Multilab Software User Guide

For use with

MultiLogPRO

July 2012

Released with MultiLab v1.6

Fourier Systems Ltd.
16 Hamelacha St., POB 11681, Rosh Ha‘ayin 48091,
Tel: +972-3-901-4849, Fax: +972-3-901-4999
www.FourierEdu.com

EXPERIENCE SCIENCE!

Contents

Introduction

MultiLab is a comprehensive software program that supports the full range of Fourier Systems data loggers. It provides everything you need in order to collect data, display the data in graphs, meters and tables, analyze the data with sophisticated analysis tools and even view online or recorded video movies of the actual experiment.

MultiLab includes four displays: Graph, Table, Video, and a navigation display called the Data Map. You can view all four displays simultaneously or view any combination of the four.

Unique video and audio features allow students to view online or recorded movies. Using a Web cam, students can film their experiment process, while the experiment data is displayed in graph, or meter format. Students can participate in e-learning projects, since MultiLab allows them to produce complete multimedia lab reports with real-time, synchronized annotated graphs and video.

Another useful feature is the Workbook tool. This gives students a step-by-step preview of the Lab activity to be conducted, and then automatically configures the MultiLab program and sets up the data logger so that the student can begin collecting the data. You can open an existing Workbook or create a new one.

MultiLab also includes a Video Motion Analyzer module that enables you to capture position and time from video movies and analyze the data with MultiLab's analysis tools.

MultiLab Software

Installing the Software

MultiLab Installation on Windows computer

Note to Windows users: MultiLab needs to modify its folder when running. Ask your system administrator for permission to gain access to all files in the MultiLab folder.

Instructions for installing and uninstalling MultiLab on your computer are provided below.

Installing MultiLab on Windows XP/Vista/7

1. Close all open programs.
2. Insert the CD labeled **MultiLab** into your computer's CD/DVD drive.
3. The Install Wizard will open. Click **Next** to continue the installation.
4. Select the option for agreeing to the terms of the MultiLab license agreement. Click **Next**.
5. Enter user name and company name in the Customer Information window.
6. Select the option regarding for which user the application should be installed. Click **Next**.
7. The default folder where MultiLab will be installed is C:\Program Files\Fourier Systems\MultiLab\. Click **Browse** to change this folder location, or click **Next**.
8. The default folder where the MultiLab data files will be stored is My Documents\Multilab Data. Click **Browse** to change this folder location, or click **Next**.
9. MultiLab installation will proceed, followed by installation of associated drivers, Windows Encoders and Quicktime ®.
10. When installation is complete the Install Wizard prompts you to restart the computer. It is recommended to restart the computer now.

Installing USB Drivers

The drivers for the USB Link are installed automatically as part of the installation process of Multilab

Uninstalling MultiLab

Note: Uninstalling MultiLab removes program files only. Project files and worksheets you created or modified in MultiLab will remain on your system and there is no need to create backups.

To uninstall the software on Windows XP:

1. From the **Start** menu select **Settings** then click **Control Panel**.
2. Double click **Add or Remove Programs**.
3. Select **MultiLab** from the list of programs.
4. Click **Remove**.
5. In the following dialog box, click **Yes** to confirm that you want to remove MultiLab.
6. When uninstall is complete you will be prompted to restart the computer. Select **Yes**.

To uninstall the software on Windows Vista/7:

1. From the **Start** menu select **Control Panel**.
2. Select **Programs > Uninstall a program**.
3. Select **MultiLab** from the list of programs and click **Uninstall** from the top of the window.
4. In the following dialog box, click **Yes** to confirm that you want to remove MultiLab.

MultiLab Installation on Mac OS computer

1. Insert the CD into your CD drive.
2. Open the CD drive folder and double-click the **MultiLab Installer** icon, then follow the on-screen instructions to complete the installation process.

Installing USB Drivers

The drivers for the USB Link are installed automatically as part of the installation process of Multilab

Overview

MultiLab On-screen Layout

MultiLab is a comprehensive program that provides you with everything you need in order to collect data from Fourier data loggers. You may display the data in graphs, meters and tables, analyze it with sophisticated analysis tools and view online or recorded video movies of the actual experiment.

The Video Motion Analyzer module enables you to capture position and time from video movies and analyze the data with MultiLab's analysis tools

The program includes four windows: A graph window, table window, video window and a navigation window called the Data Map. You can display all four windows simultaneously or any combination of the four.

The most commonly used tools and commands are displayed on three toolbars. Tools that relate to all aspects of the program and tools that control the data logger are located in the main (upper) toolbar. Tools specific to the graphs are located on the graph toolbar and tools specific to the tables are located on the table toolbar.

Another useful feature is the Workbook. A workbook is a lab activity that gives the student a step-by-step preview of the activity and then automatically configures the MultiLab program and the sets up the data logger so that the student can begin collecting the data. You can open an existing workbook or create a new one from the Workbook menu.

MultiLab Window Layout

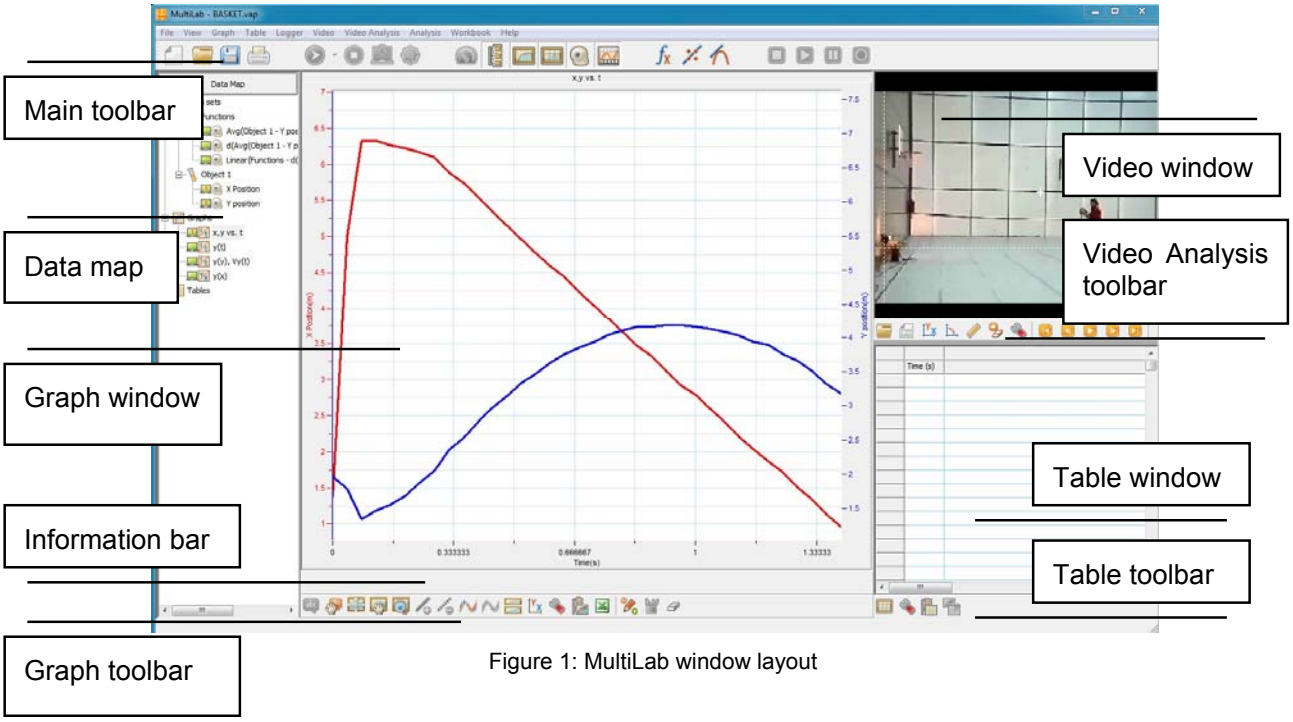


Figure 1: MultiLab window layout

Working with Projects

Every time you start a new experiment, MultiLab automatically creates a new project file. All the information you collect and process for a given experiment is stored in a single project file. Each of these files contain all the data sets you collect with the data logger, the analysis functions you've processed, the video files you've recorded, specific graphs and tables you've created, and the MultiLab settings for the experiment.

Note: All data sets in a single project must be with the same sampling rate.

Getting Started

Set up a Recording Session

Prepare the MultiLogPRO

Connect MultiLogPRO to the PC.

Turn on MultiLogPRO.

Launch the MultiLab software.

Enable Video Recording

Note: *Do not* plug in the PC camera before you have installed the camera driver.

To display the video window and enable online video and video recording:

Connect the camera to the PC

Click **Video** on the menu bar, then click **Enable video**

MultiLab will remember the video settings until the next time you change them.

Note: if more than one video source is connected to your computer MultiLab will use the default source. To select another source click **Video** on the menu bar and click **Select video device**, then select the desired source from the drop list that opens and click **OK.**

Click **Video**  on the main toolbar to display the video window.

Point the camera to the experiment setup.

Setup the MultiLogPRO

Click the **Setup Wizard** button on the main toolbar  .

Follow the instructions in the **Setup Wizard** (see page 29).

Start Recording

Click **Run**  on the toolbar to start recording.

- For the MultiLogPRO, if the recording rate is 100 measurements per second or less, MultiLab automatically opens a graph window displaying the data in real-time, plotting it on the graph as it is being recorded. If the recording rate is higher than 100/s, the data will be downloaded and displayed automatically once the data recording is finished.

You can stop recording anytime by clicking **Stop**  on the toolbar.

Data Recording Options

To set the behavior of the data display when you start a new recording session, click on the **down arrow** next to the **Run** button  , and select one of the following:

Single Measurement

MultiLab will open a new project file every time you start a new recording session.

Replace

MultiLab will display the new data set in place of the old one. The project's old data sets will still be available in the same project file. They will be listed in the Data Map and you can add them to the display at any time.

Add

MultiLab will add the new data set to the graph in addition to the old ones.

Note: A maximum of eight data sets can be displayed on the graph at the same time.

Manual Sampling

Use this mode for:

- Recordings or measurements that are not related to time.

- Situations in which you have to stop recording data after each sample obtained, in order to change your location, or any other logging parameter (**Note:** During the experiment *no changes* can be made to the MultiLogPRO's configuration).

To start an experiment using manual data logging, set the rate to manual and click **Run** once to start the data recording, and then press the **Enter** button each time you want to collect a sample.

You can also perform manual logging via MultiLogPRO (see page **Error! Bookmark not defined.**).

Online Video

MultiLab enables you to view online video of your experiment along with the data display, as well as to record video movies of the experiment and to replay them simultaneously with the data graph. You can also add voice comments to the video recordings.

To work with the video window your system should be equipped with a video camera.

To be able to record and play voice comments your system should be equipped with a sound card, a microphone and a speaker.

Online Video

The online video will start playing when you open the Video window. Click **Video** on the main toolbar to display the video window and enable video

Recording

From the **Video** menu select **Enable video**.

Click **Record** on the right-hand side of the toolbar to start video recording.

Click **Run** to start data recording.

Once the data logging has finished, click **Stop Video Recording** .

Click **Replay** to replay both the data graphing process the video recording simultaneously.

Note: Saving the file will automatically save the video as well.

Replaying an Experiment

To simultaneously replay the video and the data graphing process of a saved file:

Open the file.

Display the data on the graph.

Click **Replay** 

Video and Audio Properties

To set the video properties click **Video** on the menu bar, then click **Video properties** to open the video properties dialog box:

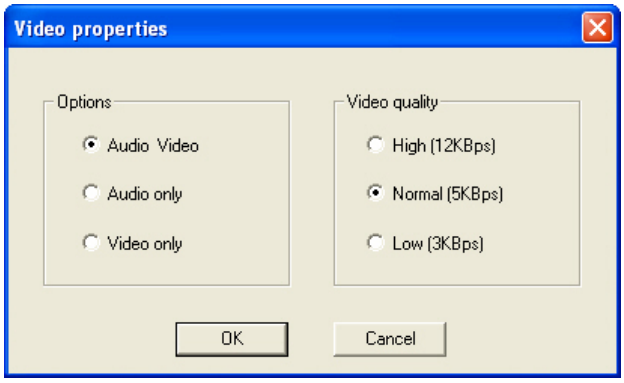


Figure 2: Video properties dialog box

Changing Video Quality

The video default quality is **Normal**; click another option to select either higher quality (12 KBps) or lower quality (3 KBps). Changing the video quality not only affects the image quality, but the recorded file space and the computer's performance as well. A higher video quality means more disk space is used, which can slow down the computer's performance.

Audio Visual Options

The default option is **Audio_video**. That means that both video and audio are enabled and you can record voice comments along with the video. If you plan to record video only, select the **Video only** option. If you plan to record voice comments only select the **Audio only** option.

Download Data from the MultiLogPRO

Whenever data is received from the MultiLogPRO, it is accumulated and displayed automatically by MultiLab. There are two modes of communication: Online and Post-Experiment.

Online communication

When MultiLogPRO is connected to the PC and is programmed to run at sampling rates of up to 100/s, MultiLogPRO transmits each data sample immediately, as it is recorded, to the PC. The software thus displays the data in real-time in both the graph window and the table window.

When MultiLogPRO is connected to the PC and programmed to run at a sampling rate faster than 100/s, data is accumulated in MultiLogPRO's internal memory but is not transmitted directly to the PC until the recording period has ended. Once the recording has ended, the data is automatically downloaded to the PC and displayed.

Off-line data logging

To download data that was recorded offline, or while MultiLogPRO was not connected to a PC, connect MultiLogPRO to the computer, run the MultiLab program and click **Download**  on the toolbar. This will initiate the Post-Experiment Data Transfer communication mode. Once the transfer is complete, the data will be displayed automatically in the graph window and in the table window. If there are several experiments stored in the MultiLogPRO, the first download will bring up the most recent experiment; the second download will bring up the earliest file, the third download will bring up the second earliest file, and so on.

To download a particular experiment, choose **Selective Download** from the **Logger** menu, then select the experiment's number in the Download dialog box.

Click **Cancel** in the Download progress window at any time to stop downloading the data.

The Timer Module

MultiLab enables you to measure timing events with the data logger's Timer module (see page **Error! Bookmark not defined.**).

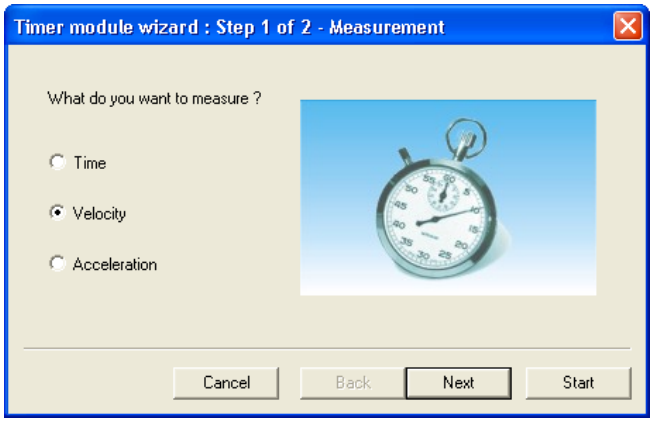
To use the photogates together with other sensors operate the data logger and then analyze the photogates data with the aid of MultiLab's Timing Wizard (see page 65).

Working with the Timer Module Wizard

Connect the data logger to the computer

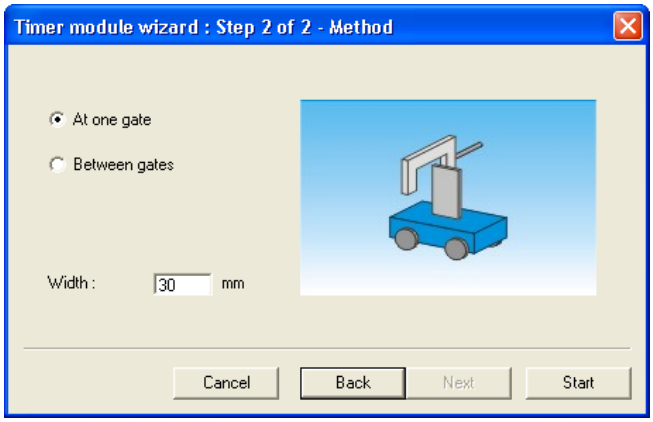
Connect one photogate to input 1 of the data logger or two photogates to inputs 1 and 2 (according to the event method)

Click **Logger** on the menu bar, and then click **Timer Module Wizard** to open the Timer module wizard:



Click an option to select measurement: **Time**, **Velocity** or **Acceleration**

Click **Next** to move to step 2 of the wizard:



Click an option to select the measuring method

If required enter the body's width in mm (an integer between 0 to 59), or the distance between the gates in cm (an integer between 0 to 99) in the appropriate edit box (in velocity and acceleration measurements only)

Click **Start** to enter to a timing standby mode

Timing begins each time a body blocks the photogate in input 1 and ends when unblocking the photogate in input 1 or input 2 (according to the event method). MultiLab displays the results in a bar graph and in the table.

