded data. The program also provides convenient browsing through the data files, by showing a preview of traces, plots and graphs.

The system can be configured to provide both a generic user interface to control any digital or analog signal, as well as to control common types of instruments, such as known brands of electrophysiological amplifiers. These settings can be easily shared between multiple users. Settings for several common instruments are included with the system.

Demo Download Insturctions:

You can download a demo version of this application along with sample protocols and recordings. First, download the “demo.zip” file using URL: *http://biosciencetools.com/catalog/links/demo.zip*. Unzip the *demo.zip* file. This will create a folder “Bioscience Tools”. Open the folder and double-click on file “runDEMO”. Load simulation settings: click on menu “Device” and select “Test ADI Device1”; from the bottom of the pop-up window click “Load...”. Select file “Settings” in subfolder “Bioscience Tools/Settings”. Click “OK” to close the pop-up window. Then, click on menu “Protocol” and select “Load Protocols...”. Choose subfolder “Bioscience Tools/Settings” and select file “Sample-Protocols”. The “Bioscience Tools” folder also contains this User Guide as a PDF file.

Quick Tour

How to Start

The data acquisition system is shipped installed on a computer. It can also be preconfigured and pre-programmed according to your experiment requirements and instruments used. Some experience with Windows, or similar, operating systems is required to run this application. After powering the computer, you can start the application by double-clicking the "Bioscience Tools" icon located on the Desktop.

Creating the Protocol - Protocol Editor

When the application starts for the first time it presents a window, which is mostly empty (Fig. lA). This window is the control center where the acquisition protocols are created and activated: ***Protocol Editor***. To create your first protocol, select "Protocols ->New protocol" from the menu located on the top of the Protocol Editor. A new item, named "Untitled Protocol", will appear in the top right panel of the window. The protocols will be shown in this panel as a tree structure.

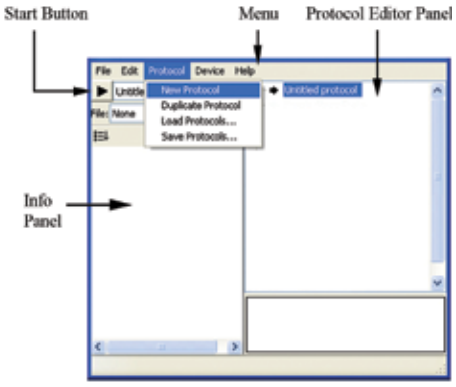


Fig. 1A

Let's first change the name of your new protocol. Click on the Untitled protocol to highlight its name and then click the second time to edit the name (similar to the way you would rename a file in a file browser). Alternatively, press the "SPACE" bar to start editing the label of any selected item. Type in the new name "Test" and press "ENTER" on your keyboard. Note that the name of the currently selected protocol is also selected in the pop-up list in the top-left corner of the window, Fig. 1B. This list provides quick access to your protocols. Try adding more protocols to see how it works.

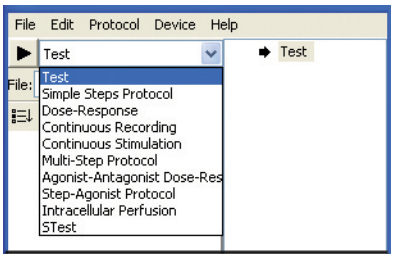


Fig. 1B

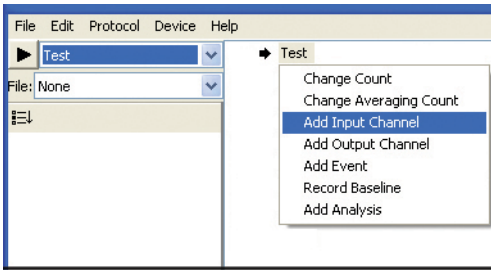


Fig. 1C

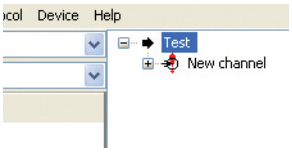


Fig. 1D

Adding Input Channels

Now let's add some content to the protocol. First we need to add a channel that will provide input data. Right-click on the protocol name and select "Add Input Channel" from the pop-up menu, Fig. 1C. A new item named "New channel" will be added to the protocol, Fig. 1D. Initially, it is marked with a red exclamation point indicating that the program is not able to associate this channel with any device. We will fix this later.

Adding Events

To do anything with this channel the protocol has to specify the time to perform an event and the type of action. These, like everything else, are specified by adding corresponding items to the protocol. Right-click on the protocol name and select “Add Event”, Fig. 2A. The event specifies the time during protocol execution. The first event, which we just created, can mark the beginning of the recording, for example. We can change the event name to reflect this fact. Click on the event to select it, press the “Space” bar and type the new name “Start”. Right-click on the event and select “Add Action On New channel”, Fig. 2B. Since “New channel” is an input channel, the created action is “Start recording”. This concludes the creation of the simplest executable protocol. At this point your protocol editor panel should look similar to Fig. 2C.

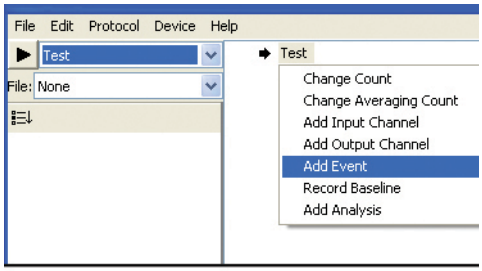


Fig. 2A

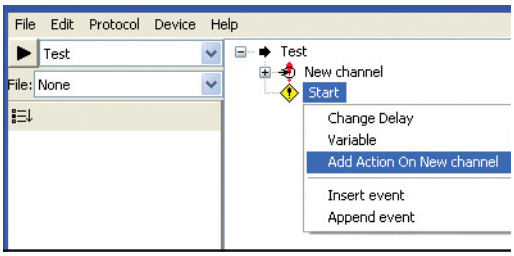


Fig. 2B

Fig. 2C

Start Recording

Recording can be started by clicking the start button at the top-left of the Protocol Editor. If you click this button now, you will get an error message stating that protocol channel “New Channel” does not match any device. To make a match the protocol channel has to have the same name as a channel of one of the open devices, which will be described in the following sections.

Input/Outpot interface -Device Window

Let’s open a device now. Click on the “Device” menu. You will see a list of devices found in your system, Fig. 3A. In the rest of the tutorial we will use “Test ADI device1” - ADI stands for analog/digital interface. This is a virtual device that simulates the behavior of a real analog data interface with input channels connected to output channels. This device can be used to create your demo protocols. If you have a real AD

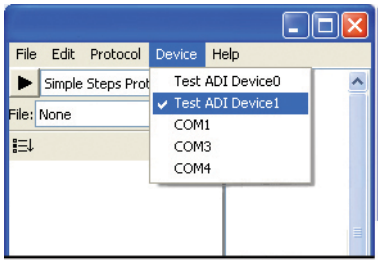


Fig. 3A

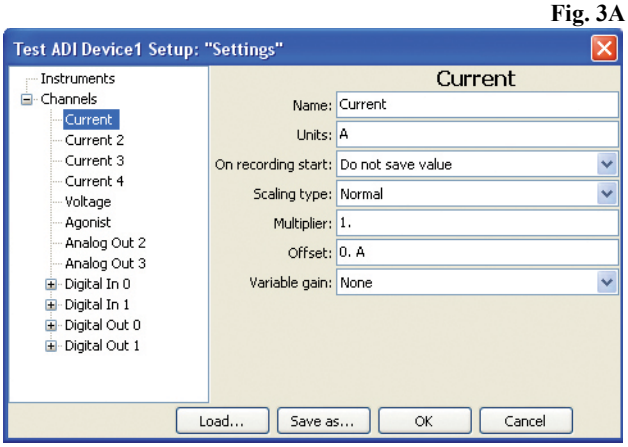


Fig. 3B

interface installed in your system, you can use it instead. To open a device, simply select it from the menu.

Input/Output Channels - Settings Window

When you open the device for the first time the program presents a set-up dialog, where you can describe different interfaces and instruments used, Fig. 3B. We will use the default settings for now, so click "OK" to close the window. The program will then ask you to specify a new file where the settings should be saved. You can have multiple settings files for different experiments, you can also share settings files with other users who already configured systems similar to yours. Your system might also come preconfigured. Give some descriptive name to this settings file, such as "Test setup" and save it in the recommended folder. The next time you start the program and open the device again, the settings will be loaded automatically from the file that you used the last time. Furthermore, all the changes you do to the settings will be saved in this file automatically, without going through the SAVE dialog again.

Once settings are saved an empty DEVICE Window will open, Fig. 3C. For our ADI device this window will work like an oscilloscope or monitor. Initially the window is empty. Now we will associate our protocol channel with the input channel of the opened device. This association is done by giving the protocol channel the same name as one of the inputs used in Settings. You do not need to memorize the names and type them in the protocol: right-click on the protocol channel and choose "Select" from the menu, Fig. 3D. The dialog lists all input channels available, Fig. 3E. Simply select a device channel and click "OK" to rename the protocol channel automatically. Note that the channel in our protocol is not marked with a red exclamation point any more. The protocol can now be executed. Click the "Start" button in the Protocol Editor and observe the input data appearing in the Device Window. Our simple protocol does not specify stop action so data acquisition will continue until you stop it manually. In order to stop the protocol, click on the same button you used to start the protocol. This button toggles between "Start" and "Stop", as indicated by the change of the button image.

Device Window

Before we proceed to more advanced protocols let's first explore properties of the device. The "Tools" menu of this window lists the controls that are available for the current device. The "Channel monitor" tool is always available for any ADI device. Depending on device capabilities, other tools may be also available. Some instruments can add extra tools to the menu.

