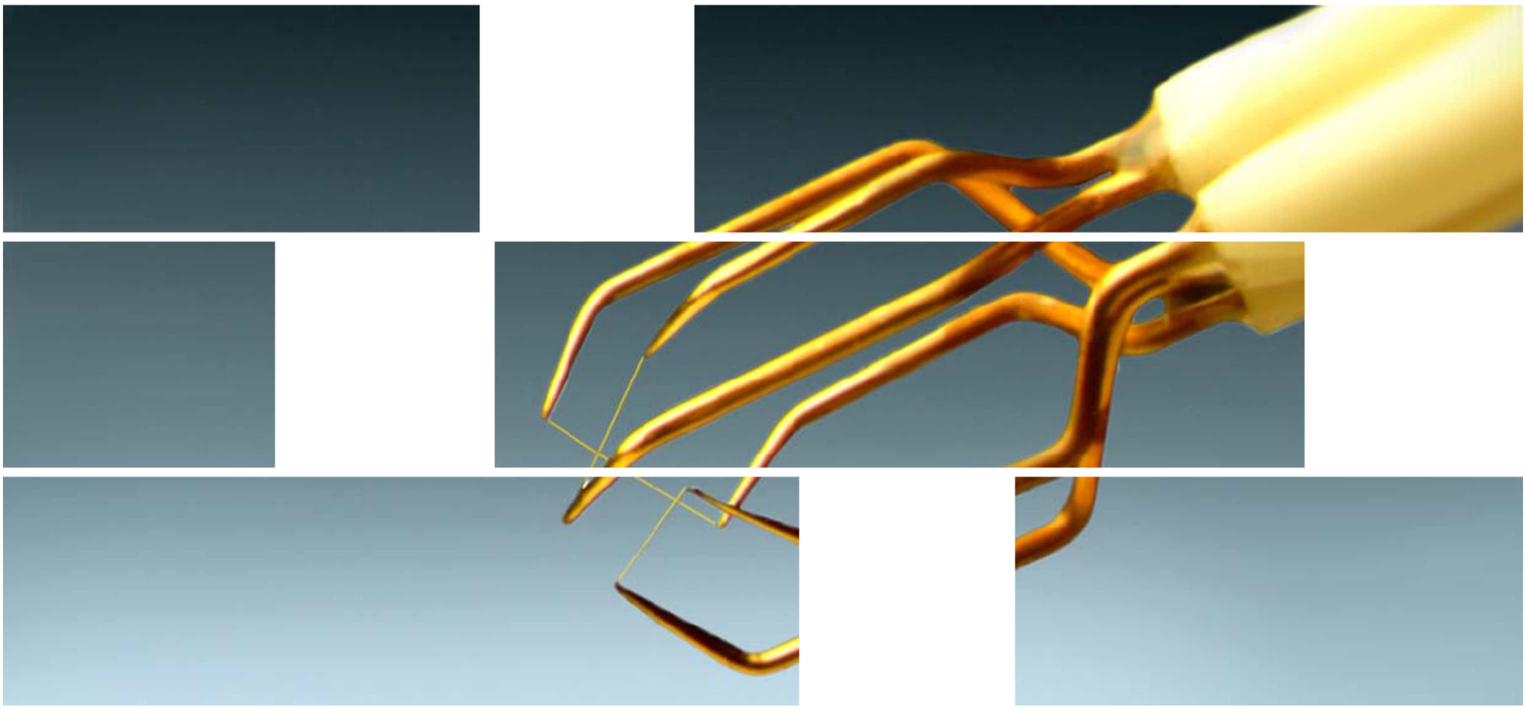
DANTEC DYNAMICS



MiniCTA Software v4.10

Installation and User's Guide

Build no.: 4.10.00.46. Publication no.: 9040U3947. Date: 24-09-2010.

Table of Content

1 Introduction

1.1 The MiniCTA Software

The MiniCTA Software is an application software designed for use primarily with the Dantec Dynamics MiniCTA anemometer. It can also be used together with any other non-computer-controlled anemometer.

With the MiniCTA Software System you can perform measurements of instantaneous fluid velocity and its statistic derivatives: Mean Velocity, Turbulence Intensity, Higher order Moments and Spectra etc., in one or more points in space.

It supports probe calibration with curve fitting and provides a Setup platform, where you can define probe setup and execute automatic measurement sequences with probe traversing, data acquisition and data reduction. A built-in Project Manager gives full documentation of all measurement procedures and results.

The MiniCTA Software System can be used with hot-wire and hot-film probes, Dantec Dynamics standard- as well as custom designed probes. It accepts any combination of 1-, 2- and 3-D probes. The software can also be used in combination with other transducers or instruments for parameters other than fluid velocity, e.g. temperature or pressure.

1.1.1 Documentation

- MiniCTA Manual - This manual, see below.
- How to Measure Turbulence with Hot-Wire Anemometers - A Practical Guide - 9040U6154 This guide describes the fundamental of measuring turbulence with the hot-wire technique. Including measurement equipment, planning of the experiment, setup and configuration, calibration and data acquisition and conversion using the software.
- OEM Documentation - A number of OEM supplied manuals and guides are delivered together with the devices delivered for your system. This includes A/D devices, etc.

Contents of this Manual

This manual consists of sections presenting the system and software and how to use it.

- "Software Installation" on page 10
  Describes what you need to run the MiniCTA Software application software, and how it is unpacked and installed.
- "Hardware Installation" on page 12
  Describes how to install the CTA hardware in connection with the MiniCTA software.
- "Operating the System" on page 14
  A Step-by-step description of three Sample Projects.
- "MiniCTA Software Guide" on page 36
  User's Guide that gives you an overview of the software and describes how to use it and interpret the results.
- "Software Reference" on page 143
  A Software Reference with definitions of terms, formulas and messages.

On-line Help

The On-line Help is a context sensitive Help system built into in the MiniCTA application software. It provides you with a quick reference for procedures, keystroke sequences and commands that you will need in order to run the MiniCTA System. The Help can be used within the Application Software.

1.1.2 Assumptions

It is assumed that you have a basic knowledge about measurement techniques in Fluid Mechanics and that you are familiar with the concept of Constant Temperature Anemometry.

It is also assumed that you are familiar with Windows Terminology.

2 Software License Agreement

This software end user license agreement ("License Agreement") is concluded between you (either an individual or a corporate entity) and Dantec Dynamics A/S ("Dantec Dynamics"). Please read all terms and conditions of this License Agreement before installing the Software. When you install the Software, you agree to be bound by the terms of this License Agreement. If you cannot agree to the terms of this License Agreement you may not install or use the software in any manner.

Grant of License

This License Agreement together with the Software package including eventual media, user's guide and documentation and the invoice constitutes your proof of license and the right to exercise the rights herein and must be retained by you.

One license permits you to use one installation at a time of the Dantec Dynamics software product supplied to you (the "Software") including documentation in written or electronic form and solely for your own internal business purposes.

You may not rent, lease, sublicense or otherwise distribute, assign, transfer or make the Software available to any third party without the express consent of Dantec Dynamics except to the extent specifically permitted by mandatory applicable law.

Updates

Updates, new releases, bug fixes, etc. of the Software which are supplied to you (if any), may be used only in conjunction with versions of the Software legally acquired and licensed by you that you have already installed, unless such update etc. replaces that former version in its entirety and such former version is destroyed.

Copyright

The Software (including text, illustrations and images incorporated into the Software) and all proprietary rights therein are owned by Dantec Dynamics or Dantec Dynamics' suppliers, and are protected by the Danish Copyright Act and applicable international law. You may not reverse assemble, decompile, or otherwise modify the Software except to the extent specifically permitted by mandatory applicable law.

You are not entitled to copy the Software or any part thereof except as otherwise expressly set out above. However you may make a copy of the Software solely for backup or archival purposes. You may not copy the user's guide accompanying the Software, nor distribute copies of any user documentation provided in “on-line” or electronic form, without Dantec Dynamics' prior written permission.

License and Maintenance Fees

You must pay any and all licensee and maintenance fees in accordance with the then-current payment terms established by Dantec Dynamics.

Limited Warranty

You are obliged to examine and test the Software immediately upon your receipt thereof. Until 30 days after delivery of the Software, Dantec Dynamics will deliver a new copy of the Software if the medium on which the Software was supplied is not legible.

A defect in the Software shall be regarded as material only if it has a material effect on the proper functioning of the Software as a whole, or if it prevents operation of the Software in its entirety. If until 90 days after the delivery of the Software, it is established that there is a material defect in the Software, Dantec Dynamics shall, at Dantec Dynamics' discretion, either deliver a new version of the Software without the material defect, or remedy the defect free of charge or terminate this License Agreement and repay the

license fee received against the return of the Software. In any of these events the parties shall have no further claims against each other. Dantec Dynamics shall be entitled to remedy any defect by indicating procedures, methods or uses ("work-arounds") which result in the defect not having a significant effect on the use of the Software.
Software is inherently complex and the possibility remains that the Software contains bugs, defects and inexpediencies which are not covered by the warranty set out immediately above. Such bugs, defects and inexpediencies etc. shall not constitute due ground for termination and shall not entitle you to any remedial action including refund of fees or payment of damages or costs. Dantec Dynamics will endeavour to correct bugs, defects etc. in subsequent releases of the Software.
The Software is licensed "as is" and without any warranty, obligation to take remedial action or the like thereof in the event of breach other than as stipulated above. It is therefore not warranted that the operation of the Software will be without interruptions, free of bugs or defects, or that bugs or defects can or will be remedied.

Indemnification

Dantec Dynamics will indemnify you against any claim by an unaffiliated third party that the Software infringes such unaffiliated third party's intellectual property rights and shall pay to you the amount awarded to such unaffiliated third party in a final judgment (or settlement to which Dantec Dynamics has consented) always subject however to the limitations and exclusions set out in this paragraph.
You must notify Dantec Dynamics promptly in writing of any such claim and allow Dantec Dynamics to take sole control over its defense. You must provide Dantec Dynamics with all reasonable assistance in defending the claim.
Dantec Dynamics' obligation to indemnify you shall not apply to the extent that any claim comprised by this paragraph is based in whole or in part on (i) any materials provided directly or indirectly by you; (ii) your exploitation of the Software for other purposes than those expressly contemplated in this License Agreement; and/or (iii) combining of the Software with third party products. You shall reimburse Dantec Dynamics for any costs or damages that result from such actions.
If Dantec Dynamics receives information of an alleged infringement of third party intellectual property rights or a final adverse judgment is passed by a competent court or a final settlement consented to by Dantec Dynamics is reached regarding an intellectual property right infringement claim related to the Software, Dantec Dynamics may (but shall not under any obligation to do so), either (i) procure for you the right to continue to use the Software as contemplated in this License Agreement; or (ii) modify the Software to make the Software non infringing; (iii) replace the relevant portion of the Software with a non infringing functional equivalent; or (iv) terminate with immediate effect your right to install and use the Software against a refund of the license fees paid by you prior to termination. You acknowledge and agree that Dantec Dynamics is entitled to exercise either of the aforesaid options in Dantec Dynamics' sole discretion and that this constitutes your sole and exclusive remedy in the event of any infringement of third party intellectual property rights.

Limitation of Liability

Neither Dantec Dynamics nor its distributors shall be liable for any indirect damages including without limitation loss of profits and loss of data or restoration hereof, or any other incidental, special or other consequential damages, and even if Dantec Dynamics has been informed of their possibility. Further, Dantec Dynamics disclaims and excludes any and all liability based on Dantec Dynamics' simple or gross negligent acts or omissions. In addition to any other limitations and exclusions of liability, Dantec Dynamics' total aggregate liability to pay any damages and costs to you shall in all events be limited to a total aggregated amount equal to the license fee paid by you for the Software.

Product Liability

Dantec Dynamics shall be liable for injury to persons or damage to tangible items caused by the Software in accordance with those rules of the Danish Product Liability Act, which cannot be contractually waived. Dantec Dynamics disclaims and excludes any liability in excess thereof.

Assignment

Neither party shall be entitled to assign this License Agreement or any of its rights or obligations pursuant this Agreement to any third party without the prior written consent of the other party. Notwithstanding the aforesaid, Dantec Dynamics shall be entitled to assign this License Agreement in whole or in part without your consent to (i) a company affiliated with Dantec Dynamics or (ii) an unaffiliated third party to the extent that such assignment takes place in connection with a restructuring, divestiture, merger, acquisition or the like.

Term and Termination

Subject to and conditional upon your compliance with the terms and conditions of this License Agreement may install and use the Software as contemplated herein.
We may terminate this License Agreement for breach at any time with immediate effect by serving notice in writing to you, if you commit any material breach of any terms and conditions set out in this License Agreement. Without limiting the generality of the aforesaid, any failure to pay fees and amounts due to Dantec Dynamics and/or any infringement of Dantec Dynamics' intellectual property rights shall be regarded a material breach that entitles Dantec Dynamics to terminate this License Agreement for breach.

Governing Law and Proper Forum

This License Agreement shall be governed by and construed in accordance with Danish law. The sole and proper forum for the settlement of disputes hereunder shall be that of the venue of Dantec Dynamics. Notwithstanding the aforesaid, Dantec Dynamics shall forthwith be entitled to file any action to enforce Dantec Dynamics' rights including intellectual property rights, in any applicable jurisdiction using any applicable legal remedies.

Questions

Should you have any questions concerning this License Agreement, or should you have any questions relating to the installation or operation of the Software, please contact the authorized Dantec Dynamics distributor serving your country. You can find a list of current Dantec Dynamics distributors on our web site: www.dantecdynamics.com.

Dantec Dynamics A/S
Tonsbakken 16-18 - DK-2740 Skovlunde, Denmark
Tel: +45 4457 8000 - Fax: + 45 4457 8001
www.dantecdynamics.com

3 Software Installation

This section defines the hardware and software environment that you need in order to install the MiniCTA Software Application software and it describes the installation procedure. The MiniCTA Software user interface and their related Icons are explained.

Installation and setup of A/D devices supported by MiniCTA is described in "A/D Devices" on page 159.

3.1 Installation of the MiniCTA Software

3.1.1 Unpacking

The MiniCTA Software Application software is delivered in a box containing a CD-ROM together with a dongle and a license agreement.

3.1.2 System Requirements

- PC with a modern multi-core processor.
- Installation must be performed by an account with admin privileges.
- Microsoft® Windows© 7 x86/x64 with latest updates.
  - or -
  Microsoft® Windows© Vista x86/x64 with latest updates.
  - or -
  Microsoft® Windows© XP with latest updates.
  - or -
  Microsoft® Windows© 2000 with latest service packs.
- Microsoft® Windows© Installer v3.0 or later.
- Microsoft® Internet Explorer 6 or later with latest security updates.
- 512 MB of RAM minimum.
- 200 MB of available hard-disk space minimum; 1 GB for running acquisitions.
- DVD/CD-ROM drive.
- Super VGA (800x600) or higher-resolution monitor with 65.000 colors or more.
- USB or Parallel port for dongle.
- Mouse or compatible pointing device.

Microsoft® Windows© 95, 98, 98SE, Me, and NT operating systems are not supported.

3.1.3 Installation

To install the MiniCTA Software software, insert the CD-ROM into the CD-ROM drive of the PC and execute the SETUP.EXE file. This will start the installation program. If your PC supports the autostart feature, the setup program will start automatically. You will be asked for the hardware key (dongle) password during the installation. This is shown on a page supplied with the software.

It is recommended that you follow the guidelines in the installation routine. Select the path where you wish to install the files. The default directory path is:

C:\ PROGRAM FILES\DANTEC DYNAMICS\MINICTA SOFTWARE\

When you have finished the installation, you can start the program by double clicking on the MiniCTA Software icon in the Start Program -> Dantec Dynamics group.

3.1.4 Software Options

Programmer's Toolkit

The software structure allows interfacing with external devices and software code. If you want to use OEM A/D Devices or Traverse Systems that are not supported by Dantec Dynamics or you want to create your own Extended Processing code, the Programmer's Toolkit provides you with a set of templates for writing customized translation drivers. The writing of drivers requires experience in programming MS Windows drivers. The Programmer's Toolkit has to be ordered separately.

4 Hardware Installation

This section describes how to connect an anemometer to the MiniCTA software via an A/D device.

4.1 Anemometer Systems

4.1.1 MiniCTA

The MiniCTA anemometer is a self-containing unit, where all settings are done by means of dip-switches and jumpers on the CTA board. The anemometer is powered from a separate power adaptor. The probe is connected to the MiniCTA and the overheat ratio, gain and offset are adjusted in accordance with the MiniCTA manual. The output bushing is connected to the input channel of the A/D device normally via a connector box delivered together with the A/D device.
For information see the MiniCTA User's Guide.

4.1.2 Multichannel CTA

The Multichannel CTA is a self-contained unit, and the set-up of the individual CTA channels are done as in the MiniCTA. The output signals are available from a multi-pin connector via a flat cable to the A/D device input.
The Multichannel CTA is available in two versions with 8 and 6 CTA channels, respectively. The 6-channel version is equipped with a temperature probe (output channel 6) and an optional velocity reference probe (output channel 7) for in situ probe calibration in e.g. a wind-tunnel. For further information see the Multichannel CTA User's Guide.

4.1.3 Other CTA Systems

The MiniCTA Software may be used with any CTA anemometer, which has an analog output. The connection to the A/D input is done as in the case of the MiniCTA.

4.1.4 Temperature Probe

If you want to utilize the automatic temperature correction feature in the software, an input from a temperature probe measuring the ambient temperature is needed. This may be any type of transducer, e.g. a thermistor or a Pt-100 element, as long as it has an analog output.

4.1.5 Velocity Reference Probe

The velocity reference probe type 54T29 can be connected to the 54N81 Multichannel CTA and used for multichannel calibration of hot-wire probes at velocities up to 30 m/s.

4.2 Connecting Probes and making Grounding Precautions

Grounding the Probe(s)

Probes for measurement in air or other gases should not be grounded. If you have installed a film probe in liquid then connect the shield of the probe input BNC connector at the CTA to a grounding electrode placed in the liquid as close to the probe as possible. See See "Configuring the System" on page 79.

Warning
The BNC connectors on the Probe Cables are electrically part of the cable shield. Therefore avoid contact

between the transducer/cable BNC connector assembly and electrically conducting structures of the experimental equipment (windtunnel or the like).

4.3 Connecting Anemometer Output to A/D Input

The output voltage(s) from the anemometer(s) and outputs from possible temperature and pressure transducers can be connected to the A/D input in different ways.
As a rule of thumb you can do as follows:

- If the power supplies of the anemometers and additional transducers are insulated from each other use Single-ended Referenced input.
- If the power supplies are not insulated from each other use Differential input.

In all cases it is recommended to check the actual A/D device manual.

If you are using a Notebook computer, it is advised to connect the reference ground to a suitable ground. This is due to the fact that a Notebook does not have a ground chassis.

5 Operating the System

This section demonstrates how to operate the MiniCTA System by way of three sample projects:

- 1-D measurement with default setup, calibration, data acquisition and data reduction.
- 1-D measurement with temperature correction.
- 1-D measurement with traverse.

5.1 System Startup

At this point it is assumed that you have installed the system as described in See "Hardware Installation" on page 12.

Starting the MiniCTA

This section directs you to the working platform of the MiniCTA application software.

1. Create a folder for the database and project.
2. Double click on the MiniCTA Icon in the MiniCTA Group.

MiniCTA opens, and the system is now ready for operation.

Note
It recommended to have only one database with only one project in each folder. This makes it much easier to move the project to another PC for further manipulation.

Switching on the MiniCTA

Note
You should **not** connect the power adaptor to the MiniCTA, before the probe is connected and the decade is adjusted to proper overheat ratio.

5.2 Sample Project I
1D Measurement in a Point

This section describes a system with one probe. The anemometer output is connected to an input channel of the A/D device in the PC. See See "A/D Devices" on page 159.

5.2.1 Project Description

The project demonstrates:

- Default setup
- Probe calibration
- Data acquisition
- Data reduction and display.

It is assumed that the ambient temperature remains constant or nearly constant during the project. If temperature variations more than a few °C are expected, or you require high accuracy, please refer to Sample project II, where procedures for temperature correction is outlined.

5.2.2 Hardware list for Sample Project I

- 55P11 Probe
- 55H20 Probe Support
- 9055A1863 Probe Cable, 4-m.
- 55T30 MiniCTA anemometer.
- Cable and Connector Box for the A/D converter board.
- PC with A/D device installed.
- A fan or blower to create an air flow.

5.2.3 Physical Configuration of a 1-D CTA

Physical configuration covers the interconnections of the PC , MiniCTA Frame and probe. It consists of the following steps:

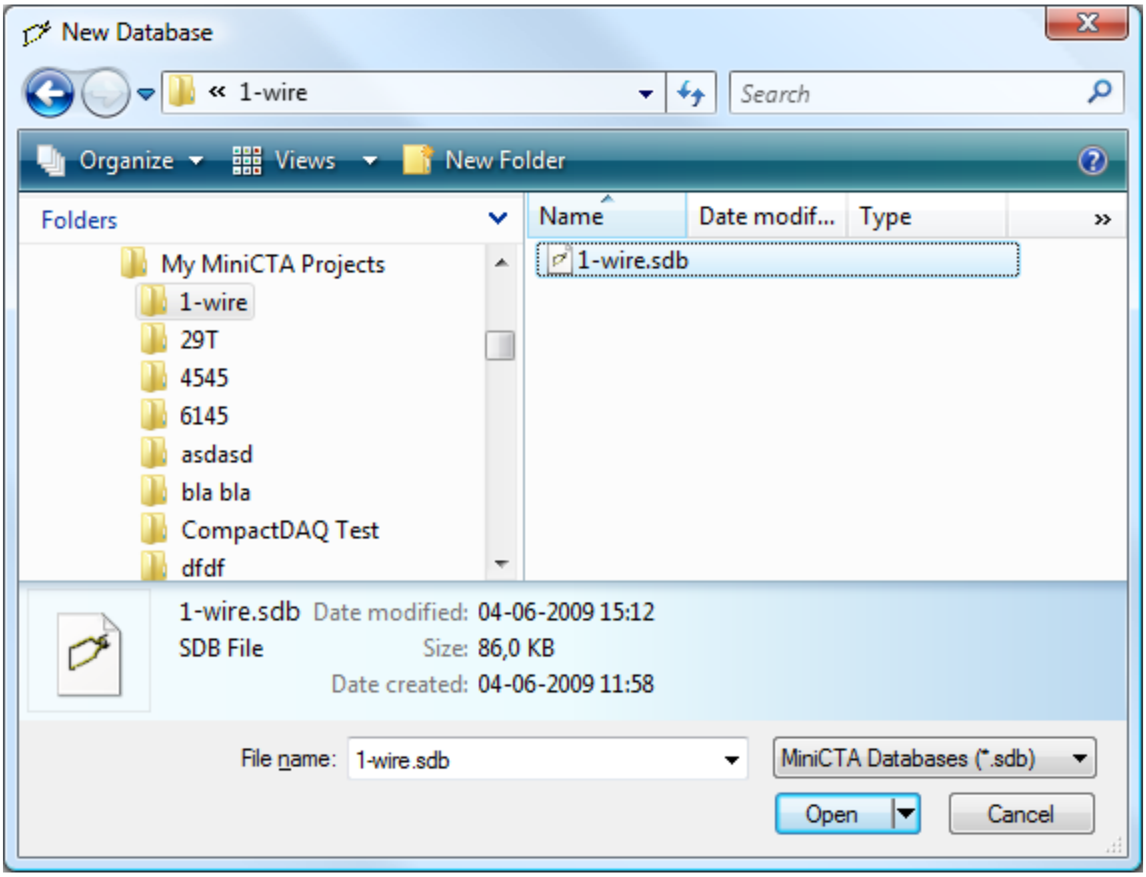
1. Connect the Analog Output bushing on the MiniCTA to the A/D device input channel no. 0 on the connector box for the A/D device with a 50 ohms BNC cable (max. 50 m).
2. Connect Probe and Support with a 4 m Cable to the Probe BNC-connector on the MiniCTA.
3. Place the probe in front of the fan.
4. Switch power on to both systems.

5.2.4 Open MiniCTA Software

Open MiniCTA software by double clicking on the MiniCTA Icon in the Program Manager.

You are now prompted to create a new database.

5.2.5 Create a Database



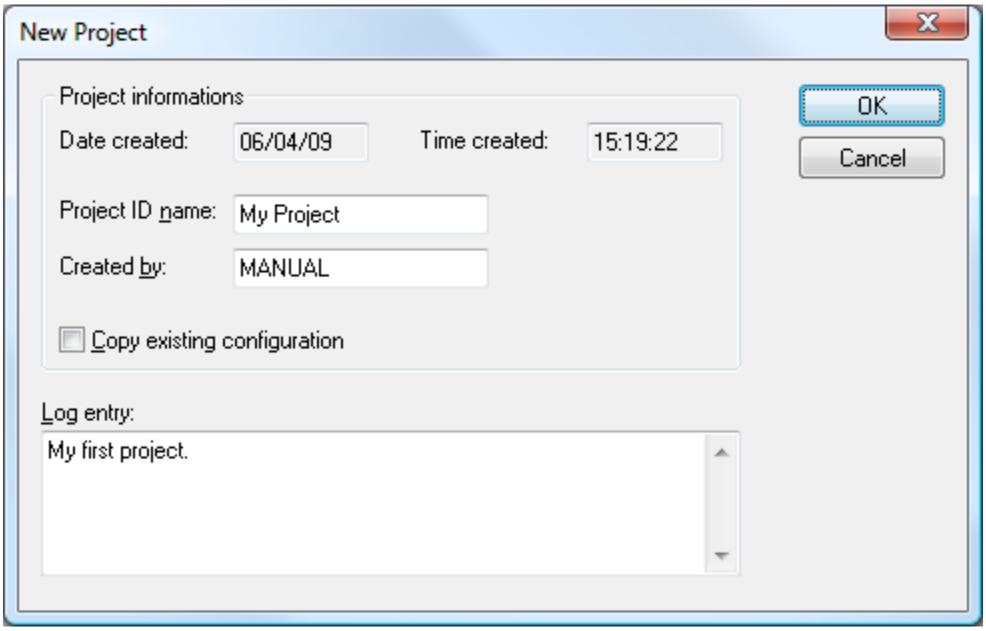
1. Choose Database/New from the File menu.
   New Database dialog box opens.
2. Select the folder that you already have created for the purpose.
3. Type in the name of your new data base, e.g. 1-wire.sdb.
4. Choose OK.

5.2.6 Create a Project

You are now prompted to create a Project. Select Yes. A New Project dialog box opens. (You can also create a new project by selecting Project/New in the File menu).

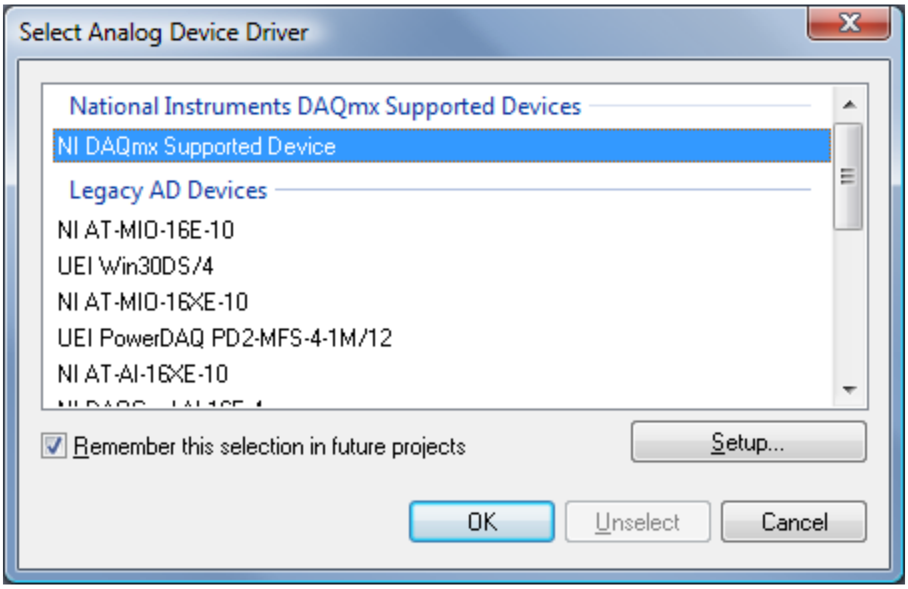
The project will contain information about all actions that you carry out from now on.

1. Enter the name of your project and your initials in the Created by field.
2. Add your comments in the Log entry, if wanted.
3. Choose OK.

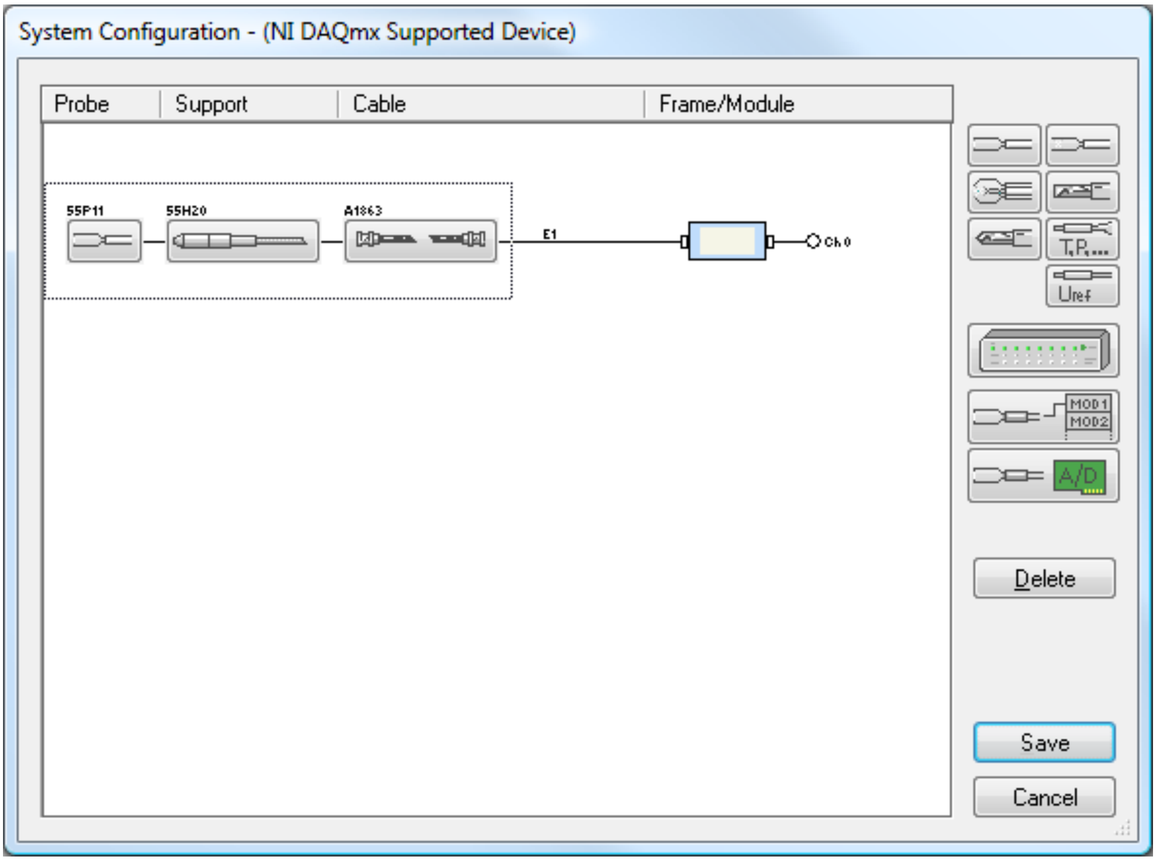


5.2.7 Define Devices

You are now prompted to select an A/D device. Say Yes and the Select AD driver dialog box opens. (You can also define a A/D driver by selecting the Devices, A/D drivers from the Configuration menu.)