You can repeat as many measurements as you want. After each event MultiLab adds the results as a new bar in the graph and as a new raw in the table.

To exit the timing mode click Stop  on the main toolbar.

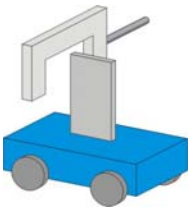
Measuring Methods

The Timing Wizard offers you various methods of analyzing the different measurements. In some measurements you will be asked to enter the dimension of the moving body, or the distance between the two photogates to allow for the calculation of velocity and acceleration.

The methods depend on the selected measurement:

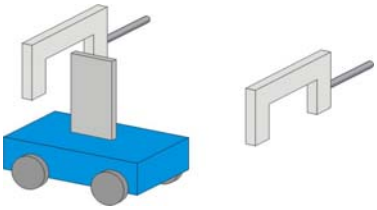
Time

- At one gate



Measures the time it takes the body to cross the photogate (between blocking and unblocking the infrared beam)

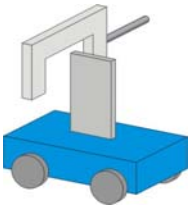
- Between gates



Measures the time it takes the body to move from one photogate to the second photogate (between blocking the first and blocking the second infrared beams)

Velocity

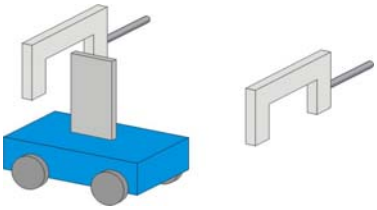
- At one gate



Measures the time it takes the body to cross the photogate (between blocking and unblocking the infrared beam) and returns the velocity.

You should enter the body's width in mm

- Between gates

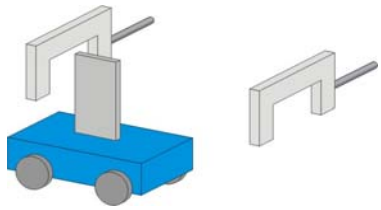


Measures the time it takes the body to move from one photogate to the second photogate (between blocking the first and blocking the second infrared beams) and returns the average velocity.

You should enter the distance between gates in cm.

Acceleration

- Between gates



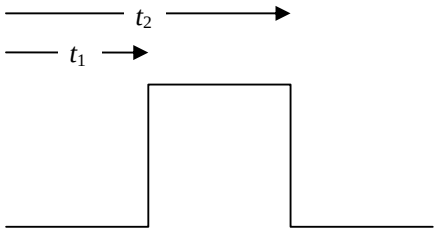
Measures the crossing time at the first gate, the time it takes the body to move from one gate to the second gate and the crossing time at the second gate and returns the average acceleration.

You should enter the body's width in mm.

Time Schemes and Calculations

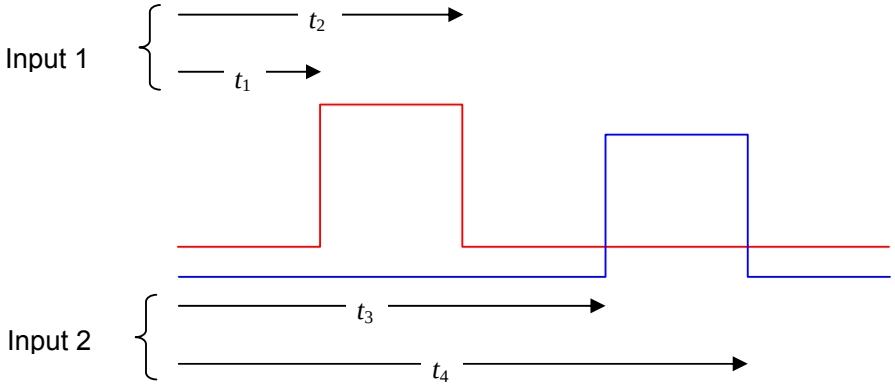
Time measurements

At one gate



Result: $\Delta t = t_2 - t_1$

Between gates

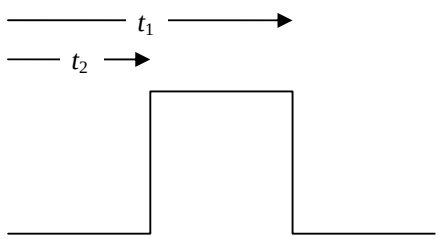


Result: $\Delta t = t_3 - t_1$

VELOCITY

At one gate

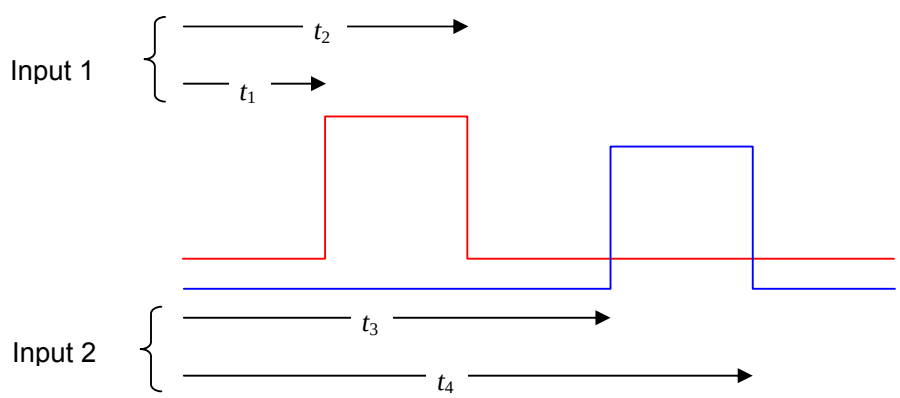
Required parameters: w – The body's width



Result: $v = \dfrac{w}{\Delta t}; \quad \Delta t = t_2 - t_1$

Between gates

Required parameters: L – The distance between gates

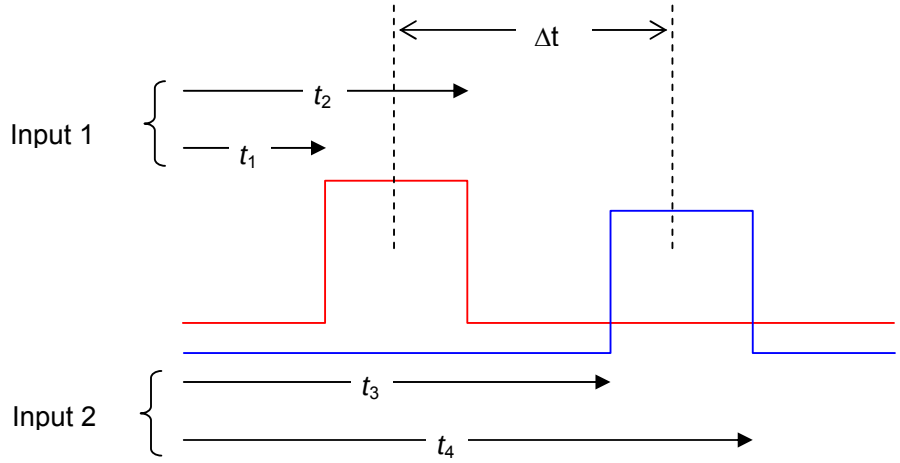


Result: $v = \dfrac{L}{\Delta t}; \quad \Delta t = t_3 - t_1$

ACCELERATION

Between gates

Required parameters: w – The body's width



$$v_1 = \frac{w}{t_2 - t_1}; \quad v_2 = \frac{w}{t_4 - t_3}$$

Result:

$$\Delta t = \frac{t_4 + t_3 - t_2 - t_1}{2}$$

$$a = \frac{v_2 - v_1}{\Delta t}$$

Save Data

Click **Save**  on the main toolbar to save your project. This will save all the data sets, graphs, tables and video movies under one project file.

Saving the project will also save any special formatting and scaling you did.

If you made any changes to a previously saved project, click **Save** to update the saved file or select **Save as…** from the file menu to save it under another name.

Note: To delete a specific data set, graph or table from the project, use the Data Map.

To remove unwanted data from a specific data set, apply the crop tool.

Open a File

Click **Open**  on the main toolbar.

Navigate to the folder in which the project is stored.

Double click the file name to open the project.

MultiLab opens the project and displays the first graph on the graph list. If the project does not include saved graphs, the file opens with an empty graph window. Use the Data Map (see page 25) to display the desired data set.

Create a New Project

There are three ways to create a new project:

Open the MultiLab program, which will open a new file each time.

When working in Single Measurement mode, a new project is opened every time you click on the **Run** button to start a new recording

Any time you click **New** button  on the toolbar.

Import Data

Any file that is in comma separated values text format (CSV) can be imported into MultiLab.

To import a CSV file:

Click **File** on the menu bar, then click **Import CSV file**.

In the dialog that opens, next to **Look in**, navigate to the drive and folder that contains the CSV file.

Select the file.

Click **Open**.

Tips:

To create a text file in a spreadsheet:

Open a new spreadsheet.

Enter your data according to the following rules:

The first row should contain headers. Each header includes the name of the data set and units in brackets, e.g. Distance (m).

The first column should be the time. The time interval between successive rows must match the time intervals accepted by MultiLab. You can export MultiLab files to Excel to learn about these time formats.

See for example the table below:

	A	B	C
1	t(s)	x(m)	y(m)
2	0	0	5
3	0.1	0.95	4.05
4	0.2	1.8	3.2
5	0.3	2.55	2.45
6	0.4	3.2	1.8
7	0.5	3.75	1.25
8	0.6	4.2	0.8

On the **File** menu, click **Save As**.

In the **File name** box, type a name for the workbook.

In the **Save as type** list, click the **CSV** format.

Click **Save**.

To import files that were previously exported from MultiLab open MultiLab and import the file as described above as they are already in CSV format.

Print

Print a Graph

Click **Print**  on the main toolbar.

Select the **Graph 1** option (when in split graph mode you can choose between Graph 1 and Graph 2).

Click **Print** to open the print dialog box.

Click **OK**.

MultiLab will print exactly what you see in the graph display.

Print a Table

Click **Print**  on the main toolbar.

Select the **Table** option.

If you want to print only a specific range, uncheck the **Print all data** check box and type the desired row numbers into the **To** and **From** edit boxes

Click **Print** to open the print dialog box.

Click **OK**.

Viewing the Data

Display Options

The MultiLab program screen consists of four parts: Graph window, table window, video window and Data Map window. You can display all four parts simultaneously (the default view) or any combination of the four.

The graph window is the main window by default and is and displayed in the center of the application window. To specify other window as the main window:

Click **View** on the menu bar, and then click **View selection** to open the View selection dialog box:

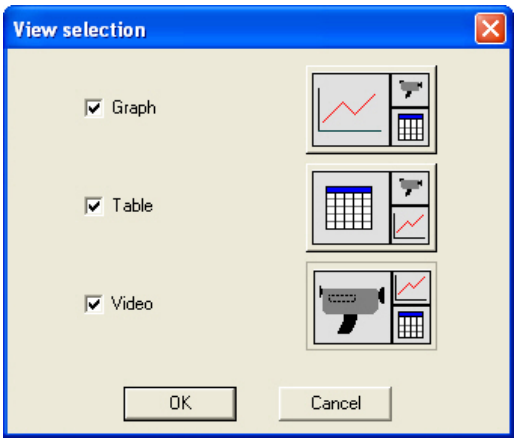


Figure 3: View selection dialog box

Check the checkbox next to any window you'd like to include in the view.

Click the window display type you want.

Click **OK**.

In addition to these sections, you have the option to display an on-screen meter for each of the sensors (see page 25).

Graph Display

Click **Graph**  to display or hide the graph. The default graph display is the data set or sets plotted vs. time, but you can change the X-axis to represent any of the individual data sets (see page 20).

The graph usually displays all the data sets of a given recording, but you can use the Data Map to remove one or more of the sets from the graph (see page 25).

In order to keep the graph clear and simple, only two Y-axes are shown on the graph at once. If there are three curves in the graph, one of the Y-axes is hidden. To make this axis visible, select the corresponding plot with the cursor (see section 0 below).

You can identify the Y-axis by its color, which matches the plot color.

Split Graph View

MultiLab enables you to display your data in two separate graphs within the graph window.

Click **Split graph** on the graph toolbar to split the graph window into two separate graphs.

Click **Edit graph** on the graph toolbar to open the Edit graph dialog box.

Choose which data sets to display on each of the graphs (or use the Data Map to do so – see page 25).

To return to the single graph display, click **Split graph** a second time.

The Cursor

You can display up to two cursors on the graph simultaneously.

Use the first cursor to display individual data recording values, to select a curve or to reveal the hidden Y-axis.

Use two cursors to display the difference between two coordinate values or to select a range of data points.

To display the first cursor:

Double click on an individual data point or click **1st Cursor** on the graph toolbar. You can drag the cursor with the mouse onto any other point on the plot, or onto a different plot. For finer cursor movements use the forward and backward keys on the keyboard.

The coordinate values of the selected point will appear in the information bar at the bottom of the graph window.

To display the second cursor:

Double click again anywhere on the graph area or click **2nd Cursor** on the graph toolbar.

The information bar will now display the difference between the two coordinate values. If the x-axis is time 1/dt will also be displayed.

To remove the cursors:

Double click anywhere on the graph area, or click **1st Cursor** a second time.

To remove the 2nd cursor:

Click **2nd Cursor** a second time.

To display the cursors in split graph mode:

To display the cursors on the upper graph, use the same method as for single graph mode.

To display the cursors on the lower graph, you must first remove them from the upper graph and then double click anywhere on the lower graph to display the first cursor. Double click a second time to display the second cursor, and double click a third time to remove the cursors.

Zooming

To zoom in to a specific data point

Select the point with the cursor (see above).

Go to **Graph > Zoom in** on the main menu.

To reverse the operation, go to **Graph > Zoom out** on the main menu or click **Autoscale** on the graph toolbar.

To zoom in to a range

Select the range with both cursors.

Go to **Graph > Zoom in** on the main menu.

To reverse the operation, go to **Graph > Zoom out** on the main menu or click **Autoscale** on the graph toolbar.

To zoom in to a specific area

Click **Zoom to selection** on the graph toolbar and drag the cursor diagonally to select the area you want to magnify. Release the mouse button to zoom in to the selected area.

Click **Zoom to selection** a second time to disable the zoom tool.

Autoscale

Click **Autoscale** on the graph toolbar to view the full data display, or double click on an axis to auto scale that axis alone.

Manual scaling

Click **Graph properties** on the graph toolbar to open the Graph properties dialog box.

Select the **Scale** tab, and choose the axis you want to scale in the **Select axis** drop down menu.

Uncheck the **Autoscale** check box and enter the new values in the edit box.

Click **OK**.

To manually scale a specific axis, right click on the axis to open its Properties dialog box.

To restore auto scaling click **Autoscale**  .

The Stretch/Compress Axis tool

Move the cursor onto one of the graph axes. The cursor icon changes to the double arrow symbol (↔), indicating that you can stretch or compress the axis scale. Drag the cursor to the desired location. Repeat the procedure for the other axis if necessary.

Double click on the axis to restore auto scaling.

Panning

Use the pan tool after zooming in to see any part of the graph that is outside the zoomed area.

To do this, click **Pan**  on the graph toolbar, then click anywhere on the graph and drag the mouse to view another area.

Click **Pan** a second time to disable the Pan tool.

Edit the Graph

Use the Edit graph dialog box to select which data sets to display on the graph's Y-axis and to change the X-axis from time, to one of the data sets.

Click **Edit graph**  on the graph toolbar to open the Edit graph dialog box:

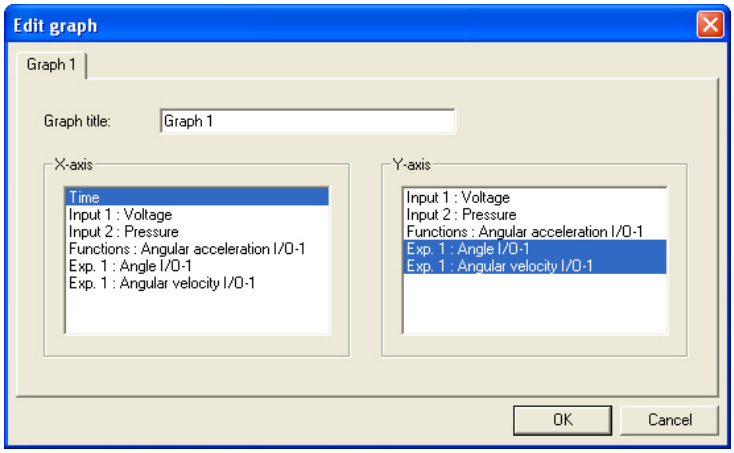


Figure 4: Edit graph dialog box

To select a data set to display on the Y-axis, click on the data set's name in the **Y-axis** list. To display more than one curve, click on the data sets you want.