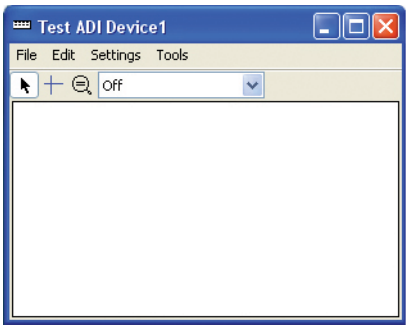


Fig. 3C

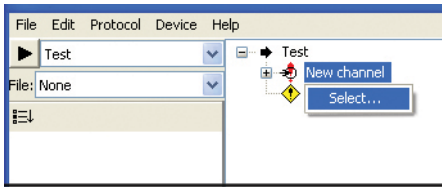


Fig. 3D

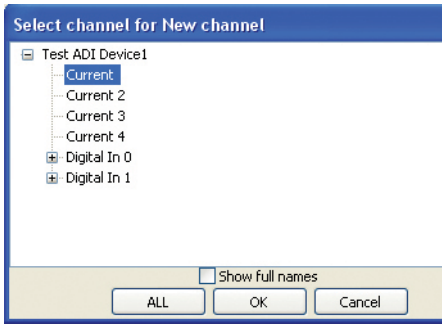


Fig. 3E

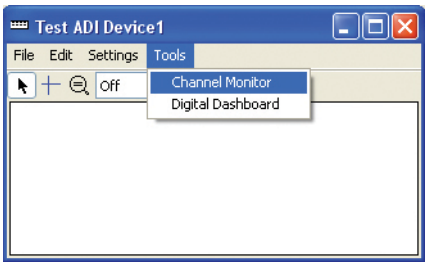


Fig. 4A

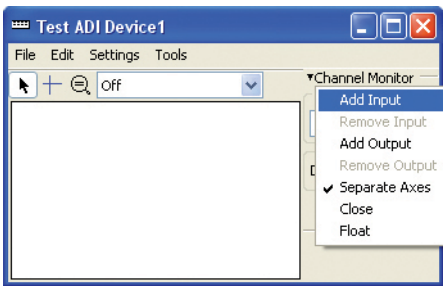


Fig. 4B

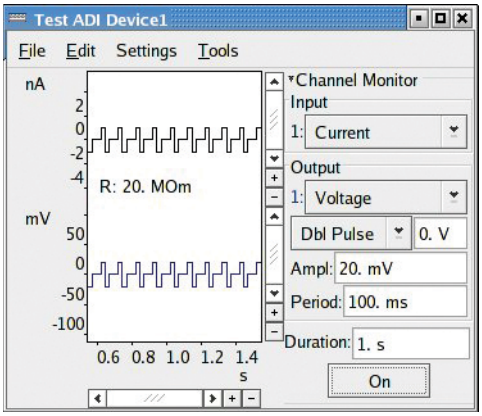


Fig. 4C

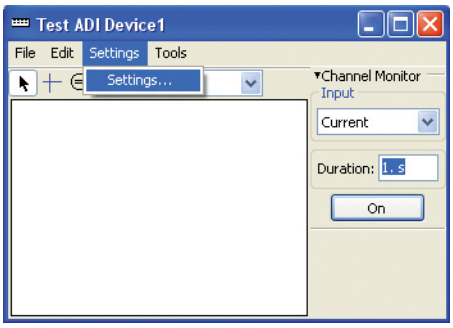


Fig. 4D

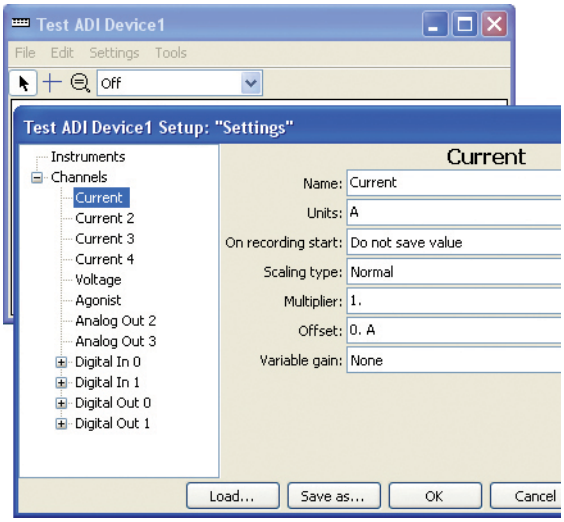


Fig. 4E

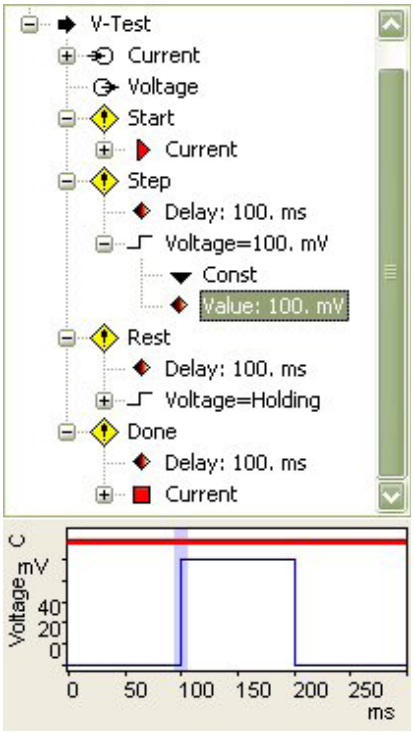


Fig. 5

Select “Channel monitor” from the Tools menu, Fig. 4A, and examine the controls that appear. The tools are shown to the right of the oscilloscope area, Fig. 4B. The name of the tool also provides access to the tool menu unique for each tool. Menus for all tools have a “Close” option to remove the tool and a “Float” option that causes tools to detach from the device window into a separate floating window.

The channel monitor tool can be used to test the functioning of inputs and outputs of the ADI devices. The tool menu contains commands for adding additional channels to the display. To activate the tool click the “On” button - note that it stays depressed after you click on it. Many other tools have this button - when it is toggled the tool takes control of the device, making all other tools inactive. To STOP the tool, click the depressed “On” button again, the button will pop up to indicate that the tool is inactive, Fig. 4C

Open the device setting dialog by selecting the “Settings...” command from the “Settings” menu of the Device Window, Fig. 4D. This dialog allows you to specify physical properties of signals connected to your recording system. Select first INPUT channel and type in a new name - “Current” and type in physical units - “A”, Fig. 4E. Select the first analog OUTPUT channel and name it “Voltage” and define units as “V”. Click “OK” to accept these changes.

Creating Voltage Steps Protocol - I-V Curve

Now we will create a more complicated protocol for the amplifier instrument that will produce the stimulus and perform analysis automatically. Make sure not to close the device

window while creating this protocol. Create a new protocol, rename it to V-Test. Right-click on its name and select “Add new input channel”. This time, the channel selection dialog will open automatically - this always happens upon the addition of new channels to a protocol, when there is at least one device open. Select “Current” for input. Add an output channel, select “Voltage”. Add four events, rename them to “Start”, “Step”, “Rest” and “Done”, in that order. Add actions on current to “Start” and “Done” events. Add actions on voltage to “Step” and “Rest” events. Right-click on the first action on voltage and select “Set Value”. Type “0.1” into the editor that appears. Your protocol editor should now look similar to Fig. 5.

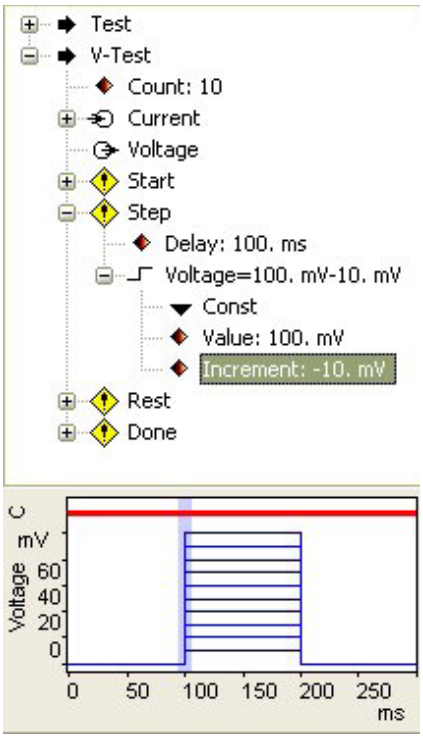


Fig. 6

Note that the plot view in the lower part is dynamically updated as you edit the protocol. The position of the currently selected event is highlighted on the plot to help you locate the event, which is currently being edited. You can also use the plot to navigate to the event - simply click anywhere inside the plot area and the closest event will be selected on the tree view.

Note that we only specified an output value for voltage in the “Step” event. We did not specify starting “holding” value or the value in the action inside the “Rest” event. The value here is determined by the default value set using the Channel Monitor tool. Try changing the voltage using the Channel Monitor tool in the Device Window and note that the protocol plot is updated to reflect the changes.
This illustrates the general rule of protocol editing: many properties do not have to be specified explicitly because default values are used. Some properties do not have default values, such as the sampling period for the input channel.

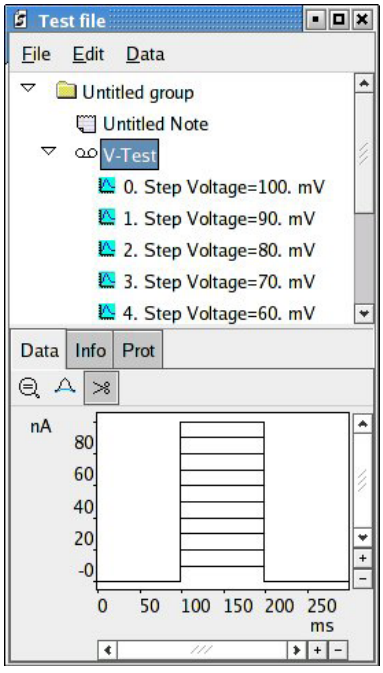


Fig. 7

By default a protocol is executed once during recording. The program allows multiple executions of the same protocol with changes to some of its properties during every execution. Let’s explore this. Right-click on the V-Test protocol you just created, select “Change Count” and type in a value of 10. Now after starting the recording the protocol will be executed 10 times, but every execution will be the same. To add variation right-click on voltage action that is inside “Step” event and select “Set Value Increment” command. Type value -0.01 into the newly created item. At this time your protocol editor should look similar to Fig.6.

You can now press the start button to observe protocol execution in the Device Window. If you use a virtual test device you should see step traces, since this device directly connects voltage OUTPUT to current

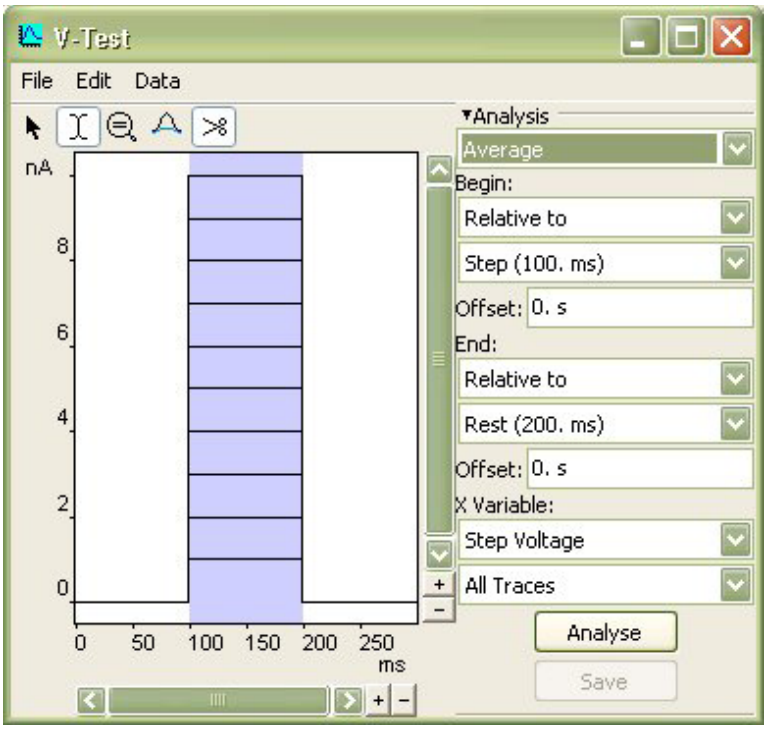


Fig. 8

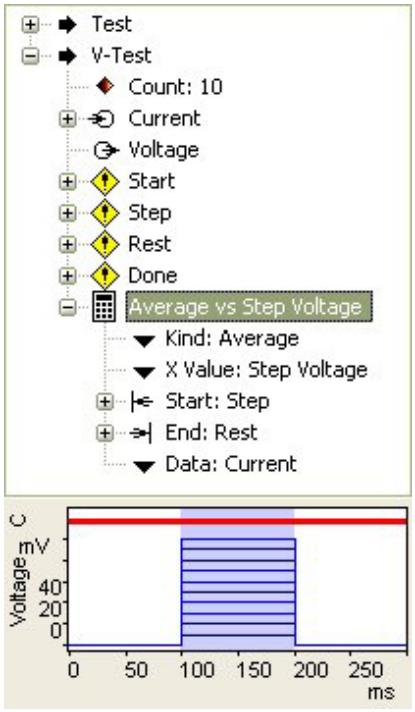


Fig. 9

INPUT. If you are using a real device you will need to provide some feedback from voltage to current channels to see the effect of variations in voltage.