1. Select the A/D device that is installed in the PC.
2. Choose OK.



A path is now made from the project to the translation driver.

5.2.8 System Configuration

You are now prompted to configure the system. Select Yes and the System Configuration dialog box opens.

Now select probe, cable/support and A/D input channel.

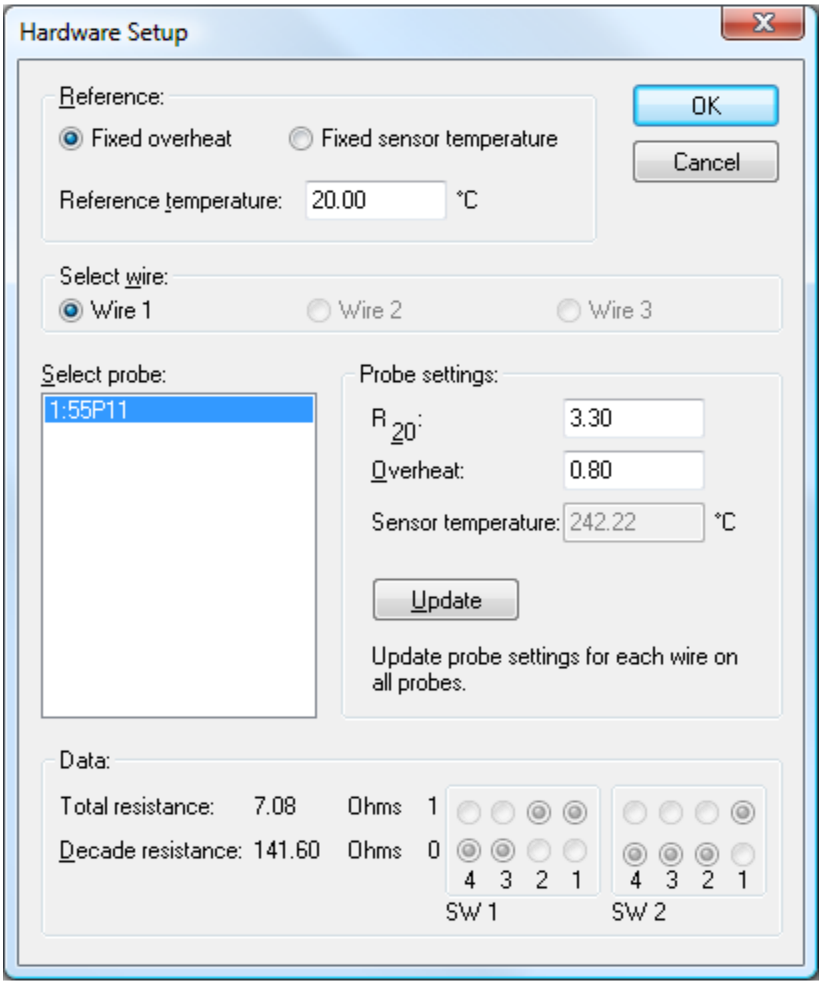
1. Click on the Single Wire sensor probe Icon.
2. Select Probe, Support and Cable dialog box from the Probe library opens.
3. Select 55P11 Wire, 55H20 Support and a 1864 BNC/BNC Cable in the list boxes.
4. Choose OK.
5. The Probe, Support and Cable are now added to the map. It is by default connected to A/D input channel 0.
6. Choose OK.

The CTA configuration is now finished and saved into the project.

You are now prompted to define the Setup.

5.2.9 Define Hardware Setup

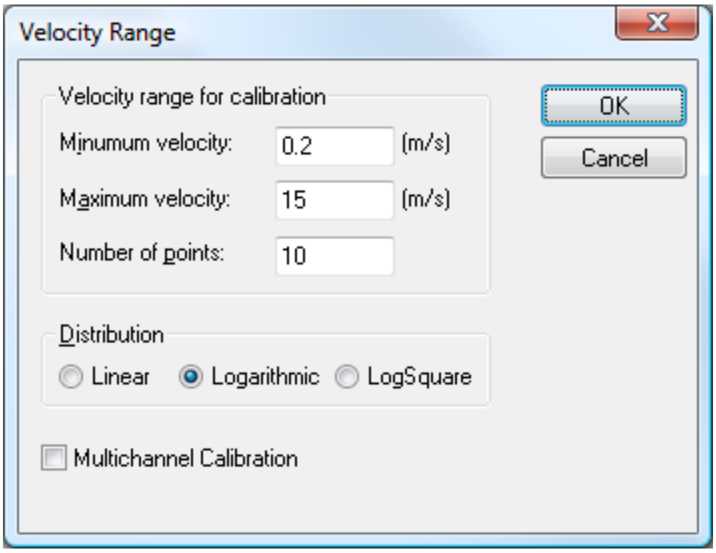
You are now prompted to define the setup of the CTA.



Reference temperature and overheat from the probe library are displayed.

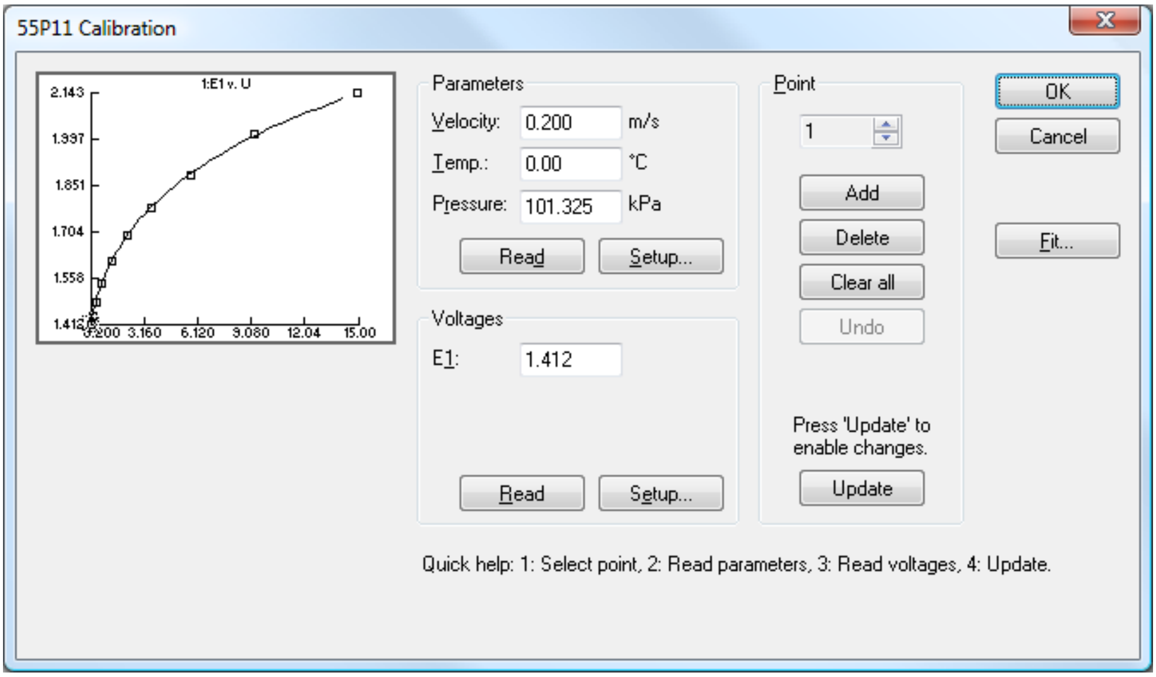
1. Enter the sensor cold resistance R20 at 20 °C from the label on the probe container.
2. Click Update and the probe operating data and the corresponding dip switch settings in the MiniCTA are displayed.
3. Click on OK and you are now prompted to carry out a velocity calibration of the probe.

5.2.10 Velocity Calibration



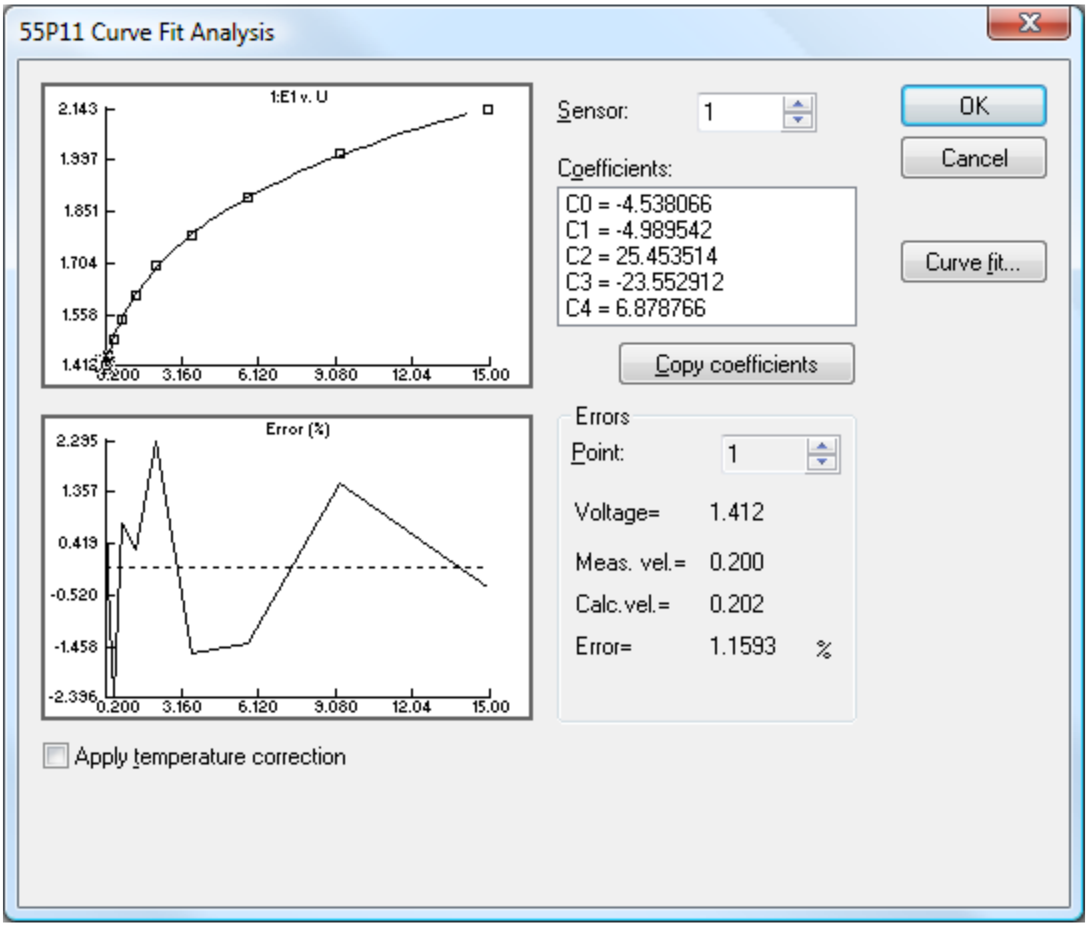
It is assumed that you have some calibration means, e.g. a free jet or a windtunnel with e.g. a pitotstatic tube as velocity reference.

1. The Velocity Range dialog box opens.
2. Enter min. and max. velocity and number of calibration points.
3. Leave the Log distribution, as it gives the best linearization accuracy over a wide velocity range.
4. Select OK. The Calibration dialog box opens.



1. Select Point 1. Create the velocity displayed in the Velocity field. If you obtain a different, but acceptable, velocity, you can enter the actual value from the keyboard.

2. Enter the temperature into the Temperature field from the keyboard.
3. Click on the Parameters Read button.
4. Click on the Voltages Read button. The probe voltage from the CTA is now read via the A/D device.
5. Click on the Update button. The velocity, temperature and probe voltage are now placed in the data sheet.
6. Select next point and continue until all points are done.



Curve Fit Analysis dialog.

7. Click on the Fit button. The Curve fit analysis dialog box opens. It displays the curve fit coefficients and the linearization errors. In this project please neglect the Temperature correction check box.
8. If you accept the quality of the calibration click on the OK button. The completed calibration data sheet is now displayed.

1:55P11 Velocity Calibration Data - [UNTITLED]

	U	E1	T(C)	P(kPa)	E1corr	U1calc
1	0.200	1.412	26.800	0.000	1.412	0.202
2	0.323	1.444	26.900	0.000	1.444	0.322
3	0.522	1.486	26.700	0.000	1.486	0.510
4	0.843	1.546	26.900	0.000	1.546	0.850
5	1.363	1.614	26.800	0.000	1.614	1.367
6	2.202	1.698	26.900	0.000	1.698	2.252
7	3.557	1.783	26.900	0.000	1.783	3.500
8	5.747	1.888	27.000	0.000	1.888	5.665
9	9.284	2.014	26.900	0.000	2.014	9.424
10	15.000	2.143	26.800	0.000	2.143	14.942
11						
12						
13						

Close and save the calibration event

1. Double-click in the Menu Control box in the upper left corner of the data sheet window. A Save event dialog box opens.
2. Type in an identification and select OK. The dialog box closes and you are prompted to create a Conversion event on basis of the probe calibration.
3. Say Yes. A Save event dialog box opens.
4. Type in an identification and select OK. The dialog box closes and you are prompted to make the Conversion event default. Say Yes.

A Calibration event and a Conversion/reduction event (both with the default stars) are added to the Project Manager.

You have now a complete Default setup with a Hardware setup and a Conversion/Reduction setup that can be used to acquire, linearize and reduce probe voltages into first and second order moments (mean and standard deviation).

Linearization on Basis of Previous Calibration Data (Voltage vs. Velocity)

If you have a set of calibration data for the probe created at earlier with the same overheat ratio and ambient temperature, you can skip the input dialog and simply enter them into the data sheet.

1. Select the Velocity calibration window in the Setup menu.
   An empty Worksheet opens with columns for Voltage, Velocity, Ambient temperature and Ambient pressure.
2. Type in your calibration data.
3. Select the Fit command in the Edit menu or the f(x) button in the toolbar. The best curve fit is made and presented in the Curve fir dialog box.

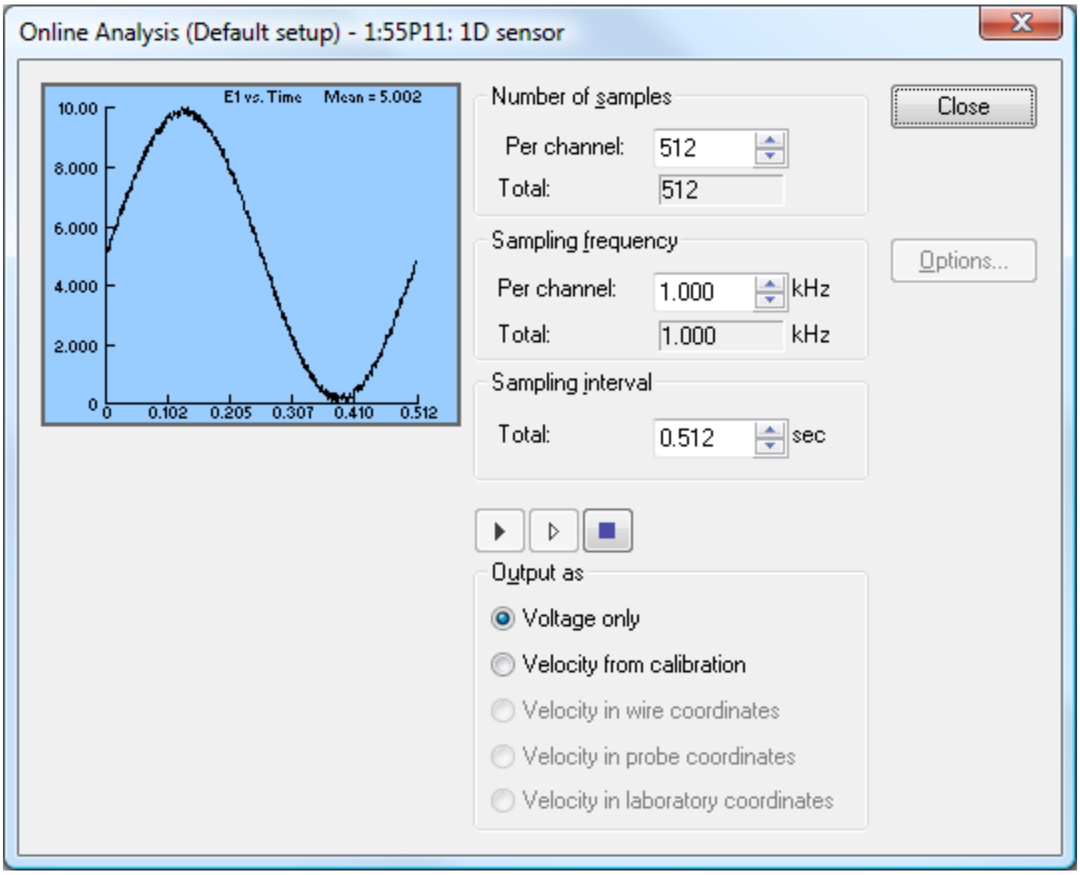
From here on you can proceed and create the Calibration event like explained above.

5.2.11 Run Online

You can now check the performance of the system in Online.

1. Choose Online analysis from the Run menu.
   Online dialog box opens.
2. Choose the Start button.

The Data display is now updated for each acquisition until you choose Stop or leave the dialog box.



You can display velocity instead of voltage by selecting Velocity from calibration in the "Output as" field.

The axis on the graph can be changed and you can Zoom out by clicking in the graph with the right mouse button. You can stop the acquisition and print the graph after Zoom out. No data are stored during Online.

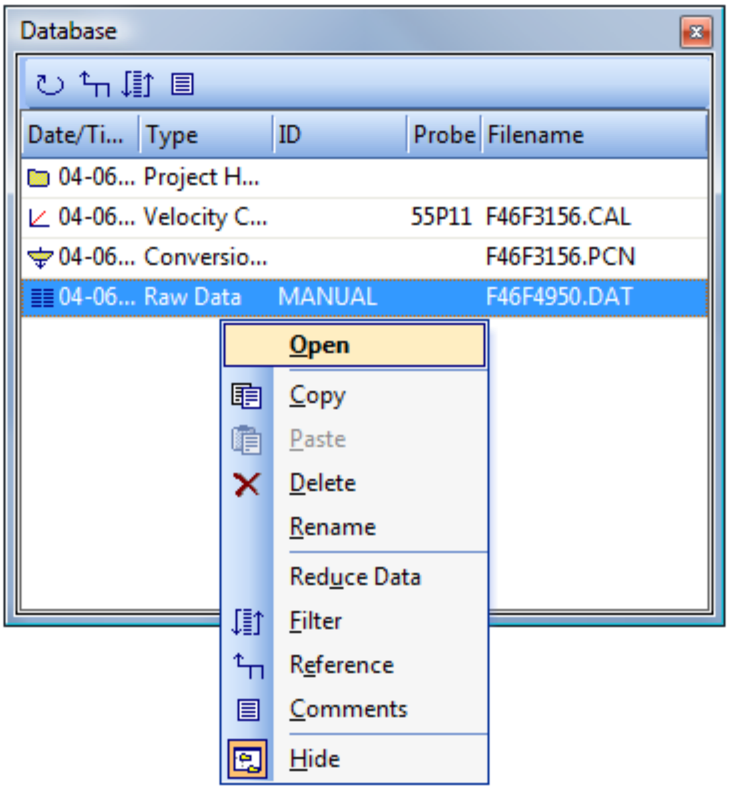
5.2.12 Run Default

You can store acquired data in a file by running Default instead of Online, see See "MiniCTA Software Guide" on page 36, To display probe default parameters if you want information about the parameters.

1. Choose Run default setup from the Run menu or click on the Acquire data Icon in the Main toolbar.
2. An Acquire data to disk event dialog box appears
3. Type in the identification for the Raw data event that will be the result of the default run.
4. Close the dialog box by choosing OK.

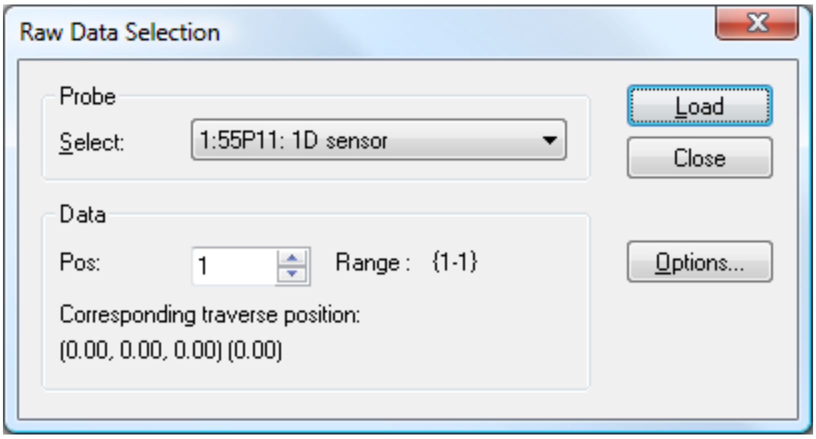
When finished, a Raw data record is added to the Project Manager.

5.2.13 Load Raw Data



Raw data are stored in a file arranged with a database structure. In order to present the data, they have to be unpacked.

1. Point at the Raw data event in the Project Manager and click with the right mouse button and select Load. Raw Data Selection dialog box appears.

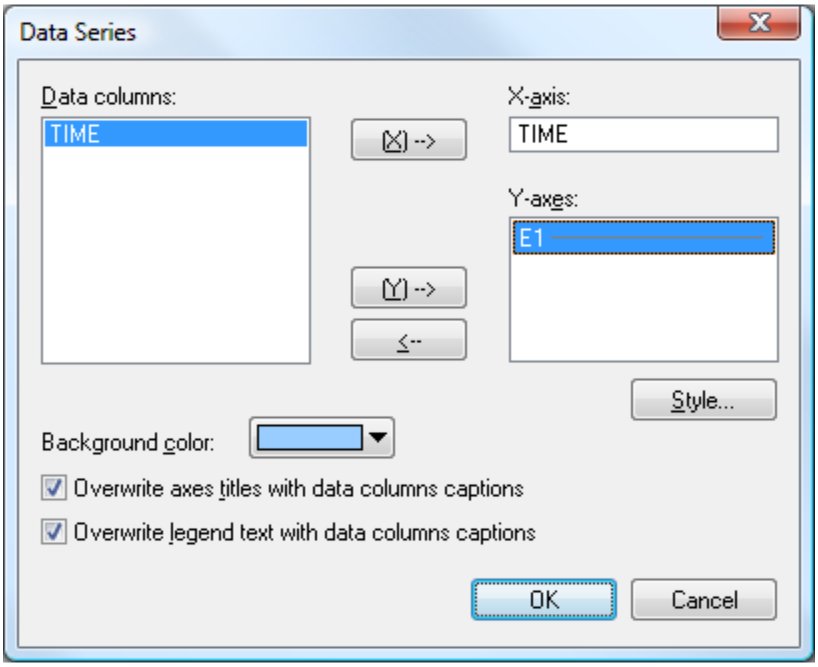


Raw Data Selection dialog.

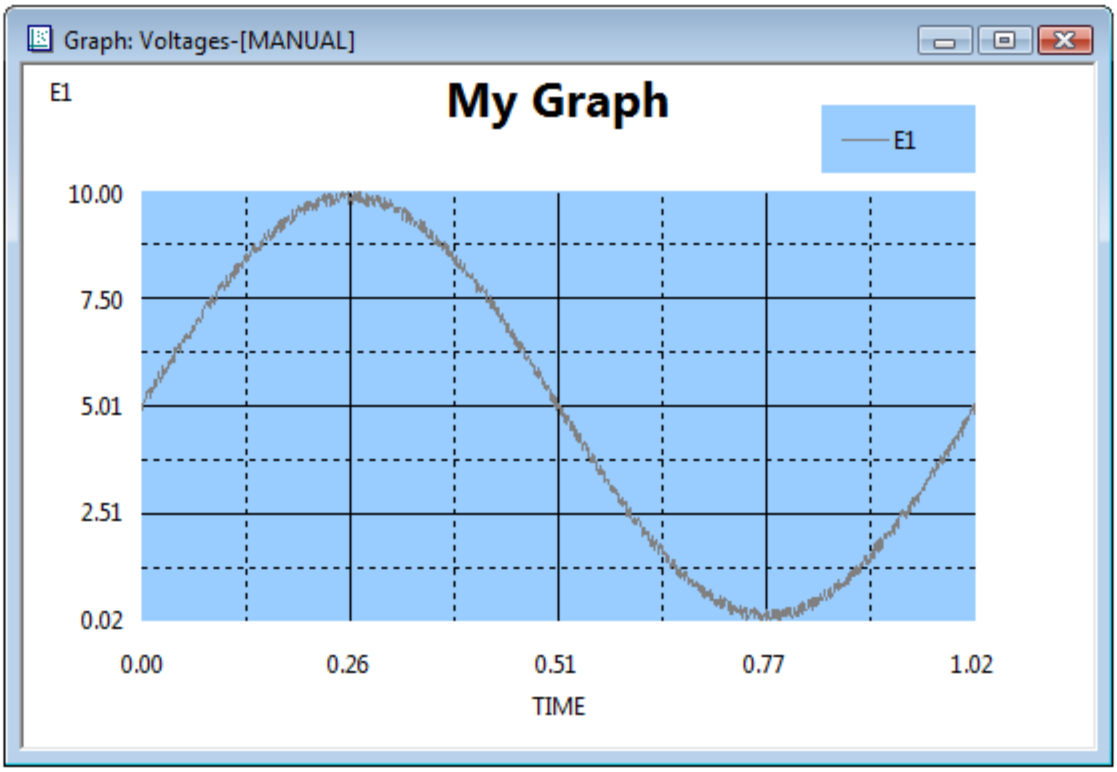
2. Choose Load.
   A data sheet with the acquired data opens.
3. Choose Close. The dialog box closes and leaves the data sheet open.

Present Data in a Graph

1. Choose Graph from New Window in the Window menu or click on the Graph Icon in the Main toolbar.
   Select Data Dialog box appears.
2. Select U1cal in the Data columns box.
3. Click on the Y--> button.
   U1cal is now moved to the Y-arguments box.
4. Choose OK.



A Graph is now created with voltages as function of acquisition time.



See See "Graphs" on page 73 how you can change axes, labels etc.

5.2.14 To Reduce Data

Create Reduced Data Event

1. Point at the Raw data event in the Project Manager and click with the right mouse button and select Reduce data.
   A Save event dialog box opens.
2. Type in an identification and choose OK.

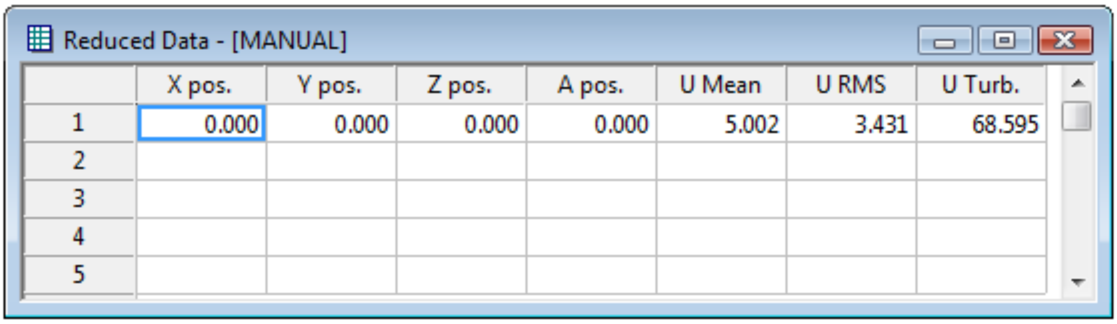
The Data reduction process is now carried out and a Reduced data event is added to the Project manager.

Note
The default Conversion event automatically will be used to convert and reduce the data.

Load Reduced Data

1. Point at the Raw data event in the Project Manager and click with the right mouse button and select Load.

Double click on the Reduced data event with the left mouse button. A Data sheet showing the positions (0,0,0), the U_{mean} and U_{RMS}. opens. As you have only one set of data it makes no sense at this point to make a graph of the reduced data, as they only represent one point.


Reduced Data - [MANUAL]

	X pos.	Y pos.	Z pos.	A pos.	U Mean	U RMS	U Turb.
1	0.000	0.000	0.000	0.000	5.002	3.431	68.595
2							
3							
4							
5							

5.3 Sample Project II
1D Measurements with Temperature Correction

This project demonstrates how to incorporate temperature correction into a project. For simplicity a project with only one single-sensor hot-wire probe has been configured. The procedure, however, is identical for any number of probes and for dual -and triple-sensor probes as well.

5.3.1 Project Description

If a high absolute accuracy on mean velocity is required it is always recommended to include temperature correction in order to avoid systematic errors from even small fluid temperature variations. If not compensated for a temperature change of 1 °C gives approximately 2 % error in velocity for a wire probe operated at the default overheat ratio 0.8.

Temperature correction requires that temperature is acquired together with the anemometer voltage. This is done by means of a temperature probe with an analog output connected to the A/D device and assigned as a probe in the System configuration. The CTA probe and the temperature probe should be mounted close to each other in the flow, so that they are both exposed to the same temperature.

The basic procedure in the MiniCTA software is:

1. Assign the temperature probe in the System configuration.
2. Choose it as temperature probe.
3. Apply temperature correction to the Data conversion/reduction event.

Prior to that the temperature must be placed in the Probe library. For details see See "Files" on page 41.

5.3.2 Hardware List for Sample Project II

- 55P11 Probe
- 55H20 Probe Support
- 54T30 MiniCTA anemometer.
- 55P32 Thermistor Probe
- 54T40 Thermistor Amplifier
- 2 pcs. 9055A1863 Probe Cable, 4-m.
- Cables and Connector Box for the A/D converter board.
- PC with A/D device installed.