A list entry that begins with an Experiment (Exp.) number denotes a recorded data set. A list entry that begins with an input number denotes the next recording and will be displayed on the graph the next time you start a recording.

To deselect a data set click on the set a second time.

To select a data set for display on the X-axis, click on the data set's name in the **X-axis** list. You can only select one data set at a time for the X-axis.

Type the title of the graph in the **Graph title** edit box (optional).

Click **OK.**

Format the Graph

You can change the data line's color, style and width. You can also add markers that represent the data points on the graph and format their style and color.

The color of the Y-axis matches the corresponding plot's color and will automatically change with any change made to the color of the corresponding plot.

Click **Graph properties** on the graph toolbar to open the **Graph properties** dialog box.

Select the **Lines** tab, and then select the plot or axis you want to format in the **Select plot** drop list.

From here you can format the line's color, style and width, as well as the markers' color and style. To remove the line or the marker, uncheck the corresponding **Visible** check box.

Click **OK**.

To restore the default formatting, click **Restore default**.

Change the Graph's Units and its Number Format

Click **Graph properties** on the graph toolbar to open the graph properties dialog box

Select the **Units** tab, and then select the plot or axis you want to format in the **select plot** drop list.

Choose the prefix option you want.

Select the desired number of decimal places.

To display numbers in scientific format, check the **Scientific** check box.

Click **OK**.

Add Annotations to the Graph

You can add annotations to the graph. An annotation is always connected to a certain data point.

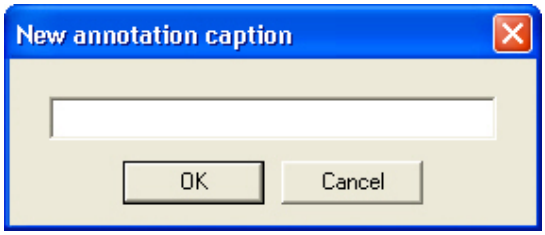
To view annotations:

Click **Graph** on the menu bar and then select **Show annotation** (this option is usually selected by default).

To add an annotation:

Place a cursor on the desired point.

Click **Add new annotation** on the graph toolbar to display the new annotation caption edit box.



Type in the text, and then click **OK**.

To move an annotation:

Click **Move annotation** on the graph toolbar.

Click on the annotation you want to move, then drag if to the new position.

Click **Move annotation** a second time to exit the move annotation mode.

To edit an annotation:

Place a cursor on the point to which the annotation is connected.

Click **Graph** on the menu bar, and then click **Edit annotation**.

Edit your annotation, and then click **OK**.

To delete an annotation:

Place a cursor on the point to which the annotation is connected.

Click **Graph** on the menu bar, then click **Delete annotation**.

To hide all annotations:

Click **Graph** on the menu bar and then click **Show annotation** to uncheck it.

Add a Graph to the Project

MultiLab displays new data in the graph window every time you start a new recording. You can always display previous data using the **Edit graph** dialog box or by double-clicking on the data's icon in the **Data Map.** If you want to save a graph that you created to your project, or to update a saved graph with changes you made, use the **Add to project** tool:

Click **Add to project**  on the graph toolbar.

The Table Display

Click **Table**  to display or to remove the table window.

When you start a new recording, MultiLab displays the new data on the table.

To add or remove columns from the table, use the **Edit table** tool (see below).

Editing Data Values

You can change the value of any data point. When any value is changed, MultiLab automatically creates a copy of the data set and leaves the original data unchanged. To do this:

Select a data cell by clicking on it.

Type in the new value.

Press **Enter** on the keyboard to accept (you can also press Tab or the arrow keys to accept and move to the next cell).

Editing the Table

To add or remove data sets from the table:

Click **Edit table**  on the table toolbar to open the Edit table dialog box:

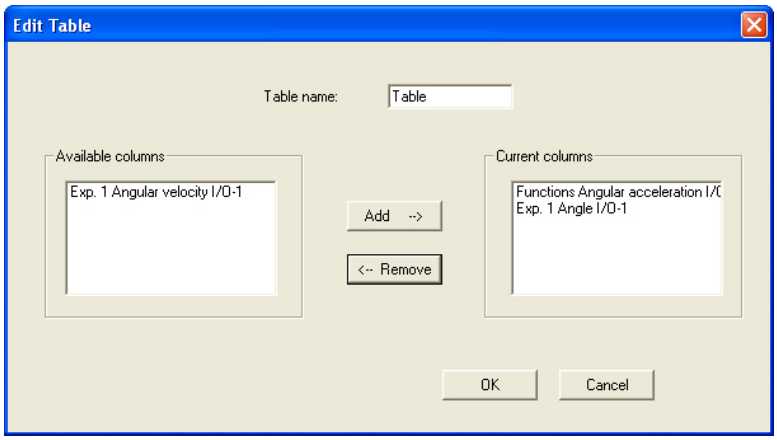


Figure 5: Edit table dialog box

To add columns to the table:

Select one or more of the data sets from the **Available Columns** list.

Click **Add**.

To remove columns from the table:

Select one or more of the data sets from the **Current Columns** list.

Click **Remove**.

Click **OK**.

Formatting the Table

Changing column width

Drag the boundary on the right side of the column heading until the column is the desired width.

Changing row height

Drag the boundary below the row heading until the row is the desired height.

Formatting the fonts

Click **Table properties** on the table toolbar.

Select the **Font** tab.

Format the font, as well as the font style and size.

Click **OK**.

Changing units and number format

Click **Table properties** on the table toolbar.

Select the **Units** tab, and then select the plot you want to format from the **select plot** drop list.

Choose the prefix option you want.

Select the desired number of decimal places.

To display numbers in scientific format, check the **Scientific** check box.

Click **OK**.

Add a Table to the Project

MultiLab displays new data in the table window every time you start a new recording. You can always display previously recorded data using the **Edit table** dialog**.** If you want to save a table that you created to your project, or to update a saved table with changes you made, use the **Add to project** tool:

Click **Add to project**  on the table toolbar.

Meters

MultiLab enables you to view data in meters format on the screen (one meter for each sensor), with up to four meters showing at once. The meters can display live data while MultiLab is recording, or saved data when a saved file is replayed.

When a cursor is displayed, the meter shows the measured values that correspond to the time of the point at which the cursor is positioned.

There are three meter types: analog, bar and digital. The meter's scaling automatically matches the graph's scaling.

To set up the meters:

Click **Meter Setup**  on the main toolbar:

Select the meter type, and the data set to be displayed.

A list entry that begins with a graph number denotes a displayed data set. A list entry that begins with an input number denotes the next recording, and will be displayed on the meter the next time you start a recording.

Repeat this procedure for up to four meters.

To remove the meters click **Meter Setup**  , and click **Remove all**.

Data Map

Click **Data Map**  to display or remove the Data Map.

The data map is a separate window that displays the list of data sets that were recorded or downloaded in the current session, as well as the lists of all the saved

graphs and tables. Use the Data Map to navigate through the available data sets and to keep track of the data that is displayed in the graph window.

Control the Display with the Data Map

The items in the Data Map are sorted into three main categories:

- Data sets (including analysis functions)
- Saved graphs
- Saved tables

Double click on a category to bring up the full list. Double click a second time to collapse the list. You can also use the plus (+) and minus (-) signs next to the icons to expand or collapse the categories.

The Data sets list expands to sub-categories of experiments and functions. To display the complete list of measurements, or the complete list of analysis functions performed on the measurements for any individual experiment, double click the experiment's icon or click the plus sign (+) next to it.

To collapse a list under an individual experiment, double click the experiment's icon or click the minus sign (-) next to it.

To display a data set or a saved graph in the graph window, or to display a table in the table window, double click its icon. Double click a second time to remove it.

You can also use a shortcut menu to display or remove a data set from the graph. Simply right-click an icon, then click **display on graph #1** or **display on graph #2.**

To delete an individual item from the Data Map and from the project, right-click on its icon and click **Delete**. To permanently delete the item, click **Save** on the main toolbar.

To remove all data sets from the Data Map, right-click the Data sets icon and click **Clear All Data**.

Understanding Data Map Icons

Data sets list

Functions – includes all the analysis functions of the project

Individual function – Currently not on display

Individual function – Currently displayed on graph 1

Individual function – Currently displayed on graph 2

Individual function – Currently displayed both on graphs 1 and 2

Function

Experiment – Includes all recorded data of the experiment

Individual data set – Currently not on display

Individual data set – Currently displayed on graph 1

Individual data set – Currently displayed on graph 2

Individual data set – Currently displayed both on graphs 1 and 2

Data recorded without video

Data recorded with video

List of saved graphs

Saved graph – Data sets vs. time

Saved graph – Data sets vs. data set

Graph currently not on display

Graph currently displayed on graph 1

Graph currently displayed on graph 2

Graph currently displayed both on graphs 1 and 2

List of saved tables

Saved table

Table not currently displayed

Table currently displayed

Export Data to Excel

Click **Export to Excel** on the graph toolbar to export the displayed data to Excel. You will be given the option of saving the data in a specific location, and then MultiLab will open a new Excel workbook displaying the data.

Export File Settings

If MultiLab fails to export the data properly, try to change the export file settings:

Click **File** on the menu bar, and then click **Export file settings**.

Check the **Ignore regional settings** check box.

Click **OK**.

Copy the Graph as a Picture

You can copy the graph to the clipboard as a picture and then paste it to other Windows programs such as Word or PowerPoint:

In the Graph menu, click **Copy graph**.

Open the destination file.

In the destination file, right click and select **Paste**.

Programming the Data Logger

Setup

Quick Setup

Use the Setup wizard to guide you through the setup process.

Click the **Setup wizard** icon  on the main toolbar to launch the setup process for the data logger.

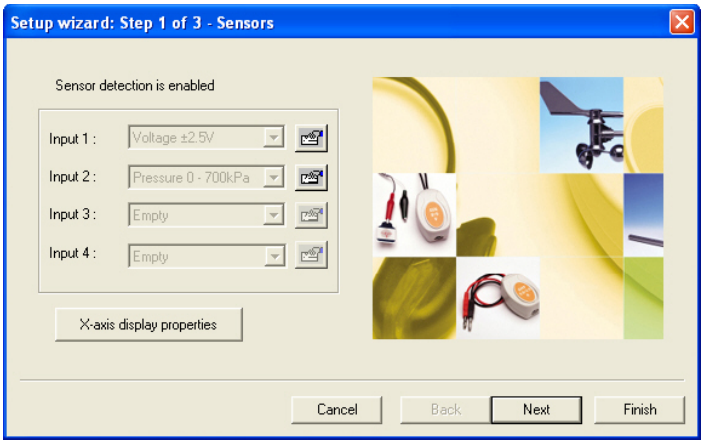


Figure 6: Setup wizard step 1 of 3

The first step is to assign sensors to each input. If you are working with the data logger in Auto ID mode, the sensors are selected automatically as you plug them in to the data logger. Otherwise, you can manually select the sensors you are using from the corresponding input drop down menus.

Click **Next** to move to the second step:

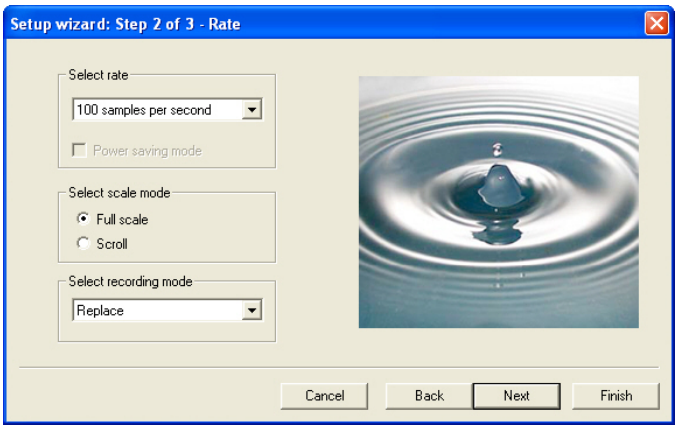


Figure 7: Setup wizard step 2 of 3

Select the desired recording rate in the **Select rate** drop down menu.

Select the **Scroll** option if you want the graph window to scroll as the data recording progresses.

Select the desired data recording mode in the **Select recording mode** drop down menu.

Click **Next** to move to the third step:

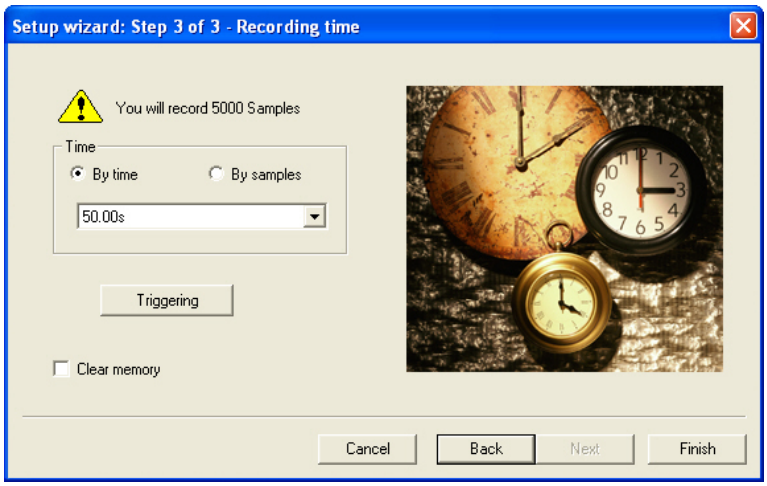


Figure 8: Setup wizard step 3 of 3

Select the desired length of the recording period in the **Select recording time** drop down menu.
The recording period is displayed as a unit of time by default. To display the recording period as the number of recording samples, select the **By samples** option. The number of samples is calculated with the following formula:

Total recording samples = Recording rate x Recording period

To start the data recording only when a specific time or measurement condition has been met click **Triggering** to open the triggering dialog box (see page 33).

Click **Finish** to complete the setup and to send the setup command to the data logger.

Defining Sensor Properties

Some of the sensors can produce more than one measurement. For example, the distance sensor measures distance, but MultiLab also calculates the velocity and acceleration of the measured body in real-time. MultiLab automatically displays the basic measurement of distance, but enables you to display any combination of distance, velocity and acceleration. Other sensors may only give one measurement, but can display the data in different units. For example, the acceleration sensor can display the acceleration in multiples of the gravity acceleration, or in the standard unit m/s^2.

The Distance and Force sensors properties include definition of the positive direction and the distance sensor properties enable online averaging as this sensor is comparatively noisy.

You can define the properties of each sensor individually in the first step of the Setup wizard**,** or by selecting **Calibrate sensors** from the **Logger** menu.

Click **Setup Wizard** on the main toolbar to open the Setup wizard

Click **Properties**  next to the input you want to set.

Click the check boxes next to the desired measurements.

To change more properties, click the corresponding tab at the top of the dialog box.

Click **OK**.

Setting the Zero Point of a Sensor

MultiLab enables you to rescale some sensors and to set the current readings to zero for subsequent loggings. This feature applies to the following sensors:

- Distance
- Force
- Magnetic field
- Light sensors
- Pressure sensors

To set the current readings of a sensor to zero:

Click **Setup Wizard** on the main toolbar to open the Setup Wizard.

Click **Properties** next to the input you want to set.

Click the **Set Zero** tab.

Check the **Set the current reading to zero** checkbox.

Click **OK**.

Click **Finish**

MultiLab takes a single measurement and set the reading to zero.

To change the zero point:

Click **Setup Wizard** on the main toolbar to open the Setup Wizard.

Click **Properties** next to the input you want to set.

Click the **Set Zero** tab.

Click **Reset zero**.

Click **OK**.

Click **Finish**.

To return to the default zero point:

Click **Setup Wizard** on the main toolbar to open the Setup Wizard.

Click **Properties** next to the input you want to set.

Click the **Set Zero** tab.

Uncheck the **Set the current reading to zero** checkbox.

Click **OK**.

Click **Finish**.

Presetting the Display

You may want to define the graph's settings, such as formatting and scaling the graph and selecting the axes, and to define the meters' settings prior to beginning a recording session. You can define the settings of each sensor input individually in the first step of the **Setup Wizard**:

Click **Setup Wizard** on the main toolbar to open the Setup Wizard.

Click **Properties** next to the input you want to set.

Click the **Display properties** tab.

Select a measurement from the **Select measurement** drop down menu.

Select the graph or graphs in which you would like to display the data, from the **Display on graph** drop down menu.

Select a meter in the **Meter type** drop list if you want to display the data in a meter as well as on the graph.

Format the line and markers in the **Plot** section.

If you want a specific scale, uncheck the **Auto scale** checkbox and enter the desired minimum and maximum values of the axis.

Click **OK**.