Let's save the recording into a file for later analysis. In the "File" menu of the Protocol Editor select "New" and type the name and select location for your new file. By default the program gives the current date as the name of the file. The file is immediately created on the disk since the program does not keep all recorded data in its memory during recording but streams data to the file on the disk. A new File Window will appear. Before we start exploring the file window let's put some data into the file. In the Protocol Editor choose the file name using the File list selector that is located under the protocol selector and is marked with "File:" label. This selector specifies which open file, if any, will receive the recorded data. Select your newly created file. Now make sure that V-Test is selected in the protocol selector and start recording by pressing the start button. Observe data traces appearing both in the Device Window and in the File Window. After recording is finished the File Window should look similar to Fig.7.

The upper panel of this window shows recorded data in a hierarchical tree structure. Multiple recordings can be stored together inside a group of items that look similar to folders on a computer disk. Data recorded from every input channel during single execution of the protocol are represented by a separate entry under the main recording (labeled "V-Test" in Fig. 7).

The lower panel of the File Window (Fig. 7) contains several overlapping tabs that show various types of information about the currently selected item of the file structure. The "Data" tab graphically displays recorded data for the selected item. Click on the "Info" tab to see various text information about he selected item. Most of these Info fields will be the same as those in the Info area of the Protocol Editor, but you might also have some extra fields added during recording, such as the date and time when the recording was made. Click on the "Prot" tab to see a plot of the protocol used to record the selected item. When you double-click on the item or select it and press "Enter" a

separate Plot Window showing the item data will open. This window allows you to perform some additional operations depending on type of data. For example, open the Plot Window for recording item “V- Test”. The Plot Window allows you to perform analysis of traces on it. Select “Analyze” from the “Data” menu of this window. The analysis panel will be added to the window (Fig. 8). Press the “Analyze” button on the panel to perform the analysis according to the settings in the control panel. A new window will appear showing the results of analysis. For details about the program’s analysis capabilities see the corresponding section in the manual.

In order to make the program automatically analyze the traces during recording we need to add analysis requirements to the protocol. Right-click on “V-Test” in the protocol panel and select “Add analysis” from the menu that appears. An analysis item will be added following the last event, as shown on Fig.9. You can specify a type of analysis to be performed and select the data range for analysis. Note that the program automatically selected a time between Step and Rest events, which is a logical choice for this protocol.

Also note that when the analysis item is selected in the editor its range is marked on the plot view.

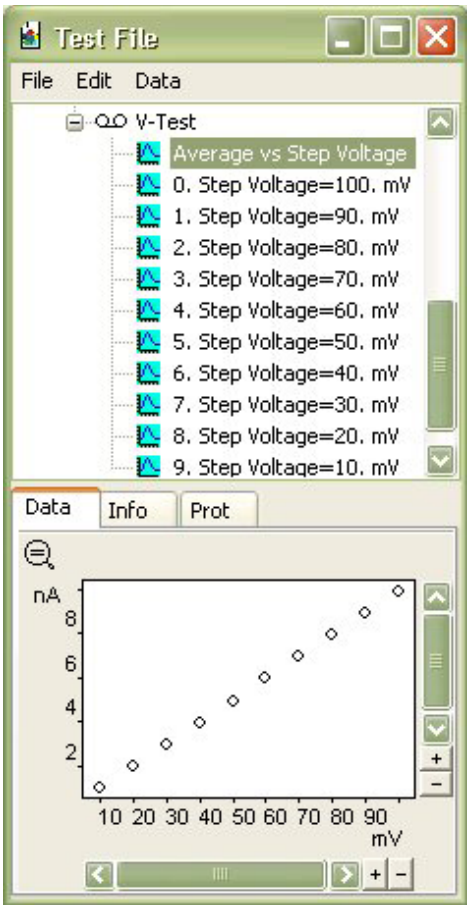


Fig. 10

The next time you start recording, the analysis will be done automatically right after each trace is recorded and results will be stored in a separate item within the data file. Note that automated analysis is only performed when data is recorded into a file. To see the results simply select the analysis item in the file structure and the results will be shown on the plot in the “Data” tab in lower part of the File Window, Fig. 10. You can also double click this item to display the plot in a separate window.

The program can also automatically display analysis results in a separate window during the recording. This allows simultaneous observation of recorded traces and results of analysis. To achieve this, right-click on the analysis item in the protocol editor and select “Autoshow”. The next time you run the protocol, the analysis results window will automatically open. To prevent automatic display, either edit the “Autoshow” item and select “No” from the menu or delete this item using “Delete” key on your keyboard.

This concludes the guided tour to the program. The following sections will describe various features of the program in detail.

Chapter 2

In addition to creating and editing the protocols, the main window of the application - Protocol Editor, provides controls for executing the data acquisition protocols. Closing the window will result in exiting from the program and is equivalent to selecting the “Quit” command from the “File” menu. The Protocol Editor window is divided into three panels:

- Menus and Recording Control - in the upper left
- Information Panel - in the lower left
- Protocol Editor - covers all the right side of the window

Recording Control

The Recording control panel has three controls: Start button, protocol selector and file selector. The Start button activates execution of the currently selected protocol. Pressing this button during the execution of the protocol will interrupt the recordings. The protocol can be selected either using the protocol selector menu located next to the start button, or by selecting the protocol name in the editor panel. The file selector specifies the opened data file where the data from input channels will be saved during protocol execution. If none is selected, the input data will be displayed during execution but will not be saved.

Information Panel

The Information panel shows a list of properties and information that will be saved to the recorded data file. The properties and information are provided by the active interface devices, instruments and tools associated with the devices. Their values are updated automatically during the experiment. You can also define your own custom information items.

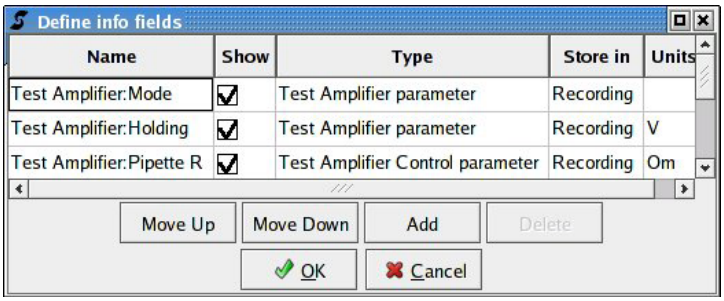


Fig. 12

The program allows you to specify which items you want to be displayed in the Information panel. This is done through a “Define info fields” dialog (Fig. 12), which is activated by clicking the button located on the top of the Information panel. You can create custom information items through this dialog by clicking “Add”. To remove a previously created custom info item, select it in the table and click “Delete”. Items provided by devices, instruments or tools cannot be deleted using this dialog. Closing the device window or tool, or removing the instrument from the device setup will automatically remove the information items associated with these devices and tools.

To prevent an item from being displayed in the Information panel, uncheck the corresponding check box in the second column labeled “Show”. Note that the unchecked items will still be saved to the data file together with data during recording and can be viewed in the Information panel of the data file. To change the order of the items use

“Move Up” and “Move Down” buttons that move the currently selected item.

The name of a custom Info item, displayed in the first column of the “Define Info fields” table, can be modified by editing. These names, however, have to be unique. Since similar devices, instruments or tools may provide Info items with the same names, the name of these items is qualified by prefixing it with the name of the parent device, instrument or tool name followed by colon. Because the colon character has a special meaning in separating the name of the Info item from the name of its parent entity, this character can not be used in the names of custom Info items. Although the names of automatic items are always shown with a prefix in the “Define Info” dialog, the prefix will be omitted in the Information panel if the item is unique in the current system.

The column labeled “Type” describes the data stored in the item. Automatic items provide a short description of the item that cannot be modified. For custom info items it is possible to specify the type of data by double-clicking on the Type field and using the up and down keys on your keyboard to make a menu selection. Currently available Types are “Text” - a single line of text, “Integer” - an integral number, and “Real” - a real number. The units specified in the last column of the table will be used for items containing real numbers, but are ignored for all others types.

The “Store in” column shows the association of the info item with the recorded data item. Association of items provided automatically is determined by their providers and cannot be altered here. For custom info items, it is possible to choose between “Group”, “Recording” and “Unit”. The items with “Group” association will be stored only once for each group in the data file at the moment the group is created either manually or with the first recording. The info items associated with the group will be used to describe the whole experiment, such as a type of specimen used during all recordings. The info items associated with “Recording” will be saved in the beginning of the protocol’s execution. The “Unit” association specifies that the info item will be save at the start of every iteration of the protocol.

Protocol Editor Panel

The Protocol Editor allows you to create, modify and view the experimental protocols. A protocol is represented by a hierarchical structure that defines the data input and output channels and the time sequence of events performed on the channels. A detailed description of the protocol hierarchy and its editing can be found in a separate chapter “Experimental Protocol”.

The editor is divided into two parts. The upper part shows a hierarchical representation of the protocol, where all the editing is performed. All defined protocols are listed on the top. The lower part shows a timeline of the currently selected protocol. This plot provides an overview of the whole protocol and simplifies navigation within the protocol hierarchy. Clicking inside the plot will highlight the event closest to the mouse pointer and will select this event in the hierarchical view above.

Selecting an item in the hierarchy within the protocol will highlight the event on the plot. Selecting any item within the protocol will select the protocol for execution and its name will be selected in the protocol selector of the Recording Controls panel. Alternatively, choosing the protocol in the protocol selector will select the protocol in protocol editor, allowing faster navigation through the protocol structure.

Menus

File

New: Create new data file.

Open: Open an existing data file.

Quit: Exit the program

New Timer: Creates a new timer window. See section “Timer” for usage of timers in the program.

Protocol

New protocol: Creates new empty protocol.

Duplicate protocol: Creates a copy of currently selected protocol.

Load Protocol: Loads a previously saved set of protocols from a file. Note that this file becomes the default storage file for the protocols. All changes made to protocols in the editor will be automatically saved in this file. Also the protocols will be automatically loaded from this file the next time the program is started.

Save Protocols: Saves the current set of protocols into a new file. Note that this file becomes the default storage file for the protocols. All changes made to the protocols in the Editor will be automatically saved in this file. Also the protocols will be automatically loaded from this file the next time the program is started.

Device

This menu lists all the interface devices that the program finds in the system. Note that this list can change as a result of the connecting or disconnecting of external devices. Selecting the device in the menu results in opening a new device window, if it was not already opened. This window becomes active and is shown on top of the other windows. The devices that have their windows opened are referred to as opened devices. Opened devices are checked in the “Device” menu. Protocols only work with channels from opened devices. Upon exiting from the program a list of opened devices is remembered and the program will attempt to open these devices the next time it starts.