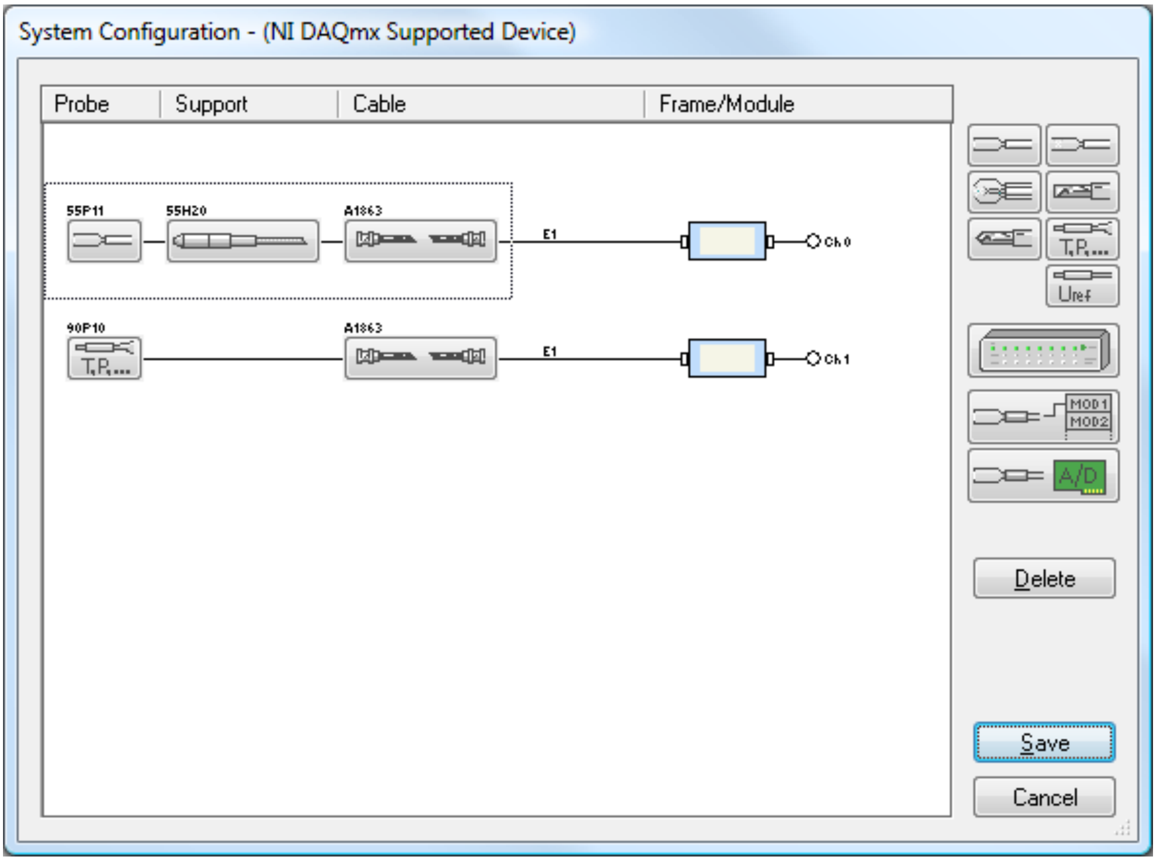
5.3.3 Adding a Temperature Probe to the Probe Library

If you are not using the 90P10 Thermistor probe, which is in the Probe library, you have to add your own temperature probe to the library. Open the miscellaneous probes and add the temperature probe.

Remember to enter the calibration constants that converts volts to °C in the Coefficients dialog box.

5.3.4 System Configuration

1. Configure the system with the hot-wire probe and the temperature probe. This is done by clicking on the Single sensor probe icon in the Configuration dialog box and select the 55P11 from the Single-sensor probe Icon and the 90P10 probes from the T,P... Icon, respectively.
2. Choose OK. The dialog box closes.

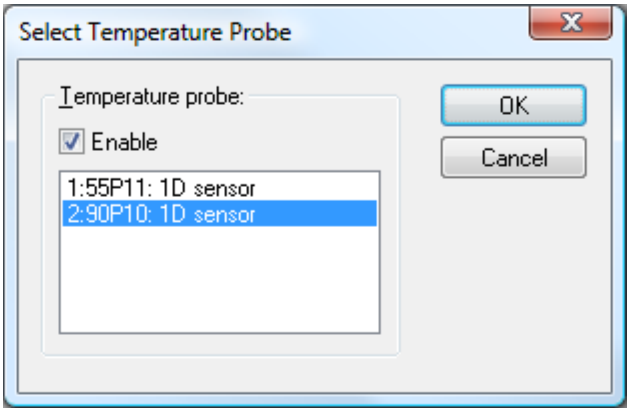


System Configuration dialog.

5.3.5 Selection of Temperature Probe

When you close the Configuration dialog, you are prompted to select a temperature probe. Say Yes. Select probes dialog box opens.

1. Click Enable and select probe 2: 90P10: 1D sensor.
2. Click Ok to close the dialog box.



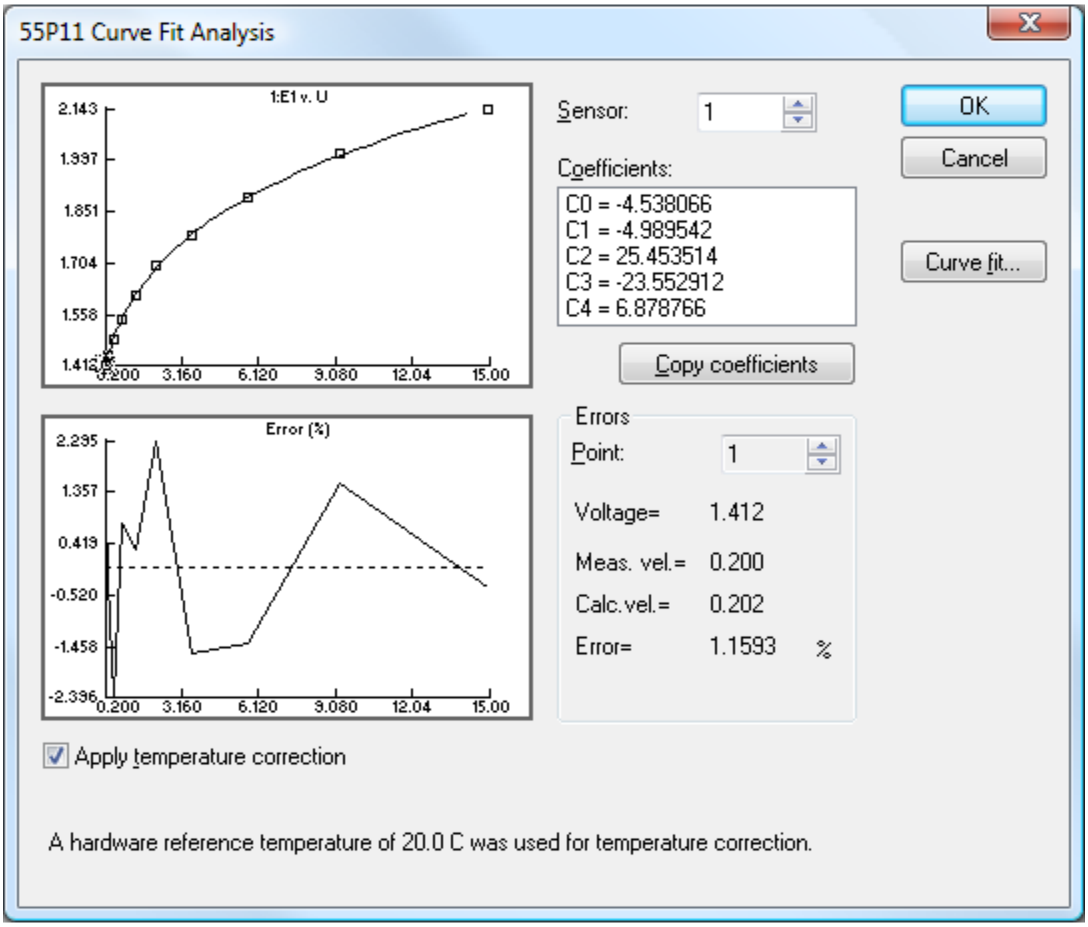
Select Temperature Probe dialog.

5.3.6 Hardware Setup and Velocity Calibration

Perform hardware setup and velocity calibration as described in Sample Project I. Now it is most important to enter the proper temperature. Make both events default.

The temperature can automatically be read as a Parameter during the calibration, if the temperature probe is placed in the flow next to the hot-wire probe. In this case choose Parameter/Setup/Temperature/A/D/ch1 in the Calibration dialog box. Type in the linearisation (polynomial constants from the keyboard). For more details see See "Running the System" on page 126.

Important
Select Apply temperature correction in the Curve Fit dialog box, when you have finished the velocity calibration.



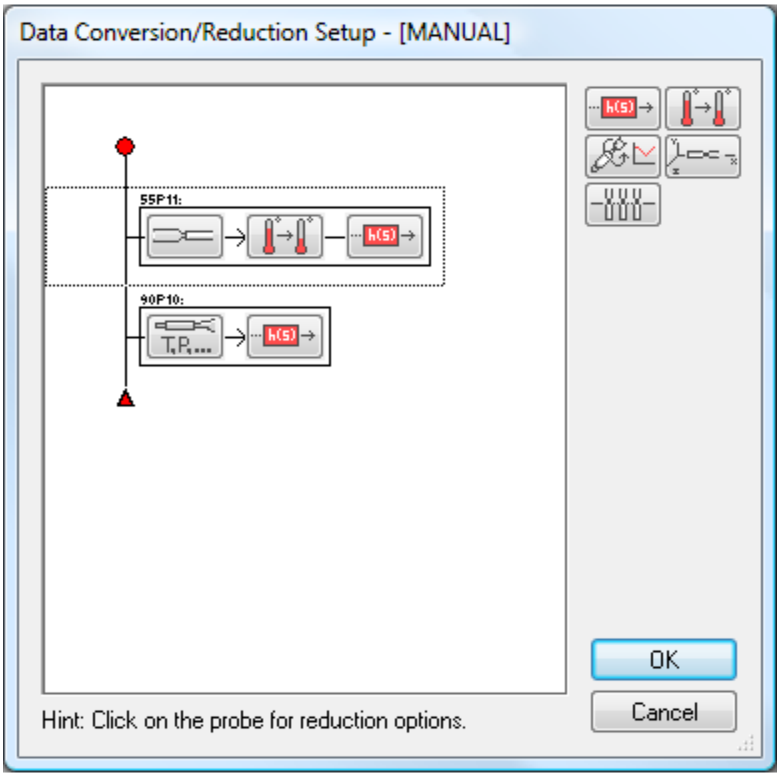
Curve Analysis dialog with Temperature Correction enabled.

When you close the Calibration data sheet you are prompted create a data Conversion/Reduction. Say Yes to that and say Yes to the “Make the event default”. You will now have the two events marked with default stars listed in the Project Manager

5.3.7 Data Conversion/Reduction with Temperature Correction

It contains all necessary information for performing temperature correction, linearisation and data reduction into mean and RMS values. You need not make any further manipulations in it. To see what it looks like:

1. Open the Default Conversion/Reduction in the project manager (right mouse button). The Data conversion/reduction setup dialog box opens.
2. Close the dialog boxes by clicking on Cancel.



Data Conversion/Reduction Setup dialog.

5.3.8 Run Default Setup

Run the default setup as described in Sample project I.

5.3.9 Run Data Conversion and Reduction

Run the data reduction as described in Sample project I.

5.4 Sample Project III
Mapping of a Velocity Profile with Probe Traversing

5.4.1 Project Description

This project demonstrates how to measure the velocity profile in a free jet with a single sensor wire probe. It expands Sample Project I project with a Traverse device and a Traverse Grid. When you run the new Default setup, the probe is traversed across the jet, data are acquired in each position and saved in a file. The data are then converted on basis of the probe calibration and reduced into U_{mean} and U_{RMS}.

5.4.2 Hardware List for Sample Project III

In addition to the hardware in Sample Project I you will need the following:

- A free Serial Comport installed in the PC.
- 41T41-D Traversing Mechanism
- 41T72 1-D Traverse Controller

5.4.3 Getting Started

Start MiniCTA. If you have not made any new projects in the meantime MiniCTA opens Sample Project I, where you have to load the Traverse driver, which gives you access to the Traverse controller.

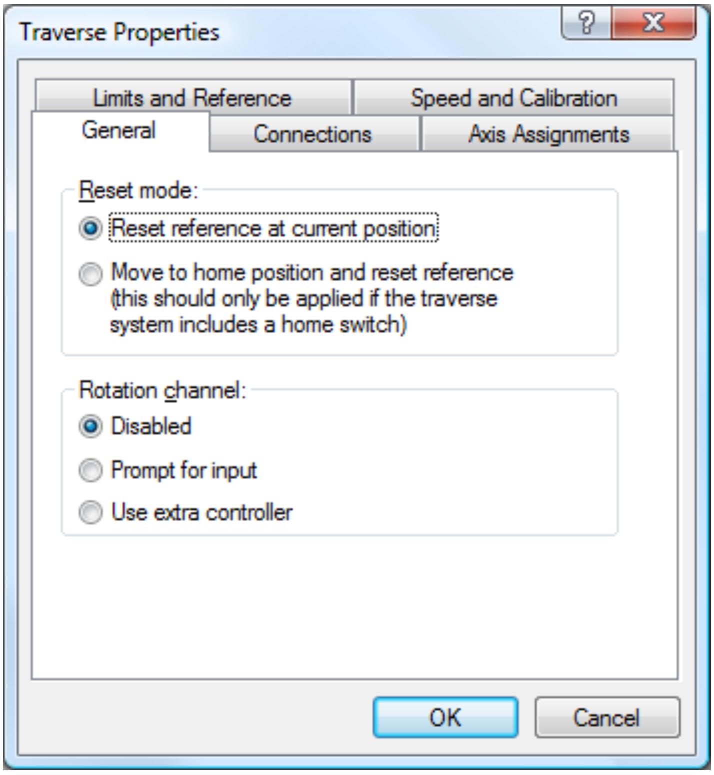
5.4.4 Load Traverse Driver

Traverse Driver

1. Choose Devices from the Configuration menu.
2. Choose Traversing from the Devices sub menu.
   Select Traverse Driver dialog box opens.
3. Select the Lightweight Traverse from the driver list.
   Choose Setup. Select Traverse Properties dialog box opens.

Traverse Setup

1. Choose the Setup button.
   A Traverse Properties dialog box appears where you select the following:



General Traverse Properties dialog.

General

Reset mode at current position.
Rotation channel Disabled.

Connections

Comport 2

Axis assignment:
Assign X as Axis 1. (Disable Y, Z and Rot.)

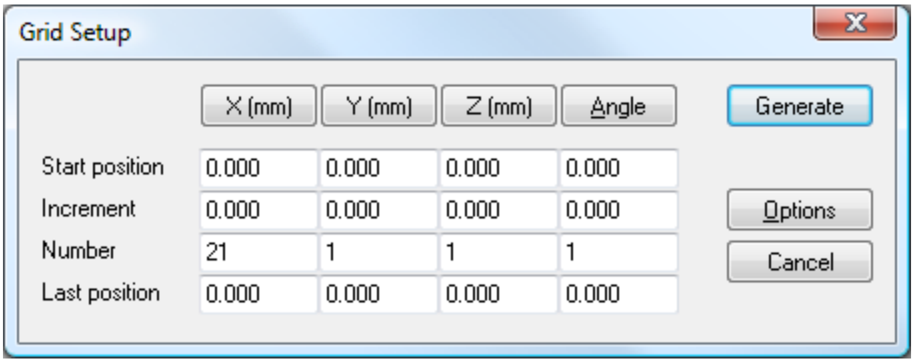
Limits and Reference

X_{pos} Min: 0 mm
X_{pos} Max: 540 mm

Speed and Calibration

Speed: 25 mm/s
Cal. Fact.: 80 pulses/min

5.4.5 Define Traverse Grid



Grid Setup dialog.

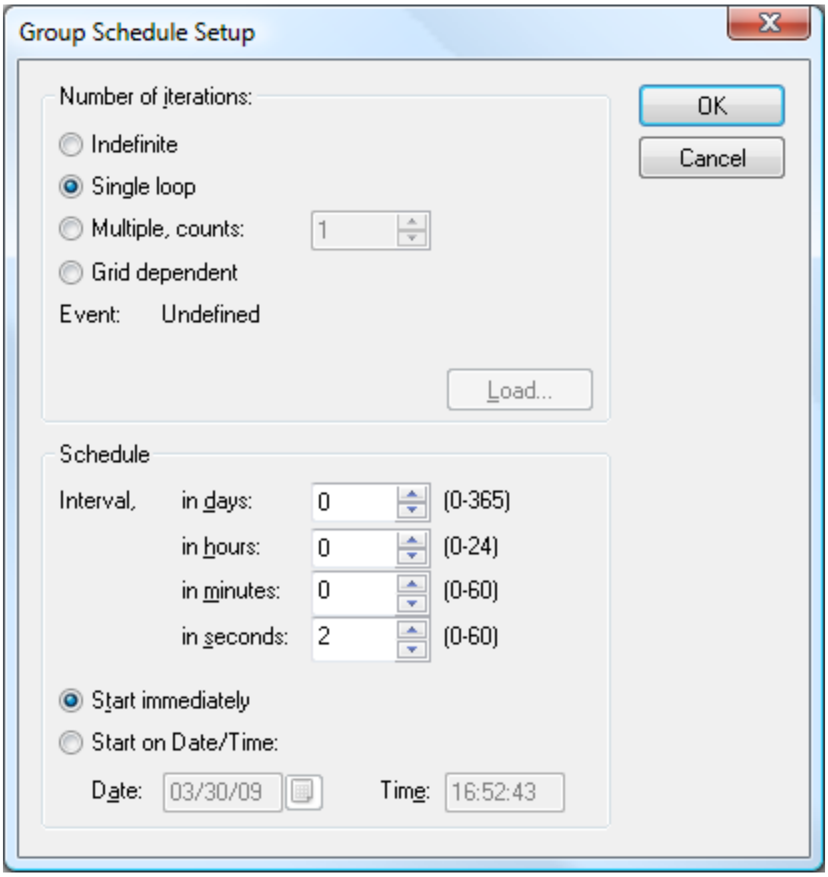
1. Choose Traverse grid from the Setup menu or click on the Traverse Icon in the Main toolbar.
2. A Grid Setup dialog box opens.
3. Enter Start position, Increment and Number of positions.
4. Choose Generate.
5. The dialog box disappears and the 21 positions from 0 to 20 are written into the X-column.
6. Double click in the Menu-control box.
7. A Save event dialog appears.
8. Enter an identification.
9. Choose OK.
10. The Traverse Event is now added to the event list in the Project manager.

5.4.6 Load the Traverse grid into the Default Setup

In order to move the probe the Default Setup must include the Traverse grid.

1. Choose Default setup from the Project menu.
2. A Default setup dialog box appears.
3. Click on the Group Schedule Icon. 
   A Group Schedule dialog box appears.
4. Select the Grid dependent radio button and click on the Load button.
   A Load event dialog box appears with the Traverse event in the event list.
5. Select the Traverse event and choose OK.
   The event identification is now written into the Group Schedule dialog box.

6. Close it by choosing OK.
   You are prompted to use the traverse. Select Yes and you will be back in the Default setup dialog box.
7. Click on the Position Input Icon. 
   A Position Input Setup dialog box opens. The "Move traverse directly to position specified in grid" option is automatically selected, when you have assigned the Traverse system.
8. Choose OK.
   The dialog box disappears and the Default has been updated.



Group Schedule Setup dialog.

5.4.7 To Traverse Manually

You can include a manual traverse in the Default setup, if you do not have an automatic Traverse system. First deselect any Traverse driver that may have been loaded earlier. This is done in the Device menu. Then select the "Display prompt to move position from keyboard" option in the Position Input Setup dialog box. You will then be able to move the probe manually before data are acquired.

5.4.8 Set Traverse to Starting Position

Place the probe in the Traversing System and move it to the starting point of the Traverse grid (X = 0):

1. Choose Traverse Control from the Run menu or click on the Move Traverse icon in the toolbar:
   

2. A Traverse Control dialog box appears.
3. Move the probe to the starting point by means of the X-position box and the Move command
4. button.
5. Click on the Reset button.
   The position is now defined as the Traverse grid starting point.

5.4.9 Run Default Setup

1. Choose Default setup from the Run menu or click on the Acquire data to disk Icon in the Main toolbar.
2. An Acquire data to disk dialog box appears.
3. Type in an identification and choose OK.

Data are acquired in the first traverse position. Then the Traversing system moves the probe to the next point and acquires data in that and so on, until all points are done.
During execution the mean velocity in each point is plotted in an Acquisition status window.
When all points are done, the Raw data event is added to the Project Manager.
In case you move the probe manually, you are prompted to move the probe before data can be acquired.

5.4.10 Reduce Data and Present them in a Data Sheet and in a Graph

You can now convert the Raw data and reduce them to U_{mean} and U_{RMS} in each traverse position.

5.4.11 To Reduce the Raw Data

Data are reduced as in Sample Project I.

5.4.12 To Present the Velocity Profile in a Graph

A Graph of the velocity profile is created as in Sample Project I.
See See "Graphs" on page 73 how you can change axes, labels etc.

6 MiniCTA Software Guide

This guide describes the MiniCTA application software and how to use it in Fluid Dynamics investigations. It is assumed that you have performed the system configuration as described in See "Hardware Installation" on page 12 and See "Configuring the System" on page 79.

The computer equipment and software environment that you need in order to run MiniCTA are described in See "Configuring the System" on page 79.

The Software Guide goes through all commands as they appear in menus and dialog boxes and explains all actions that can be carried out and their functional meaning.

6.1 MiniCTA Software Interface

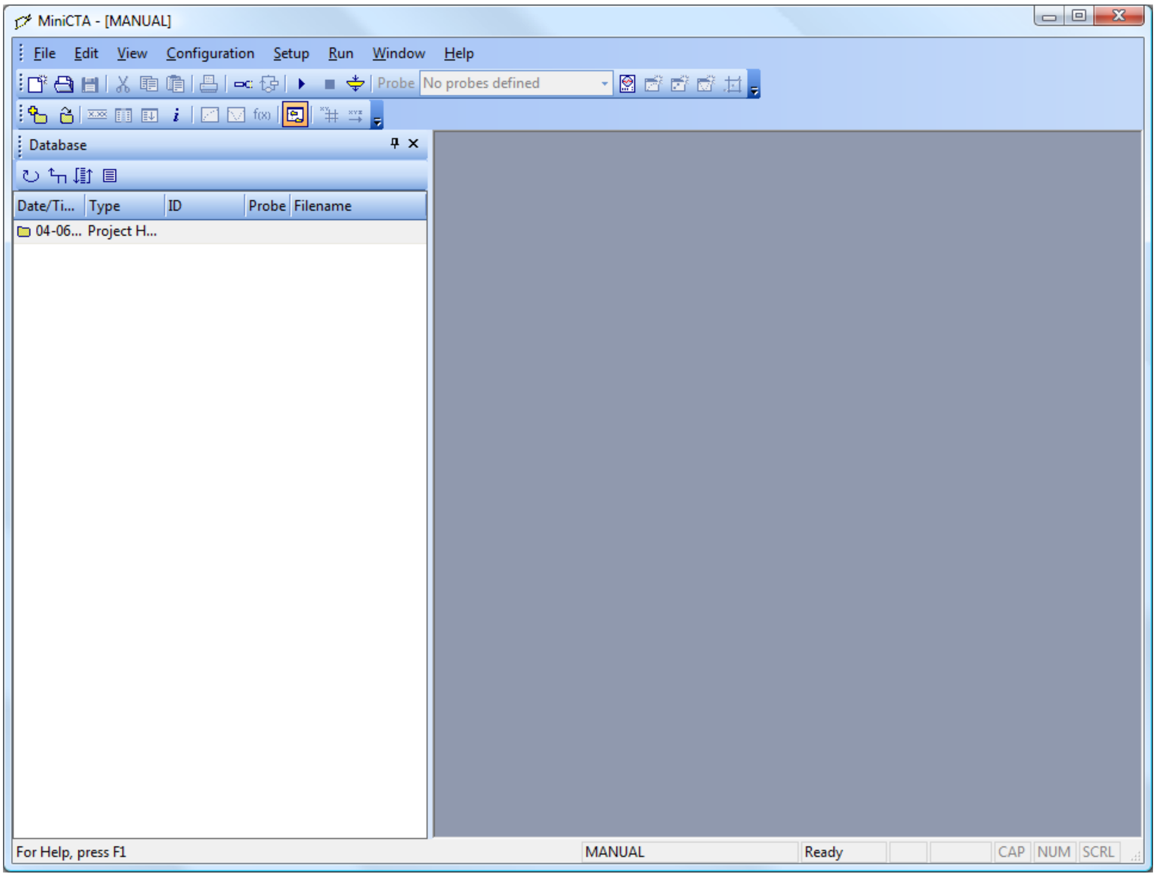
MiniCTA Software interfaces with the anemometer via an A/D Converter Board in the PC for acquisition of measuring data.

Working with MiniCTA Software takes place in a Windows environment with the ordinary Windows elements:

- Separate windows.
- Menu bars.
- Toolbars with icons.
- Dialog boxes.
- List boxes and option and check boxes.

It also contains Data sheets for presentation of data. All manipulations follow Windows standards.

The MiniCTA Workplace



The MiniCTA Software window with an opened Project Manager.

The main window has a menu bar with the following commands:

- File for handling Databases, Projects, Libraries and for data exchange (Export/Import)
- View for customization of the user-interface.
- Configuration for defining system configuration, A/D device, traverse system and temperature probe for temperature correction purposes.
- Default Setup for defining hardware setup (reference temperature and overheat ratio), calibrations, data acquisition and data conversion/reductions.
- Run for starting on-line display, data acquisition and reduction of data.
- Window for arranging dialog boxes and data sheets.

The most common commands are accessible from the toolbars.

At the bottom of the window a Status bar shows which project is open, the operation mode and the status of that mode together with the present time.

6.1.1 MiniCTA Software Icons

The MiniCTA Software Icons are command buttons placed in toolbars or in toolboxes inside dialog boxes. When placed in toolboxes, they are the only means of getting access to the action or process they represent.

The Icons are activated by a single click with the left mouse button.

Toolbar Icons

Main Toolbar

 New Database

 Open Database

 Save Event

 Print

 Cut

 Copy

 Paste

 Configuration

 Setup

 Run

 Stop

 Convert

 Online

 Velocity Calibration

 Two-point Calibration

 Directional Calibration

 Graph

Project Toolbar

 New Project

 Open Project

 Refresh Database View

 Seek Event Dependencies

 Filter Events

 Event Log

 Project Information and Log

 Show/Hide Database View

 Format Data Columns

 Tile Data Columns

 Data Series

 Edit Velocity Calibration

 Edit Directional Calibration

 Calibration Fit

 Generate Traverse Grid

 Traverse Controller

Dialog Box Icons

Configuration


Assign 1D wire/fiber probe


Assign 2D wire/fiber probe


Assign 3D wire/fiber probe


Assign 1D film probe


Assign 2D film probe

Assign Misc. probe (temperature, pressure etc.)

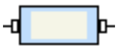
Assign Reference probe (velocity reference)

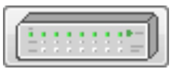

1D Support (automatically assigned)


2D Support (automatically assigned)


3D Support (automatically assigned)


Cable (automatically assigned)


MiniCTA Box (automatically assigned)


Assign Multichannel CTA Frame

Assign A/D channels

Default Setup

 Select Hardware Setup (reference temperature, overheat ratio etc.)

 Select Group Schedule

 Select Traverse Setup

 Select Data Acquisition Setup

 Select Data Reduction Setup

 Run Default Setup

Data Reduction

 Define Data Reduction (moments, mean, RMS etc.)

 Define Linearization of Velocities from Calibration

 Define Temperature Compensation of Probe Voltages

 Define Decomposition of Velocity Components in Probe Coordinates

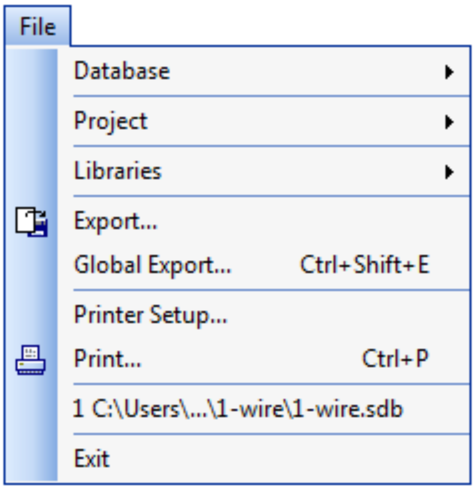
 Define Coordinate Transformation of Velocity Components

 Define Probe Array

6.2 Files

6.2.1 Definitions

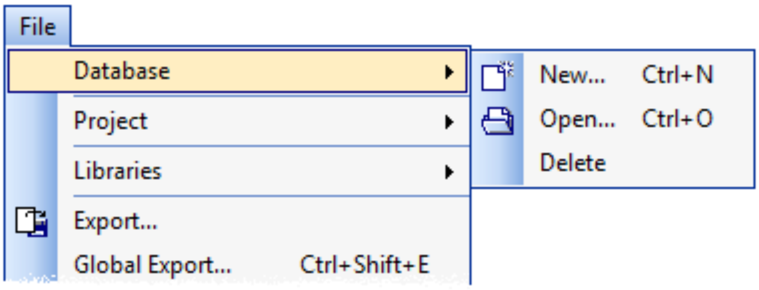
In the File menu you can create or open databases, create or open projects, access libraries and import or export data.