Presetting the Graph's X-axis

Click **X-Axis display properties** in the first step of the Setup Wizard to open the dialog box:

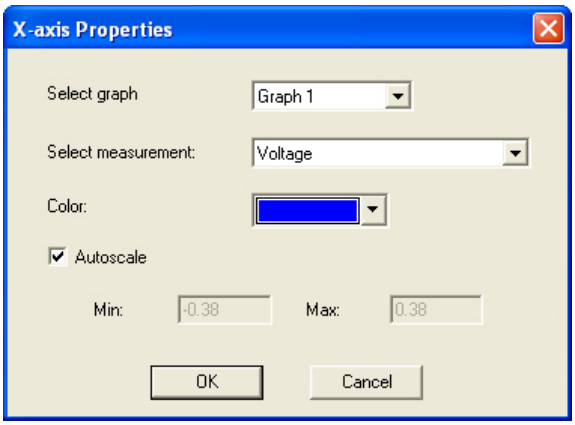


Figure 9: X-axis properties dialog box

Select the graph you wish to modify.

Select the measurement you'd like to display on the X-axis.

If you want a specific scale, uncheck the **Auto scale** check box and enter the minimum and maximum values of the axis.

Click **OK**.

Triggering

To start the data recording only when a specific time or measurement condition has been met, click **Triggering** in step 3 of the Setup Wizard to open the triggering dialog box:

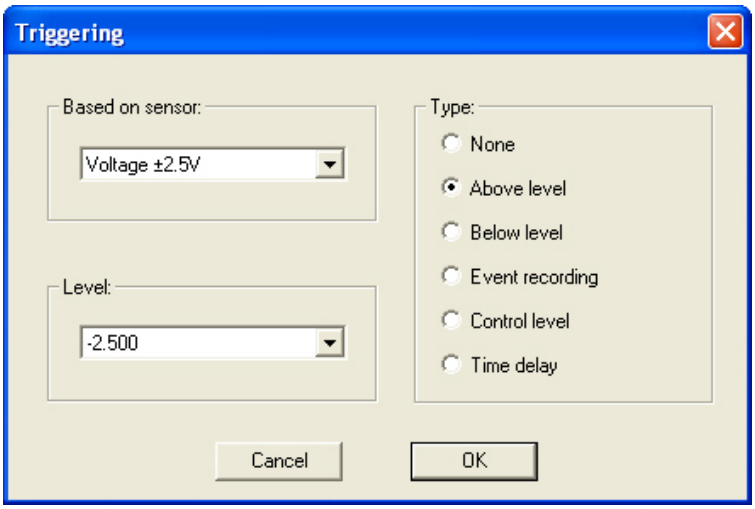


Figure 10: Triggering dialog box

Select the triggering sensor in the **Based on sensor** drop down menu.

Choose one of the following from the **Type** options:

None - Trigger is disabled

Above level - Start logging only once the measured value is ***higher*** than the trigger level.

Below level - Start logging only once the measured value is ***lower*** than the trigger level.

Event Recording - This function enables you to record the exact time and date at which a certain phenomenon occurs: The trigger level set for this option is actually a threshold setting. Each time the threshold is crossed, the data logger will record the exact time and date of the occurrence, and will continue to do so until the desired number of samples has been obtained.

Note: The trigger acts on analog measurements only (not on the Distance sensor). The trigger condition must be fulfilled for at least 300 μS.

Control Level - The Control Level trigger allows you to create an automatic sense & control system. This means that you can connect a sensor measuring a certain phenomenon (for example, temperature) and connect a device that will start operating when the recorded data from the sensor falls above or below a certain level (for example, a fan that will start operating when the temperature measured by the sensor rises above 30 °C). This function requires the

use of a Splitter cable and a Control sensor. The cable divides each input into a sensor cable and a controller cable. After setting the control level and starting the data logging process, the sensor will sample and record the data as usual. However, when the measurement from the sensor rises above the predetermined threshold value, the controller cable will send a pulse of 5 V to the Control sensor, and will continue to do so until the sensor measures a value below the threshold level. When receiving the 5 V pulse, the Control sensor will close/open a relay capable of switching 110/220 V to any load.

Time delay - This trigger type enables you to set a timer that will start the logging after a predetermined amount of time. After setting the trigger to **Timer Delay**, click the down arrow on the **Level** combo-box, and select from the 17 different time options. The timer will start its countdown when you click **RUN**, and the actual recording will start once the countdown has ended.

Select the trigger level in the **Level** drop down menu.

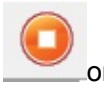
Click **OK**.

Note: When you turn off the MultiLogPRO, it will save the setup for the next session.

Start Recording

Click **Run**  on the main toolbar. Click the **down arrow** to change the recording mode.

Stop Recording

Click **Stop**  on the main toolbar.

Clear MultiLogPRO's Memory

To erase all experiment data currently stored in the MultiLogPRO, click **Clear memory** in the **Logger** menu.

Edit MultiLogPRO's Experiment Notes

Connect MultiLogPRO to the PC.

Select **Edit notes** from the **Logger** menu to open the **Notes** dialog box with the MultiLogPRO's current experiment's notes downloaded to the window.

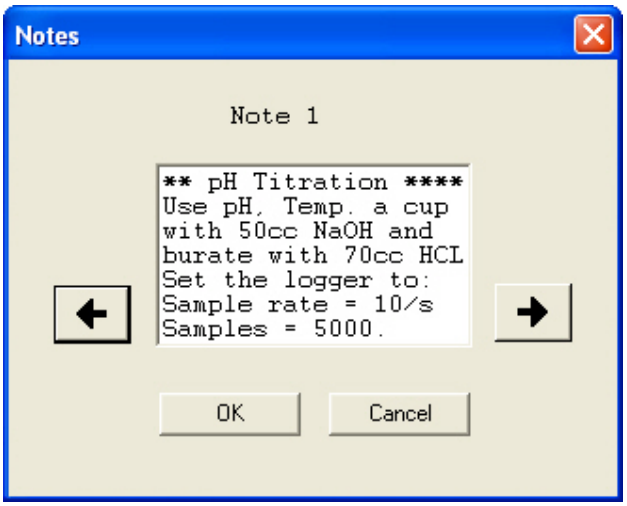


Figure 11: Edit experiment notes dialog box

From this dialog box, you can edit, delete, or write new notes.

Note: An experiment note is limited to 140 characters and a maximum of 5 notes can be stored in MultiLogPRO at any time.

Use the arrow buttons to move to the next note, or to the previous one.

Click **OK** to upload the edited notes to MultiLogPRO.

Calibrating the Sensors

MultiLab enables you to calibrate any of the sensors manually. This calibration is much more accurate than the calibration performed by NOVA LINK or MultiLogPRO. With MultiLab, you can calibrate the sensor using two points instead of the *one point automatic offset calibration* used by the data logger:

Connect the data logger to the PC.

Select the **Calibrate sensors** option from the **Logger** menu.

Choose a sensor from the **Select sensor** drop down menu and click **OK**.

Click the **Calibration** tab.

Enter a distinct real value in each of the **Real Value** edit boxes and the corresponding measured values in each of the **Measured Value** edit boxes (The measured values are the values displayed by MultiLab when measuring the two real values).

Click **OK**.

The calibrated sensor parameters will be saved, so there is no need to calibrate the data logger every time you run the MultiLab program.

To reset to the default calibration for any sensor, select the sensor and click **Restore defaults**.

Define a Custom Sensor

Ordinarily, you will not need to use this option, but MultiLab enables the user to define additional custom sensors. This is a useful tool for when the data logger is communicating with many sensors from different vendors. See section **Error! Reference source not found.** on page **Error! Bookmark not defined.** for a list of supported sensors.

Any additional sensor that you would like to connect to NOVA LINK or MultiLogPRO must comply with the following restrictions:

- The sensor's output must be greater than or equal to **0V** and less than or equal to **5V**. Remember that all sensors transform actual data into electrical data, so the electrical output should remain between 0 and 5 V.
- The sensor Transfer Function (sensor output voltage changes vs. the sampled phenomenon changes) must be a linear Transfer Function.
- The sensor must have a code resistor in order to be automatically identified. If the sensor does not have a code resistor, you will have to work in **8 inputs** mode and select the sensor manually.

  To determine whether your sensor has a code resistor or not, simply plug it in to the data logger (while in **Auto ID** mode) and see if MultiLogPRO displays the setup menu and the sensor name.

  Refer to the appendix to learn how to add a code resistor to your custom sensor.

  To define a new sensor:

  Connect the data logger to the PC.

  Select **Define new sensor** from the **Logger** menu to open the **Define new sensor** dialog box:

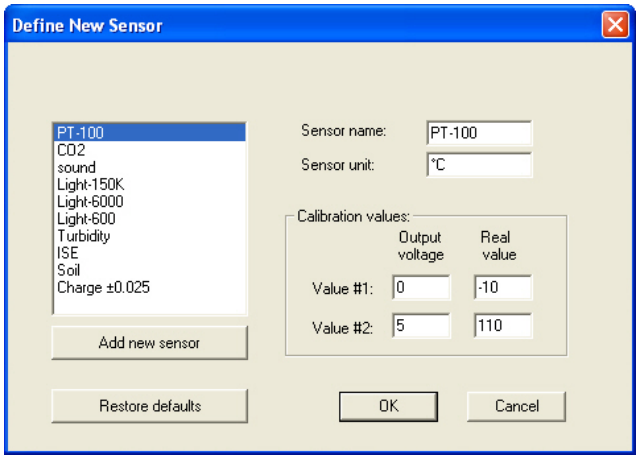


Figure 12: Define new sensor dialog box

  Click **Add New Sensor**.

Type a sensor name and a sensor unit into the relevant fields.

Type in two calibration values (two real values and the corresponding output voltages of the sensor).

Click **OK**.

Communication Setup

The MultiLogPRO communicates via a USB or Serial communication port. Some users may find it difficult to select the correct port, in which case MultiLab can perform an automatic port selection, and can also report on the status of all available ports.

Automatic COM Port recognition

The computer may have several communication ports. One of these ports is usually available for external communication. It is not necessary to know which port is available, since **MultiLab will find it for you**:

Connect the communication cable from the data logger to one of the computer COM port connectors.

Turn on the data logger and launch the MultiLab software.

MultiLab automatically attempts to communicate with the data logger. Once communication is established, MultiLab reports the connection in the status bar at the bottom of the application window.

If communication fails:

Working in Windows OS

Select **Comm Setup** from the **Logger** menu. MultiLab will display a dialog box reporting what COM ports are already in use, and which port is recommended for use with the NOVA LINK or MultiLogPRO:

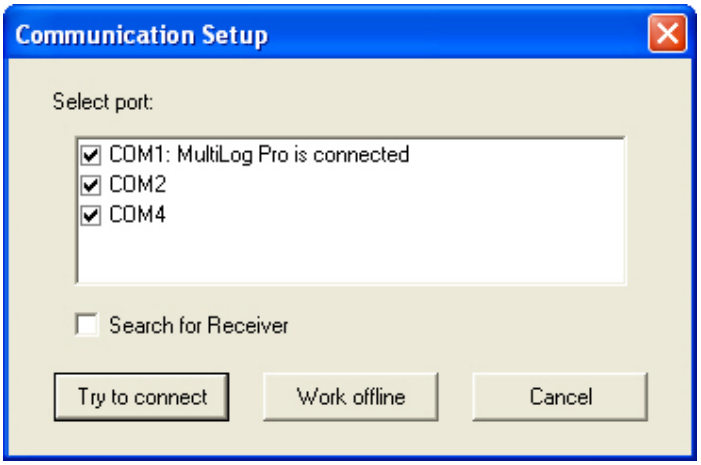


Figure 13: Communication setup dialog box (Windows)

Click **Try to connect** and MultiLab will try to communicate with the logger. If communication is achieved, MultiLab will designate the selected port as the specific logger's communication port.

You can override the MultiLab recommendation and choose the COM port yourself by clicking the checkbox button to the left of the desired port, and clicking **Try to connect**.

Note: In the event of any communication malfunction, please refer to 0: Troubleshooting Guide.

You can click **Work offline** if you wish to work without the logger. While working offline, you can still open saved files to view and analyze them.

Working in Mac OS

Select **Comm Setup** from the **Logger** menu. MultiLab will display a dialog box listing the available ports:

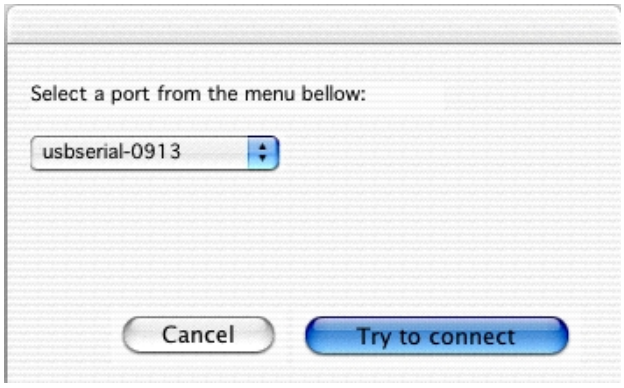


Figure 14: Communication setup dialog box (Mac OS)

Select a port from the drop down menu.

Click **Try to connect** and MultiLab will try to communicate with the data logger. Once communication is established, MultiLab reports the connection in the status bar at the bottom of the application window.

Click **Cancel** if you wish to work without the data logger. While working offline, you can still open saved files to view and analyze them.

Analyzing the Data

Reading Data Point Coordinates

Position the cursor (see page 18) on a point to display its coordinates on the information bar at the bottom of the graph window.

If more than one curve is displayed, and you want to read the Y- coordinates of all of the curves simultaneously, bring up digital meters (see page 25) for each of the

curves. When you position a cursor on one curve, the meters will show the corresponding Y-coordinates for the other curves.

Reading the Difference between two Coordinate Values

Position one cursor on the first point and a second cursor on the second point to display the difference between the two coordinate values on the information bar at the bottom of the graph window.

Working with the Analysis Tools

The analysis tools can only be applied to data sets that are displayed in the graph window.

Use the cursors (see page 18) to select the graph and the data range to which you want to apply the analysis.

Select the analysis function you wish to use.

The analysis function will be added onto the graph, with the exception of the smoothing (averaging) function, which will replace the original data set.

To measure and analyze time events with photogates use the Timing Wizard (see page 65).

Smoothing

The smoothing tool is very useful in reducing random noises, especially if you want to apply any analysis functions to the data. The smoothing process replaces every data point with the average of its neighboring points.

Use the cursor to select the graph that you want to smooth.

Click **More smoothing** on the graph toolbar.

You can repeat the procedure to further smooth the data.

Click **Less smoothing** to reduce the amount of smoothing.

Statistics

Use the statistics tool to display statistics of a selected data set or a range of data.

The statistics include:

Average The average of all the numbers in the range

Median The median is the number in the middle of a set of numbers; that is, half the numbers have values that are greater than the median, and half have values that are less. If there is an even number of numbers in the set, then Median calculates the average of the two numbers in the middle

StDev. The standard deviation

Minimum	The smallest value in the range
Maximum	The largest value in the range
Sum	Adds all the numbers in the range
Area	The area between the graph and the x-axis in the range
Samples	The number of data points in the range
Rate	The recording rate

To display statistics:

Use the cursors to select the graph and the data range to which you want to apply the statistics.

Click **Analysis** on the menu bar, and then click **Statistics**.

MultiLab will open a statistics window and will display the results in it.

Most Common Analysis Functions

Linear Fit

Click **Linear Fit**  on the main toolbar to draw a line of linear least square fit

$$y = ax + b$$

and to display the line's equation.

Note: If you want the automatic curve fit equation to start at $t = 0$, apply the crop tool before applying the linear fit (see page 65).

Derivative

Click **Derivative**  on the main toolbar to construct a graph in which each point is the slope of the three consecutive points on the source graph.

Integral

Choose **Integral** from the **Analysis** menu to construct a graph in which each point is the integral of all the preceding points on the source graph.

The Analysis Wizard

Using the Analysis Wizard

The analysis wizard will guide you through the various analysis functions available in the MultiLab program. The analysis functions are grouped into three main categories: curve fit (regression), averaging, and mathematical and trigonometric functions.

To apply an analysis function to a data set:

Use the cursors to select the graph or data range to which you would like to apply the analysis (optional – you can select the desired data set directly from the Analysis Wizard).

Click **Analysis Wizard**  on the main toolbar.

Click a category tab:

Curve fit

Averaging

Functions

Curve Fit

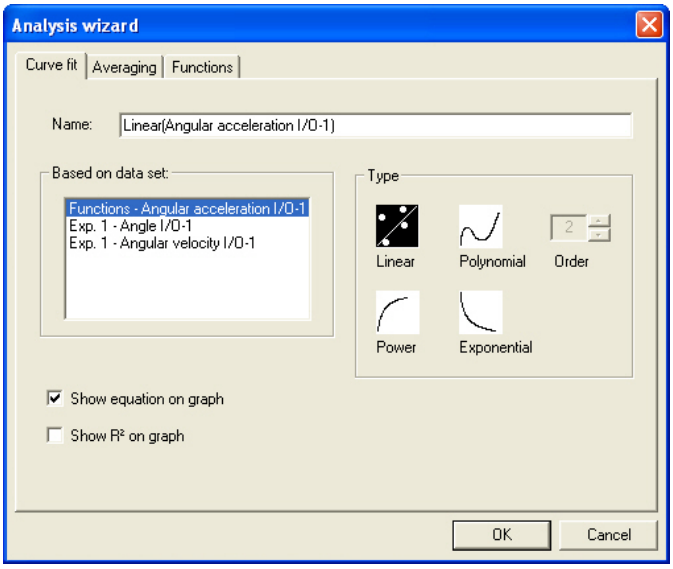


Figure 15: Analysis wizard – curve fit

Select a fit type by clicking its icon (If you choose polynomial fit, select the order you want).