Device Windows

Each active interface device in the system is represented by its own device window. The program does not acquire data from or control devices that are not activated and do not have their window opened. Most of the Device window area is used for monitoring data acquired from the device. In the case of the analog/digital interface the window behaves like an oscilloscope showing signal traces.
The area along the right site of the device window may contain a number of control tools. These tools can be used to control the state of the device and instruments connected to the device. A list of available tools depends on the

type of device as well as on the instruments configured within the device. Currently available tools are listed in the "Tools" menu, which is used to open the tools.

When a Device Window is opened for the first time a device setup dialog is displayed automatically. Approving the settings by clicking "OK" will result in the program asking for a file name to store the settings. Any further changes to the device settings will be saved to that file. The next time the device opens the setting will be automatically loaded from this file. The Settings dialog can be opened from the "Settings" menu of the device window.
Further details about device settings and tools are in the chapter "Devices" and your device reference manuals.

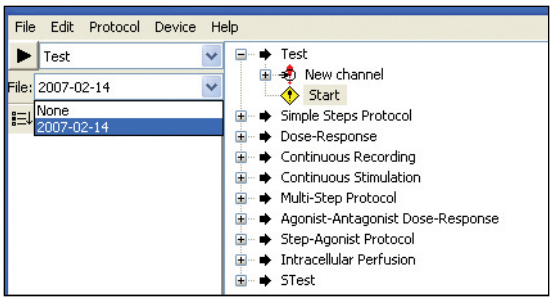


Fig. 13A

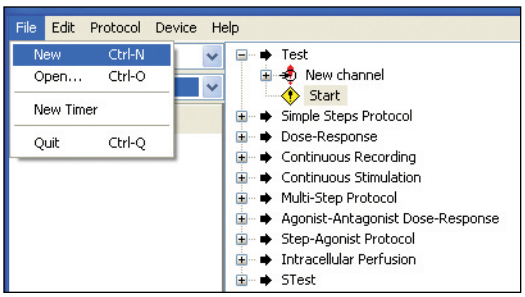


Fig. 13B

Data File Window

The File Window shows the data file structure and contents. The name of this window is the actual data file name. All data and information assotiated with the protocol are stored in a single file. The program may have several files opened simultaneously, but only one file can be selected for saving the data during protocol execution. The file for recording is specified by the file selected in the Protocol Editor, Fig. 13A. To create a new data file or open an existing file use the "File" menu, Fig 13B.

The File Window (Fig. 13C) is devided into two parts. The top part of the window shows the file structure. The lower part contains tabbed graphical views of recorded data, under tab "Data", experimental conditions, "Info", and protocol sequence, "Prot". Each data recording is represented by an icon and a text label. The text label may be edited by selecting the item and then clicking on it again or pressing the "Space" key on your keyboard. The text label may be prefixed with unchangeable text provided by the program that further describes the item. This prefix text, if present, is separated from the user-specified text by a colon, and does not appear during label editing.

The data are organized into a hierarchical structure similar to file structure on the computer hard disk. Grouped data items (hereafter called "group items"), similar to folders on a hard drive, are on top of the structure. The group items can only be on the top level of file hierarchy and cannot contain other group items. Data recorded during a single execution of the protocol, are further grouped together under recording items, "V-Test" in Fig. 13C. The recording item is labeled by the name of the protocol. A copy of the protocol used during recording is saved within the recording item.

To view the protocol hierarchy, all the protocols in the file can be made visible by selecting from the menu "Data --> Show Protocols" on the top of the File Window, Fig. 13D. The graphical representation of the protocol for the cur-

rent recording item is also shown under the “Prot” tab in the lower section of the window. You can also drag the event item from the protocol hierarchy in the Protocol Editor onto the Prot tab window, and the event will be shown as a marker on the plot. If the event time varies between iterations of the protocol, the program will ask if you want the index of iteration to display on the plot.

The data recorded from every input channel during each iteration are represented by a separate entry within the recording item. The program supplies labels for recordings made with iterated protocols including the iteration number and possibly a value of the output channel that varied with every iteration. For protocols with multiple input channels and for single execution protocols the label contains the name of the input channel that was the source of the data.

The lower part of the File Window contains tabbed graphical views showing various properties of the selected item.
“Data” tab: Displays a preview of the data contained within the selected item or its nearest predecessor containing the data. If a recording item is selected, data from multiple channels and all iterations of the protocol are displayed, Fig. 13C. The recorded data can also be displayed in a separate Plot window by double-click on the item. To add another recording to this Plot window drag-and-drop the item into the window.

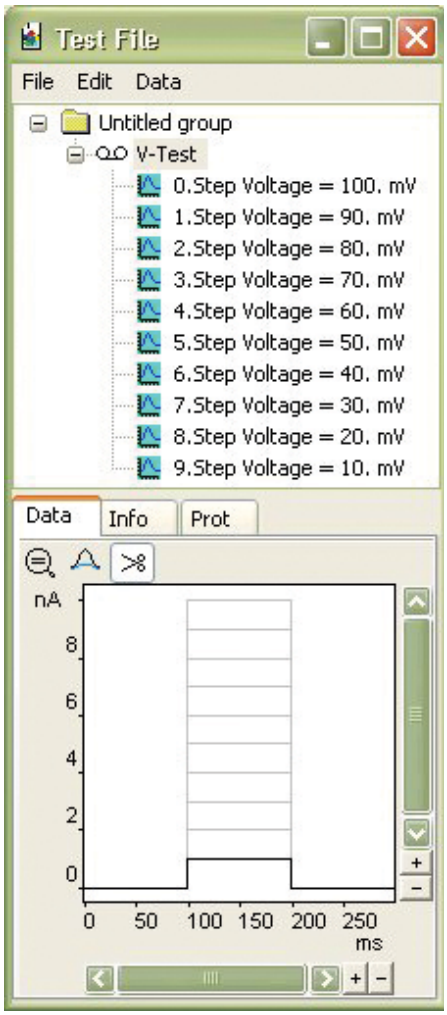


Fig. 13C

“Info” tab: Displays information items describing various experimental conditions at the time of recording. For additional details refer to section “Information Panel”. The Info items associated with the selected data and all of its predecessors in the hyerarchy will be shown here. This panel allows you to specify which items you want to be displayed, the same way as in the Information Panel of the Protocol Editor window. Filtering and order settings are unique for group items and data items so the same information may be displayed in a different order for individual recordings and for group items.

“Prot” tab: Displays time sequence of the protocol in a way similar to the Protocol Editor. In particular, if the hierarchical view of the protocol is enabled from the “Data” menu, clicking on the plot will select the nearest event in the protocol view. Selected events are highlighted.

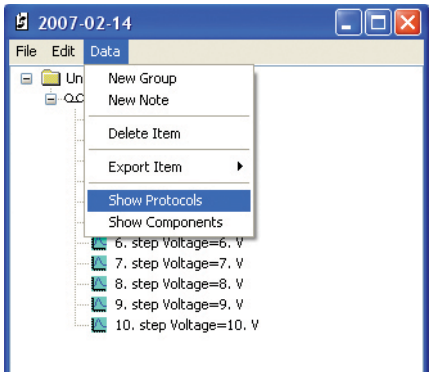


Fig. 13D

In addition to automatically recorded data and info items the file may contain various types of extra information. One example is text notes. To create a new text note select “New Note” from the “Data” menu.

The note will be created as the last child of the currently selected item. Text notes can be placed within any kind of item, allowing extensive description of the file. To edit the text of the note double click on its entry in the hierarchy - this will open a text editor in a separate window. The note does have to contain text - its label may provide sufficient space for information that needs to be seen first. The note items can be moved by dragging. To make a copy of a note hold down the “Shift” key on the keyboard while dragging.

File Window Menus

File

Save: Saves changes to the file. Although the data are written directly to the memory disk during the recording, the data properties, including the location of various data within the file, are not saved automatically.

Data

New Group: Creates a new group item as the last item in the hierarchy. All subsequent recordings will be placed into the last group item created. Note that if the recording is started into a newly created file that does not have group items, a new group will be created automatically.

New Note: Creates a new text note as the last child of the currently selected item. If no item is selected the note is created on the top level. You can drag a note to a different item or even to a different file. You can create a copy of the note by dragging with the “Control” key pressed. To edit text inside the note double-click on the note, or select it and press the “Enter” key.

Delete Item: Removes currently selected item from the file.

Export Item: Exports currently selected item to an external data format as selected from the sub-menu. This command is disabled if the selected item does not contain recorded data.

Text (CSV): Exports data as a text file with comma separated values. This format is suitable for import into spreadsheet software.

Igor Text: Exports data in text format for import into an Igor Pro data analysis software package.

Show Protocol: Displays or hides protocol used during recording. This menu item has a check mark next to it if protocols are currently displayed. The protocol items are shown as children of recording items. This command only controls hierarchical display of the protocol items. Graphical representation of the protocol for the selected item can also be shown under the “Prot” tab in the bottom part of the File Window. Selecting the event items in the protocol hierarchy will highlight the event reperesented graphically in the Prot tab.

Show Components: Displays or hides items representing separate recordings of averaged items and baseline items. This item has a check mark reflecting the current settings. Activating this option allows you to remove a recorded baseline or some of the recordings used for averaging.

Plot Window

Plot windows can display recorded data and results of analysis from multiple files. This window also hosts controls for performing interactive analysis.

To display data in a new Plot Window, double click on a data item in the File Window or select it and press the "Enter" key. Opening a trace will produce a window with a single trace on it. It is also possible to open a group item - this will display a window with all traces from channels recorded during the protocol iterations. This is used for a quick overview of the recorded data. Channels that have different units will be plotted separately. To add data to an already opened plot, simply drag-and-drop an item on top of the Plot Window.

The Plot Window allows you to compare data from different files. If vertical or horizontal units of the dropped data items do not have corresponding axes on the plot, the new axes will be added automatically. Every axis has its own scrollbar. In addition to common controls used to scroll traces, each scrollbar has two buttons to ZOOM IN and OUT. The full scale on all axes can be restored by clicking on an auto scale button on the toolbar. In addition to zoom buttons the axis scale can be changed using a selection marquee. The selection marquee can be created on a plot by dragging the mouse when the mouse pointer is an arrow. Right-clicking inside the marquee opens a menu with options to expand or shrink the scale relative to the marquee.

Right-clicking with the mouse cursor inside a plot area opens a pop-up menu containing commands to auto scale all axes or only the vertical or horizontal axis. If the mouse pointer is on top of the trace while right-clicking, the menu will also contain commands to reveal the data item corresponding to the trace in the file window, to remove the trace from the plot, and to change trace color. For recorded data trace the menu also contains a command to create a timer marker with a value corresponding to the horizontal position of mouse pointer. See section "Time Marker" for additional details.

The Plot window's toolbar may contain various buttons depending on type of data shown on the plot:

Autoscale: Restores the scaling of all axes of the plot to show data maximums and minimums.

Arrow cursor: Activates the most commonly used mouse pointer mode - dragging the mouse over the plot area produces a selection marquee. Right clicking within the marquee selection results in a popup menu containing commands for rescaling the plot using the marquee as a reference.

Crosshair button: Activates crosshair pointer mode. In this mode the values of the cursor positions are displayed. Clicking and dragging the mouse in crosshair mode displays the value difference between start and end positions of the cursor.

Beam: This button is visible only when interactive analysis is active. With the beam pointer you can select a range on the horizontal axis that will be used for analysis.