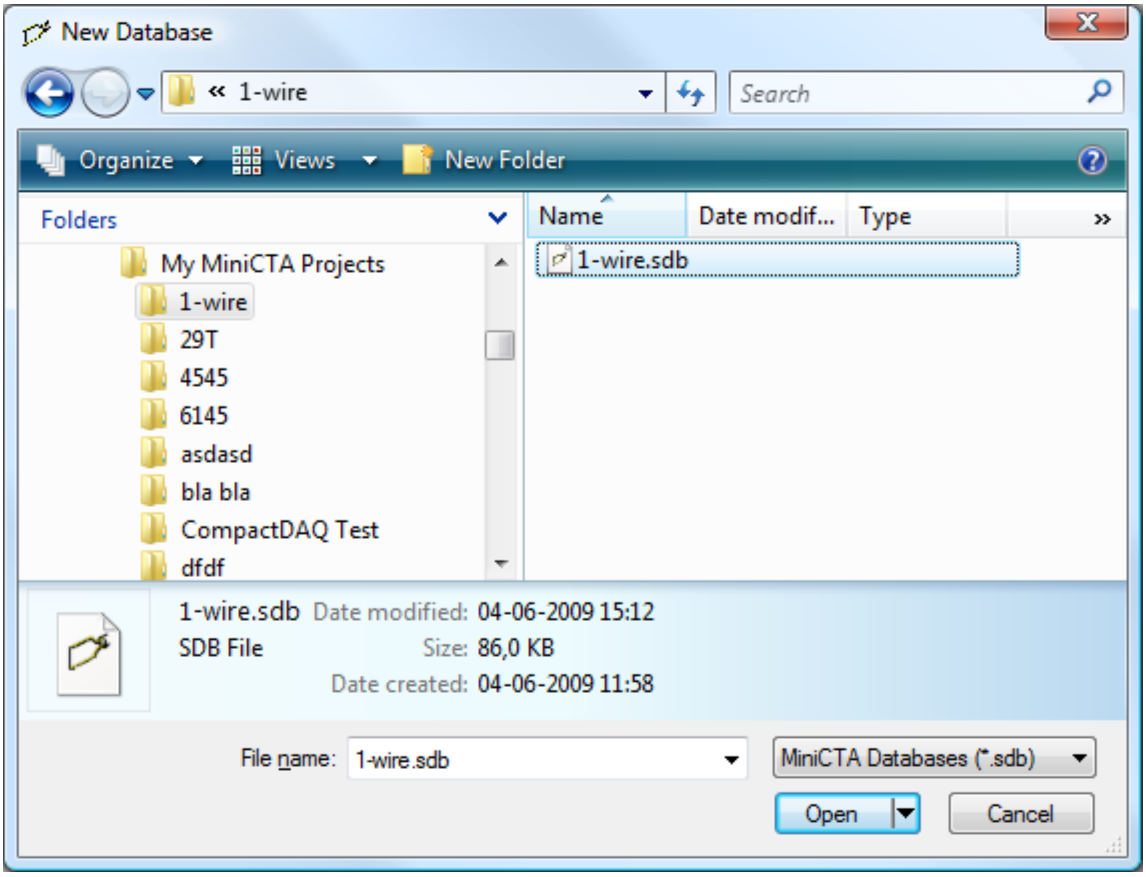
6.2.2 Handling Databases

All work in MiniCTA is done in a project placed in a MiniCTA database. First time you start up MiniCTA you must therefore create a new database. After that the software always opens the Project and shows the Project Manager that you worked in latest.

To Create a New Database



1. Choose the New Database button  in the Main toolbar or choose New from the File/Database menu.
   New Database dialog box opens.



2. Select drive and directory.
3. Type the name of new database in the File Name box. Follow the rules for file names: max. 8 characters. Extension is always .sdb. If another extension is typed in, it will automatically be overwritten with .sdb when saved.
4. Choose OK.
   The dialog box closes.

To Open a Database

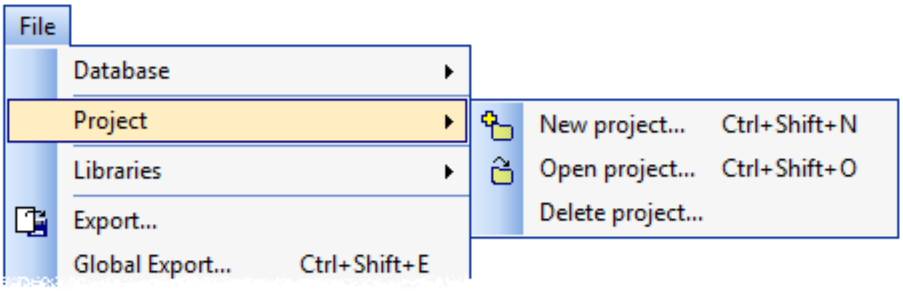
1. Choose the Open button in the main toolbar  or choose Open from the File/Database menu.
   Open Database dialog box opens.
2. Select drive and directory.
3. Select the database in the File box or type in the name in the File Name box.
4. Choose OK.
   The dialog box disappears and is replaced by the Project Manager from the last opened project.

To Delete a Database

1. Choose Delete in the File/Database menu.
   Delete Database dialog box opens.
2. Select drive and directory.
3. Select the name of the database in the File box or type in the name in the File Name box.

4. Choose OK.
   The database is deleted and the dialog box closes.

6.2.3 Handling Projects



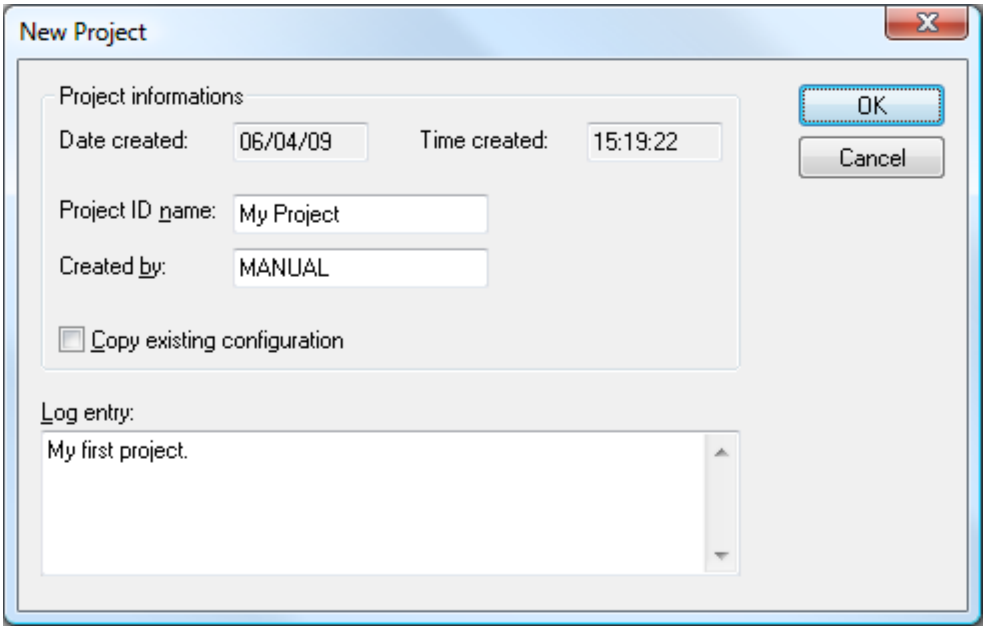
A Project is a list of records containing full information about configurations, setup, data acquisitions, data reductions etc. This information is time stamped and listed in a Project manager.

To Create a New Project

Before you make a new project make sure that you are in the right database.

Note
It is recommended to have only one project per database per directory. In this way the data files from different projects will not be mixed up. This makes it much easier to move a project from one PC to another.

1. Choose the New project button from the Project toolbar  or choose New project in the File/Project submenu. New Project dialog box opens:

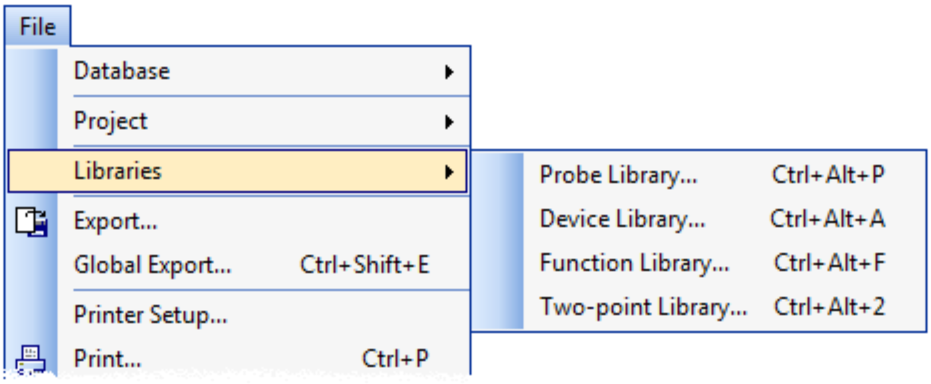


2. Enter the name of the new project.
3. Enter your name or initials.
4. Enter any comments that you want to tie to the project in the Log entry.
5. Choose OK.
   The dialog box closes and the Project manager opens in the left side of the screen. The project name is displayed to the left in the Status bar at the bottom of the screen.

To Open a Project

1. Choose the Open project button  in the Project toolbar or choose Open in the File/Project sub-menu.
   Select Project dialog box opens with a list of the projects in the actual database.
2. Select the project that you want to delete.
3. Choose OK.
   You are now prompted to confirm the command.
   Confirm and the dialog box closes.

6.2.4 Handling Libraries



Libraries can be accessed from the File/Libraries menu. There are three libraries with information about probes, A/D devices and traverse systems and external functions for data reduction and a Two-point Library with typical calibration data to be used with the Dantec Two-point Calibrator.

Libraries are arranged as databases from which relevant parameter are loaded into a project. New items can be added and parameters can be changed.

Probe Library

The Probe Library contains parameters for all Dantec standard probes, supports and cables. It has an additional group, Miscellaneous, where you can place other than CTA probes, e.g. temperature or pressure probes.

A Probe library always has the extension *.pdb.

The parameters in the probe library is used as Configuration defaults for CTA Module assigned to the selected probe.

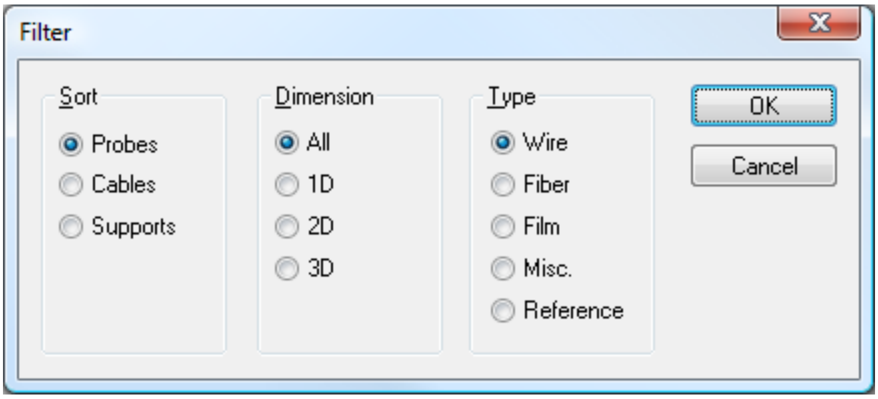
Note
You can only assign probes from a Probe Library to CTA Modules.

To Open a Probe Library

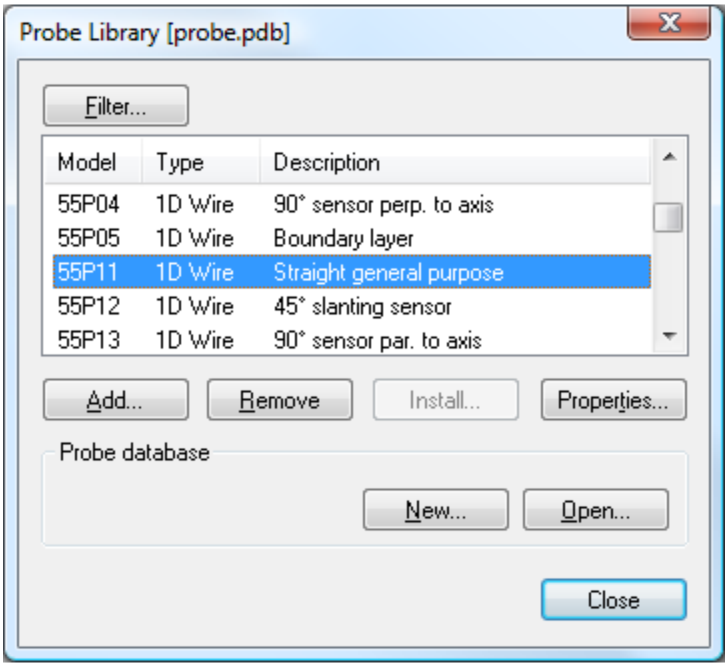
1. Choose Probe Library from the File menu.
2. Probe Library dialog box opens with the last used library. Here you can see its content sorted after item type, and you can do editing, copying, adding and deleting.
3. The dialog box also gives you access to other Probe libraries or to load new libraries.

To Filter in the Library

Click on the Filter button and a Filter dialog box opens, where you can sort on Probe types, Supports and Cables.

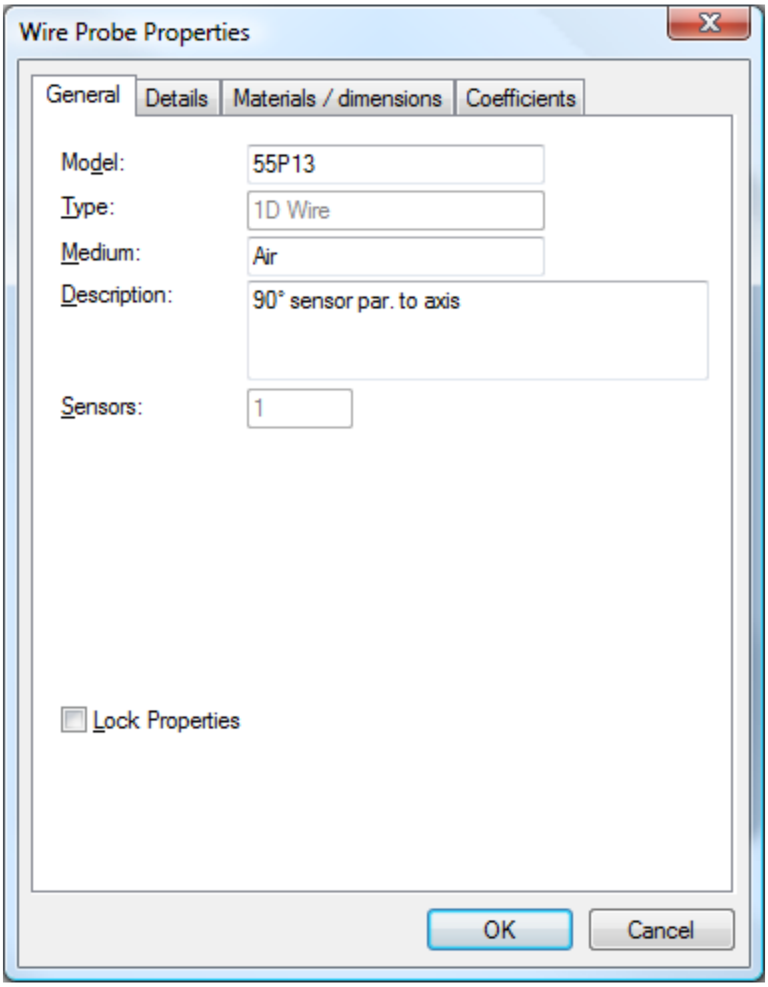


Make your selection and click OK. The related dialog box opens.



Probe Properties

Select Properties button. Probe properties dialog box opens.



General

All fields can be edited except Type, as this indicates the number of sensors on the probe.

Details

Shows typical sensor resistance, accepted resistance variation, temperature coefficient of resistance. Also recommended overheat ratio at room temperature, maximum sensor temperature and maximum ambient temperature are shown.

Materials and Dimensions

Lists sensor material, plating, dimension and prongs/substrates.

Coefficients

Linearization coefficients for each sensor: The configuration defaults are 0,1,0,0,0 in a 5th order polynomial. This means that voltages are not converted unless you enter the actual values for the probe. Linearization can be either polynomial or power law. If you select power law, you can enter A, B and exponent n.

Note
If you change the default linearisation constants, you are advised to copy and paste the probe into a custom probe first in order to keep the original probe library intact.

Pitch and yaw coefficients for each sensor (only for X-array and Tri-axial probes).
Wire to probe coordinates:
Angles between each sensor and the axis in the probe coordinate system.

Miscellaneous Probes

These are typically pre-calibrated probes, which can be used as references for temperature.
Misc. probes can be linearized by the following functions:

- Polynomials
- Power law
- Steinhart-Hart (recommended for thermistor probes)
- Logarithmic polynomial (alternative for thermistor probes)

Velocity Reference Probes

These are pre-calibrated probes, which can be used for calibration of other probes, for example in a wind tunnel.

- Logarithmic power law (for velocity calibration reference)

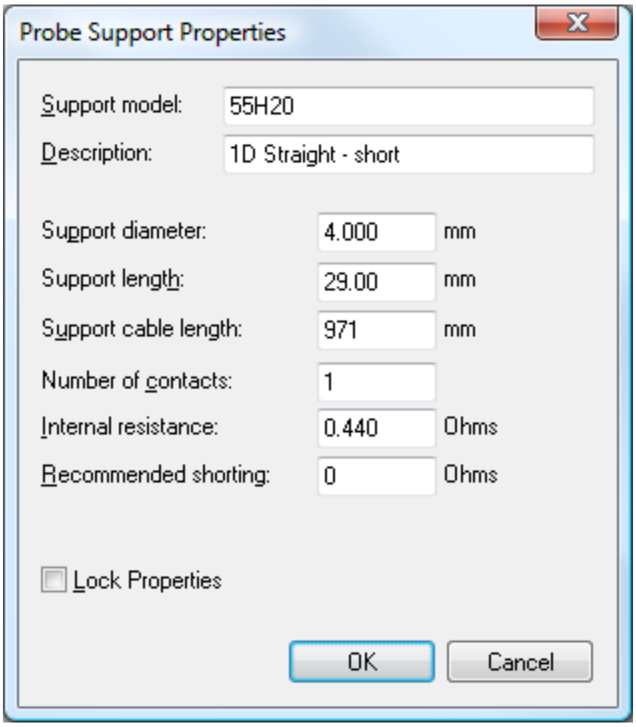
The calibration data are loaded from a file delivered together the velocity reference probe. See To add "View" on page 63a probe ... pg. 5.1-9

See "Software Reference" on page 143 for details about linearization functions.

Probe Supports and Cables

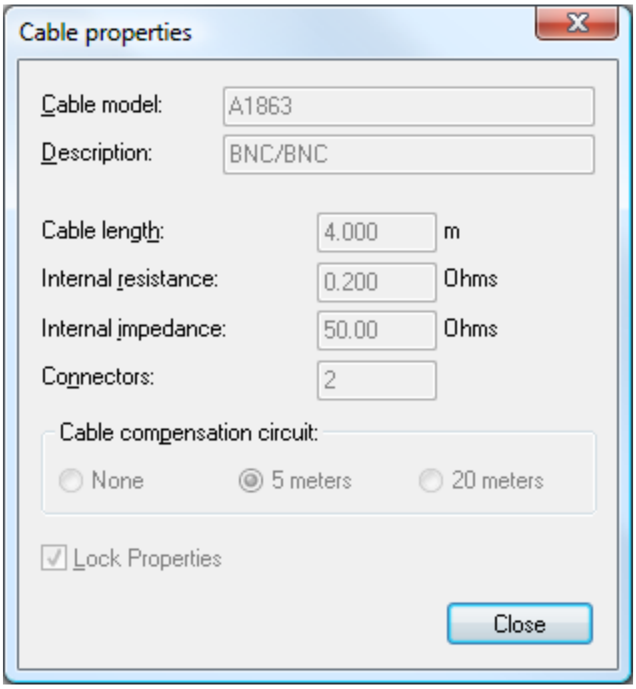
Probe Supports

1. Select Supports. A Probe Support dialog box opens, where you can select the wanted support.
2. Click OK and a Probe Support Properties dialog box opens.



Cables

1. Select Cables. A Cable dialog box opens, where you can select the wanted cable.
2. Click OK and a Cable Properties dialog box opens.



The dialog box contains information about cable length, resistance and impedance. It also determines the cable compensation circuit that will be used in the bridge for that cable. It is important to leave the default configuration unchanged (e.g. 4 m cable ® 5 meter compensation (=1 m probe support + 4 m probe cable).

To Create a New Probe Library

1. Choose the New button in the Probe Library dialog box.
   Open dialog box opens.
2. Select drive and directory in the Drives and Directories list boxes.
3. Type in the name of the new library in the File Name box.
   Note that the extension must be .pdb.
4. Choose OK.
   The dialog box disappears, and a new Probe Library dialog box opens.

To Edit Parameters

You can edit in the probe, support or cable parameters, if you use the Edit command in the Probe Library dialog box and overwrite the existing parameters.

1. Choose the Sort button.
   Sort dialog box opens.
2. Select the wanted Probe, Cable or Support.
3. Choose OK.
   The dialog box closes and a sorted list appears in Probe Library dialog box.
4. Select the wanted item.
5. Choose the Edit button.
   Parameter dialog box opens showing default parameters.
6. Overwrite the parameters that you want to change with your own values.
7. Choose OK.
   The dialog box disappears.

You are advised never to change Configuration Default Setup parameters for an original Dantec item. Create instead a Custom probe (support or cable) by coping and pasting (can be done within the same Library), changing its name and modifying its parameters as described above.

To Add or Delete a Probe, Probe Support or Probe Cable

Before you can add or delete an item , you must select the type it belongs to.

1. Choose the Sort button.
   Sort dialog box opens.
2. Select Probe, Cable or Support.
3. Choose OK.
   The dialog box disappears, and the Add button corresponding to the selected type is enabled.

To Add from New

1. Chose the Add button.
   Probe dialog box opens with empty parameter boxes.
2. Type in the relevant parameters.
   When you add a probe remember to fill in parameters for Bridge settings, Signal Conditioner settings and Coefficients.
   If you do not have values for all parameters, you can leave these boxes empty. The software will then automatically fill in expected defaults, before the probe is added to the database.

When you later on carry out processes requiring defaults that you have not entered yourself, you will be warned that the results may be wrong.

3. Choose OK.
   The dialog box closes and the new item is added to the list in the Probe Library dialog box.

To Add by Copying and Modifying an Old Item

You can also add an item by means of the Copy and Paste commands using the right mouse button either in the same or in another Probe Library. If the new item has parameters different from the one it is copied from, you can correct them using the Edit command.

To Remove an Item

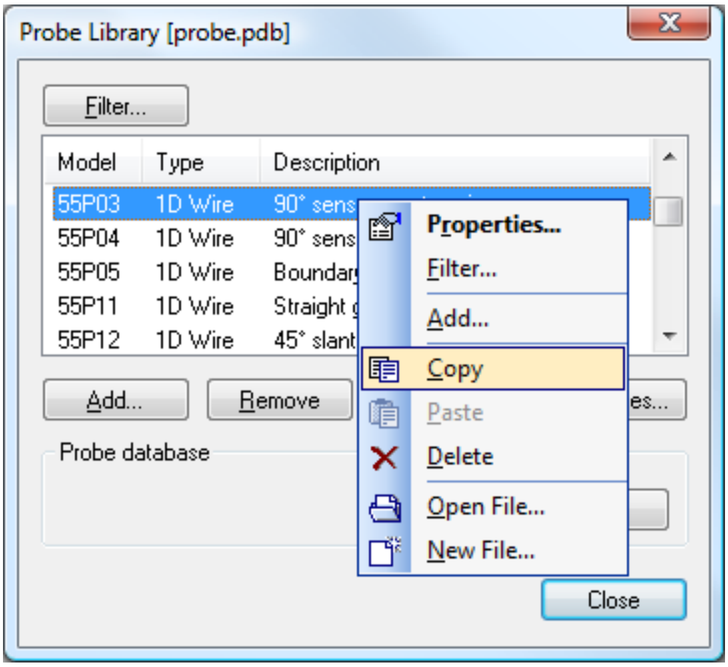
1. Select the item in the list box.
2. Choose the Remove button.
   The item is removed from the list.

To Copy and Paste an Item

You can copy and paste items within the same library or from one library to another.

To Copy and Paste in same Library

1. Click on the item with the right mouse button and choose Copy.



2. Click on the item with the right mouse button and choose Paste.
3. The item is now pasted into the list box immediately below the selection.
   You can now edit in the parameters, if needed. For details, see the section: To Edit Parameters.

To Copy and Paste in Different Libraries

1. Click on the item with the right mouse button and choose Copy.
2. The item is now placed in the clipboard.
3. Choose the Open button.
   Open dialog box opens, where you select the target library. For details, see the section: To open a Library.
   The target library opens with the same item type as the source library, so that you do not have to make any sorting in the new library.
4. Select the position, where you want to place the item.
5. Click with the right mouse button and choose Paste.
   The item is now pasted into the list box.

To Add a Velocity Reference Probe

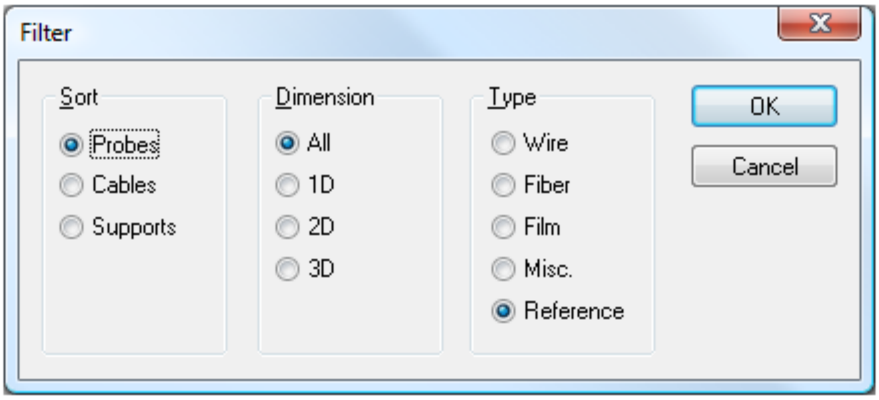
Currently only the pre-calibrated velocity-reference probe T29 is supported by the software. Please refer to the User's Guide for 54T29 Reference Velocity Probe publication number 9040U4061.

Older versions of this probe included a dedicated Calibration File CD, containing the calibration file for the given probe. Newer versions include a general calibration CD containing multiple calibration files.

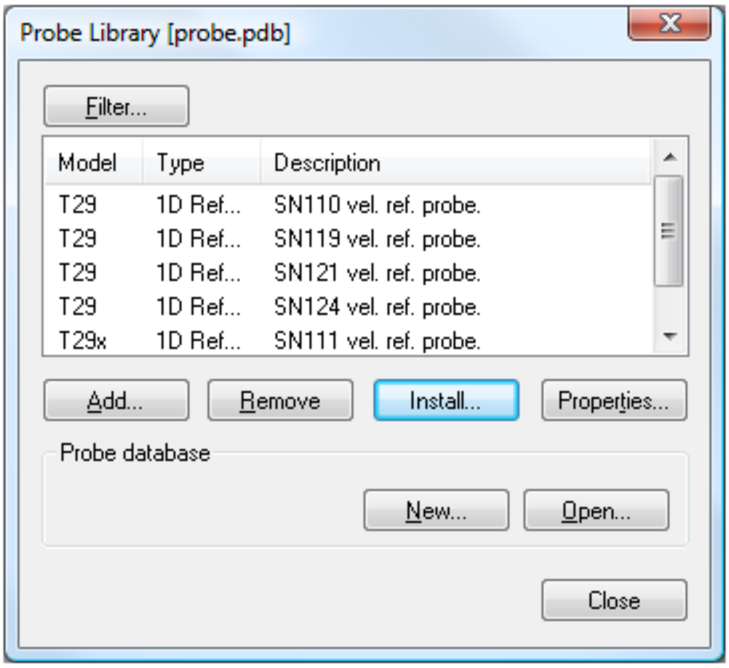
Installing Calibration Files

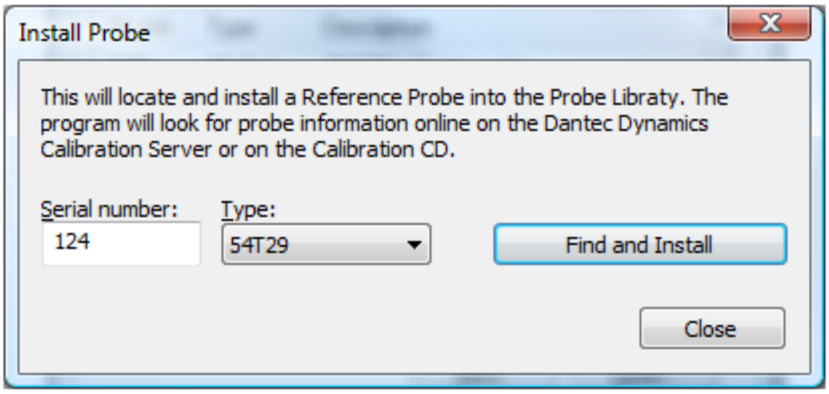
Newer systems are delivered with only one general calibration CD. This CD includes the calibration files for all reference probes.

1. To install the calibration files place the CD into a free CD drive.
2. Enter the Probe Library.
3. Filter on Reference Probes.



4. Press Install, and type in the serial number of your first reference probe.





5. Repeat this for all your 54T29 reference probes.

Legacy Systems

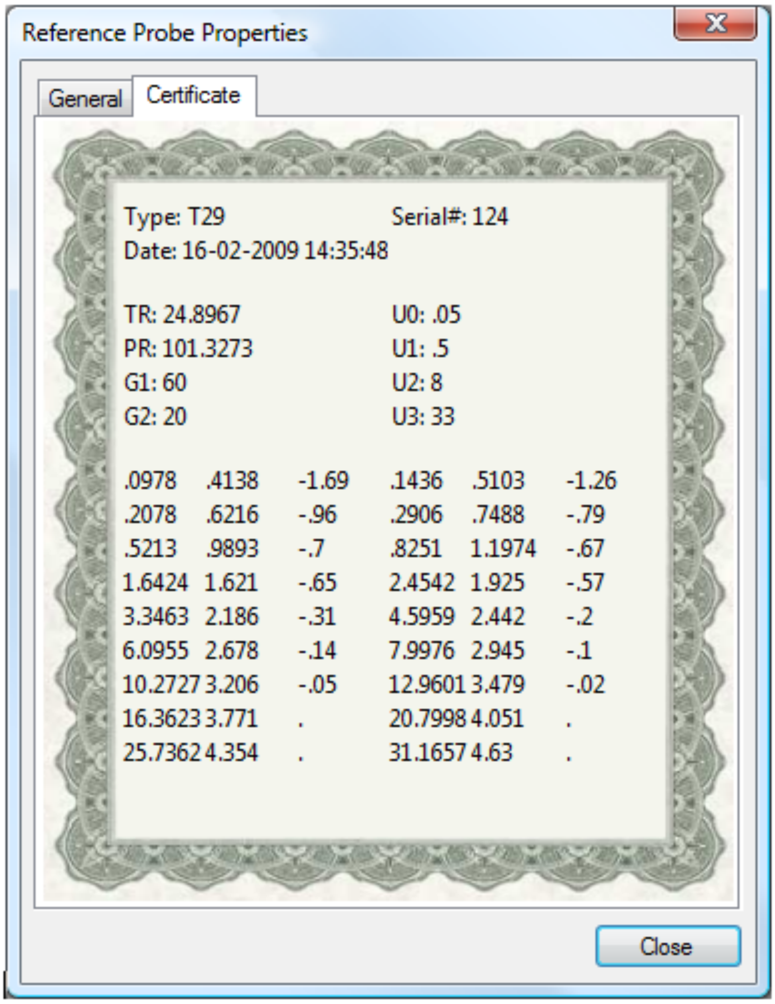
A dedicated Calibration File CD is delivered for every probe specified by the serial number of the probe. To install the Calibration File place each CD one at a time into a free CD drive and follow the auto run instruction, or manual run the setup.exe file in the root of the CD.