If you use the cursor to select a data set, it will be highlighted in the **Based on data set** list, but you have the option of selecting a different data set.

Type a name in the **Name** box (optional - the default name includes both the function and the data set names).

Click the **Show equation on graph** check box to display the line's equation on the information bar.

Click the **Show R^2 on graph** check box to display the correlation coefficient on the information bar.

Click **OK**.

Note: If you want the automatic curve fit equation to start at $t = 0$, apply the crop tool before any type of curve fitting (see page 65).

Averaging

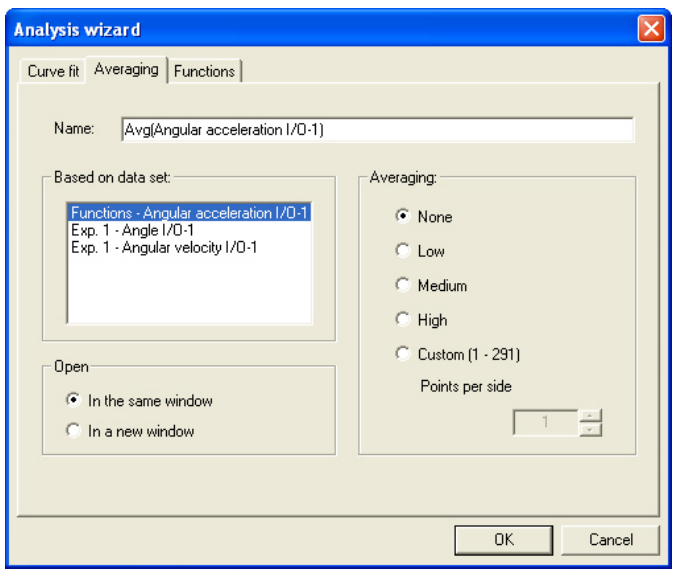


Figure 16: Analysis wizard – averaging

If you use the cursor to select a data set, it will be highlighted in the **Based on data set** list, but you have the option of selecting a different data set.

Select an averaging option (if you choose custom averaging, choose the number of averaging points in the **Points** box).

Enter a name in the **Name** box (optional - the default name include both the function and the data set names).

Select an **Open** option.

Click **OK**.

Functions

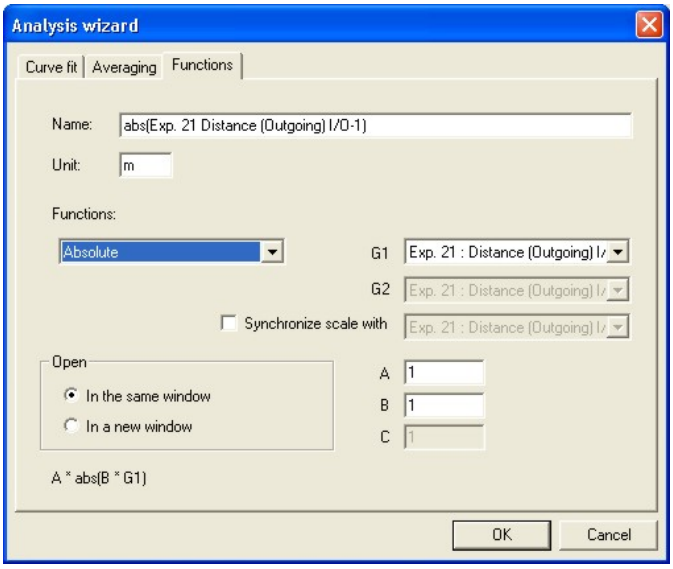


Figure 17: Analysis wizard – functions

Select a function from the **Functions** list. MultiLab displays the function's formula at the bottom of the dialog box.

If you use the cursor to select a data set, it will be highlighted in the **G1** drop down menu, but you have the option of selecting a different data set.

Tip: To create a function whose independent variable is time (e.g. to fit a mathematical function to a data series), select a time series from the **G1** drop down menu. If there are several plots on the graph the list includes a separate time series for each plot as the number of data point may vary from plot to plot.

If the analysis function involves two data sets, select the second data or time series set from the **G2** drop down menu.

To synchronize the Y – Scale with one of the plots on the graph, check the **Synchronize scale with** checkbox and select the appropriate plot from the adjacent drop down menu.

Enter a constant in each of the **A, B** and **C** boxes (optional, the default values are one for a coefficient and zero for a free term).

Enter a name in the **Name** box (optional – the default name includes both the function's formula and the data set name).

Enter a unit (optional).

Select an **Open** option.

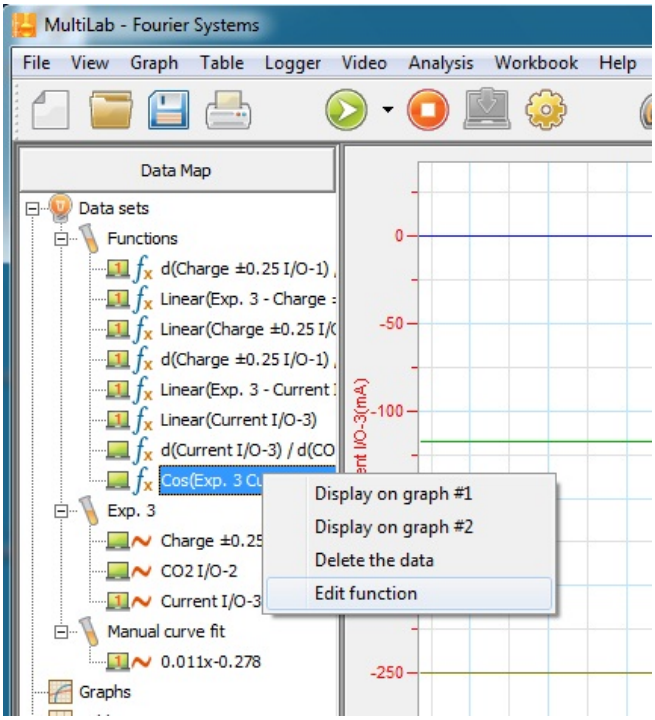
Click **OK**.

Editing a Function

MultiLab enables you to edit the functions parameters as well as its name and units after you have displayed it on the graph. The **Edit function** command is enabled for all the analysis functions created in the current session and is disabled when you open a stored file.

To edit the function:

Right click the function's icon in the Data Map to open a context menu.



Click **Edit function to** open the Function dialog.

Enter the new parameters.

Click **OK**.

Available Analysis Tools

This section includes a brief description of each of MultiLab's analysis functions.

For time, velocity and acceleration analysis see also the Timing Wizard on page 65.

Curve Fit

Linear $y = ax + b$

Draws a line of linear least square fit.

Polynomial $y = a_0 x^n + a_1 x^{n-1} + \cdots + a_n$

Draws a line of polynomial least square fit (you must select an order between 1 and 6.)

Power $y = \frac{a}{x^n}$

Draws a line of power least square fit.

Exponential $y = ae^{bx}$

Draws a line of exponential least square fit.

MultiLab displays the curve fit equation and the correlation coefficient (R^2) on the information bar below the graph.

Averaging

The average function replaces every point with the average of *n* neighboring points from both sides of the point.

Low averaging: $n = 5$, Medium averaging: $n = 11$, High averaging: $n = 41$.

Functions

In the formulas below, G_1 and G_2 represent selected data sets or time series, and A, B and C are constants that you can enter.

Absolute $y = A|BG_1|$

Draws a line of the absolute values of a data set.

Add $y = AG_1 + BG_2$

Draws a line of the addition of two data sets.

Arccosine $y = A\arccos(BG_1)$

Draws a line of the arccosine values of a data set (in radians). Arccosine is the angle whose cosine is BG_1. The argument BG_1 must be between –1 and 1.

Arcsine $y = A\arcsin(BG_1)$

Draws a line of the arcsine values of a data set in radians. Arcsine is the angle whose sine is BG_1. The argument BG_1 must be between –1 and 1.

Cosine

$$y = A\cos(BG_1 + C)$$

Draws a line of the cosine values of a data set. The argument $BG_1 + C$ must be expressed in radians.

Delta Y

$$y = G_1 - G_1(t = 0)$$

Draws a line of the difference between the Y-coordinate of every point and Y-coordinate of the first point. Use this function to move the data set along the Y-axis so that the point will intersect the Y- axis at the origin.

Derivative

$$y_n = \frac{y_{n+1} - y_{n-1}}{2\Delta t}, \Delta t = \frac{1}{sampling\,rate}$$

Draws a line of the slopes of every three consecutive points of a data set. For high recording rates and small Δt, this line may be very noisy, which is why smoothing the data set is recommended before applying the derivative function.

Divide

$$y = \frac{AG_1}{BG_2}$$

Draws a line of the division of two data sets

Envelope (lower)

Lower envelope of G_1 with tolerance of A points

Draws a line that connects the minimum values of a data set. The tolerance defines the minimum distance (in sampling points) between two minima, so that the envelope function is able to ignore random noises.

Envelope (upper)

Upper envelope of G_1 with tolerance of A points

Draws a line that connects the maximum values of a data set. The tolerance defines the minimum distance (in sampling points) between two maxima, so that the envelope function will be able to ignore random noises.

Exp.

$$y = Ae^{BG_1} + C$$

Draws a line of e raised to the power of a data set.

Fourier transform

Discrete Fourier transform of G_1.

Draws a line of the amplitudes of the harmonics of Fourier transform vs. frequency.

Frequency

The frequency of G1 (minimum of A points in one cycle).

Draws a line of the frequency of a periodic data set vs. time. The constant A defines the minimum data points in one cycle.

Integral

$$y = A + B\sum G_1 \Delta t$$

Draws a line in which each point is the discrete integral of all the preceding points in a data set.

Kinetic energy

$$y = \frac{1}{2}A(G_1)^2$$

Draws a line of the kinetic energy of a data set. The argument G_1 must be the velocity of the body, and the constant C_1 must be the mass of the body.

Linear

$$y = AG_1 + B$$

Draws a line of a linear displacement of a data set. This function is useful when you want to change the point of origin of a data set.

Ln

$$y = A\ln(BG_1)$$

Draws a line of the natural logarithm of a data set. The argument BG_1 must be positive.

Log

$$y = A\log_{10}(BG_1)$$

Draws a line of the logarithm of a data set to base 10. The argument BG_1 must be positive.

Multiply $y = AG_1 \cdot BG_2$

Draws a line of the multiplication of two data sets

Quadratic $y = AG_1{}^2 + BG_1 + C$

Draws a line of the quadratic form of a data set.

Reciprocal (1/X) $y = \dfrac{A}{G_1 + B} + C$

Draws a line of the reciprocal values of a data set.

Sine $y = A\sin(BG_1 + C)$

Draws a line of the sine values of a data set. The argument $BG_1 + C$ must be expressed in radians.

Square (X^2) $y = A(BG_1)^2$

Draws a line of the squares of a data set.

Square root $y = A\sqrt{BG_1 + C}$

Draws a line of the square root values of a data set: The argument C_2G_1 must be greater than or equal to zero.

Subtract $y = AG_1 - BG_2$

Draws a line the subtraction of two data sets.

Tan $y = A\tan(BG_1 + C)$

Draws a line of the tangent values of a data set. The argument $BG_1 + C$ must be expressed in radians.

Online Analysis

MultiLab enables you to analyze data from a live experiment and to display the analysis function in the graph window, as the data is being recorded and processed.

To execute online analysis, you will first have to use the Analysis Wizard to define the function to be used, and then edit the graph in order to display the function (see page 20).

For example, here is how to set up an online envelope (amplitude) function of data, recorded with a Voltage sensor connected to Input 1:

Click **Analysis Wizard** on the main toolbar to open the Analysis Wizard dialog box:

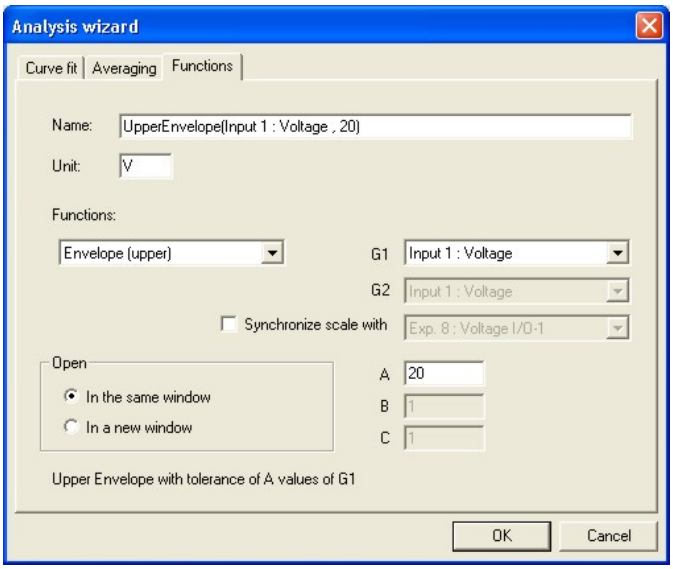


Figure 18: Analysis Wizard – online functions

Select **Envelope (upper**) from the **Functions** list.

Select **Input 1: Voltage** from the **G1** drop down menu.

Click **OK**.

Click **Edit graph** on the graph toolbar to open the Graph edit dialog box:

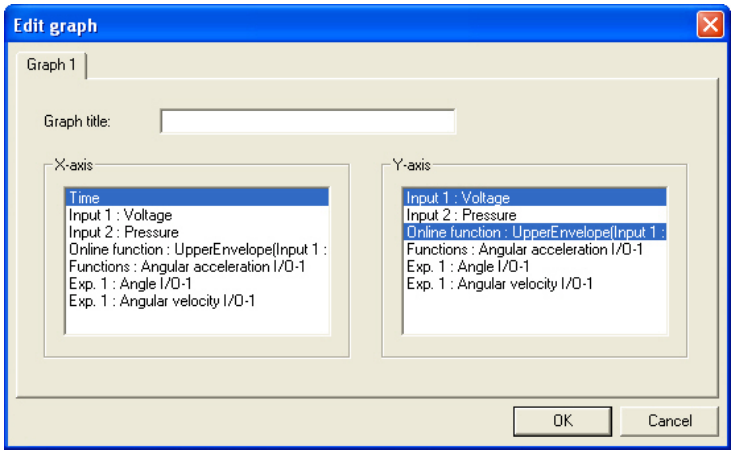


Figure 19: Using graph edit dialog to display online functions

In the **Y-Axis** list click **Input 1: Voltage** to select it and then click **Online function: Envelope (upper)** to select it.

If there are other highlighted data sets in the **Y-Axis** list, deselect them by clicking on them.

Click **OK**.

Click **Run**  on the main toolbar to start recording. MultiLab will display the original data readings coming from Input 1, as well as the data processed by the analysis function:

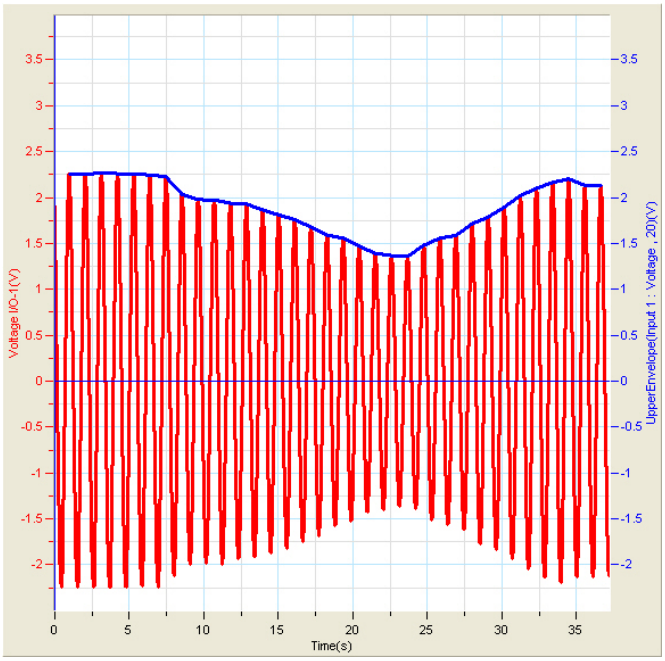


Figure 20: Online envelope function

The online analysis function will still be active in subsequent recordings.

To remove the online functions, click **Analysis** on the menu bar and then click **Clear online functions**.