Show background & Subtract background: If traces have a background signal recorded, the “Show background” button and “Subtract background” buttons are added to the toolbar. These buttons toggle between active and inactive state when pressed. The “Show background” button shows the background trace data together with the main trace data. If the subtract background button is active, the background is subtracted from the main trace, otherwise the original trace is shown as it was recorded.

Stimulus: This button is available for the traces recorded with protocol containing output channels. Activating this button will show corresponding stimulation traces that were generated during recording. These traces are similar to traces shown in the Protocol editor. Stimulus traces are shown in the same color as the corresponding recorded trace. Changing the color of the recorded trace will also change the color of corresponding stimulus trace.

Events: Allows you to mark the events on the plot by vertica lines.

Label Events: Puts event names next to the event marks.

Plot Window menus

Edit

Copy: Places an image of the plot on the system clipboard.

Data

Analyze: Starts interactive analysis of the data on the plot. Interactive analysis controls are displayed on the panel to the right of the plot. A detailed description of interactive analysis is given in chapter “Data Analysis”.

Export: Exports data to external data format as selected from sub-menu.

Text (CVS): Exports data as a text file with comma separated values. This format is suitable for import into spreadsheet software.

Igor Text: Exports data in text format for import into an Igor Pro data analysis package.

Timer

A Timer Window allows you to track time intervals and save timer marks into the data file. The program can have several timers active simultaneously. To create a new timer select the “New Timer” command from the File menu in the Protocol Editor. The timer is continuously counting time since the last reset. You can reset the timer manually by clicking the “Reset” button. The timer can also reset itself automatically on certain events as specified by the settings. Click the “Settings” button to access the dialog for the timer settings. In this dialog you can specify a name for the timer. The name is displayed in the title bar of the timer window and will become the name of the time

mark saved to the data file. You can specify when the timer will reset itself automatically by checking the appropriate check-box: it can reset when recording starts and when a new group is created in the data file.

To create a time mark in the data file with the current time, click “Save”. A dialog asking for comments will appear. The program will show a list of previous comments used for timer marks in a popup menu. Type in a new comment or select a previously used one and press “Enter” or click “OK” to save the mark in the data file. The newly created marker will reflect the time displayed on the timer at the moment the “Save” button was clicked. If the “Save” button was clicked while the recording was active, the time marker will be created inside the recording item, otherwise, it will be created inside the last group item. You can move this timer item into the trace recording item, which was recorded at the time the timer item was created, by simply dragging the marker onto the recording item in the File window. Having a time marker within a recorded trace allows you to display the marker on the same plot window. The timer marker is displayed as a vertical line with text comments next to it.

Time Marker

A time marker is an item inside the data file that specifies a moment in time during the course of an experiment. You may create the marker to designate specific moments during recordings. You may also use the marker as a container for notes and analysis results that are related to the designated moment. You can create the marker from the Timer window during recording by clicking the “Save” button. You also can create the marker later, after the recording is finished, in the plot window showing the recorded trace. To create a marker for a recorded trace, open the trace in the plot window by double-clicking on the trace in the File window, then right-click on the point you want marked and select “Add timer” command from the popup menu. A dialog will appear where you can enter a comment for the marke. The program stores a list of previous comments used for timer marks in a popup menu. Type in a new comment or select a previously used one and press the “Enter” key or click “OK” to save the marker. The item will be saved inside the trace recording. The program will also add the marker to the graph as a vertical line with the comment text next to it.

You can add an existing marker to the plot window by dragging its icon onto the graph from the File window. Note that only the markers that are located inside the trace recordings in a data file can be added to the plot window. This is because the time axis on the plot window is not absolute but relative to the start of the trace, so having a time marker within the trace allows you to relate the absolute moment of time represented by the marker to the relative time of recording. You can move and copy the marker among different traces recorded simultaneously.

Chapter 3: Experimental Protocol

This system provides a powerful tool for programming time sequences of events to control data acquisition protocol using a flexible GUI (graphical user interface). These are some of the advantages of our system:
- Different types of data acquisition interfaces, such as digital and analog I/O boards, including time-lapsed image acquisition devices, can be freely intermixed within a single protocol.
- Existing protocols can be quickly and easily reused with different combination of interfaces and instruments on the

same computer system or on a different experimental rig.
- Hierarchical representation of the protocol where you can easily define complicated time sequences, which allows you to view important features of a protocol at a glance.
- The program also allows you to specify online data analysis within the protocol itself.
- A collection of frequently used protocols can be stored in a single file.

On the first run of the program you will be asked to save the protocols into a file. You can give any name to the file and save it at any location. The next time the program starts, this file will be opened automatically and any changes made to the protocols will be automatically saved into this file. You can have multiple protocol files for different experiments, or several users can share a single protocols file by keeping it in the location that is accessible by all users. To save protocols into a new file select the “Save Protocols...” command from the “Protocol” menu of the main window. To load protocols from a different file select “Load Protocols...” from the “Protocol” menu. Once the protocol is loaded from the new file, this file becomes the default protocol file and will be used the next time the program is started.

Concept

The program represents the experimental protocol as a hierarchy of properties. The protocols are shown, created and modified in a protocol editor panel of the Main window (Fig.11). To the user the protocol is presented as a collection of expandable container items, each representing a property of the parent item that contains it. The item is shown as a labeled icon. If the item contains children, a control for displaying and hiding children is also shown.
The icon and text label describe the property represented by the item. lf an item represents a simple property then its value is also included in the text. You can change the value by directly editing the text of the label. Some properties can only be one of a limited set of choices, in which case the value is chosen from a popup menu. The item that groups other properties might not have a value.
It is common for an item to have a name property. For example, each protocol has a name. The name is often not shown as a separate child item, but as the text label of an item itself.
Many properties may have default values. By default the protocol is executed only once, for example. To avoid cluttering, properties having default values are usually not shown. Further, deleting an item is equivalent to assigning to the item a default value. To delete an item, select it and press the “Delete” key on the keyboard.
The properties are added to an item using popup menu commands that list only properties that can be added to the selected item in a given context.

Right click on the item to access its popup menu. When adding new properties the program automatically maintains a prescribed order in which the property items are displayed within its parent. This helps to avoid confusion when comparing different protocols since you will always find similar properties in the same place.
Some properties may have multiple instances. The protocol can have an arbitrary number of input channels, for example. Multiple instances of such properties are grouped together. It is possible to change the order of items by dragging them with the mouse, but the program will not allow you to drag an item outside of its group, which would violate the prescribed order.

Protocol items

The following is a detailed description of various items that can be present in an experimental protocol. Each item is

introduced in order of its appearance within its parent item.

Protocol - This item represents the protocol itself. Its label is the name of the protocol, which is also listed in the protocol selector of the main window. You can edit the text label to change the name of protocol.

Protocol menu

Change count: Adds a property that specifies the number of iterations of the entire protocol during execution.

Change Delay: Adds a property that specifies the time interval between the end of the protocol iteration and the start of the next iteration. This command is present in the menu only when the protocol has a Count property with value more than 1.

Change Averaging Count: Adds a property that specifies the number of recordings made to average recording traces.

Add Input Channel: Adds a channel item for an input channel that is associated with the device that is processing your input signals.

Add Output Channel: Adds a channel item that is associated with your device output.

Add Event: Adds an Event item to specify a moment in time during protocol execution.

Record Baseline: Requests baseline recording for the protocol by adding Baseline item.

Add Analysis: Creates new online analysis operation to be performed on recorded data after each iteration of the protocol.

Protocol property items

Count - Integral value. Default = 1. Valid values >= 1. Cannot have user-defined properties. Specifies the number of iterations of the entire protocol during the execution. To enter a different number use “Change count” from the pop-up by right-clicking on the protocol name.

Delay - Real value. Default value is 0. Valid values >=0. Cannot have additional or user-defined properties. Specifies the time interval between the end of protocol iteration and the start of next iteration. To enter a new value use “Change Delay” command, which is present in the pop-up menu only when the Count property is more the 1.

Averaging Count - Integral value. Default value is 1. Valid values >= 1. Cannot have user-defined properties. Specifies the number of recordings made to average recordings. Use “Change Averaging count” command to enter a new value.

Record baseline - No value. Compound item. When present in the protocol a baseline recording will be performed during execution of a protocol. The properties of this item describes how the baseline is performed. The baseline is

used for subtracting the background from the recorded data. To define, use “Record baseline” command.
When this item is present in a protocol, baseline recording will be performed during protocol execution. Baseline recording can be used to automatically sample the background activity of your experimental sample. A response of the system to a small stimulation, for example, can be used to subtract a linear component of the response from a non-linear “useful” signal, as it is done in conventional P/n leak current subtraction in patch-clamp recording.

Record Baseline property items

Count - Integral value. Default value is 1. Valid values >=1. Cannot have additional or user-defined properties. Specifies the number of recordings made to average traces at every protocol iteration. This property is always present.

Delay - Real value. Default value is 0. Can not have additional or user-defined properties. Specifies a time interval between the baseline recording and the actual recording. Negative values specify that the baseline recording is performed before the actual recording while positive values means that the baseline recording is performed after the actual recording. This property is always present.

Stim. Scale - Real value. Default value is 0.2. Valid values >0. Cannot have additional properties. Specifies the value of stimulus scaling. For output channels that have their Baseline Output set to “Scaled”, this value is used to scale pulse amplitude. For input channels that have their property “Scale Baseline” set to “Yes”, the recorded values will be divided by this scale factor. This property is always present.

Channel

- **Input channel.** Text value (name). No default. Can have properties.
- **Output channel.** Text value (name). No default. Cannot have additional proper1ies.

The channel item specifies the device channel used by the protocol. The text label is the name of the channel. The text label can be edited directly, or the channel name can be selected from the list of channels available in the active devices. Each protocol will have at least one channel. Use “Add input channel” and “Add output channel” commands.

The channel item defines your interfaces and instruments input or output used during the protocol. The protocol channel is a protocol item that names your device channel used during execution of the protocol. So although it is possible to have protocol channels that do not refer to any channel in the currently available devices, the protocols with such channels cannot be executed until these channels are matched with active device channels. Matching can happen in different ways: The protocol channel can be renamed to match the device channel, or a device channel in the active device can be renamed to match a protocol channel, or a new device that has matching device channels is opened.

When a protocol has channels that do not match your device channels, it cannot be executed, and the program marks such channels with an exclamation point. Another situation where the program cannot execute the protocol is when different devices have channels with the same name. When this happens, the program marks the icon of such channels with two exclamation points.

In order to distinguish device channels with the same name, the channel can be prefixed by the device or instrument name. In this case, the name of the channel is separated from the device name with a colon. Because of this rule, none of the devices, instruments and channel names can include the colon symbol.

The program provides a convenient alternative to manually typing names of protocol channels. The channel item popup menu has a “Select” command, which lists all channels from currently opened devices, allowing you to unambiguously choose the channel name. By default the dialog shows short names for the channels and adds a device or instrument name only to tredundant channel names. To show all names with device qualifiers check the checkbox “Show full names”. To change the name of a selected channel click the “OK” button. To leave the name unchanged click “Cancel”. To rename all protocol channels selected, click “All”. This is particularly useful when transferring a large set of protocols to a different experimental system.