The calibration files will automatically find your software and install the files for the reference probes.

The calibration files will be copied from the calibration CD onto your system, in the following location: "..\ProgramData\Dantec Dynamics\Libraries", corresponding to the Windows All Users profile.

Reference Probe Properties

The calibration data for each probe can be seen in the properties for the reference probe.



To Close a Probe Library

You leave the Probe Library by:

1. Choose Close in the Probe Library dialog box.
   The dialog box disappears.

It is important to note that a change in Probe Library data is saved, as soon you have typed it in. Therefore there is no Cancel button, which can undo your changes and save the library as it was, before you started manipulating. If you regret, you have to go back and reconstruct the data.

Device Library

A Device Library contains a list of translation drivers for Interface boards. The MiniCTA application software is delivered with a number of standard Device drivers. The Device translation drivers are independent software programs. This means that you can write and add your own drivers, if you want.

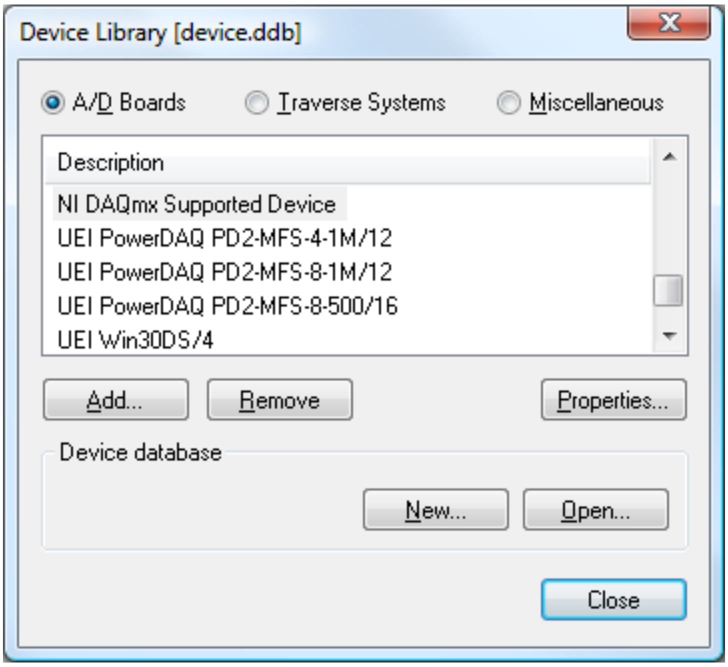
A Device Library always has the filename*.ddv.

It can contain three driver types:
AD drivers. Filename*.adv
Traverse drivers. Filename*.tdv

"A/D Devices" on page 159

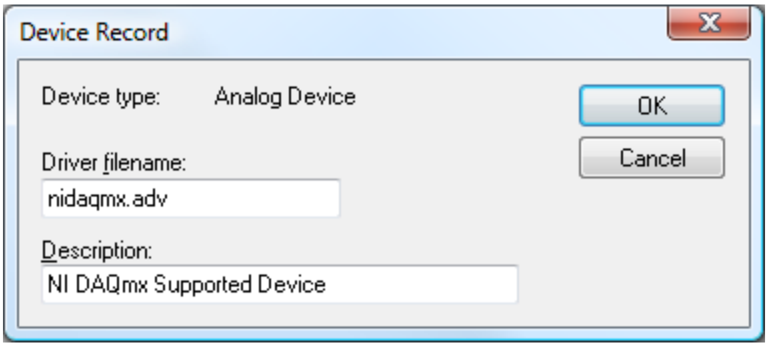
To Open a Device Library

1. Choose Device Library from the File menu.
   Device Library dialog box opens, where you can see the content of the last opened library.
2. Select driver type radio button: A/D devices, Traverse or Misc.
   The drivers list shows the drivers available in the library.



Driver Properties

Select properties to see the file name of the selected driver. A Device Record dialog box opens:



To Add and Remove Drivers

Same procedure as with probes in Probe Libraries.

To Create New or Open Another Device Library

Same procedure as with probes in Probe Libraries.

To Copy and Paste a Driver

Same procedure as for copying and pasting in Probe Library.

To Close a Device Library

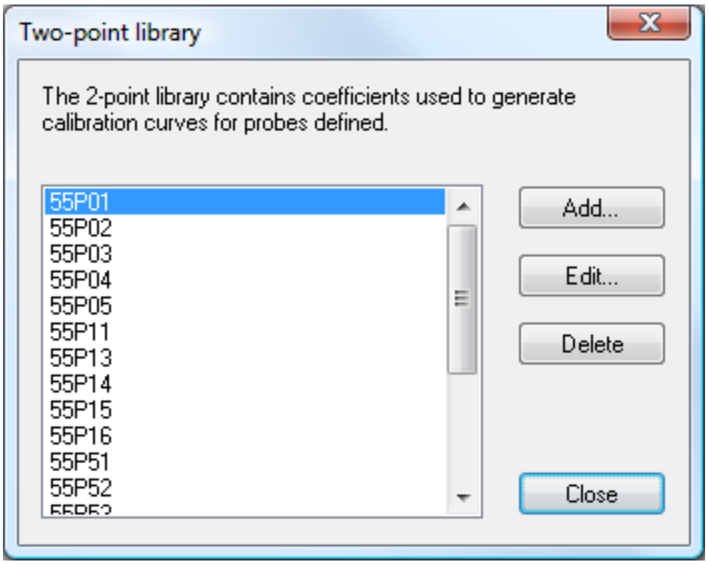
1. Choose the Close button in the Device Library dialog box.
   The dialog box closes.

Note
The last closed Device library will open, when you use the Device command in a project hardware configuration.

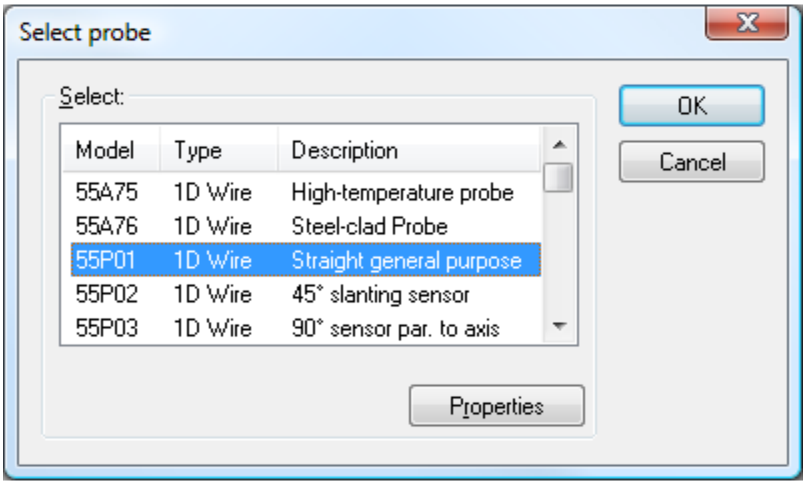
Two-point Library

The Two-point Library contains calibration data for families of probes (miniature wire probes, gold-plated wire probes etc.). Based on the family calibration a transfer function for a specific probe in the family can be made from a calibration in two points only.

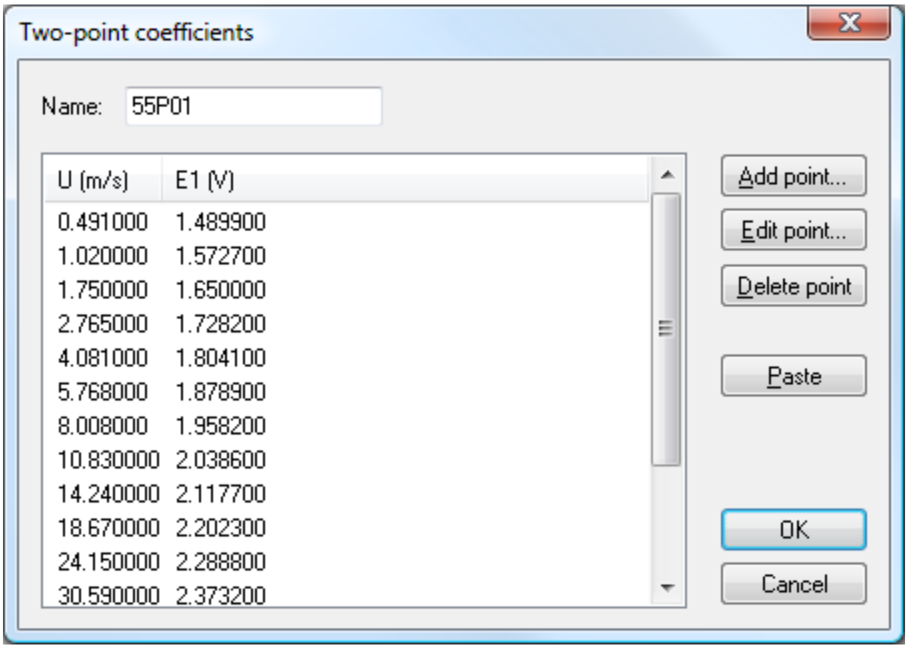
To Add a Probe Family



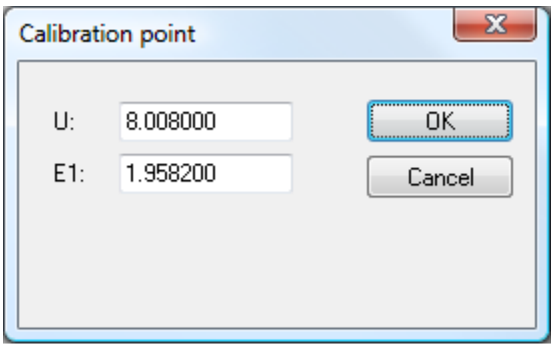
1. Choose Two-point Library from the File menu.
2. The library opens displaying the list of probe families.
3. Click on the Add button. A Select probe dialog box opens.



4. Select a probe from the family that you want to add and click OK.
5. A Two-point coefficients dialog box opens.



7. Now click Add to enter the calibration data for the family. A Calibration point dialog opens.



8. Enter the velocity U and the voltage E, click OK and repeat Add until all points are added.

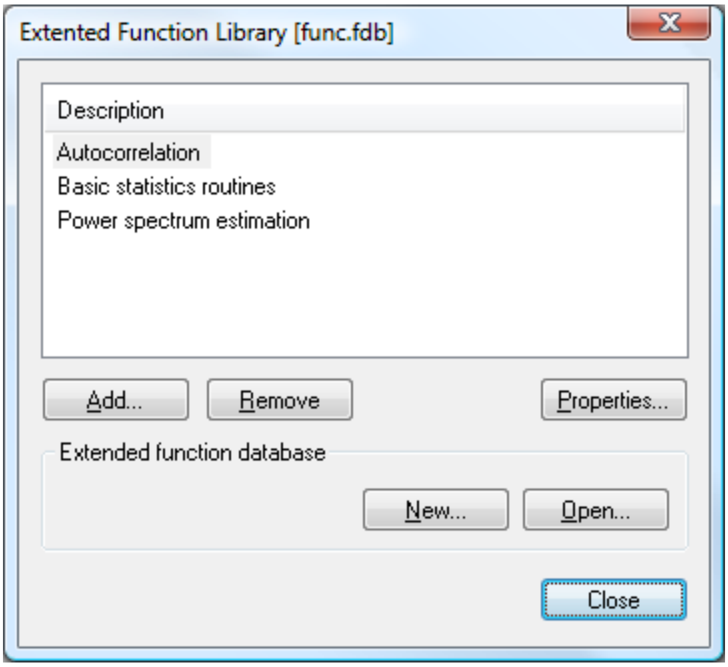
Important
The algorithms used to create the family transfer function require 15 calibration points, preferably with a log-square distribution.

Function Library

A Function Library contains translation drivers for different types of data reduction. The StreamWare software will be delivered with a number of standard Function drivers. The drivers are independent software, and you can add new Function anytime by writing a translation driver. Function Libraries always have the filename*.fdb. The individual Function drivers all have the filename *.fdv.

To Open a Function Library

1. Choose Function Library from the File menu.
   Function Library dialog box opens, where you can see the content of the current library.



6.2.5 Import/Export

The Import/Export commands allow you to exchange data with other Window software applications.

To Import Data

Before you can import data they must exist as tab -or comma-separated values in a file in the project database or in a directory with a path to it, and you must have created and opened a Data sheet for the data to be imported into.

1. Choose User-defined Worksheet from the Windows menu.
2. Choose Import from the File menu.
   An Import From File dialog box opens.

3. Select drive and directory in the Drives and Directories list boxes, if the data file is not in the project database, and you do not have a path to it.
   Note that the dialog box opens with only tab -and comma-separated files listed. You can get a list of all files by selecting All files in the List Files of Type list box.
4. Select the file you want to import in the File Name list.
5. Choose OK.
   The dialog box closes, and the Data sheet is filled out with the wanted data.

To Export Data

Before you can export data they must be present in a Data sheet.

1. Choose the Data sheet by means of the Load Event button in the Main toolbar or from the Project Manager.
2. Select the columns you want to export.
3. Choose the Export command from the File menu.
   An Export To File dialog box opens.
4. Select drive and directory, where you want the data file to be placed, in the Drives and Directories list boxes.
5. Type in a file name in the File Name text box.
6. Select the file type in the Save File as Type list box.
   TXT saves the data with tab separation, while CSV saves them with comma separation (or any other list separator that you may have selected in your Windows Control Panel).
7. Choose OK.
   The dialog box disappears, and the data are exported in the selected format.

6.2.6 Printer Setup

This is the standard Windows command for selecting another printer than the default printer.

1. Choose Printer Setup from the File menu.
   The Printer Setup Dialog box opens with a list of all active printers.
2. Select the wanted printer from the printer list box.
   If you want to change the print options, choose Setup. For further details see Microsoft Windows User's Guide.
3. Choose OK.
   The selected printer will now be used automatically.

6.2.7 Print

If you have connected and installed a printer, you can print the content of an active window, for example Data sheets and Graphs.

1. Make sure the wanted window is displayed.
2. Choose print from the File menu.
   The Print dialog box disappears.
3. Select the options that you want to use.
4. Choose OK.
   - or -
5. Choose the Print button in the Main Toolbar.
   A dialog box appears informing you that the window content is being printed.
   For further details see Microsoft Windows User's Guide.

6.2.8 Exit

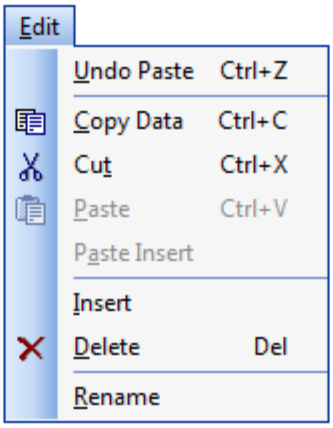
You can leave MiniCTA while it is running in three ways:

1. Choose Exit from the File menu.
   - or -
2. Choose Close from the Control menu.
   - or -
3. Double-click the Control-menu box.

The software closes and returns you to Windows. All changes that you have made since you last closed MiniCTA have been saved while you made them. You will therefore not be prompted to save any files before the application closes.

6.3 Editing

6.3.1 Definitions



The Edit menu allows you to edit in a window using either the mouse or the keyboard. The commands in the Edit menu are combinations of standard window commands. Basically you can copy, paste and delete events or data records. Some commands are duplicated with buttons either in one of the toolbars in main application window.

6.3.2 Editing Projects

To Copy and Paste an Event

Hardware setup, traverse grids, velocity and directional calibrations can be copied and pasted into another project. All other events are configuration specific and cannot be moeved to another project.

1. Select the event in the Project manager.
2. Choose Copy data from the Edit menu or the Copy button in the main toolbar.
3. Open the project, where you want to place the events and Choose Paste or click on the Paste button in the main toolbar.

To Delete an Event

1. Select the event(s) in the database.
2. Choose Delete from the Edit menu.
   Confirm Event Deletion dialog box opens.
3. Choose the Yes button.
   The first selected event is removed from the event list.
   If there are more events to delete you are promoted once more.

Or, if you want to delete all events with one keystroke:

1. Choose the Yes to All button.
   Dialog box closes.
   All selected events are removed from the event list.
   Here you can select records belonging either to specific probes, to specific events or to a combination of both.

6.3.3 Editing Data

Data appears in cells in Data sheets, where each column represents a variable.

All types of data, independent of event type and conversion level, are edited in the same way.

To Copy Data

1. Select the wanted data in rows or columns.
2. Choose the Copy button in the Main toolbar or choose Copy data from the Edit menu.

To Cut Data

1. Select the wanted data.
2. Choose the Cut button in the main toolbar or choose Cut from the Edit menu.
   The data are removed from the Data sheet and placed in the Clipboard.

To Paste Data

1. Point at the cells, where data have to be pasted. This can be in the same or in another Data sheet window.
2. Choose the Paste button in the main toolbar or choose Paste from the Edit menu.
   The new data appear in the selected cells.

To Insert Between Cells

1. Select the insertion point.
2. Choose Insert from the Edit menu.
   The cells are now shifted down.

Note
You cannot insert between columns. If so, you would have to introduce a new variable. This is not allowed, as the setup structure would then be violated.

To Delete Data

1. Select the data to be deleted.
2. Choose Delete from the Edit menu.
   The cell contents are now deleted. The cells remain active and new data can be filled in.

To Change the Content of a Cell

You can change the content of a cell in a calibration event and in a traverse grid. You cannot overwrite data in a database with raw or reduced data.

1. Select the cell by clicking in it.
2. Type in the new value.
3. Choose Carriage return, click in another cell or use the up/down arrows.
   The cell is now updated. Its previous content can be restored by choosing Undo edit from the Edit menu.

To Change the Content of a Cell Containing Date or Time

If the cell contains a date or a time, as in Data sheets with reduced data, you can use the following short-cuts when typing new dates and times:

Date	If you type 19, the cell is updated with present year, e.g. 1993 (valid 1993 - 2079)
Time	If you type 11, the cell is updated with 11:00:00 If you type 11:37, the cell is updated with 11:37:00 The cells assumes AM, if you do not type PM.

You can use any separator. The date and time will appear as you have formatted them in the Windows Control Panel.

To Close Data Sheets

1. Double click on Menu Control box of the window.
   Save Event dialog box opens, which prompts you for a new Event Identification.

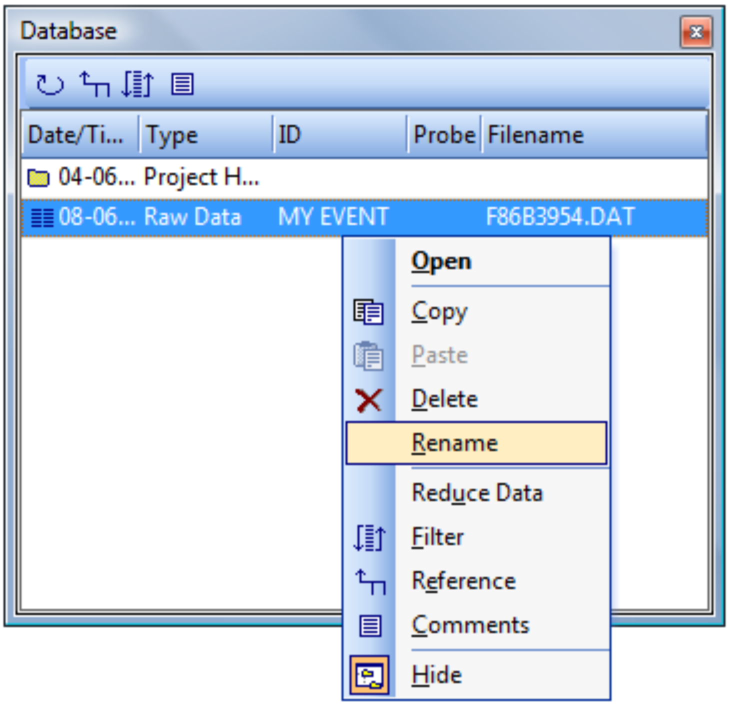
If you accept, the event is now added to the event list in the Project Manager. The original event, that formed basis for the editing, remains unchanged.

Note
You can only get rid of the original event by deleting it.

6.3.4 To Rename Events

The event identification may be changed at any time in a Project.

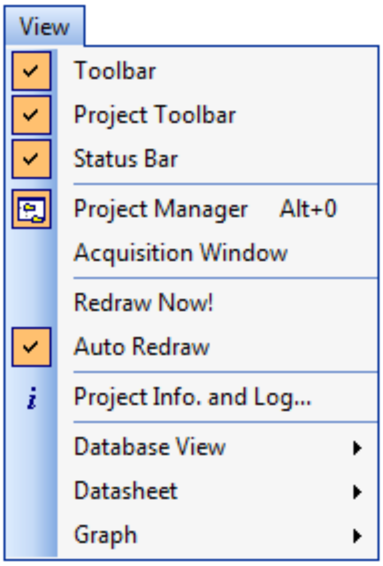
Select the event and choose Rename in the Edit mane. You can also click on the event in the Project manager with the right mouse button and choose Rename.



A Rename dialog box opens, where you can type in the new name.

6.4 View

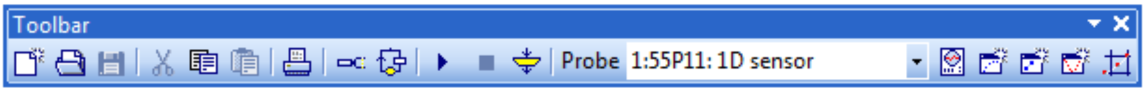
6.4.1 Definitions



The View menu allows you to select toolbars and status bars in the Windows workplace, to arrange the project manager and to view project information, to load data and finally to format data sheets and graphs.

6.4.2 Toolbars

Main Toolbar



Contains tool buttons for all major actions when creating and running a project, which are:

Create new database , open database, system configuration, default setup, data acquisition, data reduction, active probe, online, velocity calibration and graph.

Project and Event Toolbar

Contains tool buttons for all major actions concerning display and editing of events and data inside a project, which are:

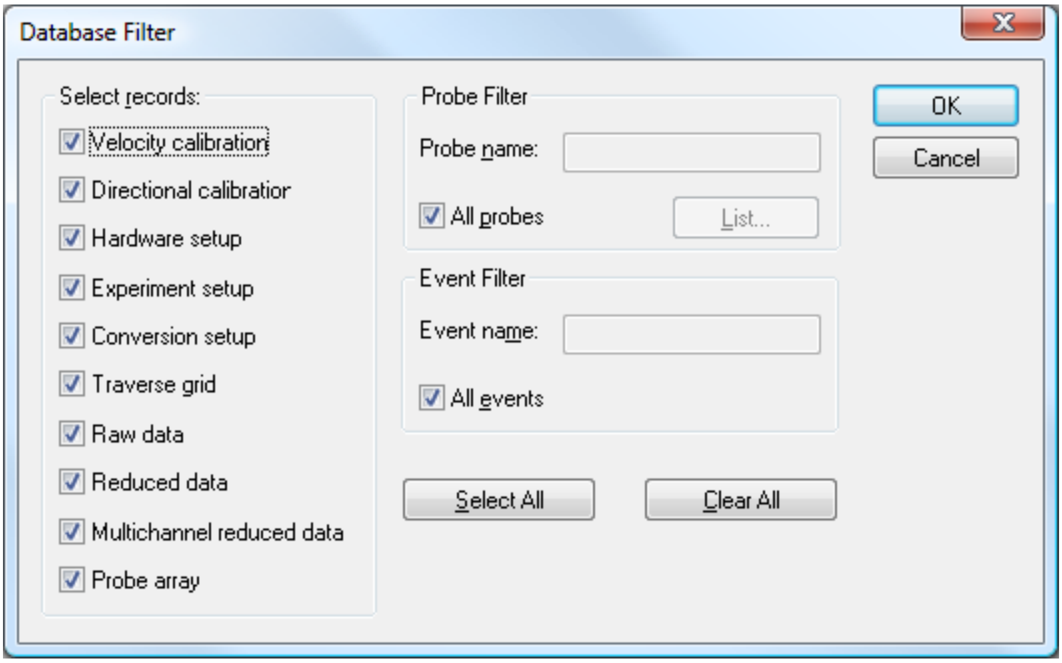
Create a new project, open a project, arrange and rearrange the project manager, load events, formatting data and data sheets, editing hardware setup, calibrations and traverse grids.

Status Bar

The status bar is placed at the bottom of the window and shows the present status. It allows you to follow the acquisition of data, which also is displayed in the Acquisition window.

6.4.3 Project Manager

To Filter Events



Database Filter dialog.
The Filter event command allows you to display only a specific type of events in the Project manager or to display events belonging to a specific probe, if more probes are assigned, or to display a combination of both.

1. Choose the Filter button  in the Project toolbar or choose Filter from the View menu.
2. Database Filter dialog box opens.

Here you can select records belonging either to specific probes, to specific events or to a combination of both.

To Find References

This command allows you to find the events related to a specific event, for example the raw data and the conversion/reduction that form basis for a reduced data event.

1. Select the event of interest in the Project manager.
2. Choose the reference button  in the Project toolbar or select Reference in the View menu.

The related events are listed in the Project manager.

To Refresh

This command recreates the full Project manager.

1. Choose the refresh button  in the Project toolbar or select Reference in the View menu.

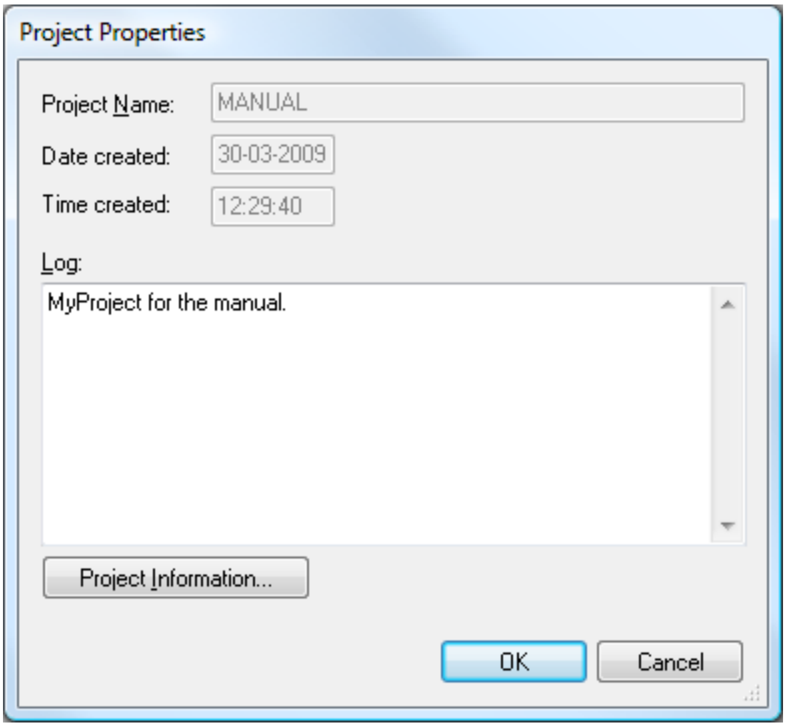
All events are listed in the Project Manager.

Project Info and Log

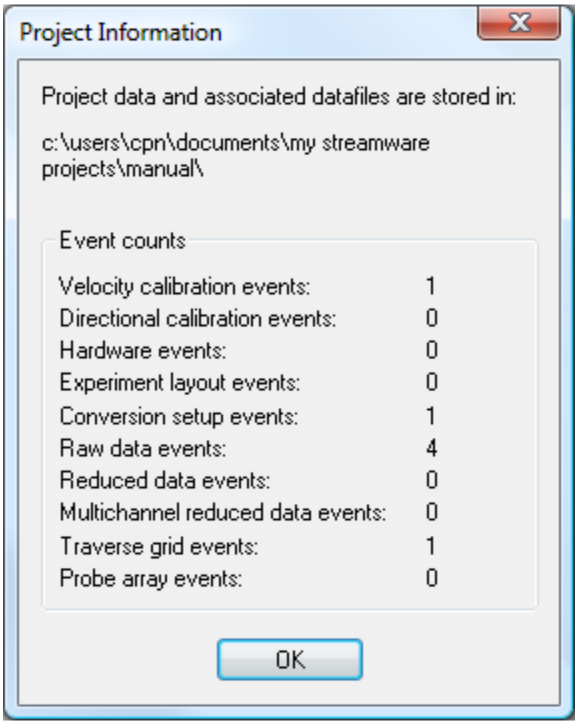
This command gives access to the Project log and to statistical information about the events.

1. Select Project info and log in the View menu.

A Project properties dialog box opens displaying the contents of the log. You can make log entries at any time.

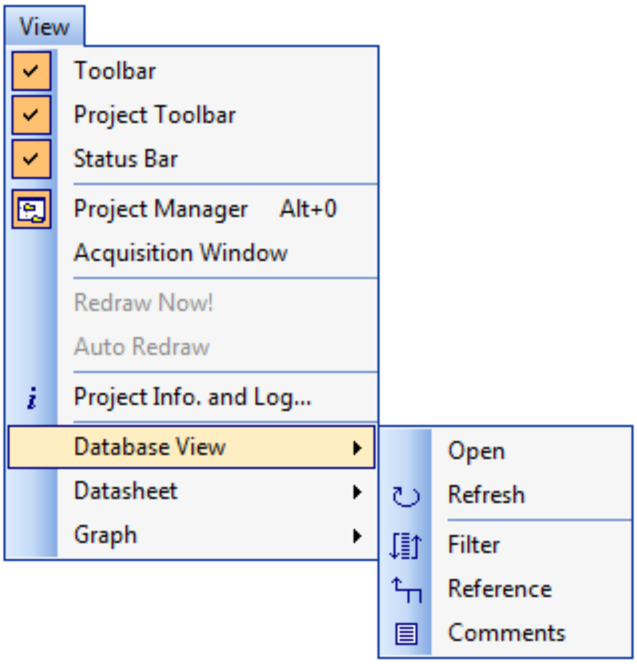


Project Properties dialog.
Select Project Information to get a list of event statistics:



Project Information dialog.