Manual Curve Fitting

The manual curve fit tool enables you to draw a mathematical curve, to change the curve's parameters until it fits into data set, and to display the resulting equation. There are four types of mathematical curves you can draw:

Linear $y = At + B$

Draws a straight line.

Quadratic $y = A(t - B)^2 + C$

Draws a parabola.

Exponential $y = Ae^{Bt} + C$

Draws an exponential line.

Power $y = At^B + C$

Draws a line of the time raised to the power of a number from –5 to 5.

Note: If you want the curve fit equation to start at $t = 0$, apply the crop tool before any type of curve fitting (see page 65).

To apply manual curve fit:

Click on the **Analysis** menu, then click **Manual curve fit** to display the Manual curve fit dialog box.

To select a data set for display on the X-axis, click on the data set's name in the **X-axis** list.

To select a data set to display on the Y-axis, click on the data set's name in the **Y-axis** list.

Click **OK**. MultiLab will display the Manual curve fit toolbox at the bottom of the graph window:

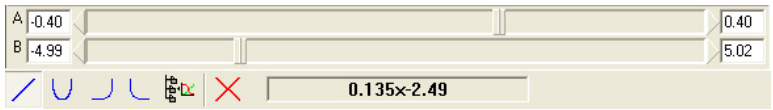


Figure 21: Manual curve fit toolbox

Choose one of the four curves by clicking its icon in the toolbox.

Use the sliders to change the A, B and C parameters until the new curve fits the sampled curve.

The lower and upper limits of each sliding bar are located on the sliding bar's left and right ends, respectively. You can change the limit values by clicking a limit box, and typing in your new value.

MultiLab will display the resulting equation on the information bar at the bottom of the toolbox.

Click **Add to project** in the toolbox to add the curve to the project.

To close the manual curve fit toolbox click **Exit** in the toolbox.

Video Motion Analyzer

Overview

MultiLab's Video Motion Analyzer enables you to capture position and time from video movies, convert this to data sets and analyze the data with all of MultiLab's analysis tools the same way you would analyze data from the data logger. You can capture one-body motion or two-body motion. You can select the origin and rotate the coordinate system.

With the Video Motion Analyzer you can also capture video movies either directly from video camera connected to the computer or from an external video source such as VCR.

Getting Started

Video Motion Analyzer Basics

Open the Video Motion Analyzer module

Click **Video Analysis** on the main toolbar

Open a movie from MultiLab's movie library or capture a new movie

Capture position and time

- **Scaling** – Scale the video to the real world dimensions
- **Coordinate system** – Set origin and axes direction

- **Step** – Set the number of frames you will capture position and time
- **Marking** – Mark the position of the body on the frames. MultiLab will automatically convert it to numerical data

Analyze the data

The captured data is automatically displayed in the graph and table windows. Analyze it as you would normally with MultiLab's projects.

Saving a New Movie

After you have finished capturing a new video movie click **Analyze movie** to begin capturing position and time.

MultiLab prompts to save the movie.

Type a name in the **File name** box and click **Save**.

Opening a Stored Movie

Click **Video Analysis** on the main toolbar.

Click **Open Movie** on the Video Motion Analyzer toolbar (lower toolbar).

Double click the movie you want to open.

Saving a Video Motion Analyzer Project

Click **Save** on the main toolbar.

Type a name in the **File name** box and click **Save**.

Opening a Video Motion Analyzer Project

Click **Video Analysis** on the main toolbar.

Click **Open** on the main toolbar.

Double click the Project you want to open.

Capturing a New Movie

There are two basic options for capturing video onto your computer from:

- A device attached to your computer

- An external source through a video capture board

From a Device Attached to your Computer

From a device attached to your computer (such as a Web cam), you can capture video directly from the camera to your computer. Web cams are intended primarily to feed video over some connection (the Internet, for example). They generally produce lower video quality. With a Web cam, you can only record when the Web cam is attached to the computer. If quality and versatility are not your main concern, this may be the best solution for you.

To capture video from a Web cam:

Connect the Web cam to the computer.

Click **Video** on the menu bar, and then click **Enable video**.

Click **Video Analysis**  on the main toolbar to open the Video Motion Analyzer module.

Click **Capture new movie**  on the Video Motion Analyzer toolbar.

To begin recording click **Record**  on the Capture new movie toolbar.

To finish recording click **Stop**  .

Click **Analyze movie**  to begin capturing position and time.

MultiLab prompts to save the movie before processing it.

From an External Source through a Video Board

Many video capture boards offer video-in/video-out features. A video capture card can be used with a wide variety of video sources and through a number of digital connections. Video from a VCR or video camera can be captured through video-out ports.

To capture video from a videotape through a VCR:

Insert the tape into the VCR. Go to the point you want to capture and then stop the tape.

Find the video-out on the back of the VCR and connect one end of the appropriate cable. Connect the other end of the cable to the video-in on your video capture board.

Click **Video** on the menu bar, and then click **Enable video.**

Click **Video Analysis** on the main toolbar to open the Video Motion Analyzer module.

Click **Capture new movie** on the Video Motion Analyzer toolbar.

To begin capturing click **Record** on the Capture new movie toolbar.

Play the video. Make sure that you start the video after you start the capture to make sure nothing gets cut off from the beginning.

To finish capturing stop the VCR and click **Stop**.

Click **Analyze movie** to begin capturing position and time.
MultiLab prompts to save the movie before processing it.

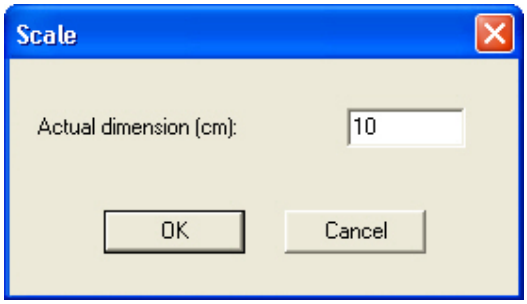
Capturing Position and Time

Scaling

In order to scale your project you must tell MultiLab the real world dimensions. When you film a new movie make sure to measure the dimension of a dominant object in the frame. Try to find an object that's dimensions are in the magnitude of the whole scene.

Scaling is done in two steps – you need to mark the object in the video frame and to type in the real dimensions:

Click **scale** on the Video Motion Analyzer toolbar to open the Scale dialog:



Click one end of the scaling object.

Click the body's other end.
MultiLab will display two markers on the points you clicked.

You can drag the markers to change their position.

Type in the body's actual dimension in the **Actual dimension** edit box (in cm).

Click **OK.**

You can change the scaling anytime by clicking **Scale** and repeating the procedure.

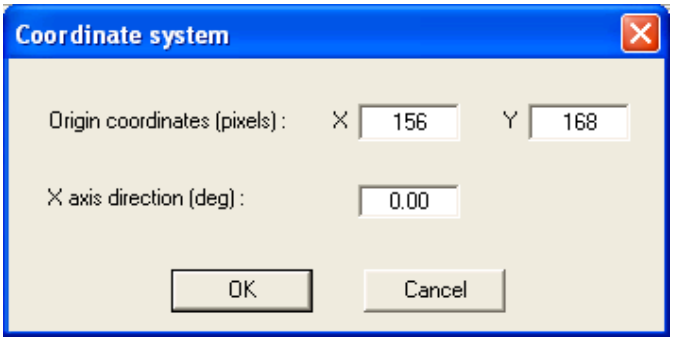
Set Coordinate System

To set the coordinate system you must specify the origin position and the x axis direction. You can do it either by a click of the mouse or by entering it manually to the coordinate system dialog.

Unless you specify otherwise MultiLab will use its default: the origin is in the lower left side of the frame and the axes are parallel to the window's sides.

Set origin

Click **Set origin** on the Video Motion Analyzer toolbar to open the Coordinate system dialog:



Click a point in the frame where you want the origin to be.

MultiLab automatically updates the X and Y origin coordinates edit boxes and moves the axes on the frame to the new origin.

You can move the origin by clicking another point in the frame.

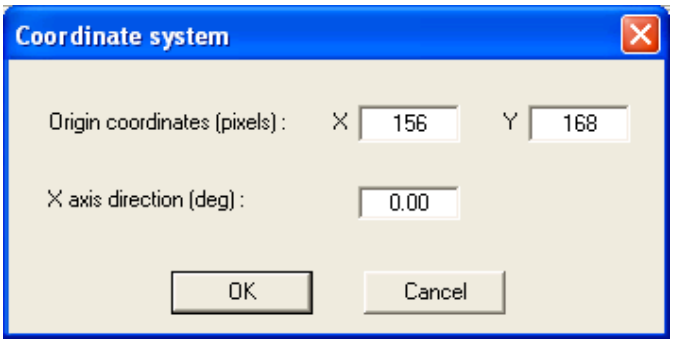
You can enter the origin coordinates manually by typing them in the corresponding edit boxes (in pixels).

You can type in the x axis direction (in degrees) in the **X axis direction** edit box.

Click **OK**.

Set X-axis direction

Click **Rotate origin** on the Video Motion Analyzer toolbar to open the Coordinate system dialog:



Click a point in the frame where you want the x axis to be placed.

MultiLab automatically updates the X axis direction edit box and rotates the axes on the frame to the new direction.

You can keep rotating the axes by clicking another point in the frame.

You can type in the x axis direction (in degrees) in the **X axis direction** edit box.

You can change the origin coordinates by typing them the X and Y origin coordinates edit boxes (in pixels).

Click **OK**.

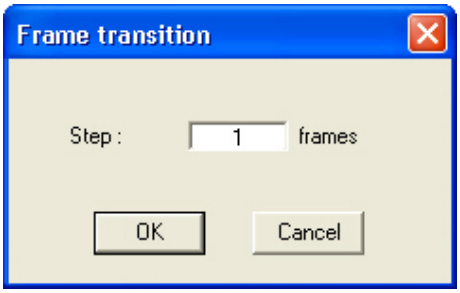
You can change the coordinate system origin and direction anytime by clicking **Set origin**  or **Rotate origin**  and repeating the procedure.

Set Step

In the marking process, MultiLab automatically steps the video to the next frame every time you click to add a marker.

To change the step to more then one frame at a time:

Click **Video analysis** on the menu bar and click **Frame transition** to open the Frame transition dialog:



In the **Step** edit box, type in the number of frames you want MultiLab to advance in one step.

Click **OK**.

Mark the Video

One-body motion

Click **Play** on the Video Motion Analyzer toolbar and observe the motion.

Decide what will be the **marking point** – the exact point on the moving object that you want to mark.

Tip: Try to choose a clear and sharp point that is visible throughout the entire movie

Click **Go to first** to roll the film back to the beginning.

Click **Next frame** until you reach the frame with which you want to start the capture.

Click the **marking point** on the moving object.

MultiLab automatically marks the point on the screen and creates two new data sets (one for x coordinates and one for y coordinates). It then adds the point's coordinates and time to the corresponding data sets and updates both the graph and the table, then automatically moves to the next frame.

Note: You don't see the mark because the display had already moved to the next frame.

Continue clicking the **marking point** on every frame until you capture all the data you need or until the video doesn't step anymore.

Observe that the graph and table are updated with each mouse click.

Note: In order for MultiLab to automatically move to the next step after one mouse click, the **Consecutive** option in two-body motion must be selected (the default option).

Two-body motion

Use the left mouse button to mark the first body and the right mouse button to mark the second body.

You can mark two-body motion in two ways: simultaneously or consecutively. In the simultaneous method you mark the two bodies alternately on every step of the marking process. In the consecutive method you finish marking the whole motion of the first body and then repeat the process with the second body.

Simultaneous marking

Click **Video Analysis** on the menu bar, point to **Two-body motion**, and then click **Simultaneous**.

Proceed as in one body motion except that in step 5 right click to mark the first body and left click to mark the second body.

MultiLab waits for both right and left mouse clicks before moving to the next frame.

Keep clicking alternately with the right and left mouse button at every step.

Consecutive marking

Click **Video Analysis** on the menu bar, point to **Two-body motion**, and then click **Consecutive**.

Proceed as in one body motion until you finish marking the first body.

Click **Go to first** to roll the film back to the beginning.

Click **Next frame** until you see a marker on the first body.

Right click the second body to mark it.

Keep right clicking on every step to mark the second body's track.

Viewing markings

You can view the markings frame by frame. You can also view the markings on the video in parallel with the graph's cursor so that the marker and the cursor are synchronized.

To view just the markings:

Use the **Next frame** and **Previous frame** buttons on the Video Motion Analyzer toolbar to view the desired frame.

To view the markings in parallel with the cursor:

Display the cursor on the graph.

MultiLab automatically displays the corresponding frame with the marker.

To scroll through the frames you can either use the **Next frame** and **Previous frame** buttons on the Video Motion Analyzer toolbar or drag the cursor on the graph or use the left and right arrow keys on the keyboard.

Editing the data

You can change the position of the marker at a specific frame. MultiLab will automatically update the corresponding point in the data set.

Display the frame you want to edit.

Click and drag the marker to the new position.

Erasing markings

You can erase a marker from a specific frame. MultiLab will automatically delete the corresponding point in the data set.

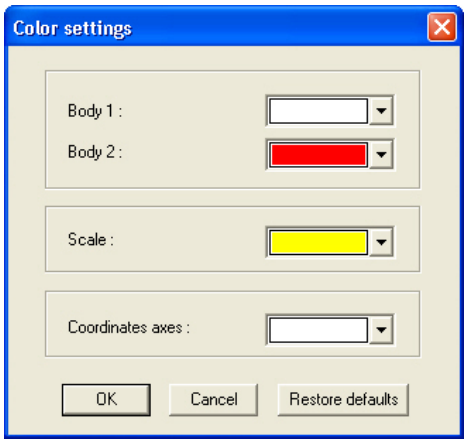
Display the frame you want to edit.

Click **Delete point** on the Video Motion Analyzer toolbar.

Click the marker.

Changing the Colors of the Marker and the Axes

Click **Color settings** on the Video Motion Analyzer toolbar to open a dialog:



Select the desired colors.

Click **OK**.

To restore default colors click **Restore defaults.**

Analyzing the Data

Once you have finished capturing position and time you can apply MultiLab's analysis tools to the data. MultiLab converts the body's track into two data sets X position (x coordinates) and Y position. By default the two data sets are displayed in the graph as a function of the time, but you can use the edit graph tool to display the track (Y vs. X).

In Video Motion Analyzer mode the video is usually displayed in the main window. To display the graph in the main window select **View selection** from the **View** menu.

Before applying analysis functions you may find it helpful to smooth the data with the smoothing tool on the graph toolbar.

At anytime you can change the origin and direction of the coordinates system

You can also change the coordinates of individual points by relocating them on the video frame.

Workbook

Working with Workbook

The Workbook is an online library of experiment manuals called *worksheets* that appear in the format of a Web page. Each worksheet includes an experiment template that automatically configures both MultiLab and the data logger at the push of a button. To begin recording, all you need to do is click **Run**.

Every time you run an experiment from a worksheet, MultiLab opens a new project file with the same predefined setup.

You can use Fourier-produced worksheets, or create your own. You can also modify Fourier's existing worksheets to meet your own specifications.

Opening a Worksheet

Click **Workbook** on the menu bar, and then click **Open worksheet**.

Navigate to the folder in which the worksheet is stored.

Double click the file name to open the worksheet.

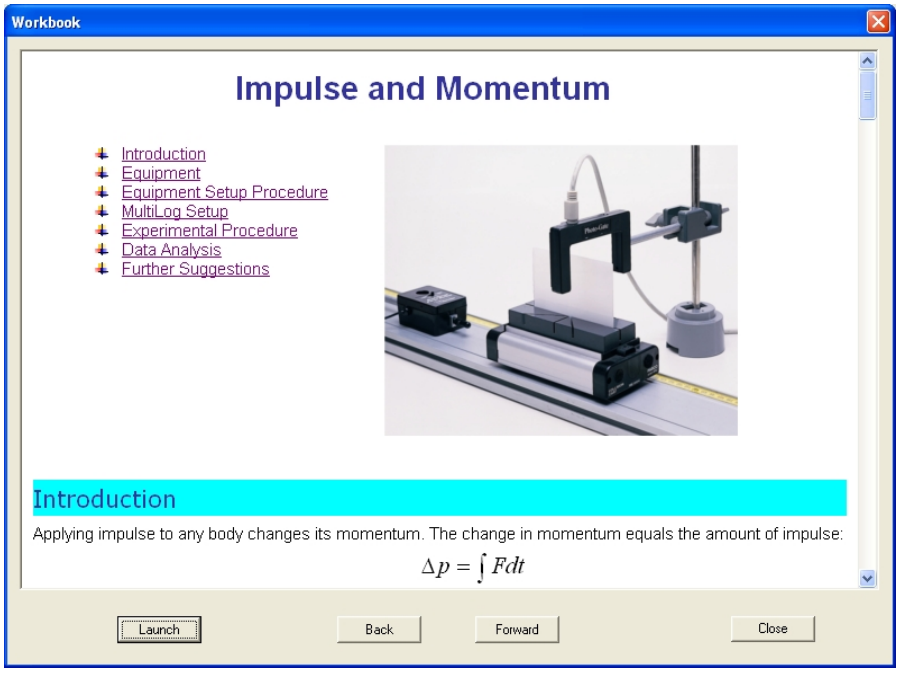


Figure 22: Worksheet

Follow the on-screen instructions and use the scroll bar, hyperlinks and the **Back** and **Forward** buttons to navigate throughout the document.