Channel property items

Sampling period - Real value. Default value is 1 ms. Valid values >0. Cannot have additional or user-defined properties. Specifies the sampling period. Present for input channels only. Note that although any positive value will be accepted by the editor, the actual hardware may impose some restrictions on possible values. The sampling specified might be too fast for the hardware to perform, for example. If the device cannot perform recording with the specified period an error will be reported when the protocol is started.

Record baseline - Boolean value. Default value is Yes. For input channels only. Present only if the protocol contains “Record baseline” property. Specifies whether data from the channel are actually recorded on the disk during the baseline recording cycle.

Scale baseline - Boolean value. Default value is Yes. For input channels only. Present only if the protocol contains “Record baseline” property. If “Yes”, the data from the channel that was used for baseline subtraction will be multiplied by the stimulus scale value specified in the “Record Baseline” property of the protocol. This setting can be used to implement P/n leak current subtraction in patch-clamp recordings, for example. If the value is “No”, unmodified data will be used for baseline subtraction.

Baseline output - Possible values: Scaled, Unmodified, Const. Default value is Scaled. For output channels only. The item determines what kind of output will be produced on the channel during the baseline recording cycle.

- Scaled: The output relative to a holding value specified by the protocol will be multiplied during baseline recording by the scale value specified in “Record Baseline” property of the protocol. The scaled output will be generated from the baseline holding value specified in the “Baseline Value” property or the current holding value. This mode can be used to implement P/n leak current subtraction in patch-clamp recordings, for example.

- Unmodified: The output produced on the channel during a baseline recording cycle will be the same as during actual recording.

- Const: The output of the channel will not change during baseline recording. The output value will be set to what is

specified by the “Baseline Value” property or the current holding value.

Baseline value - Real value. Default value is the holding value. For output channels only. The item specifies the holding value for the period of baseline recording. In order to use the same holding value as for the actual recording, delete this item by selecting it and pressing the “Delete” key on your keyboard.

Channel Menu

Select - Displays a list of all channels from currently opened devices. To select a channel click “OK“. Click “Cancel” to leave the name unchanged. To rename all the protocol channels in all protocols with the same name, click “All”.

Event - Text value (name). No default. Can have additional user-defined properties. The Event item specifies a moment in time during protocol execution. Event properties include actions performed on the channels, such as recording start and stop, and changing the value of output channels. Each protocol will have at least one event item. Use “Add Event” command.

Event Items Menu

Change Delay: Adds a “Delay” property item to the event. “Delay” property specifies the time period between the reference moment and the event.

Variable: A submenu containing commands that very the “Delay” property between iterations of protocol.

Change Delay Increment: Adds a “Delay Increment” property that specifies the value added to the “Delay” value on every iteration of the protocol. Used to introduce linearly changing timing into the protocol.

Change Delay Multiplier: Adds a “Delay Multiplier”, which specifies the multiplier for the “Delay” applied on every iteration of the protocol. Used to introduce linearly-changing timing into the protocol.

Change Increment Multiplier: Adds an “Increment Multiplier” property item to the event. This item specifies the value of the multiplier of the “Delay Increment” for every iteration of the protocol. Used to introduce exponentially changing timing to the protocol.

Change Increment Mode: Adds an “Increment Mode” property item to the event. This item determines the usage order of variable values of the delay sequence.

Set Reference Event: Adds a “Reference Event” property item to the event. This item specifies the event from which the delay of the event is measured.

Insert Event: Inserts a new “Event” item before the selected event.

Append Event: Adds a new “Event” item at the end of the event sequence.

Add Action on …: Introduces an action item to define operations performed with input or output channels during this event.

Event Properties

Delay - Real value. Default value is 0. Valid values >=0. Specifies the time period between the reference moment and the event. The reference moment for the first event is the moment of the current protocol start. The rest of the events by default specify their delay from the moment of the preceding events. The reference moment can be specified explicitly by adding a "Reference Event" property, which can refer to any event listed prior to its parent event. Use "Change Delay" command.

Delay increment - Real value. Default value is 0. Specifies the value added to the "Delay" on every iteration of the protocol. Used to introduce linearly changing timing into the protocol. Use "Variable - Change Delay Increment" command.

Delay multiplier - Real value. Default value is 1. Valid values: non-zero value. Specifies the value of a multiplier for the "Delay" to use on every iteration of the protocol. Used to introduce exponentially changing timing into the protocol. Use "Variable - Change Delay Multiplier" Command.

Increment multiplier - Real value. Default value is 1. Valid values: non-zero value. Specifies the value of the multiplier to the "Delay increment" on every iteration of the protocol. Used to introduce exponentially changing timing into protocol. Use "Variable Change Increment Multiplier" command.

Increment mode - Possible values: Increment, Decrement, Interleaved Increment, Interleaved Decrement, Alternate. Default value is Increment. Determines the order of using variable values of delay during protocol iterations. At the beginning of the protocol a sequence of variable values is computed according to the values of "Delay Increment", "Delay Multiplier" and "Increment Multiplier". During the protocol execution the value for the delay is chosen from the sequence in the order specified by "Increment Mode" as follows:

- ***Increment:*** The values are selected in the normal order beginning with the first value.
- ***Decrement:*** The values are selected in the reverse order beginning with the last value.
- ***Interleaved Increment:*** The values are selected starting from the first with odd and even values swapped.
- ***Interleaved Decrement:*** The values are selected in reverse order starting from the last with odd and even values swapped.
- ***Alternate:*** The values are selected alternatively in the normal order beginning with the first value and in the reverse order beginning with the last value.

Use "Variable - Change Increment Mode" command.

Reference Event - Possible values: Names of events preceding the parent event. This item specifies the event, from which the delay is measured. By default, when the "Reference Event" item is not present, the delay is measured from the start of protocol iteration for the very first event or from the immediately preceding event. Use "Set Reference Event" command.

Action

The Action item specifies an operation performed on a device channel at the moment specified by the event. The action is always associated with a single protocol channel. The association is established when the item is created and cannot be altered. Removing the channel from the protocol will automatically remove all actions associated with it. Each event can have as many action items as the number of channel items in the protocol. Use "Add action on Channel" command. The icon of the action item is determined by the operation specified. The text label of action items starts with the name of the channel associated with this action and may provide additional details. The text label of action items cannot be directly edited.

Action Menu

Set triggered start: Adds "Triggered" item to the action so that recording will start only upon arrival of the device-specific trigger. Only present in menu of actions associated with input channels.

Set Value: Adds "Value" for output signal. Only present in the menu of actions associated with output channels.

Set Increment: Adds "Increment" property item that specifies the value added to the "Value" property upon every iteration of the protocol. Used to introduce a linearly changing output into protocols. Present in the menu only if action has "Value" property item.

Set Multiplier: Adds "Multiplier" property that specifies a multiplier for the "Value" property upon every iteration of the protocol. Used to generate an exponentially-changing output. Present in the menu only if the action has a "Value" property item.

Set Increment Multiplier: Adds "Increment Multiplier" property that specifies the value of the multiplier for the "Value" property for every iteration of the protocol. Used to generate an exponentially-changing output. Present only if action has "Value" specified.

Set Increment Mode: Adds "Increment Mode" to determine the order of output values used. Present only if the action has "Value" specified.

Action Properties

Operation - The first property of the action specifies the operation performed on the channel. The chosen operation determines the icon used for item in the protocol hierarchy.
- Start record - For input channels only. Starts recording into the file at the moment of the event.
- Stop record - For input channels only. Stops recording into the file at the moment of the event.
- Const - For output channels only. Outputs constant value.
- Ramp - For output analog channels only. Starts producing a linearly changing signal starting at the current value and ending at the value specified by other properties at the moment of the next event. Note that if there is no event after the ramp, which is equivalent to next event being at infinity, the output will be constant. Note that recording stops with an error message if the ramp is specified for a digital channel.

- Square - For output channels only. Starts producing rectangular pulses.
- Triggered - Boolean value. Default value is No. For input channels only. Starts recording action only. Specifies that recording will not start immediately but only after a trigger. The meaning of the trigger depends on the devices used. Not all channels and not all devices have the ability of a triggered start. When not implemented an error will be reported on an attempt to execute such a protocol. Use "Set triggered start" command.

Value - Real value. For output channels only. Default value is the holding value or the output value of the channel at the start of the protocol. Valid value range depends on the hardware used (devices). Specifies the constant output value. Use "Set Value" command.

Increment - Real value. For output channels only. Default value is 0. Specifies the value added to the "Value" on every iteration of the protocol. Used to generate a linearly changing output. Use "Set Value Increment" command.

Multiplier - Real value. For output channels only. Default value is 1. Valid values: non-zero value. Specifies a multiplier for "Value" to apply on every iteration of the protocol. Used to generate exponentially changing output. Use "Set Value Multiplier" command.

Increment multiplier - Real value. For output channels only. Default value is 1. Valid values: non-zero value. Specifies a multiplier for "Increment" to apply on every iteration of the protocol. Used to generate exponentially changing output. Use "Set Increment Multiplier" command.

Increment Mode - Possible values: Increment, Decrement, Interleaved Increment, Interleaved Decrement, Alternate. Default value is Increment. For output channels only. Determines the order of output values to use during protocol iterations. At the beginning of the protocol execution a sequence of variable values is computed according to the values of "Increment", "Multiplier" and "Increment Multiplier". During the execution an output value for every iteration is chosen from the sequence in the order specified by "Increment Mode":

- ***Increment:*** The values are selected in the normal order beginning with the first value.
- ***Decrement:*** The values are selected in the reverse order beginning with the last value.
- ***Interleaved Increment:*** The values are selected starting from the first with odd and even values swapped.
- ***Interleaved Decrement:*** The values are selected in reverse order starting from the last with odd and even values swapped.
- ***Alternate:*** The values are selected alternatively in normal order beginning with the first value and in reverse order beginning with the last value. Use "Set Increment Mode" command.

Frequency - Real value. For output channels only. "Square" operation only. Valid values are determined by the device capabilities. Specifies the frequency of the rectangular periodic channel in Hz. Added automatically when "Square" operation is selected.

Duty cycle - Real value. For output channels only. "Square" operation only. Valid values between 0 and 1. May be limited by device channel capabilities. Specifies the duty cycle of the rectangular periodic channel. Added automatically when "Square" operation is selected.

Analysis - No value. Compound item. This items requests online analysis at the end of each protocol iteration. The

text label, which is provided by the program automatically, gives a brief description of analysis. Protocol can have any number of analysis items. Use “Add analysis” command. The Analysis item describes the automated online analysis operation to perform during after each iteration the protocol. The online analysis may perform the same operations as interactive analysis. The results of online analysis are saved to the same file inside the recording item. The analysis will be performed only if the recording is saved to a file. The protocol can have any number of “Analysis” items, each resulting in a separate data item in the file.

Analysis Property Items

Kind - Possible values: Average, Maximum, Minimum, This is a required property. Defines the type of analysis to be performed. The list of available analysis operations might depend on installed analysis components.
X Value - Possible values: Trace Number, and other variable properties of the protocol. A required property. Defines what values will be used on the horizontal axis of the plot showing the results. The “Trace Number” is always present if the protocol Count is greater than 1. If “Trace number” is selected for the X variable, each measurement will be identified by the sequential number of the protocol iteration. Any other variable property of the protocol will be listed as a choice. The “Running time” choice allows the analysis to run before the recording of the trace is finished, and can be used for real time analysis of continuous recording. In this case the analysis is performed on portions of the recorded data between time points defined by Start and Stop analysis properties.