6.4.4 Database View

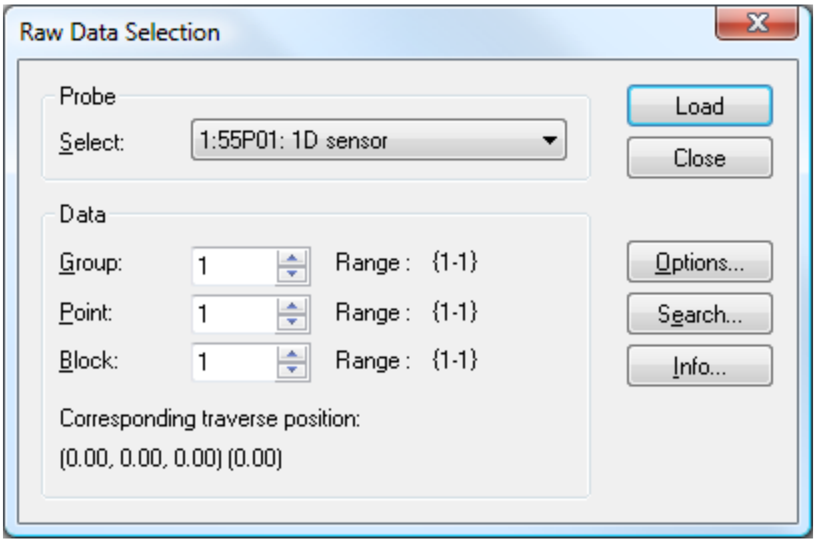


This command allows you to load the content of a database (raw data, reduced data or traverse grids).

To Load Raw Data

Raw data are placed in data blocks, each block representing one acquisition made during an iteration in a group sequence.
As you can load only one probe, one group, one point and one block at a time you must make a selection prior to loading.

1. Double click on the record in the Project Manager
 - or -
 select Database view/Open from the View menu.
 Raw Data Selection dialog box opens.

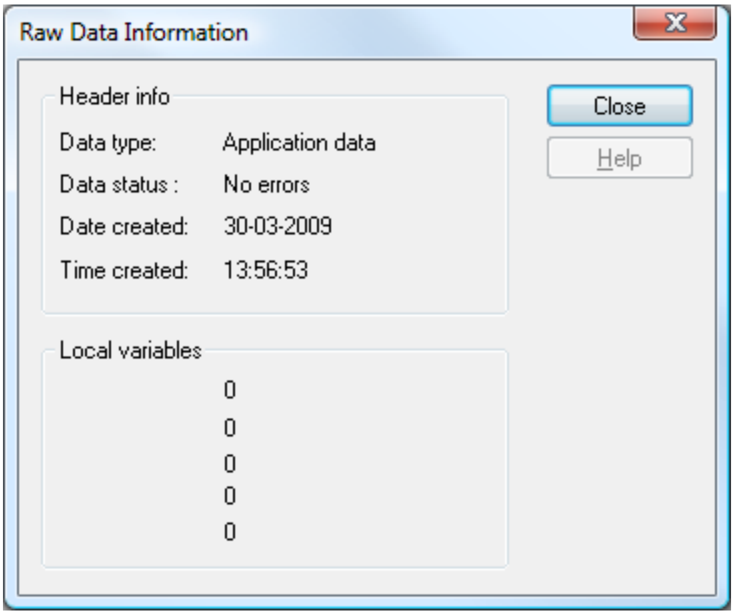


Raw Data Selection dialog.

To Select Data

1. Select the Group, Point (Iteration) and Data block.

Note
The group selection is blanked, if data are acquired from Run Default , as there is only one group in this mode.



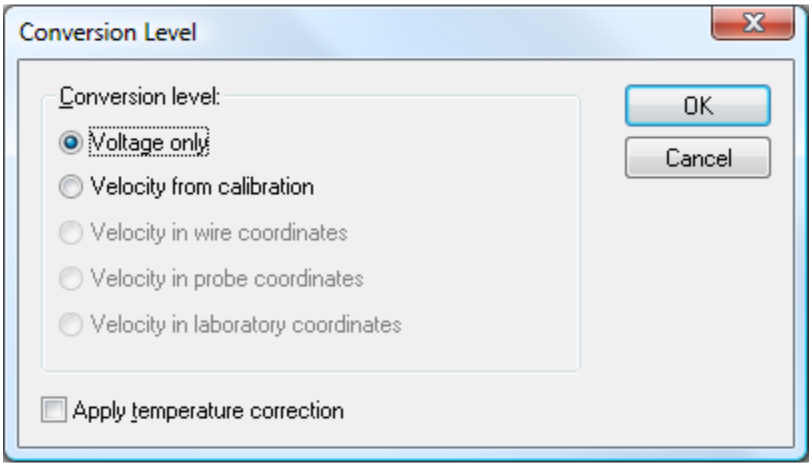
Raw Data Information dialog.
You can get further information about the data by selecting the Info button, which opens a Raw Data Information box:

To select conversion level and temperature correction

Data can be loaded as voltages or velocities (if CTA data) with or without temperature correction.

1. Choose the Option button

Conversion level dialog box opens. Data acquired in Default setup always uses the default Conversion. Data acquired in an Experiment uses the Conversion belonging to the actual group, where data were acquired.



Conversion Level dialog.

Voltage only

The Raw data are loaded as voltages.

Velocity from calibration

The velocities are calculated using the transfer function from the assigned Conversion.

Velocity in wire coordinates

The calibration velocities are decomposed into the wire coordinate system. They are calculated using the transfer function and the Yaw and Pitch coefficients defined in the assigned Conversion.

Velocity in probe coordinates

The wire velocities are decomposed into the probe coordinate system. They are calculated on basis of the wire angles from the Probe Library.

Velocity in laboratory coordinates

The probe coordinate velocities are transformed into the laboratory coordinate system as defined in the assigned Conversion. For details see See "Setting Up the System" on page 94.

2. Select OK. The dialog box closes and you are back in the Raw Data Selection dialog box.
3. Select Load.
   A Data sheet with the data in converted form opens behind the dialog box.
   You can now select a new data block and repeat the loading until all data are loaded.
4. Close the Raw Data Selection box.
   You now have a one or more Data sheets with the selected data.

Temperature correction

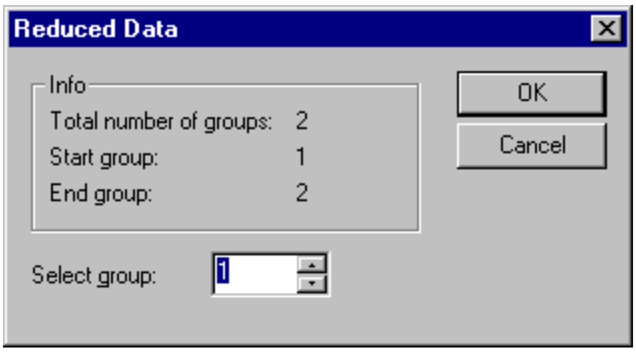
If selected, and the Conversion event includes temperature correction, the raw data are corrected for ambient temperature variations.
If de-selected, the raw data are loaded without any corrections.

To Load Reduced Data Events

The term Reduced data covers a file with a list of the Conversion/ Reduction events that are assigned to each group in a Default Setup. The file does not contain any reduced data as such. When you select Reduced data, you in fact point at the raw data via pointing at a group. They are then reduced in accordance with the Conversion/ Reduction assigned to the group, before they are loaded. Note that this structure only allows you to load reduced data from one group at a time.

To Load Data from One or More Groups

1. Double click on the Reduced Data event in the Project Manager. Or select it and Choose Load event from the Edit menu.
   A Reduced Data dialog box opens, where you can select the group.

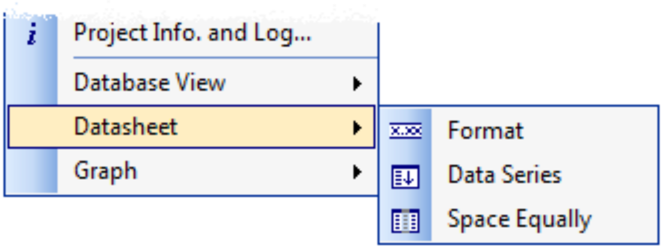


Reduce Data dialog.

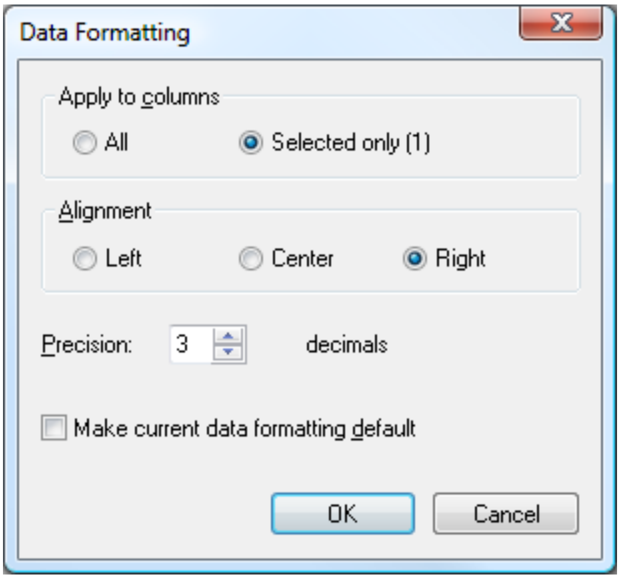
2. Choose OK and a Data sheet with the reduced data, one set for each probe position, opens. You can return to the Reduced Data dialog box and load data from other groups.

To View and Format Data Sheets

When you have loaded a Data sheet, you can format the data, create data series and adjust the column width.



To Format Data



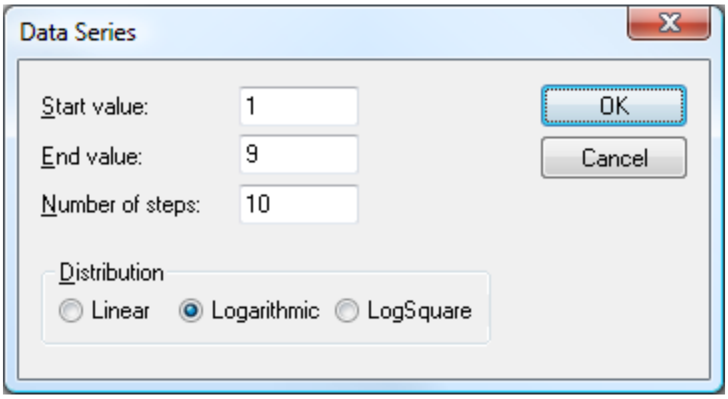
Data Formatting dialog.

Click on the Data format tool button  in the Project toolbar or select Data sheet/Format in the View menu. A Table dialog box opens, where you can choose the precision of one or more columns and alignment data in the cells.

To Create Data Series

Data series can be placed in columns in any Data sheet.
Select the starting cell in the Data sheet.

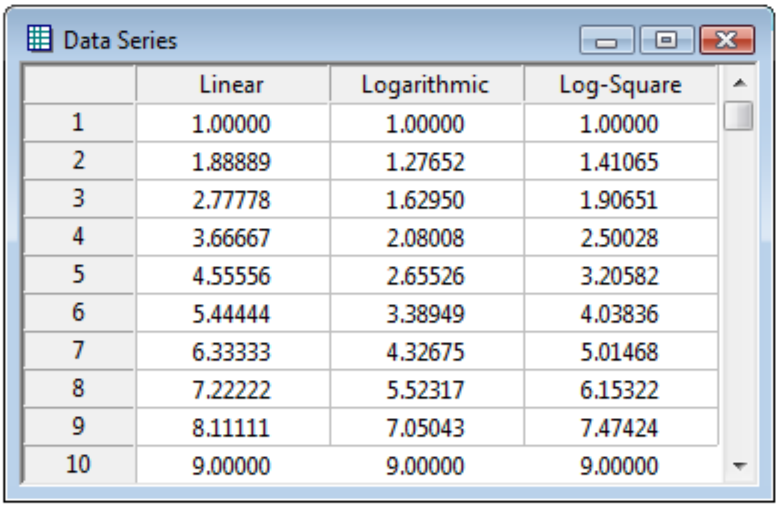
Click on the Data series tool button  from the Project toolbar or choose Data sheet/Data series in the View menu.



Data Series dialog.

A Data Series dialog box opens, where you can select start and stop value, number of steps and distribution.

The distribution may be selected with a view to the application. The Log distribution may be used for calibration points, as they give a more points in the lower part of the range, where hot-wire transfer functions change most rapidly. This is less pronounced for the Log-Square distribution, as can be seen in the below example:



Data Series

	Linear	Logarithmic	Log-Square
1	1.00000	1.00000	1.00000
2	1.88889	1.27652	1.41065
3	2.77778	1.62950	1.90651
4	3.66667	2.08008	2.50028
5	4.55556	2.65526	3.20582
6	5.44444	3.38949	4.03836
7	6.33333	4.32675	5.01468
8	7.22222	5.52317	6.15322
9	8.11111	7.05043	7.47424
10	9.00000	9.00000	9.00000

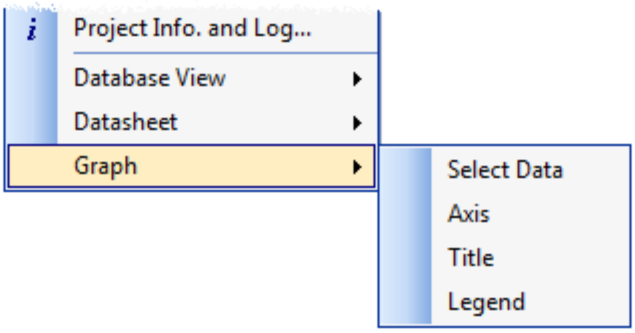
Listing of data series.

To adjust the column width

1. Click on the Space equally button  in the Project Toolbar or select Data sheet/Adjust width command in the View menu.

The columns will now be spaced equally over the entire width of the Data sheet.

6.4.5 Graphs



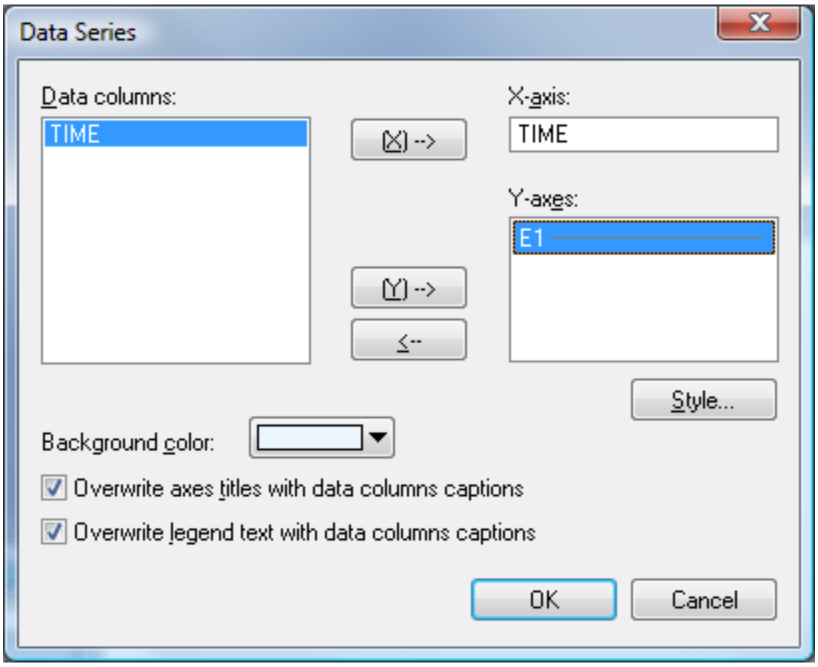
Each data sheet can have one graph attached to it showing up to five plots of data as function of the same independent variable, all from the same Data sheet.

Graphs cannot be saved as independent events, but is saved together with the Data sheet that it belongs to. This means that you can only work with graphs in connection with Data sheet windows.

To Select Data

Up to 5 sets of data , Y-arguments, can be plotted as function of one data set, X-argument.

1. Double click in the Plotting area or choose Graph/Select Data from the View menu. A Select Data dialog box opens.



Select Data Series dialog.

X-argument

1. Select a data symbol in the Data columns list.
2. Choose X button. The data symbol is displayed in the X-argument text field.

Y-argument(s)

1. Select data symbol in the Data columns list box.
2. Choose Y button. The data symbol together with a marker and line piece is displayed in the Y-arguments list box.

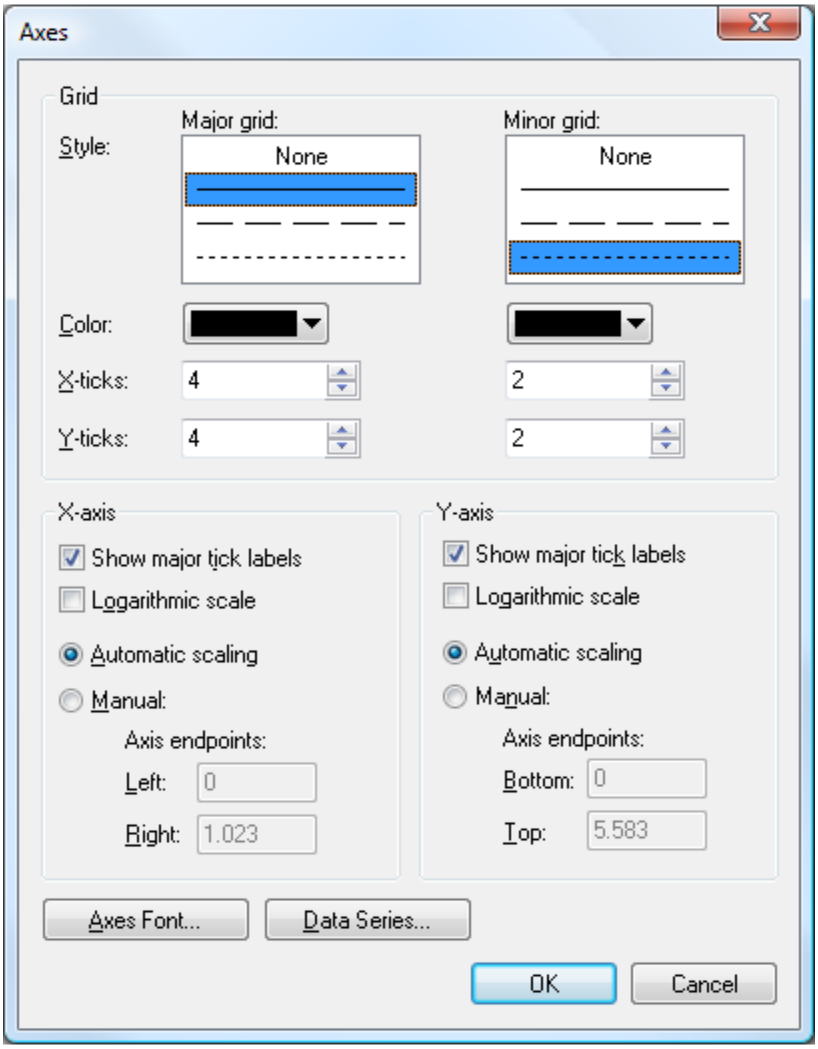
Remove data plots

1. Select the data symbol in the X(Y)-arguments list box.
2. Choose <- - button. The data symbol is removed from the Y-arguments list box and put back into the Data columns list box. The corresponding plot will be removed from the Plotting area.

2. Choose OK.
   The dialog box disappears and the plots are drawn into the Plotting area as specified.

To Define Axes

1. Double click on one of the sides of the plotting area or choose Graph axis options from the Edit menu.
   Axes Properties dialog box opens.



Axis dialog.

Grid

Select Line- and Color style for Major and Minor axis:

1. Click on Full line/ Dot-and -dash/ Dotted or None.
2. Select wanted grid color in Color list box.
   You can change background color in the Plot area by choosing Background.
   For details, see Graph Layout Options section.

Tick marks

Select number of Tick marks for Major and Minor Grids on X-axis and on Y-axis.

Tick Labels

Font for Label or Axes is changed by choosing Fonts. For details, see Graph Layout Options section.

Scaling of Axis

Select between Linear/ Logarithmic scales and between Auto scaling/Manual scaling.

Axis endpoint

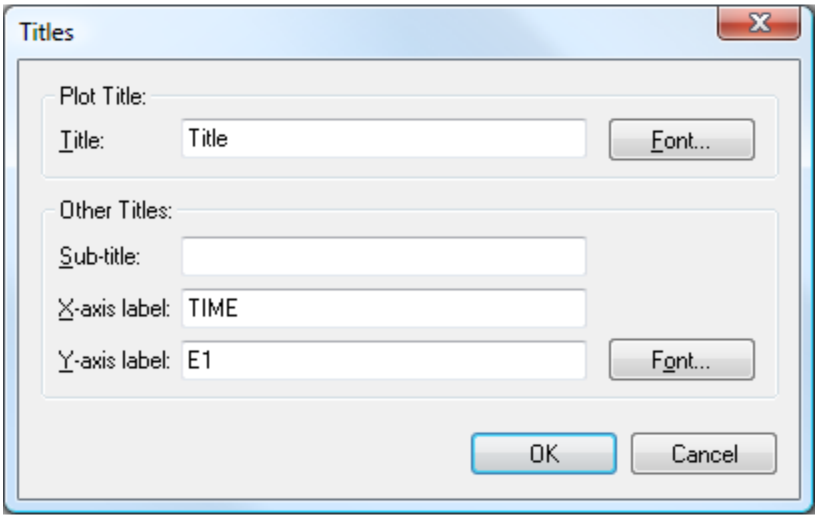
Type in values for Left and Right endpoints, respectively.

Note that the X and Y-axis are edited separately.

2. Choose OK.
   The dialog box disappears, and grid, axis and labels are drawn in accordance with the selections.

To Add Titles

1. Double click on the Title, on one of the sides of the plot area or choose Title from the Edit menu.
   Title dialog box opens:



Titles dialog.

Graph title

1. Type in the Title and Subtitle (if wanted).
   Title font can be edited by choosing Fonts.

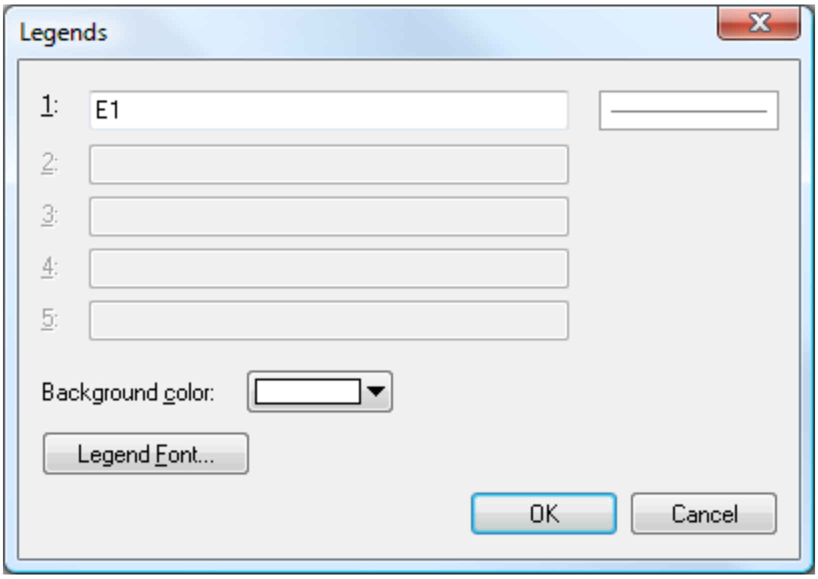
X- and Y-axis titles

1. Type in the wanted titles.
   Title font can be changed by choosing Font.
   For details see Graph Layout Options.
2. Choose OK.
   The dialog box closes and the title is displayed at the top of the Graph.

To Add Legends

You can add legends to each plotted data set.

1. Double click in the Legend field or choose Legend from the Edit menu.
   Legend dialog box opens.



Legends dialog.

2. Type in the wanted legend in the text box to the left of the data marker.
   Legend font and background color in the Legend can be changed by choosing Font and Background.
   Plot style (line, marker and color) can be changed by choosing Style.
3. Choose OK.
   The dialog box disappears and the legends are written into the Legend field.

Graph Layout Options

You can edit the graph layout with respect size, fonts and colors.

To change the size of the plotting area or the legend box

You can change size and shape of the Plotting area or the Legend box using standard Window commands:

1. Point to a boarder or corner that you want to move.
   The pointer changes to a two-headed arrow.
2. Drag the border or corner until the area has the size you want.
3. Release the mouse button.

You can cancel the resizing by pressing ESC before you release the mouse button.

To zoom in a plot detail

You can select a part of the Plotting area to be displayed in enlarged form.

1. Point inside the Plotting area, press the left Mouse button and drag diagonally across the part to be enlarged.
   A frame defining the Zoom area is drawn while dragging.
2. Deactivate the Mouse button.

The selected Zoom area now fills out the entire Plotting area.

To cancel Zoom

You can cancel the zoom and restore the original Plotting area by pointing inside the area and press the right Mouse button.

If the graph has been saved with a zoom plot, the zoom cannot be canceled after reopening. In this case you have to re select the plot data.

To change fonts

1. Choose Font in the actual dialog box.
   Font dialog box opens, where you can do the following selections:
2. Select font name in the Font list.
3. Select font style from the Font Style list.
4. Select size from the Size list
5. Select Strikeout or Underline in the Effects check boxes.
6. Select color in the Color list.
   Each time a selection is made a sample is shown in the Sample box.
7. Choose OK.
   Font dialog box disappears, and the selected text is written with the new font.

To change background color

You can select the background color for the Plotting area and for the Legend separately.

1. Choose Background in the actual dialog box.
   Color dialog box opens, where you can choose between:

Basic colors

1. Select the wanted color in the Basic colors palette.

Custom colors

1. Choose Custom color.
   Color dialog box opens.
2. Drag the color refiner cursor to the area of the color refiner box that shows the color you want.
   Then drag the arrow up or down in the right luminosity bar to adjust the luminosity of the color.
   During selection the color shows in the left side of the Color/Solid box. The right side of the box shows the solid color closest to your choice.
   If you want the solid color then double click in the right field of the box.
3. Select a box for the new color in the Custom color palette. If you choose a box already filled in, the current one will be replaced with the new one.
   If you do not choose a box, subsequent custom colors fill the palette boxes from upper left to lower right.
4. Choose the Add to Customs Colors button.
   The color is filled into the selected box in the Custom color palette.
5. Define any other color you want to add.
6. Choose the OK button.
   The dialog box disappears and the Custom color palette in the previous Color dialog box are filled out with the new colors.
7. Select the wanted color in the Custom color palette.
8. Choose OK.
   The dialog box disappears, and the selected area in the graph is updated with the new color.

To define plot style

The plot style, defined through lines, markers and colors, can be edited for each plot individually.

1. Double click on the Plotting area or choose Select data from the Edit menu.
   Select Plots dialog box opens.
2. Select the wanted data in the Y-argument list box.
3. Choose the Style button.
   Plot Style dialog box opens where you can select and combine the following:
   1.Line.
   2.Marker Style
   3.Color
   in their respective list boxes.
   A sample box shows the selected style on the actual background color.
4. Choose OK.
   The dialog box disappears and the plot is drawn in accordance with the selected style.
   Define any other plot style you want.
5. Choose OK.
   The dialog box closes, and the graph(s) shows in its updated form.

Graph Execution Hints

When large amount of data is dealt with, redrawing of the plots on the screen in between editing processes may take quite some time. Speeding up editing can be done using the Auto-redraw, Draw now facilities and the X-axis data sorted.

Auto-redraw

When you choose Auto-redraw from the View menu (tick mark is added), plots having been hidden by a dialog box are automatically redrawn, when the dialog box closes. When you disable the command (tick mark is removed) the plot is not redrawn until you ask for it. The Plotting area will appear empty, when the dialog boxes close.

Draw now

When you choose Draw now from the View menu, the selected plots are drawn into the graph.

6.5 Configuring the System

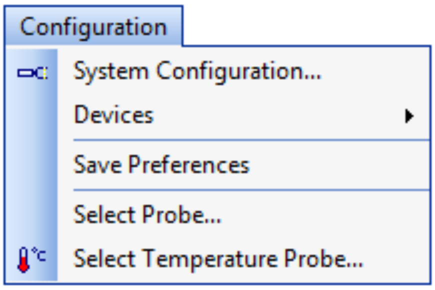
6.5.1 Definitions

Configuration covers the physical layout of the MiniCTA systems, the A/D device and Traverse systems employed.

System Configuration: Defines probes, CTA modules, MiniCTA Frames, A/D input channel connections (assignments) and if employed External probes and their assignments.

- Devices: Defines A/D devices and Traversing Systems
- Save Preferences: Saves the screen setup (toolbars etc.) for next start up.
- Local variables: Defines one or more A/D input channel(s) to acquired and saved as a mean value together with acquired raw data.
- Select current probe: Select one probe for Online display, data reduction etc.

It is important to note that the project cannot be modified with respect to hardware configuration, once an event that relates to the configuration, has been saved.



System Configuration

System Configuration involves the complete selection and connection of Probes, CTA modules, MiniCTA Frames, Comports and A/D input channels. It contains the following actions:

Selecting probes, supports and cables from the Probe library.
Assigning A/D channels to the CTA outputs.
Adding temperature probe, if temperature correction is required.