Click **Launch** to configure MultiLab and the data logger.

Click **Close** to return to MultiLab.

Click **Run**  on the main toolbar to begin recording.

Create Your Own Worksheet

Creating a worksheet consist of two steps. First, create an HTML document using your HTML editor (For example, Word or Front Page). This file should include the Lab manual and the experiment instructions (see below). The second step is to use the MultiLab software to create a configuration file, which is a file that will store the specific settings you wish to define for the experiment. These include the NOVA LINK or MultiLogPRO setup, MultiLab's layout, the recording mode, the graph format desired, special preset graph scaling, sensors measurement, etc.

Create an HTML Document with Word

Create a new Web page

Open Word.

In the **File** menu, click **New**.

Do one of the following:

- On the **General** tab, click **Web Page** to open a blank Web page.
- On the **Web Pages** tab, click the template that you would like to base your Web page on.

Enter your texts and pictures.

Click **Save** on the toolbar.

Navigate to C:\Program Files\Fourier Systems\MultiLab\Workbook.

Click **Save**.

Create a Web page from an existing Word document

Open the existing file with Word.

In the **File** menu, click **Save as Web Page**.

Navigate to C:\Program Files\Fourier Systems\MultiLab\Workbook.

Click **Save**.

Create a Configuration File

Open the MultiLab program.

Click **Workbook** on the menu bar, and then click **create new worksheet**.

Double click the file you want to configure to open it:

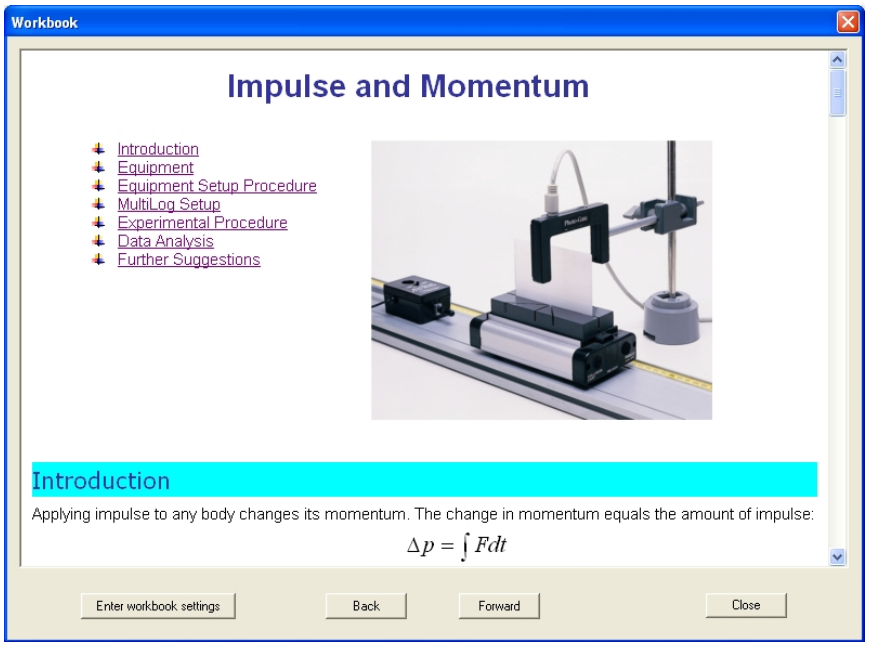


Figure 23: Creating a worksheet

Click **Enter experiment settings** to open the Setup wizard.

Use the Setup Wizard to preset MultiLab and data logger just as with the normal setup process (see page 29).

When you've finished entering the settings, click **Finish** to update the worksheet.

Click **OK**.

Click **Close**.

Special Tools

Predicting

The Predict tool enables you to draw predictions directly on the graph, prior to displaying the real data:

Click **Run**  on the main toolbar to start recording.

Click **Pause/Continue**  on the graph toolbar to freeze the graph.

Data transmitted from the data logger while the graph is frozen, is stored onto the PC but is not displayed.

Click on the end of the curve and move the mouse according to your predictions. Click to draw a straight-line segment. Move the mouse again and click where you want the second segment to end, and so on.

Click **Add prediction**  on the graph toolbar to create another prediction.

Click **Pause/Continue**  a second time to resume live data display and to compare your predictions with the real data.

The Timing Wizard

Overview

The Timing Wizard enables you to easily measure and calculate many types of time events, including velocity and acceleration with one or two photogates.

Connect one photogate to input 1 or connect two photogates to input 1 and input 2 of the MultiLogPRO and perform the experiment. The Timing Wizard will then guide you through the analysis.

With the Timing Wizard you can measure sequences of time events at gate one and/or at gate two, or time events between the two gates. MultiLab can then calculate the velocity and acceleration. Special options make it easy to measure velocities in collisions and the time period of a pendulum or any other oscillating body.

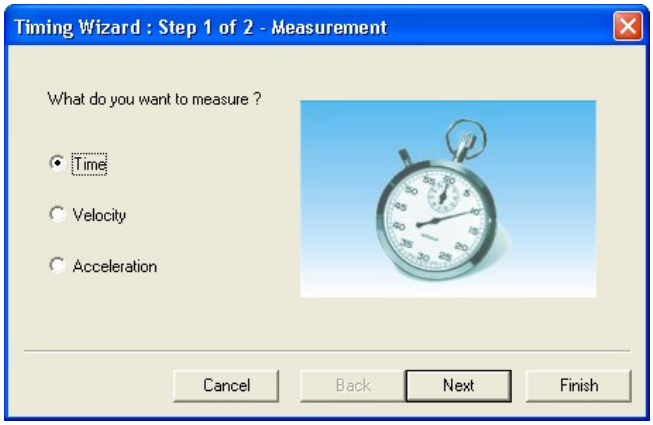
The Timing Wizard can handle multiple events. For example, when a body is crossing a photogate several times, applying the Timing Wizard will result in a series of measurements that match the number of crossings.

Note: If you wish to measure fast timing events with photogates only use MultiLogPRO's accurate timer module (see page 9).

Working with the Timing Wizard

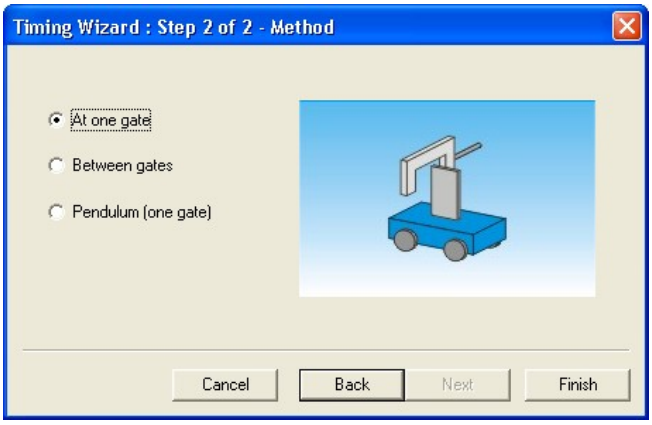
Display the data that you wish to analyze in the graph window.

Click **Analysis** on the menu bar, then click **Timing Wizard** to open step 1 of the Timing Wizard dialog:



Click an option to select measurement: **Time**, **Velocity** or **Acceleration**.

Click **Next** to move to step 2:



Click an option to select the measuring method.

If required enter the body's width, or the distance between the gates in cm in the appropriate edit box (in velocity and acceleration measurements only).

Click **Finish** to display the results.

If you want to change the method or measurement, click **Back** to return to the Timing Wizard. To exit the Timing Wizard click **OK**.

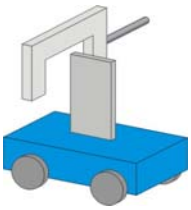
Measuring Methods

The Timing Wizard offers you various methods of analyzing the different measurements. In some measurements you will be asked to enter the dimension of the moving body, or the distance between the two photogates to allow for the calculation of velocity and acceleration.

The methods depend on the selected measurement:

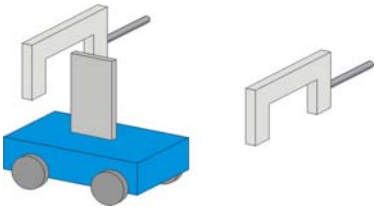
Time

- At one gate



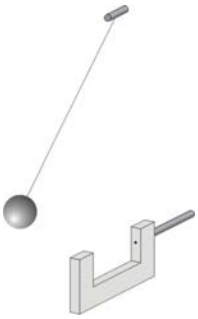
Measures the time it takes the body to cross the photogate (between blocking and unblocking the infrared beam).

- Between gates



Measures the time it takes the body to move from one photogate to the second photogate (between blocking the first and blocking the second infrared beams).

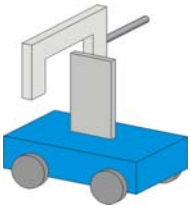
- Pendulum



Measures the time period of an oscillating body (the time interval between the first and the third blockings of the beam).

Velocity

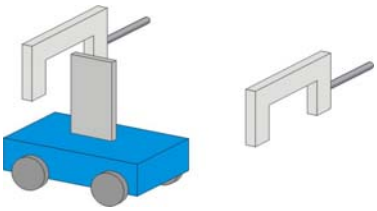
- At one gate



Measures the time it takes the body to cross the photogate (between blocking and unblocking the infrared beam) and returns the velocity.

You should enter the body's width.

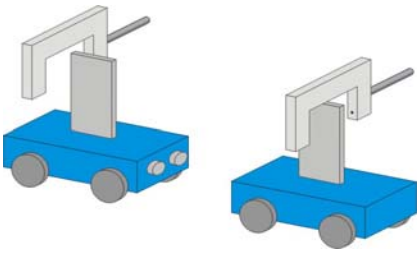
- Between gates



Measures the time it takes the body to move from one photogate to the second photogate (between blocking the first and blocking the second infrared beams) and returns the average velocity.

You should enter the distance between gates.

- Collisions

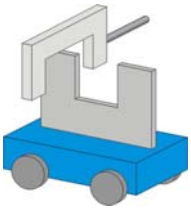


Measures the crossing time intervals at each gate and returns the corresponding velocities.

You should enter the bodies' width (the width of the two bodies must be identical)

Acceleration

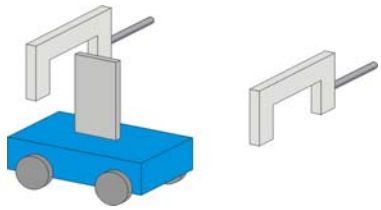
- At one gate



A card with two flags must be attached to the moving body (see figure to the left). The Timing Wizard measures the crossing time intervals of the two flags and returns the acceleration.

You should enter the flags width.

- Between gates



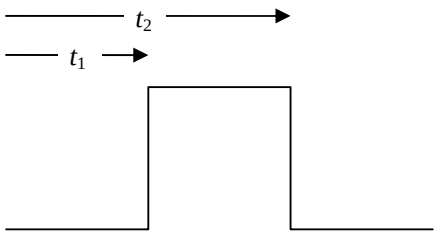
Measures the crossing time at the first gate, the time it takes the body to move from one gate to the second gate and the crossing time at the second gate and returns the average acceleration.

You should enter the body's width.

Time Schemes and Calculations

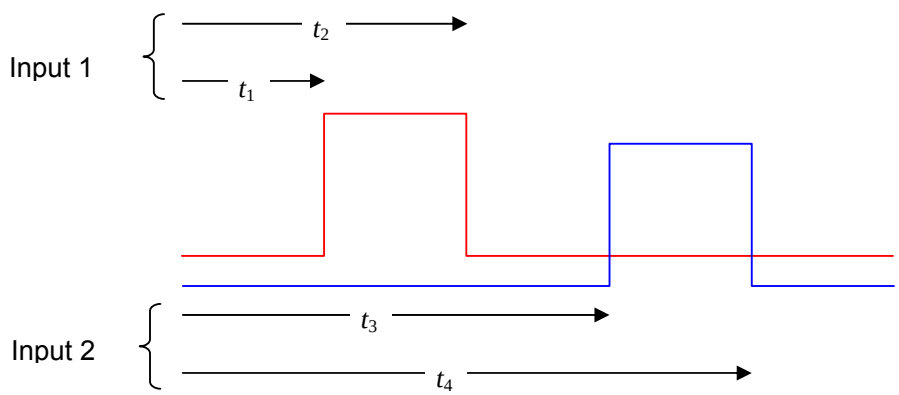
Time measurements

At one gate



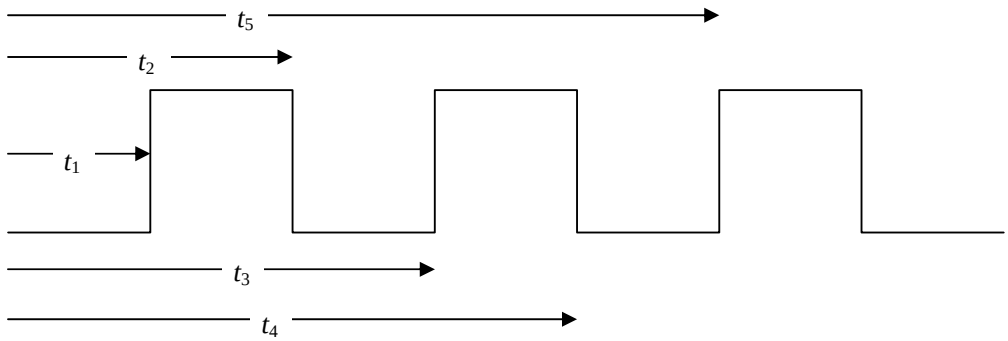
Result: $\Delta t = t_2 - t_1$

Between gates



Result: $\Delta t = t_3 - t_1$

Pendulum (one gate)

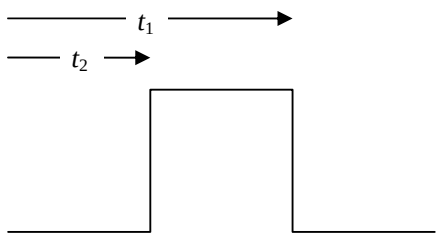


Result: $\Delta t = t_5 - t_1$

VELOCITY

At one gate

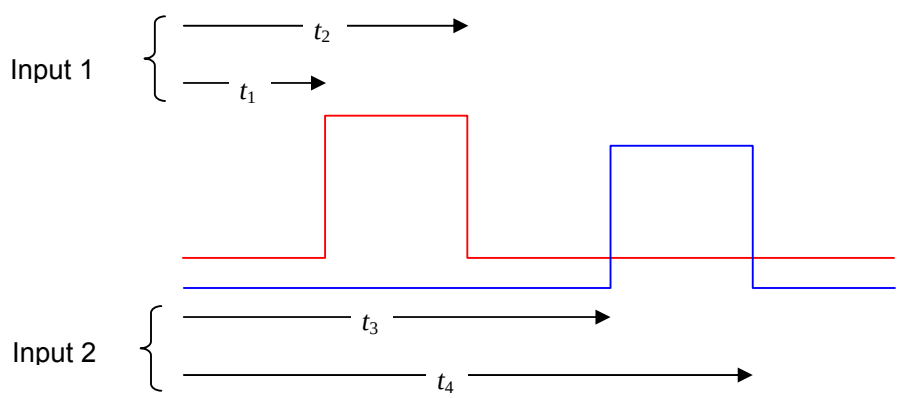
Required parameters: w – The body's width



Result: $v = \frac{w}{\Delta t}; \quad \Delta t = t_2 - t_1$

Between gates

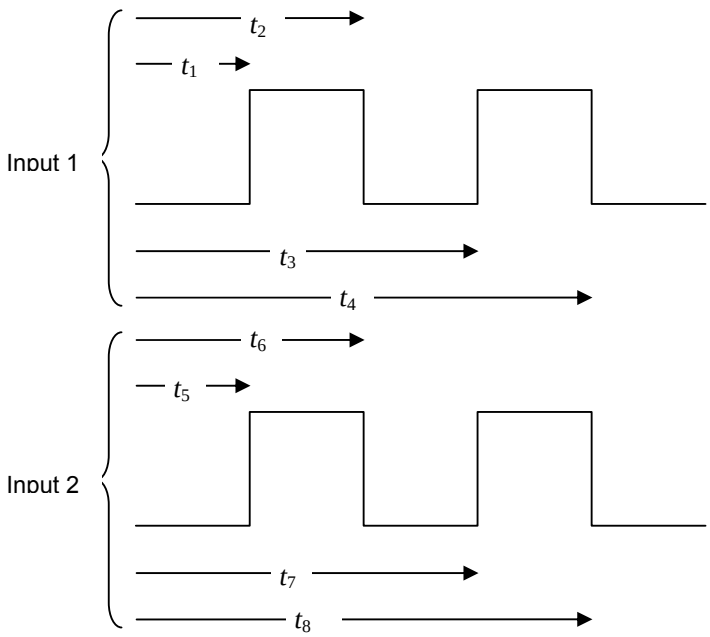
Required parameters: L – The distance between gates