Start and End - These two properties specify the time interval that will be used for analysis. Each has two properties:
- ***Value:*** Real value. Defines the value for the time offset. The offset is measured from a specified reference moment or from the start of the protocol iteration. This is a required item, which is always present.
- ***Reference:*** Possible values: All events of the protocol. The item specifies the reference moment from which the offset, specified by the “Value” property, is measured. If this item is absent, the offset is taken from the start of the current iteration. Use “Set Reference” command.

Data - Possible values: All suitable input channels. This item is specific to the type of analysis selected. Specifies the input channels for the analysis. Some types of analysis may require selection of more than one input channel. In this case some extra properties will be added automatically.

Chapter 4: Devices

The program can work simultaneously with multiple user devices: interfaces and instruments. The devices, which are connected to the computer and recognized by the system, are listed in the Device menu of the main window. Before the program can use the device, it has to be activated (opened). To open the device, select its name in the “Device” menu of the main window. This will open a separate device window. Every opened device has a check mark next to its name in the “Device” menu. The main area of this window displays the data coming from the device through its input channels during protocol execution. In the case of an analog/digital interface, the data display area behaves like an oscilloscope. The window can also host various tools, which are device-specific, providing various means to control the device.

The program saves all settings related to a single device in a separate file. The first time the device is opened the

program automatically displays a setting dialog for the device. You can modify the default settings and save the settings file in the location of your choice. Once the settings file for the device is saved, all changes to the device setup will be saved in this file automatically. The next time the program starts and the device is opened, the settings will be automatically loaded from this file. You can have multiple settings saved in different files tailored for different kinds of experiments. To save settings into a new file, open the “Settings” dialog and click the “Save as...” button. To load a different settings file, open the settings dialog and click the “Load...” button. Again the last loaded settings file will be automatically used the next time the program starts. Several users can share the same settings by saving the settings file into a location that is accessible by all users.

Device Settings

Select the “Settings...” command from the “Settings” menu of the device window to access the setup dialog. Note that if the device is opened for the first time, this dialog opens automatically in order to create the file where the settings will be stored.

The setup dialog has two panels. In the left panel all the resources of the device that can be configured by the program are listed. This list is specific to each device but most types of resources are common, such as analog input and analog output channels. This chapter describes the settings for such common resources. Settings for resources that are unique for some devices are described in corresponding device driver manuals. The right panel of the setup window shows controls applicable to the resource, channel or instrument, selected in the left panel. Fig.15 shows example settings for an analog input channel.

The “Name” field is common for all types of channels. The default name is device specific and is usually a generic name for the channel plus its number to distinguish it from other similar channels, for example “Analog In2”. The experimental protocol refers to the channel by its name. Changing the name of the channel that is already used in protocol will invalidate these protocols and will require changing the corresponding channel name in the protocols as well. The protocol editor provides a way to change the name of the corresponding channel in all protocols at once through the “Select...” command (see chapter “Channels”).

The “Units” field provides a physical description of the signal acquired by the channel. If the unit name is provided, the measured values displayed by the program will be accompanied with the units using engineering notation with an exponential prefix preceding the unit name. Otherwise, the values are displayed using scientific notation with the mantissa followed by an exponent separated by the character “e”.

The next control specifies whether the input value of the channel is saved before recording. This can be used for channels that read your hardware configuration like telegraph signals linked to the amplifier gain, for example. Available settings are:
- ***Do not save value:*** The value will not be saved
- ***Save value:*** The value will be saved in an appropriate info item.
- ***Save value for every recording unit:*** The value will be sampled on every protocol iteration and saved in an appropriate

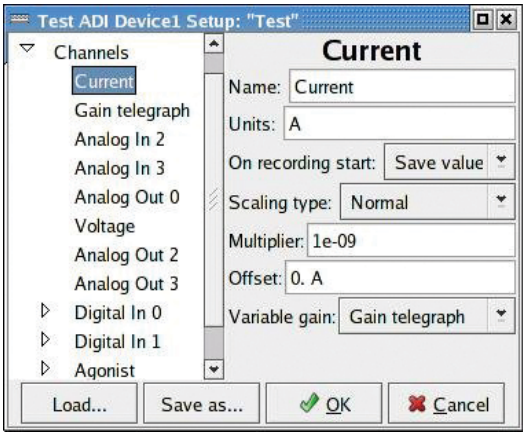


Fig. 15

info item. If the value of the channel is selected for saving, it is also displayed in the info area of the main window.

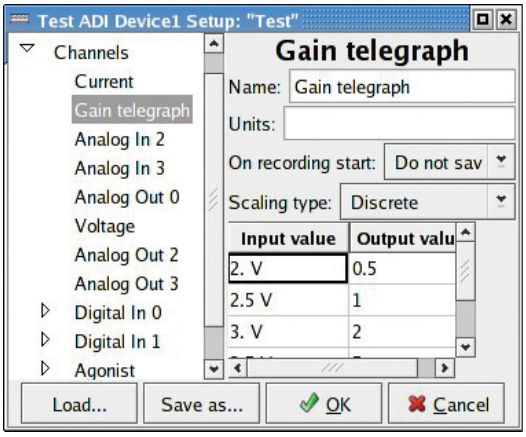


Fig. 16

"Scaling type" specifies how the voltage on the terminals of the device is converted into the physical units. The scaling type can be chosen to be either "Normal" or "Discrete". Normal scaling, as shown in Fig.15, requires a multiplier and an offset. To convert to the calibrated value, the voltage measured on the device terminal is multiplied by the multiplier and the value of offset is added. The multiplier may be calculated using several components: the multiplier specified by settings, a value measured from another input channel selected in the "Variable gain" menu, and, on some devices, an additional gain provided by hardware built into the device.

In the case of "Discrete" scaling, the calibrated value is obtained from a look-up table of thresholds, which is specified by the user, as shown in the example on Fig.16. If the channel reading is less than or equal to the value in the first column in the first row of the table, the calibrated value is taken from the corresponding OUTPUT column in the first row. Otherwise, readings are compared to values from the second row and so on, until the appropriate row is located.

Not all settings described above are present for all types of channels. The digital input channels, for example, do not have scaling fields. Digital output channels may have a discrete scaling type, however, which may be useful in some systems that are able to generate a response to a digital code.

The program might treat digital channels as compound channels containing multiple channels corresponding to the individual bits and represented by integer numbers. The protocol can refer to these channels by their names so that individual bits of the digital channels can be used as if they are separate channels.

Device Tools

The program allows you to control various functions of devices through a set of tools. The device tool is a control panel. The tool panels are shown either along the right side of the device window or inside separate floating windows bound to the device window. Each device window has a "Tools" menu that lists all the tools available for the device. Some tools are specific for particular devices. Some tools may work only with specifically configured devices: when a device is configured to work with a particular user instrument, for example. This chapter will describe common tools available for most devices.The description of device- and instrument-specific tools is provided in the device manuals.

Every tool panel has a pop-up menu activated by clicking on the title of the tool. The tool menu always contains a "Close" command to remove the tool, and a "Float" command that toggles the position of the tool between docked on the right side of the device window and floating on top of all other windows. Some additional commands specific to the tool function might also be present in the menu.

A tool, when active, takes full control of the device and the display area of the device window. Because of that only one tool can be active at any given time. Many tools have an “On” toggle button that activates the tool. Clicking the “On” button to activate the tool will inactivate any other tool that was active. Also, starting the protocol will inactivate all the tools in all devices that use channels referred to by the protocol. No tool can be activated until acquisition on all input channels is finished or the protocol is manually interrupted.

Channel Monitor

The Channel Monitor tool works with any device that has either analog or digital channels. The tool may generate signals on several output channels and continuously monitor inputs of several input channels. Fig.17 shows a device window with the monitor tool active. The monitor tool has several controls for each channel. Pop-up menu selects the channel to monitor. In the case of output channels, the tool allows you to specify the constant value for output and the shape, amplitude and period of periodic output signals. The duration control sets the maximum signal duration displayed; only recent data for the specified duration is kept in the memory.

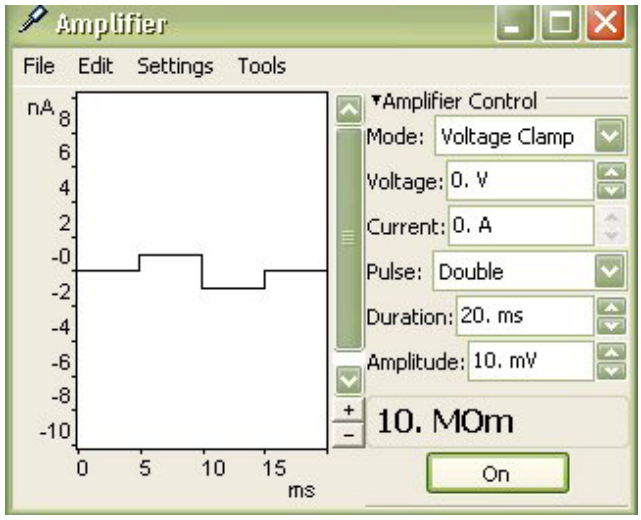


Fig. 17

The duration also sets the initial scale for the horizontal axis. It is possible to zoom in order to view the data, but zooming out beyond the specified duration will reveal empty space beyond recently acquired data. To increase displayed time range, duration needs to be increased. The tool automatically sets sampling frequency so that there are enough data points to display.

The menu of the channel monitor tool contains the following additional commands:

- ***Add Input:*** Adds another input channel to display. The tool can show up to four input channels simultaneously, given the device has that many channels. The trace for each monitored input will have a different color.
- ***Remove Input:*** Removes the last input channel.
- ***Add output:*** Adds another output channel. The tool can control up to two output channels. The output data will also be displayed in the data area in a unique color for each channel.
- ***Remove Input:*** Removes the last output channel.
- ***Separate axes:*** This is a toggle command. If selected, each trace is shown with separate vertical axes. If unselected, the traces are grouped according to the units.

Digital Dashboard

The Digital Dashboard tool works with any device that has digital output channels. Fig. 18 shows the tool in a floating window. This tool allows you to set and clear individual bits of digital output manually. The menu on top of the tool allows you to select any of the available digital output channels. The tool also allows you to simultaneously display all bits that were

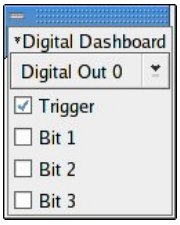


Fig. 18

given a name. Note that this tool does not have an “On” button. The tool performs its function instantaneously and does not take over control of the device for an extended period of time. This tool also does not use the data display area of the window.

Chapter 5: Data Analysis

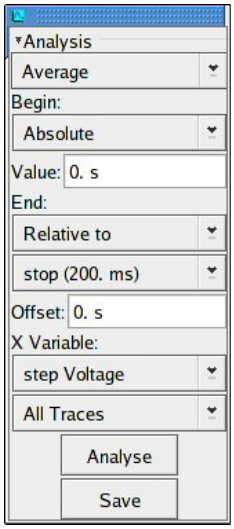


Fig. 19

The program can analyze recorded data and extract numerical characteristics, such as average values. The analysis can be performed interactively on data recorded previously, or automatically during recording at the end of every iteration of the protocol. This chapter describes mostly the interactive analysis. Setting up the protocol for automated real time analysis is described in the chapter describing the experimental protocol.