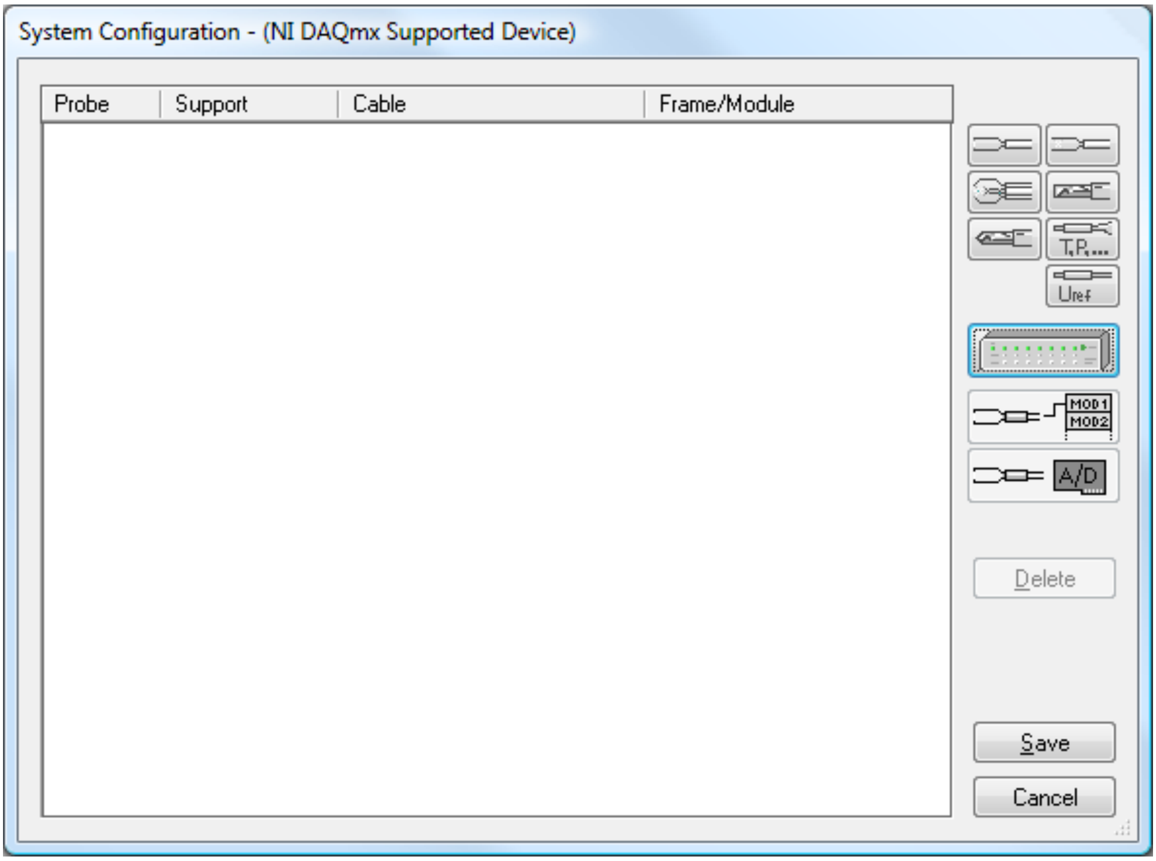
Note
An A/D device must have been selected and configured before a System configuration can be created.

Before you define the System Configuration, it is also recommended that you establish the physical setup, i.e. mount the probes, connect them to the MiniCTA's and make the connection between their outputs and the A/D device inputs.

To Open the System Configuration Menu

1. Click on the Probe button  in the toolbar or choose System configuration in the Configuration menu.

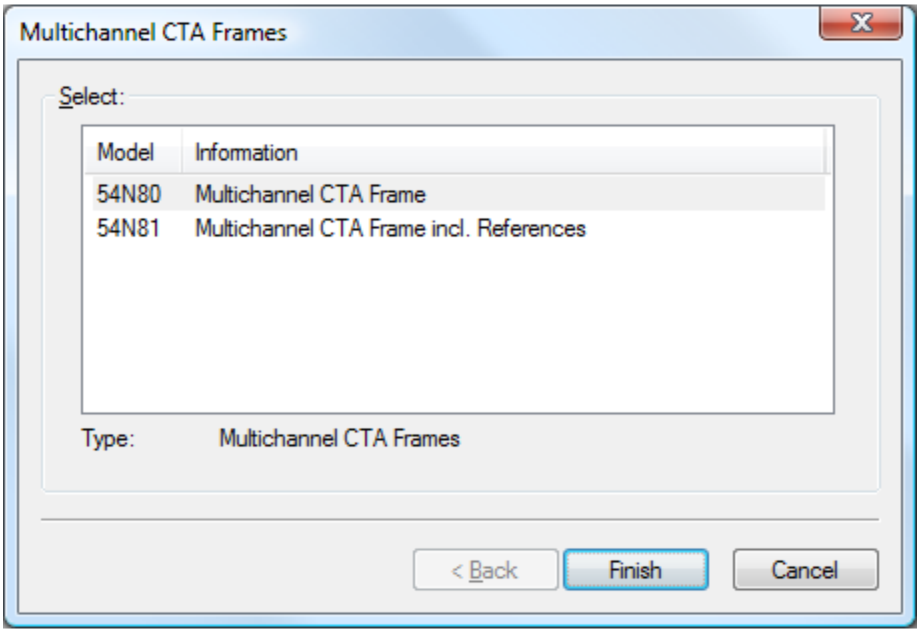
System Configuration dialog opens. It contains a Configuration map and a toolbox with icons for probes and A/D channel assignment.



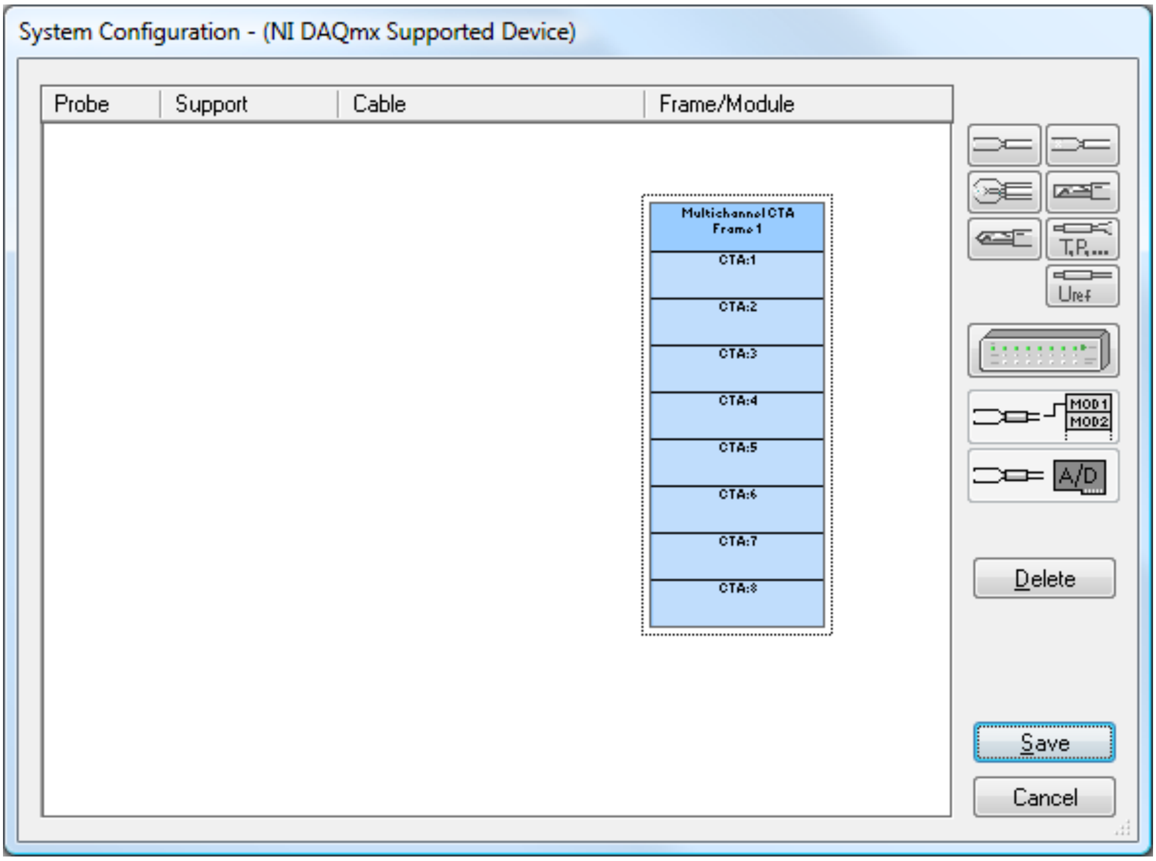
To Select MiniCTA Units

MiniCTA units cannot be selected separately. The MiniCTA is automatically added to the configuration, when a probe is selected.

To Select Multichannel CTA Frames

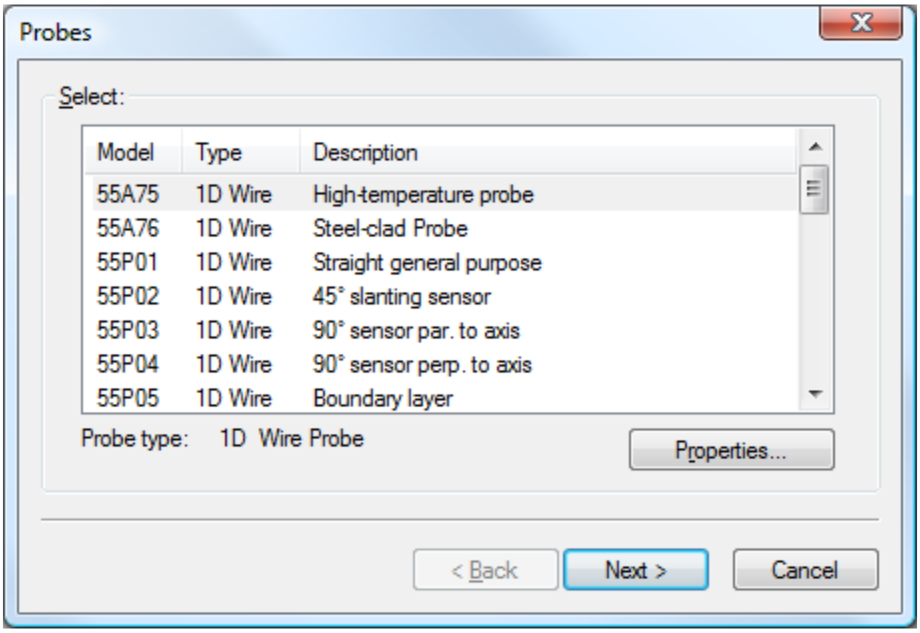


1. Choose the Frame button in the toolbox. Multichannel CTA Frames dialog box opens.
2. Select the actual frame. A frame with 6 or 8 CTA modules are placed in the Configuration map.



To Select a CTA Probe, Support and Cable

1. Choose the Probe button in the toolbox that corresponds to the actual probe type.
   Probes dialog box opens.
2. Select the Probe Type, Next the Support Type and Next the Cable Type in the list boxes. Choose Properties, if you want to see the technical data for the selected Types. For further details about the data see See "Files" on page 41.
3. Choose OK.
   The selected Probe, Support and Cable(s) are now placed in the Configuration map.
   The outputs from the CTA modules are by default connected to the first free A/D input channels.

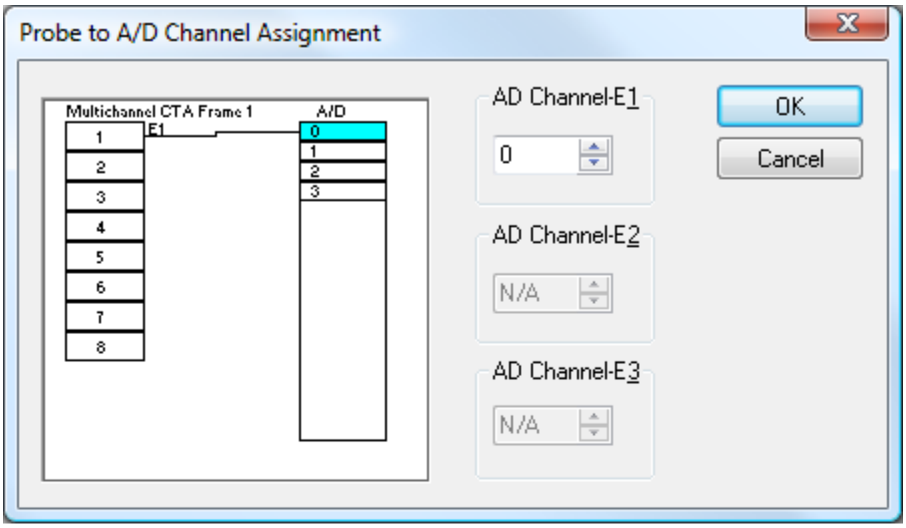


If you want to use a non-standard probe, it must be added to the Probe Library, before you perform the System Configuration. The same is the case for other supports and cables. For details see Part 5.2 Handling Files (Databases).

To Assign A/D Channels to CTA Modules

If the actual A/D to probe assignment differs from the default map, you can rearrange it.

1. Select the probe in the map by clicking with the left mouse button close to it.
2. Choose the Probe to A/D Channel Assignment button  in the toolbox.
   A Probe to A/D Channel Assignment dialog box opens.



3. Select the actual A/D channel number(s) for Channel E-1 (E-2 and E-3).
   Free channels are colored light blue, while occupied channels are red.
4. Choose OK.
   The dialog box disappears.

Select next probe, and continue until all probes signals are properly assigned.

To Select Temperature Probe (only with 54N81 Multichannel Frame or with 54T40 Temperature Amplifier)

If you want to have one temperature sample for each CTA probe voltage sample for temperature correction purposes, you must add a probe for that purpose and select it as Temperature probe.

1. Choose the Misc. probe button  in the toolbox. The probe is attached to A/D input channel no 6 by default (this corresponds to its fixed placement in the 54N81 Multichannel frame).

Note
Temperature correction requires that the temperature probe in the configuration is assigned for this purpose. This is done by the Select temperature probe command in the System configuration menu.

To Select Velocity Reference Probe (only with 54N81 Multichannel CTA)

Choose the U,ref probe button  in the toolbox. The probe is attached to A/D input channel no 7 by default (this corresponds to its fixed placement in the 54N81 Multichannel frame).

To Delete Items in the Configurations Map

1. Select the wanted item in the Configuration map.
   You do that by clicking to the left of the item button in the map.
2. Choose the Delete command button.
   The item now disappears from the map.

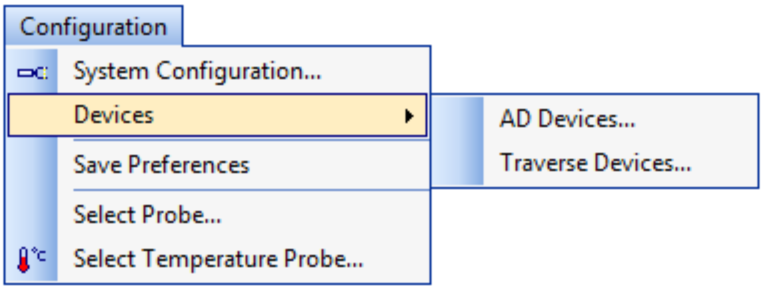
To Leave the System Configuration

1. Choose OK in the System Configuration dialog box.
   The dialog box disappears.

The defined configuration now becomes an inherent part of the project. It will not be listed in the project Manager.

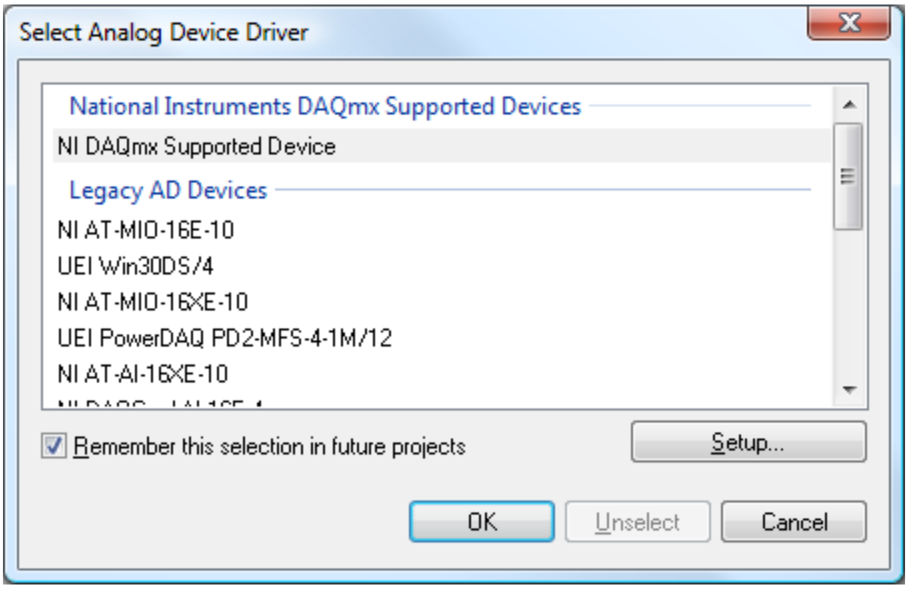
Note
You can change the CTA configuration as long as you have not saved an event, that relates to the hardware configuration. After that no changes are possible any more, unless you delete the events first.

6.5.2 Defining Devices



To Select an A/D Device

1. Choose Configuration/Devices/A/D drivers.
2. Select A/D Driver dialog box opens with a list of the supported A/D drivers.



3. Select the driver for the actual board.

To Change A/D Device Switch Setup

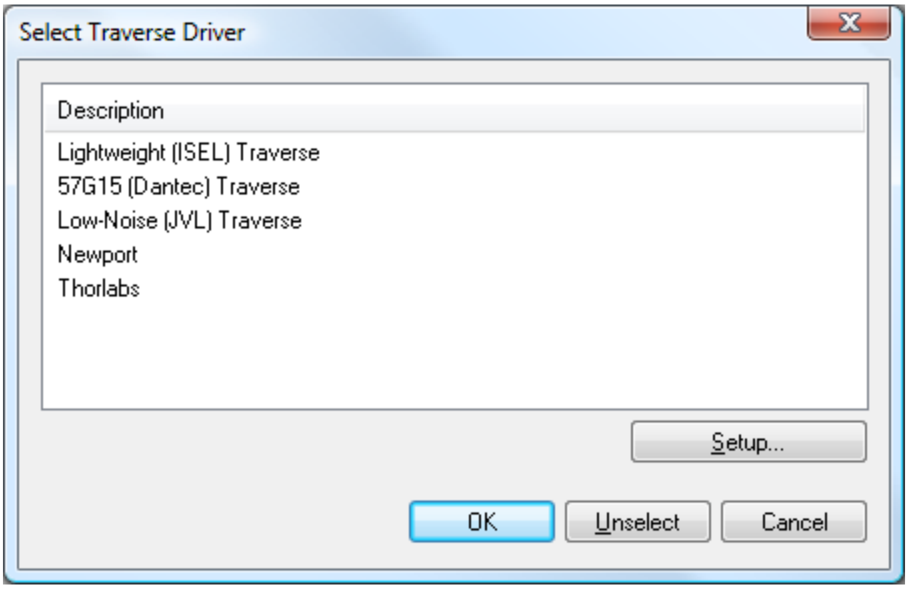
You can check the A/D device Settings by choosing the Setup button and change them if needed.

4. Choose OK.

The A/D Driver is now applied as part of the project hardware configuration.

To Select and Setup a Traverse

1. Choose Configuration/Devices/Traverse.
   Select Traverse Driver dialog box opens with a list of traverse drivers.

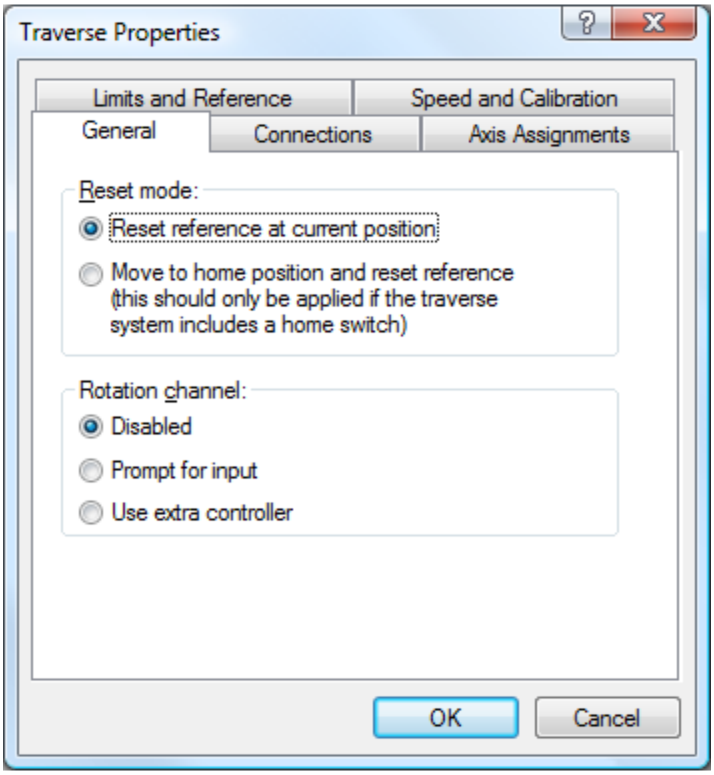


2. Select the driver for to the Traverse System attached to the PC.

Traverse Setup

3. Choose the Setup button and define the following setup parameters:

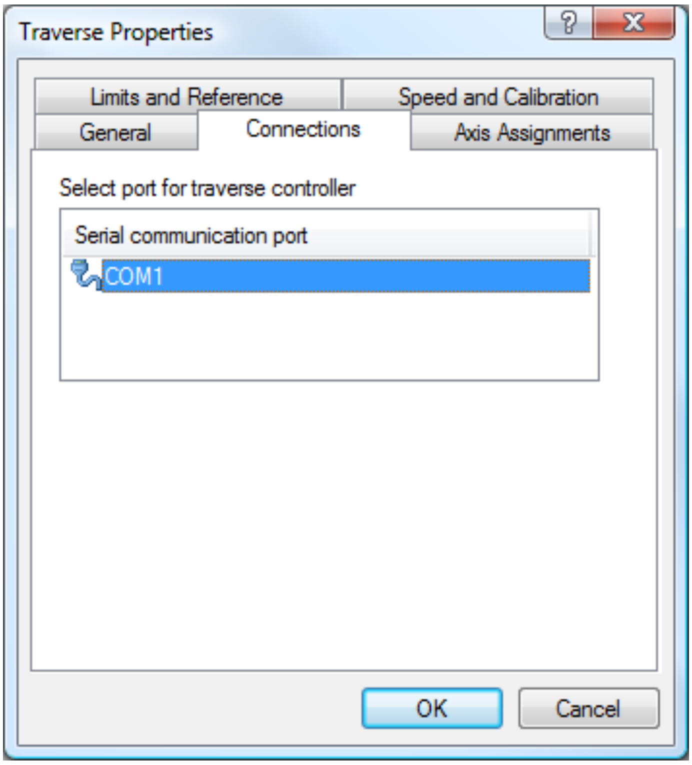
Note
The following parameters are valid for the Dantec Lightweight traverse.



General

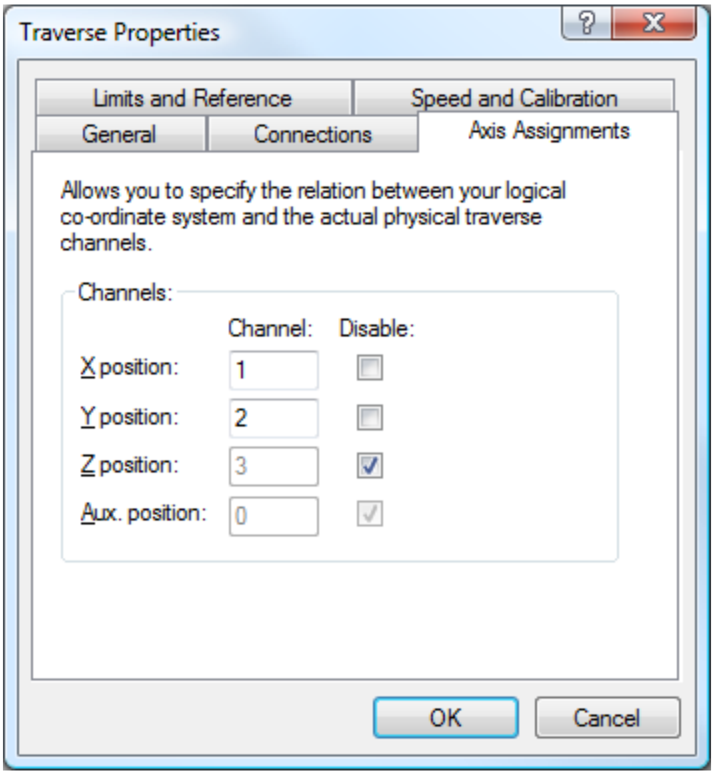
In the present example the probe is traversed without being rotated. The Rotation channel is therefore disabled.

Connections



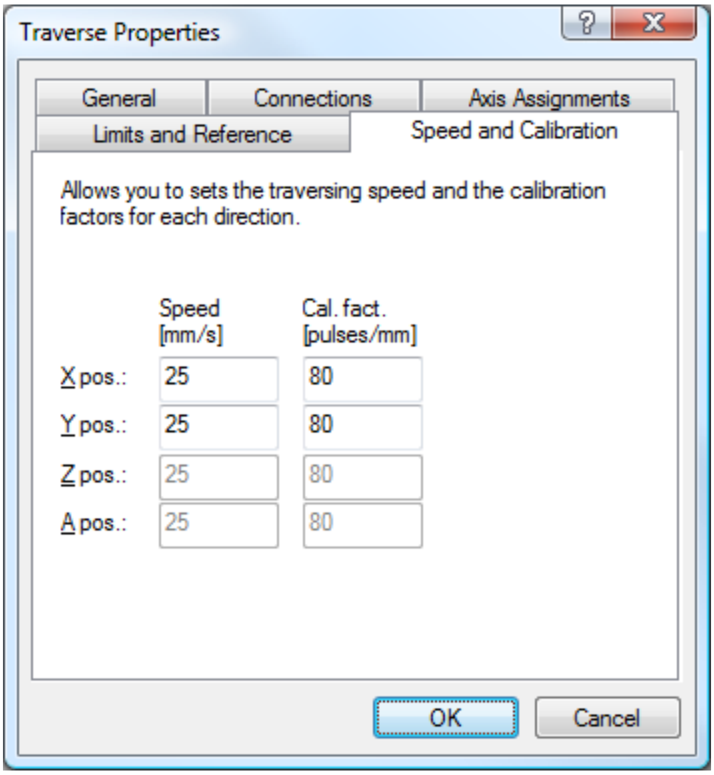
Note

The use of a Traverse requires an additional comport. In the present example COM3 has been selected. It is important to check that there is no interrupt conflicts between the boards in the PC.

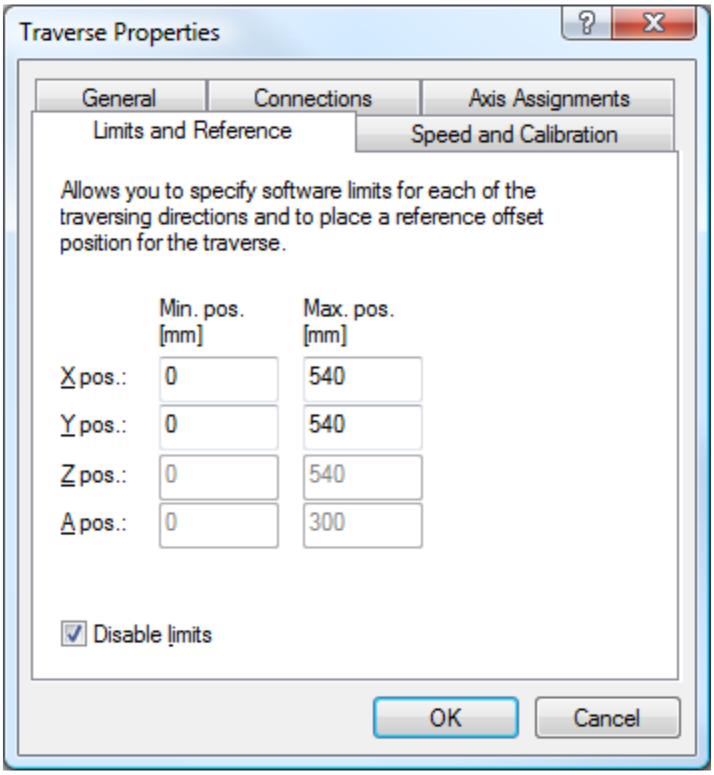


Axis assignment

Speed and Calibration (Lightweight Traverse)

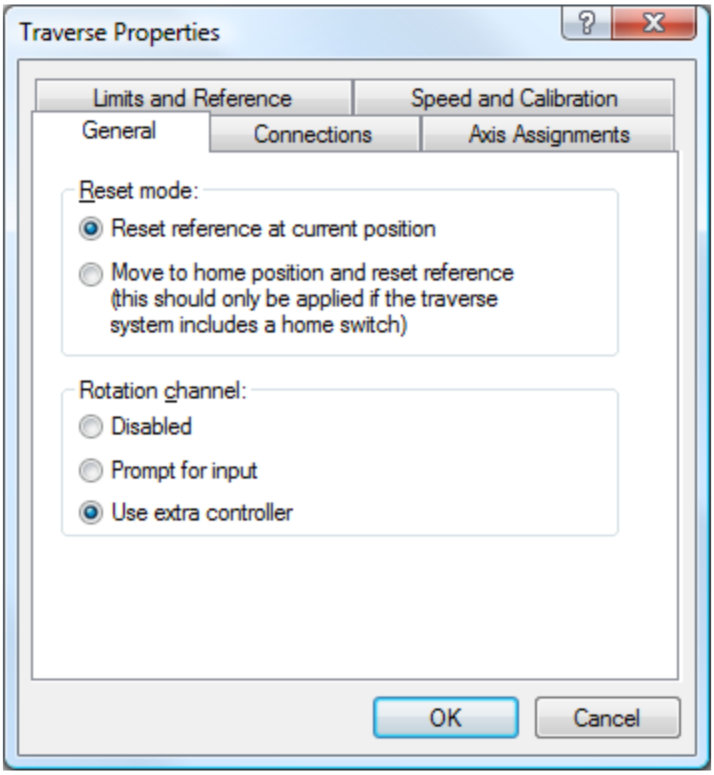


Limits and References

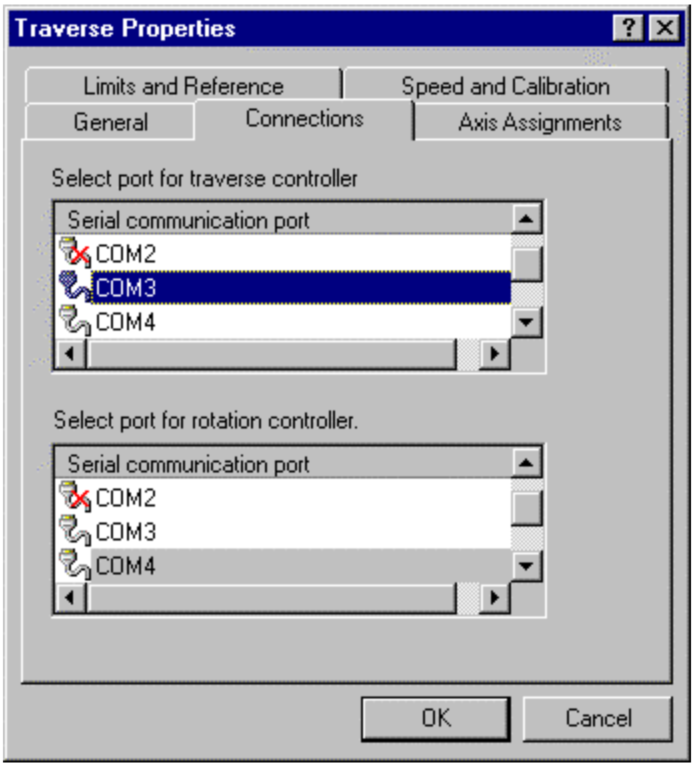


4. Choose OK to finish the setup. You are now back in the Select Traverse Driver dialog. Choose OK. The Traverse driver is now part of the project hardware configuration.