Result: $v = \frac{L}{\Delta t}; \quad \Delta t = t_3 - t_1$

Collision (two gates)

Required parameters: w – The bodies' width (must be identical)



Result:

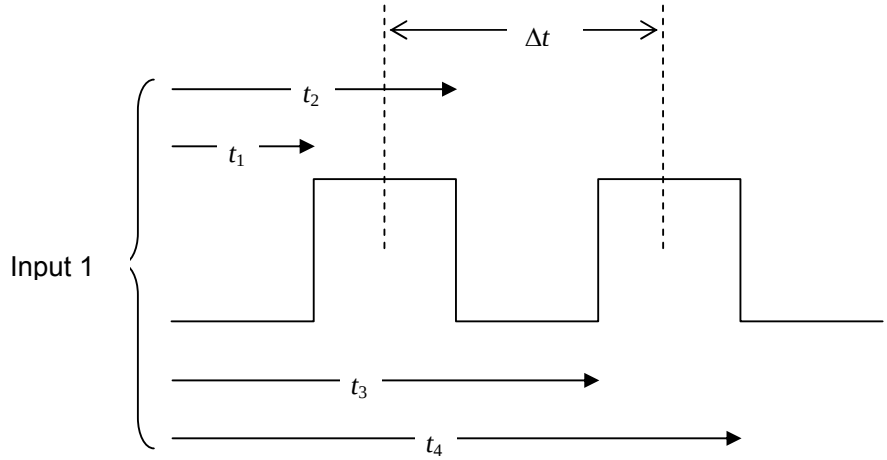
$$u_1 = \frac{w}{t_2 - t_1}; \quad v_1 = \frac{w}{t_4 - t_3}$$

$$u_2 = \frac{w}{t_6 - t_5}; \quad v_2 = \frac{w}{t_8 - t_7}$$

ACCELERATION

At one gate

Required parameters: w – The flags' width



$$v_1 = \frac{w}{t_2 - t_1}; \quad v_2 = \frac{w}{t_4 - t_3}$$

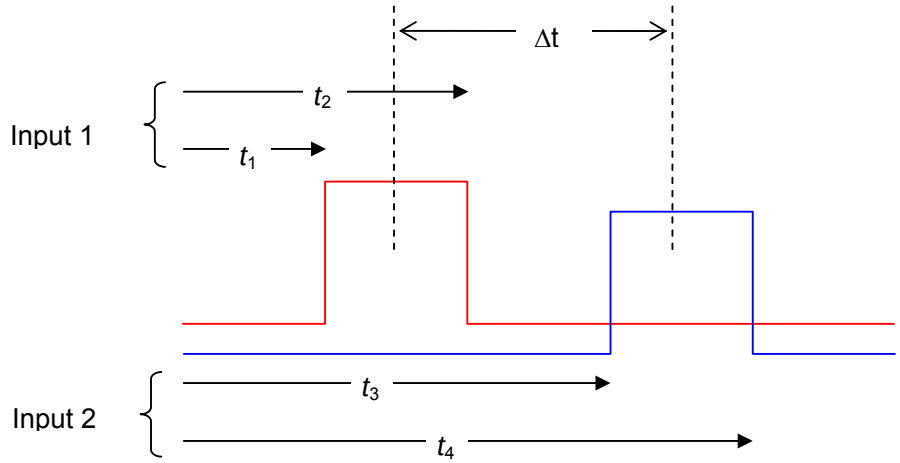
Result:

$$\Delta t = \frac{t_4 + t_3 - t_2 - t_1}{2}$$

$$a = \frac{v_2 - v_1}{\Delta t}$$

Between gates

Required parameters: w – The body's width



$$v_1 = \frac{w}{t_2 - t_1}; \quad v_2 = \frac{w}{t_4 - t_3}$$

Result:

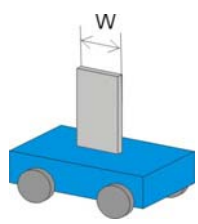
$$\Delta t = \frac{t_4 + t_3 - t_2 - t_1}{2}$$

$$a = \frac{v_2 - v_1}{\Delta t}$$

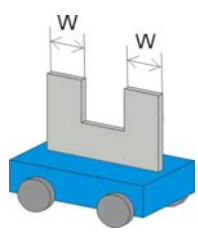
Tips on using the Timing Wizard

- **Attach a flag to the moving body**

When measuring the motion of a moving cart it is convenient to attach a vertical **flag** to the cart (see picture below). You can mount a slotted wooden block on the cart and insert the flag onto the slot, or use masking tape to attach the flag to one side of the cart.



Use a double flag to measure acceleration at one gate. The width of the two flags must be the same.



- **Use the cursors**

Use the cursors (see page 18) to select the graph and data range to which you want to apply the Timing Wizard.

- **Time resolution**

The time resolution depends on the sampling rate. Use the table below to select a rate that meats your needs.

Rate (samples per second)	Resolution
10	0.1 s
25	0.04 s
50	0.02 s
100	0.01 s

500	2 ms
1000	1 ms
2000	0.5 ms
10000	0.1 ms
20800	0.05 ms

- **Use the Trigger**

For fast events and high sampling rates use the Trigger tool (see page 33) to initiate the data logging.

Crop Tool

The Crop tool enables you to trim the edges of a data set. Use it to remove unwanted data or to apply manual curve fitting to a desired range of data points.

The time scale of the trimmed data is shifted so that it will start at $t = 0$.

If you want the automatic curve fit equation to start at $t = 0$, apply the crop tool before any type of curve fitting.

After applying the crop tool, the trimmed data set replaces the original set on the graph display and a new icon is added to the Data Map under cropped data.

To Trim all Data up to a Point

Position a cursor (see page 18) on the data point.

Click on the **Graph** menu, and then click **Crop**.

To Trim all Data Outside a Selected Range

Use the cursors to select the range you want to keep.

Click on the **Graph** menu and then click **Crop**.

Capture Tool

The Capture tool enables you to capture and record specific data points from the data logger's stream of data, like taking snapshots of the dynamic process. For example, you can design a light refraction experiment that will record the angle of the refracted beam only when the light intensity is maximal, and then plot the angle of refraction vs. the manually entered column of the angle of the incident beam.

In capture mode, when you start recording, the data in the graph window appears as it normally does, but every time you click **Capture**, MultiLab enters the most recently recorded value into a new entry in the capture table.

When you have finished capturing data, you can add manual columns to the table and display the data in the graph window.

Preparing the Capture Table

Click **Table** on the menu toolbar, and then click **Capture mode** to open the Capture dialog box:

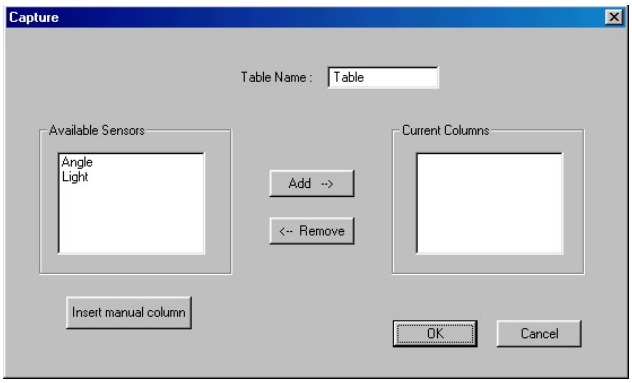


Figure 24: Capture dialog box

In the **Available Sensors** list, select the sensors you want to capture by clicking their name and then click **Add**.

If you want to insert a manual column into the table, click I**nsert manual column** to open the dialog box:

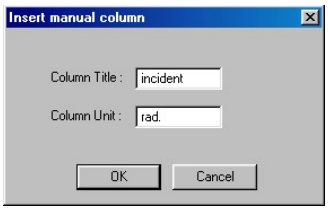


Figure 25: Manual Column dialog box

Enter the column's name and unit, then click **OK**.

The new column is added to the Current Columns list.

Note: You can always add manual columns later by clicking Add manual column on the Table menu.

Click **OK**.

MultiLab opens a new table with the columns you defined.

Enter values into the manual column by clicking a cell and typing the number, then press **Enter** or the down arrow key to move to the next cell:

	Capture 1	Capture 1	
	incident (rad)	Angle (rad)	
	0		
	0.1		
	0.2		
	0.3		
	0.4		

Figure 26: Capture table

Capturing Data

Click **Run**  on the main toolbar to start the data recording process.

Watch the online graph, and when the graph reaches a point you wish to capture, click **Capture**  on the table toolbar. MultiLab enters the momentary data to the capture table sequentially.

If logging stops, simply click **Run** to continue the data recording and capturing.

After you've finished capturing the desired data, click **Table** on the menu toolbar, and then click **Capture mode** to exit capture mode.

Displaying the Captured Data on the Graph

Click **Edit graph**  on the graph toolbar to open the **Graph edit** dialog box:

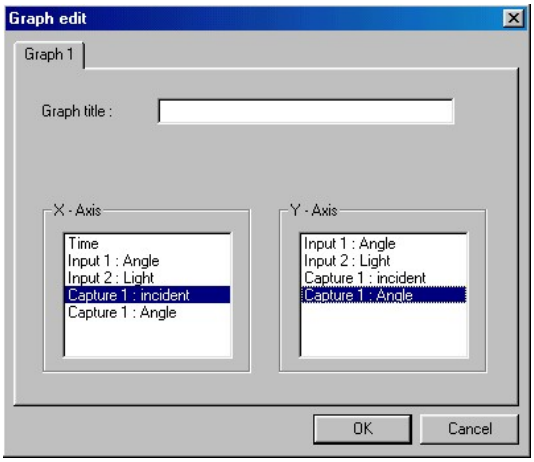


Figure 27: Using Graph edit dialog box to display captured data

Choose the data set that you wish to display on the X–axis from the **X–Axis** list by clicking its name.

Choose the data set or sets you wish to display on the Y–axis from the **Y–Axis** list by clicking its name (or names).

Click **OK**.

Toolbar Buttons

Main (Upper) Toolbar

	**New**	Start new project
	**Open**	Open saved project
	**Save**	Save project
	**Print**	Select print options
	**Run**	Start a data recording session. Click on the down arrow to select recording mode
	**Stop**	Stop recording

	Download	Download the data from the most recent recording session (for MultiLogPRO only)
	Setup	Open the Setup dialog box (for MultiLogPRO)
	Meter setup	Open the Meter setup dialog box
	Data map	Display the Data Map
	Graph	Display the graph window
	Table	Display the table window
	Video	Display and enable the video window
	Video Analysis	Open Video Motion Analyzer module
	Analysis	Open the Analysis wizard
	Linear fit	Draw a line of linear fit of the selected data set
	Derivative	Draw a line of the derivative of the selected curve
	Stop	Stop video recording or video replay
	Play	Replay video and/or data
	Pause	Pause replaying

	Record	Record video

Graph Toolbar

	Add annotation	Add new annotation to the graph
	Move annotation	Relocate the annotation on the graph
	Autoscale	Display all the data
	Zoom to selection	Zoom in to a selected area
	Pan	Pan in all directions while in zoom mode
	More smoothing	Smooth (average) the selected curve
	Less smoothing	Reverse the most recent smoothing operation
	1st Cursor	Display or remove the first cursor
	2nd Cursor	Display or remove the second cursor
	Split graph	Switch to a split graph display
	Edit graph	Select the data to display on the axes
	Graph properties	Graph formatting, scaling and units selection
	Add to project	Add the displayed graph to the project
	Export to Excel	Export the displayed graph to Excel
	Pause/Continue	Pause/continue displaying online data on the graph
	Add prediction	Add a prediction onto the graph
	Erase prediction	Erase the selected prediction

Table Toolbar

	Edit table	Add/remove columns from the table
	Table properties	Format fonts and units
	Add to project	Add the current table to the project
	Capture	Capture the momentary data

Video Motion Analyzer Toolbar

	Open	Open movie
	Capture	Capture new movie
	Set origin	Set the origin of reference frame
	Rotate axes	Set direction of reference frame
	Scale	Scale the data to real dimensions
	Delete point	Delete a point and point's coordinates
	Color settings	Select colors of markers and axes
	Go to start	Go to first frame
	Previous frame	Move to previous frame
	Play	Play the movie
	Next frame	Move to next frame
	Go to end	Go to last frame

Capture New Movie Toolbar

	Record	Start video capture
	Stop	Stop video capture
	Analyze movie	Return to Video Motion Analyzer mode

System Requirements

Windows

Software

- Windows XP, Vista, Windows 7, 32 and 64 bit
- Internet Explorer 6.0 or later
- Multilab 1.6 or higher

Hardware

- Pentium Dual Core 1.2 GHz or higher
- 512 MB RAM (2 GB recommended)
- 200 MB available disk space for the MultiLab application
- One USB port for the logger
- To work with the video window your system should be equipped with a video
- camera
- To be able to record and play voice comments your system a microphone is required

Mac OS

Software

- Mac OS X 10.4 – 10.6.8
- Safari browser 4.0 or higher
- Multilab version 2.5 or higher

Hardware

- Intel based Macintosh Dual Core 1.2 GHz or higher
- 512 MB RAM (2 GB recommended)
- 200 MB available disk space for the MultiLab application
- One USB port for the logger
- To work with the video window your system should be equipped with a video
- camera
- To be able to record and play voice comments your system a microphone is required

Troubleshooting Guide

Symptom	*Cause of Problem*
I can't see a video of the experiment	• The video window is closed. Click **Display video** on the main toolbar to open it. • The video is disabled, select **Enable video** from the **Video** menu. • The camera driver is not properly installed.
I can't record voice comments	• The microphone volume is muted. Open Window's **volume control** and uncheck the **Mute** check box at the bottom of the **Mic** volume column.

Fourier Systems Ltd.
16 Hamelacha St., POB 11681, Rosh Ha'ayin 48091,
Tel: +972-3-901-4849, Fax: +972-3-901-4999
www.FourierEdu.com

EXPERIENCE SCIENCE!

I opened MultiLab but the application immediately closed down.	• During MultiLab installation, the component Windows Media Encoder (WME) is installed. Without WME, MultiLab cannot run. • In the Control Panel, open Add or Remove Programs. Check if WME is installed on your computer. • If WME is installed, check the WME version number. • MultiLab v1.4.06 and higher supports up to WME Series 9. If you are running WME 9, make sure your MultiLab version is compatible. If not, contact your Fourier vendor for a MultiLab upgrade. • If WME is not installed then: • Open Windows Explorer and • right-click Explore on your CD drive. • Locate the Windows Media • Encoder installer file in the MultiLab • folder and double click this file. • Follow the on-screen instructions • to install WME. You will now be able • to run MultiLab. • If you have WME 9 installed on a non-compatible version of MultiLab, you can always uninstall WME 9 and install WME 7 located on

	the MultiLab CD.
I opened MultiLab. An error message opened up stating that I needed Quicktime in order to run the MultiLab software.	• Quicktime is a required component for MultiLab software to operate and is installed during initial MultiLab installation. • If Quicktime has since been removed from your system, you may install it again from your MultiLab Installation CD: • Place the CD in your computer's CD/DVD drive and click Cancel to stop automatic MultiLab installation. • Open Windows Explorer and right-click Explore on your CD/DVD drive. • Locate the Quicktime installer file in the MultiLab folder and double click this file. • Follow the on-screen instructions to install Quicktime. You will now be able to run MultiLab. • NOTE: You can also download the latest version of QuickTime from Apples web site.

Sensors

Symptom	*Cause of Problem*
I see voltage units when sampling using the microphone.	• The microphone is monitoring the sound wave shape, which is displayed in voltage units. The sound level, measured in decibel values, is another mathematical description of the sound amplitude.
The recorded data is *noisy*	• With voltage or current sensors, use short connections between the

	circuit being tested and the sensor. In some cases it is advisable to connect the (-) of the current sensor to the ground terminal • Avoid working near strong electromagnetic fields. (e.g. engines, fluorescent lights)
I turned the calibration screw but did not obtain the accurate calibration	• The calibration screw is of high-resolution type and has 15 turns. Try to continue turning the screw, or change the direction of rotation.
I am having trouble setting up my sensor properly.	• Check http://www.fourieredu.com for a list of all Fourier sensors along with detailed set up guides for each one.

Technical Support:

Fourier Help Desk:

E-mail: support@fourieredu.com

Contact information:

1 866 771 6682 (toll-free from within USA only)
1 708 478 5333
Hours of Operation:
Monday - Friday, 9AM to 5PM (UTC -06:00)

Index

S

T

U

V

W

X

Y

Z