The interactive analysis is started from the plot window displaying recorded traces. Analysis can be done on any collection of traces but usually it makes sense to analyze all traces from one channel belonging to a single recording. If the protocol has a single input channel you can double click on the recording in the file window to create the traces graph window. Select “analyze” from the “Data” menu to add interactive controls to the window. The analysis control panel behaves similarly to a device tool. It has a pop-up menu that can be accessed by clicking on the panel title. The menu allows you to close controls and to separate the panel into a floating window. Fig.19 shows the analysis control panel in a floating window. The selector on the top specifies the type of analysis to perform. The choices may include common operations such as average, minimum and maximum, and more specialized operations that may be included in some configurations.

Analysis Controls

Begin: & End: Controls specify a range of data for analysis. A selector specifies how the limit is determined. The “Absolute” selection makes the limit an absolute value as displayed by the horizontal axis. The “Begin:” limit on Fig.19 is specified using this setting. Selecting “Relative to” allows you to select the start of data recording, end of recording, or any event that is common for all the traces on graph. In this case the limit is specified as an offset from the selected reference point. The “End:” limit on the Fig.19 is specified using this setting. The numeric value for the limit can be entered manually into the text entry field, or it can be selected by dragging the mouse cursor over the graph. When dragging, both limits are selected. To change only a single limit, click on the graph near the limit while holding the “Shift” key on the computer keyboard, and drag the mouse to select the range. The selected data range will be highlighted.

X variable: Allows you to specify an independent variable for plotting analysis results. If the graph contains multiple traces you have a choice named “Trace number”, which is a sequential number of the trace. This sequential number is the order the trace was added to the graph, and not the order it was recorded. If the graph was created by double-clicking the recording item, this number is the same as the recording order. You might have other choices if the output or timing values vary during protocol execution. Note that these values have to be present in protocols used for all the traces on the graph. If you add a trace obtained with a completely unrelated protocol, it may prohibit these extra choices. Usually analysis is performed once on every trace on the plot, and the X variable relates each point of

the results to the trace that produced the result. There is one exception to this: if “Resampled” is selected by “X variable” control. In this case each resulting trace is derived from the analysis of a single trace. During the analysis each trace is divided into smaller portions and analysis is performed on each portion. The portion length is determined by the distance between the “Begin” and “End” controls. The start of the first portion is selected by the “Begin” control. The independent variable of the resulting trace is a starting horizontal coordinate of each analyzed portion. The analysis is performed from the point selected by the “Begin” control until the end of the trace.

The group of controls below the “X variable” is specific to the analysis function chosen. In simple cases like average value calculation it will be just a selector for the trace to be analyzed. This control also allows you to analyze all the traces in a single run. Other analysis functions may provide some additional parameters, for instance a number of exponential terms for exponential curve fitting. Some functions may require selection of several traces, as in the case where the ratio of two signals is used in analysis, for example.

The analysis is performed when you click the ”Analyze” button. On the first time, a new plot window for the analysis results will be created. You can change some analysis settings and click the “Analyze” button again. The results will be updated in the resulting plot. Finally, the resulting data can be saved into a file by clicking the “Save” button. A new file named “Analysis results” will be created and the resulting trace will be placed into a data item inside this file. You can save the results inside this new file or move the resulting item to the file containing the original data by dragging and dropping with your mouse. The latter approach allows you to store the analysis results together with recorded data in the same way as it would be done by an automated real time analysis.

Extending Analysis with Python Language

The program allows you to use routine analysis functions written in the Python programming language. You can learn about Python at http://www.python.org. You need to download a specific version of Python - currently it is version 2.4. If you have a different version of Python preinstalled on your computer, you will need to download and install the required 2.4 version. On MS Windows the directory where Python is installed will be included in the search path so that the program can locate the Python DLL file. The program looks for Python scripts in text files located inside the “Python” folder. The analysis functions will be located within the “Analysis” folder inside the “Python” folder. The analysis functions are regular Python modules but can use all facilities of the language, such as numeric and scientific libraries. The only requirement to analysis extension is to have a function named “Analyze” that will be called by the program when data analysis is performed. Below is a simple example of Python analysis extension that calculates the maximum of the data trace:

```
from Synapter import * def Analyze(data):
val = max(data)
return Value(‘Max Value’, val*data.YScale, data.YUnits)
```

The first line makes known all the data types provided by the program to communicate with Python. The second line defines the Python function that will be called by the system during analysis. The argument of this function, called “data”, will contain a sequence of data values from the trace to be analyzed. The third line does the actual analysis by applying the built-in Python function “max” to the data sequence and stores the result in numeric variable named “val”. The last line wraps the result into a special data type, which is returned back to the program. The first argu-

ment gives a name to the result. This name "Max Value" will be used by the program. The second argument is a floating number that contains the result. Note that the result is multiplied by a vertical scaling property of the data sequence (data..YScale). This is because the system provides the data sequence in the original format as recorded from input interface, which may be integer and not floating point numbers. The last argument provides the physical units of the result. In this case the result has the same units as the "data.YUnits" property of data.

Structure of Python Module

Each extension analysis function is represented by a separate Python module, that is a text file located in the "Python Analysis" folder. In order to access data, the module will use the statement "from Synapter import *".

Module Attributes

The module can specify several attributes as a top level variable definition that will describe the extension so that the system can correctly represent the extension the same way as it represents built-in analysis operations. Any omitted attributes will result in default behavior.

Name

String value giving the name of the performed function. This name will be included into the function selector both in the analysis panel and in the protocol analysis setup. If this attribute is omitted the base name of the file containing the extension will be used.

VectorInputs

This attribute specifies the names of data arguments supplied to the analysis function. This attribute can be specified as a string value or a list of string values. For each string value the system will create a separate trace selector in an interactive analysis panel or input channel selector in a protocol analysis setup. If this attribute is omitted the "Analyze" function of the module will take single vector input as its first argument and this input will be named "Input data".

ScalarInputs

This attribute specifies the names of analysis parameters, each represented by a floating point number. This attribute can be specified as a single string value or a list of string values. For each string value the system will create a separate field to edit a floating point number in an interactive analysis panel or numeric item in a protocol analysis setup. If this attribute is omitted the "Analyze" function of the module will not list any scalar arguments.

Analyze Function

The module defines a function named Analyze. The function can take one or two arguments. The first argument is used to pass vector data for analysis as described by the *VectorInputs* argument. If the *VectorInputs* argument is absent, the first argument will be a data sequence of type Synapter.Data, otherwise the first argument will be a dictionary with keys being strings taken from *VectorInputs*, and values of type *Synapter.Data*. The second argument is used to pass scalar parameters for analysis as described by *ScalarInputs* attribute. If *ScalarInputs* is absent, the function will not have the second argument. Otherwise, the second argument will be a dictionary with keys being strings taken from *ScalarInputs*, and values of type *Synapter.Value*. The function will return a single object of type Synapter. Value, or a list of such objects. In case of multiple objects, the program will place values from each object

into a separate trace item. If the function cannot complete the analysis for some reason, such as arithmetic overflow, the function can raise a Python exception. The description of the exception will be given to the user in a message box. If an exception is raised during analysis performed as a part of recording protocol, the execution of the protocol will be stopped immediately.

Data Types

Value

Value is an object used to pass scalar parameters into the analysis function and return analysis results. Python can create new objects of this type using constructor *Value(Name, Value, Units)*, where the arguments describe the following attributes. Attributes:

Name - String giving a name of the parameter.

Value - Floating point number

Units - String giving the physical units for the parameter.

Data

Data is a sequence of numeric values used to pass trace data to the analysis function. As with other Python sequence objects, its elements can be accessed by index and iterated over the range. The Python analysis function does not need to create an object of this type by itself; you can use standard sequence types and types from numeric extensions libraries for internal vector computations. All attributes of data are for read only and cannot be modified:

Name - String value giving a name to data.

YUnits - String value giving the physical units of data.

YScale - Floating point number giving the multiplier for data values to convert into values measured in units given in YUnits attribute. This property is required for data that were recorded as integer values from an analog interface, while actual physical value is obtained by scaling.

XUnits - For data recorded with constant samplng interval this gives the name of physical units of such interval, usually seconds.

XScale - For data recorded with constant sampling interval, this specifies a distance between adjacent data points in units given by XUnits property.

LeftX - Gives the X value for the first data point.

Serial Port Driver

The serial port device driver allows you to control external equipment by sending text commands through a serial port. The driver provides virtual channels that can be used in your recording protocols similar to other input and output channels.

Installation

Place the serial port driver inside the “Devices” folder. The next time the program starts the Devices menu of the main window will contain new devices for each serial port available in your system. Note that the program cannot use serial ports that are under the control of other applications. If some of your ports are not listed in the menu, stop other programs that might use serial ports and restart the system.

Setup

Select a serial device from the Device menu of the program main window. This will open a new device window. The first time the device opens the Settings dialog will open automatically. If you need to change the setup, open the dialog by selecting “Settings” from the Settings menu in the device window. Initially, the serial port setup will contain only one item in the properties list on the left of the dialog: Channels. Here you can specify the usual settings of the serial port. Consult your equipment manual to find the correct settings for the serial port. The equipment manual should also list available text commands and their functions. To include a carriage return symbol into the command text use “\r” (pair of symbols) during editing of the properties. On the bottom of the Channels properties panel you can find the button “Add channel”. Click this button to create a new virtual channel. All virtual channels are listed on the left list inside the Channels item. Select the virtual channel name in the list to edit its properties. The properties of the virtual channel are similar to the properties of analog output channels of your Digital/Analog interface devices. The main difference between virtual and analog channels is that the virtual channel’s properties specify the text command while the analog channel specifies the voltage value of the output. If “discrete” scaling is selected for virtual channel the command is selected from the lookup table. The closest value in the left column not exceeding the requested value will be found to send the corresponding text through the serial port. With “normal” scaling the program will apply an offset and multiplier to the requested value and convert the resulting value into a text command according to format specification given in the Command property. The command can be any text that includes the single number conversion command - either %g for general scientific representation that might include mantissa and exponential, %f for floating point representation, and %d for integer representation. Field width and precision can be specified optionally by including two numbers between the % sign and the format letter similar to this example: %6.3f. Note: To include the percent character in the command string precede it with the backslash character.

Usage

If the serial port device window is opened, all virtual channels that are defined in the setup are available for usage in the protocols. Note that the timing of the events controlling the serial port channels is derived from the computer internal clock, which might not be as precise as the dedicated clock inside the interfaces and measuring instruments. The program attempts to maintain the timing of the serial port event with 50 ms precision, but this may fail in the event of heavy load on the computer processing unit CPU.

The serial port device window provides controls to interactively apply values to the virtual channels. For each channel there is a separate control that allows either direct entry of output values with normal scaling, or selector of the

value for channels with discrete scaling. To send the corresponding command, press the small button next to the value control.

The device window can also show a simple terminal to send arbitrary text to the serial port and to monitor the response from the serial port. To use the terminal select “Show terminal” from the Settings menu. Type the text in the lower edit field of the terminal. The text is sent to the serial port only after you press the ENTER key on the keyboard. The characters received from the serial port are displayed in the upper part of the terminal window. To remove received text from the terminal display, select “Clear input” from the Settings menu.

Bioscience Tools, 4527 52nd Street San Diego CA 92115
Ph: 877-853-9755, Fax: 619-582-0919
www.biosciencetools.com