Traverse Setup with Rotation

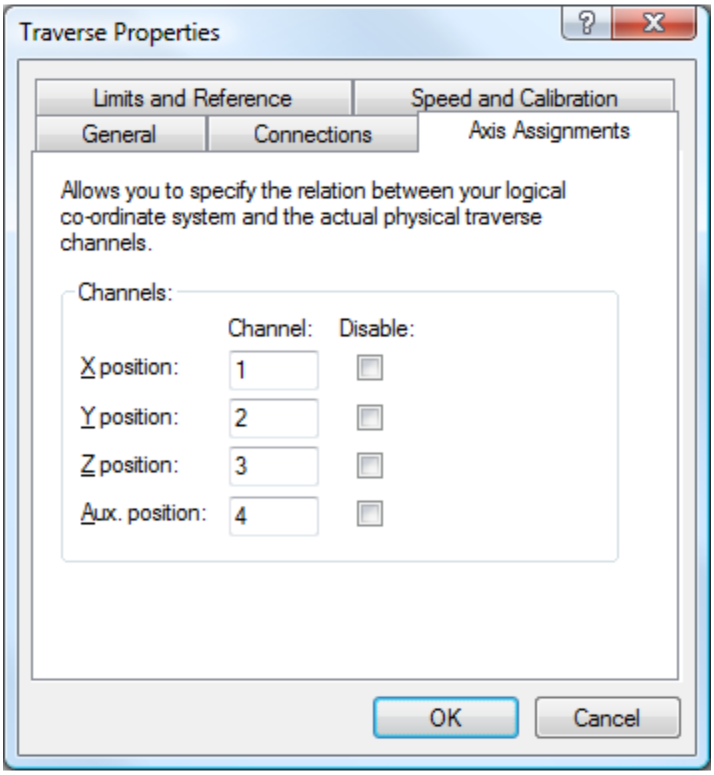


Select “Use extra controller” in Traverse Properties.

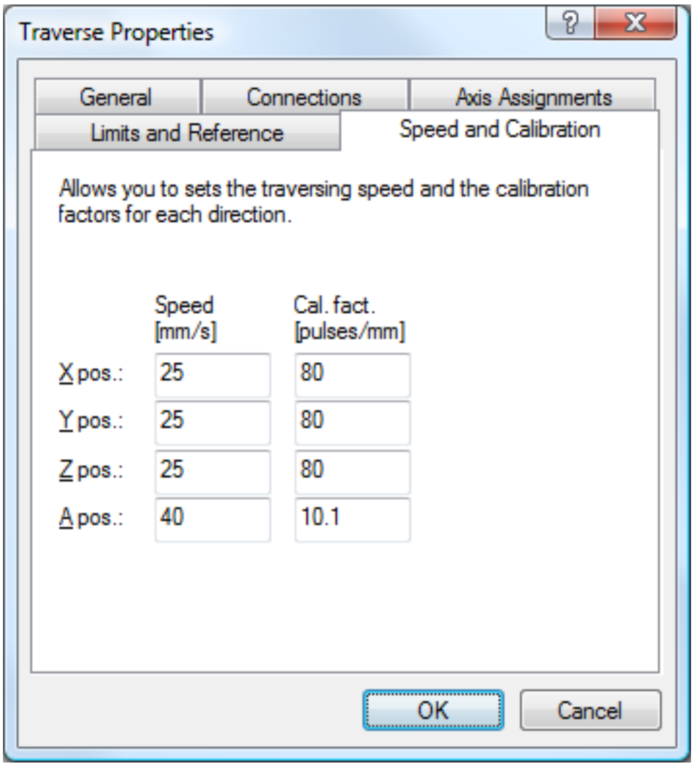


This requires an additional Comport for the Rotating Unit, as the Lightweight Controller only supports 3 axis.

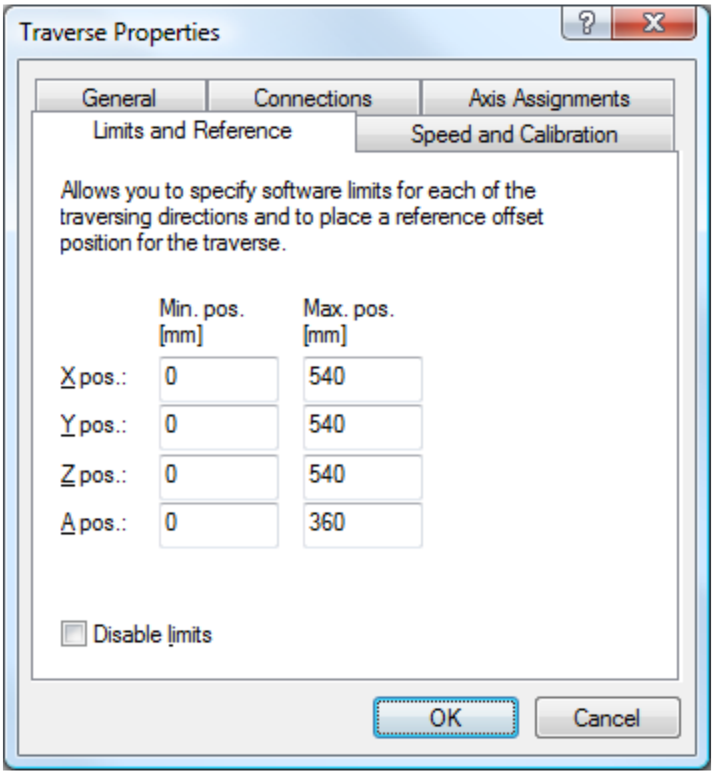
Axis Assignments



Speed and Calibration (valid for Dantec Lightweight Traverse and Probe Rotation Unit)



Limits and References



Important
If you change setup parameters specific to an A/D device or a Traverse, like for example base address or single ended/differential, it will influence the setup of earlier projects using that device. The reason is that these parameters are written into the device driver belonging to the board and shared by all projects. If you return to an old project after having changed board specific parameters remember to re-establish the original setup.

6.5.3 Save Preferences

If you want a window to maintain the toolbar selection, you have made with the Show/Hide option, then choose the Save preferences from the Options menu, before you close the window. Next time the window opens it will be with the previously made selection.

6.5.4 Selecting Current Probe

When you are going to run Online analysis, velocity calibrations or directional calibrations, you must select the wanted probe first:

1. Choose Select current probe from the Configuration menu.
   Select Current Probe dialog box opens with a list of probes defined in the hardware configuration.
2. Select the wanted probe.
3. Choose OK.
   The dialog box disappears

The probe name is shown in the probe drop down list in the main toolbar.

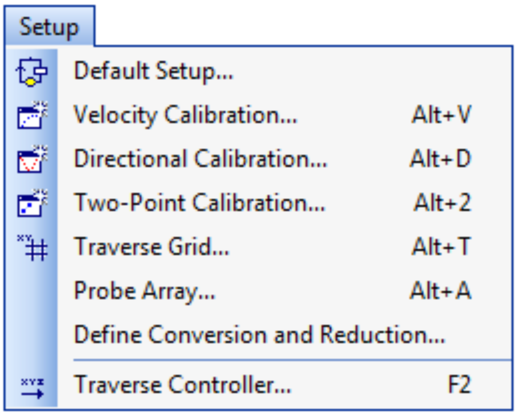
6.5.5 Selecting Temperature Probe

If you want to carry out an analytical temperature correction of the velocity data, you must define a temperature probe to provide the necessary temperature information.

1. Choose Select temp. probe from the Configuration menu.
   Select Probes dialog box opens with a list of possible temperature probes.
2. Select Enable in the Temperature probe field.
3. Select the probe that you want to use. It may either be the assigned temperature or a Local variable.
4. Choose OK.
   The dialog box disappears.

See See "Sample Project III Mapping of a Velocity Profile with Probe Traversing" on page 31 for more details.

6.6 Setting Up the System



6.6.1 Definitions

A complete Setup includes the following:

- Default Setup: Defines CTA reference temperature and overheat ratio. Also defines probe traverse, data acquisition and data reduction schemes.
- Velocity Calibration: Defines the velocity transfer function of a probe through calibration in many points.
- Two-Point calibration: Defines the velocity transfer function of a probe through calibration in two points.
- Traverse Grid: Defines traverse positions.
- Probe Array: Defines the geometry of a probe array.
- Data Conversion/Reduction: Defines how raw data are converted (linearised) and reduced to statistical quantities.

In addition the Setup menu includes:

- Traverse control: Gives access to move the Traverse equipment.

6.6.2 Define Default Setup

Definitions

With the Default Setup you can carry out measurements followed by data reductions.

The Default Setup contains 5 fixed processes that are carried out, when you run the Default setup. They are:

- Hardware setup
- Traverse
- Data Acquisition
- Group scheduling
- Data reduction

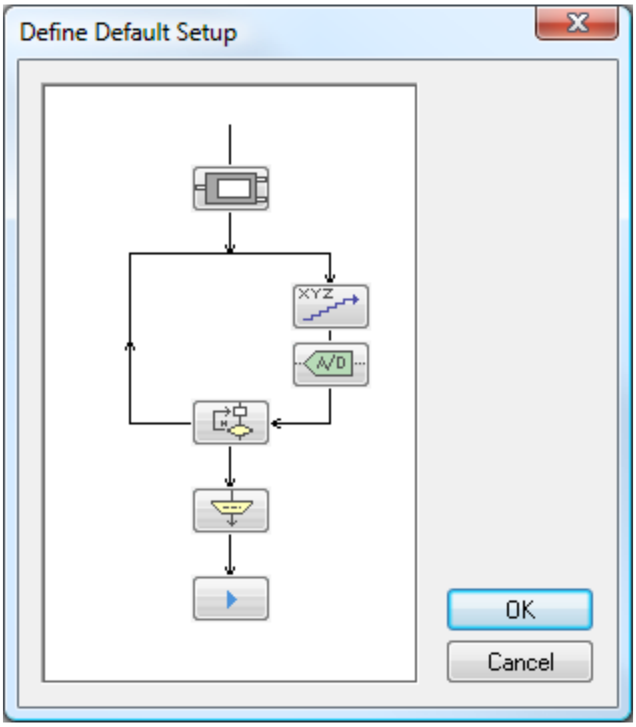
With the Default setup you can perform a hardware setup, traverse a probe, acquire data in each point, repeat the traverse and acquisition in a loop defined by the group scheduling and reduce the acquired data afterwards.

Each process is defined by means default parameters. You can replace the configuration defaults from the Probe Library with you own choice of parameters or events and thus create your own user Default Setup. When you run Default setup, the data conversion/reduction are those marked with a star (*) in the Project manager.

Default Setup is not listed as an event of its own in the Project manager. Data acquired with a Default Setup, however, are saved as events as Raw data records.

To Open Default Setup

1. Choose the Default button in the main toolbar or choose Default Setup from the Project menu.



Default Setup dialog box opens with a map with the processes in the Default Setup group. It contains buttons for Hardware setup, Position input setup, A/D conversion setup, Group scheduling and Data reduction.

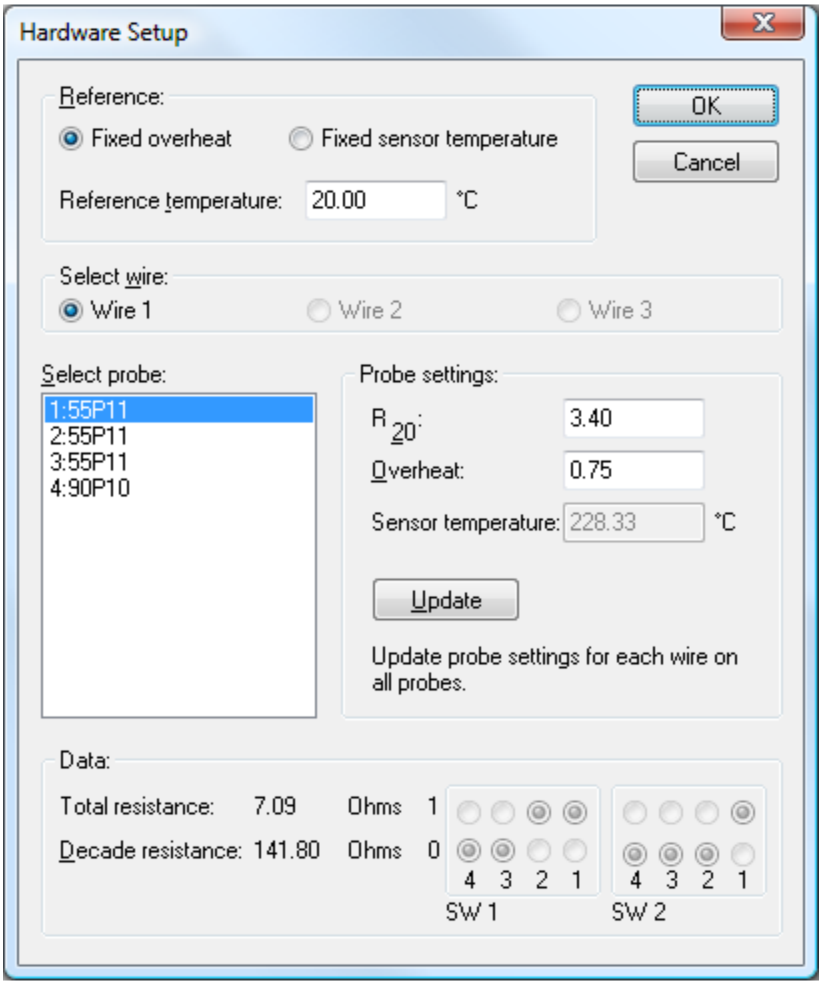
To Define CTA Reference Temperature and Overheat Ratio

The reference temperature T_{ref} is the temperature at which the probe cold resistance R_{sensor} is measured. T_{ref} has to be defined together with the overheat ratio a in order carry out accurate calibrations and temperature correction, if need be. Instead of measuring the cold directly you can use the information from the label on the probe container and set the decade in the CTA bridge accordingly:

R_{decade} = ((1+a)·R_{sensor}+R_{leads}+$R_{support}$+R_{cable})·Bridge ratio

In this case the reference temperature will be 20 °C.
First time you perform a Hardware setup, the overheat ratio and the support, cable and leads resistance are Configuration defaults taken from the Probe Library.

1. Click on the MiniCTA Box icon  in the Default Setup dialog box.
   Hardware setup dialog box opens.



2. Select the probe (and wire) number.
3. Insert the reference temperature at which the sensor cold resistance is measured.
4. Insert the sensor cold resistance in the R_{20} field.
5. You can now choose fixed overheat ratio and adjust the decade switch settings accordingly, or you can choose fixed decade resistance and accept the resulting overheat ratio.
6. Fixed overheat ratio:
   Enter the wanted overheat ratio.
   Click on Update. The probe total resistance, decade resistance (bridge ratio 1:20) and wire operating temperature are now calculated and displayed. The dip-switch settings (SW 1 and SW 2) of the MiniCTA are data are displayed.

   Fixed decade resistance:
   Enter the wanted decade resistance.
   Click on Update. The overheat ratio and the wire operating temperature are now calculated and displayed.

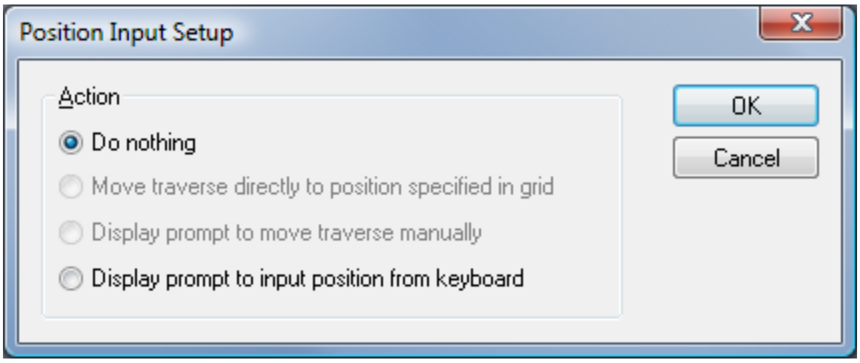
7. Continue with next probe.
8. Choose OK. The dialog box disappears and you are back in the Default setup dialog.

Note
Reference temperature is only needed for CTA probes.

Position Input Setup

The Position Input Setup defines, how the probe is moved, before the next sequence is executed.

1. Choose the Position Input button  in the sequence map.
   Position Input dialog box opens.
   Select one of the following options by means of the check boxes:



Do Nothing

The probe will stay fixed, and data acquisition will be performed immediately after start of the group iteration.

Move traverse directly to position specified in grid

The probe is moved automatically in accordance with the selected Traverse grid.

Display prompt to move traverse manually

The grid position is shown in a prompt. You must then move the probe manually and respond to the prompt.

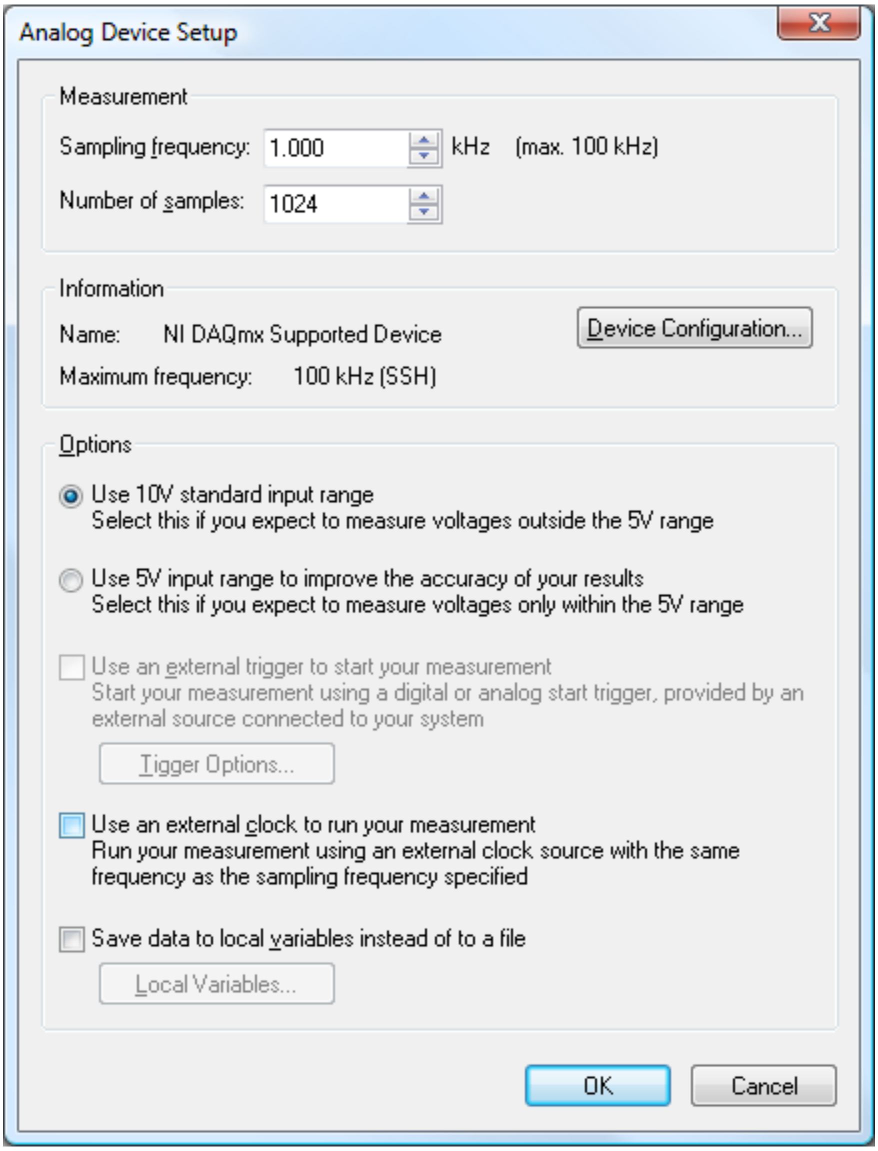
Display prompt to input position from keyboard

This option is always available. Before data acquisition is carried out you will be prompted to enter a probe position.

Data Acquisition (A/D Device Setup)

Here you define the A/D device setup to be used for data acquisition in a loop iteration. You can accept or overwrite the defaults for sampling frequency, number of samples, gain, clock and trigger.

1. Choose the A/D button  in the sequence map.
   A/D Setup dialog box opens with the last defined settings.



Sampling frequency and Number of samples

2. Select the sampling frequency by means of the up/down arrows in the Sampling frequency select box.
3. Select the number of samples by means of the up/down arrows in the Number of samples select box.

Note
The sampling frequency is shared by all active channels on the A/D device. This means that if you have e.g. a triple wire probe, each wire will be sampled with 1/3 of the selected frequency.

You can only define sampling frequency, when the sampling is triggered by the software (internal trigger).

Gain

Select the gain by means of the high 5 V or normal 10 V radio buttons. It is recommended to leave the gain on its default value.

Data storage

In Default Setup the acquired data will always be written to file and noted as raw data records in the project manager..

Trigger

Select the trigger to be used for starting the acquisition by choosing one of the following:

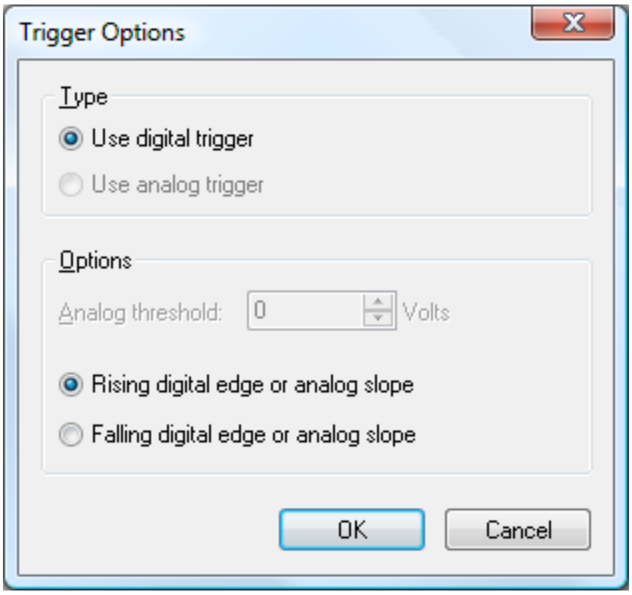
Internal trigger

Select the Internal radio button, if you want the acquisition to start as defined by the software in the Group scheduling.

External trigger

Select the External radio button, if you want to use an external voltage signal to start the data acquisition. The Options button is enabled.

1. Choose the Options button.
   External Trigger Options dialog box opens.



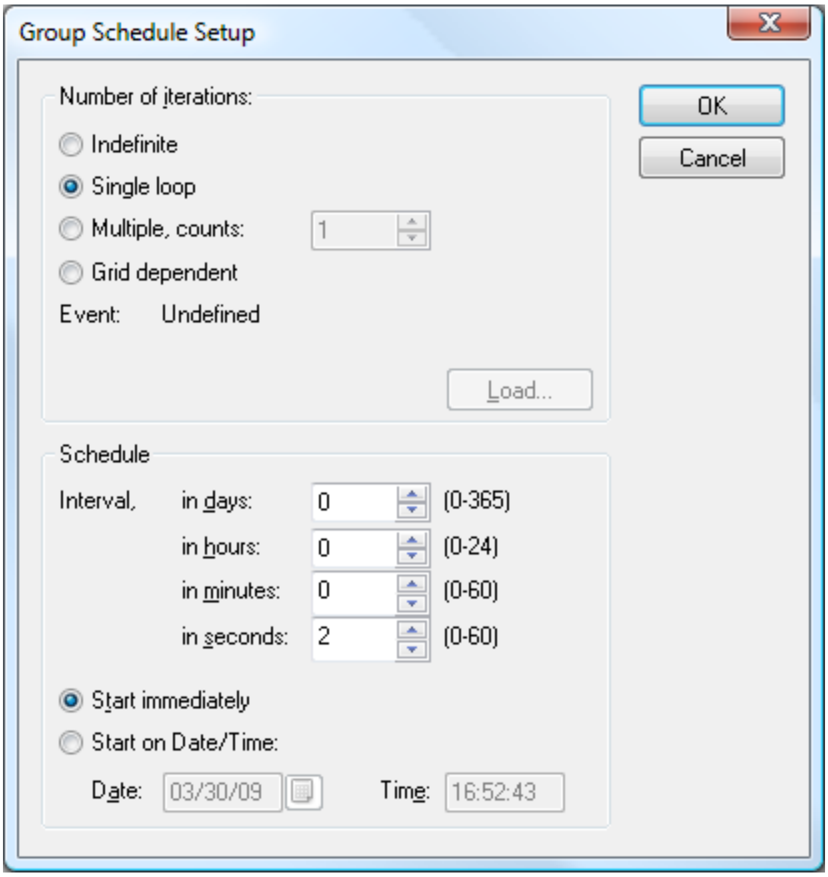
2. Select the trigger level in the Trigger level select box.

Group Scheduling

In Group scheduling you can define the number of iterations, i.e. the number of times the processes in the group are repeated, the interval between iterations, and when the group iteration starts.
You can also make the iteration dependent on a Traverse grid, so that a traverse position is attached to each iteration.

It is used in both Default Setup and in Experiment Setup.

1. Choose the Group schedule button in the sequence map.
   Group Schedule Setup dialog box opens.



You can select the following options:

Number of iterations

Indefinite: The iteration is repeated infinitely.
Single loop: Only one iteration is carried out.
Multiple counts: Enter the number of iterations in the text field.

Grid dependent

The iterations are related to a traverse grid. Select the Traverse event by means of the Load button.

Interval

In days, In hours, In minutes, In seconds: The wanted interval is entered into the text fields.

Start

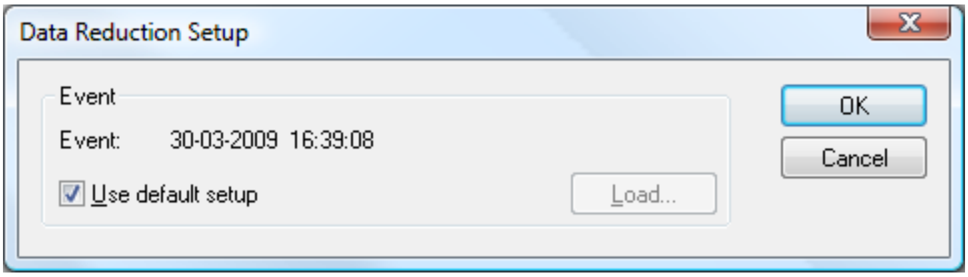
Immediately: The iteration starts as soon as the Hardware setup assigned to the group has been established.
Start on Date/time: Enter the wanted date/time in the text field. When the dialog box opens, the actual date/time is displayed.

2. Choose OK. The dialog box disappears.

Data Reduction

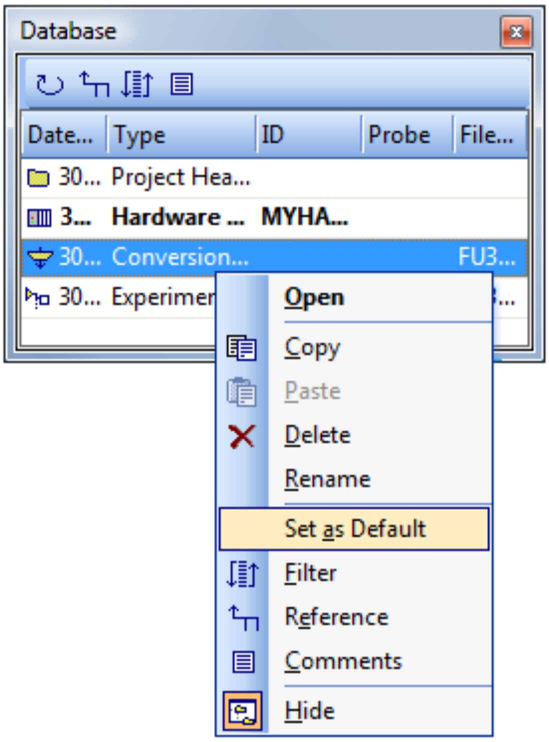
The Data reduction defines how the raw data are reduced into statistical quantities.

1. Choose the Data Reduction button  in the sequence map.
   Data Reduction dialog box opens, where Use default setup or Event can be selected.



2. Select the Use default setup.
   - or if you want to use your another setup:
   Unselect the Use default setup check box.
   A Load Event dialog box opens
3. Select the event.
4. Choose OK.
   The dialog box closes.
5. Choose OK.
   The dialog box closes. Data will now be converted and reduced in accordance with the Data Reduction selected.

To Change Default Setup



1. Select the wanted Setup event in the Project manager, click on it with the right mouse button and choose Set as default.

For details see See "Define Default Setup" on page 94.

To Leave the Default Setup

When default parameters for the processes selected have been redefined, the Default Setup is left by choosing OK. The dialog box closes, and the factory defaults are now overwritten with the new user defaults in the project database.

Note
The Default Setup is not saved as an event.

6.6.3 Calibration in General

Definitions

When you calibrate a probe, you create a dependency between the probe voltage and the velocity at a certain reference condition (temperature and pressure). The calibration software reads velocities and voltages and performs a curve fit through the calibration points. The fit forms basis for the transfer function used to convert probe voltages into velocities.

The calibration procedure in the MiniCTA Software is primarily designed for CTA probes, but you can establish transfer functions for other than CTA probes with the curve fit function in the calibration sheet.

The MiniCTA software allows you to perform calibration of one probe or more probes at a time and accepts probes with 1, 2 or 3 sensors.

Selecting Calibration Equipment

Two-Point calibrator

This represents the easiest and fastest way to calibrate a single standard wire probe. The Dantec Two-Point calibrator type 55H02 establishes two known velocities. The transfer function is then created on the basis of the probe voltages in these two points in combination with a “family” transfer function for the probe type in question.

Wind tunnel with velocity reference

You can use any laminar air flow equipped with a velocity reference as for example the 54T29 Velocity reference probe (only in connection with the 54N81 Multichannel CTA) or a Pitot tube connected to a micro manometer with or without electrical output. Or you can simply enter the velocity via the keyboard. This requires a suitable number of calibration points, e.g. 10 or more.
A wind tunnel also allows Multichannel calibration, where more probes are calibrated simultaneously.

Orientation of the Probe during Calibration

Place the probe with the X-axis of the probe coordinate system in the flow direction. The probe coordinate system is defined in See "Probe Coordinates" on page 153.

Selecting Probe

The probe to be calibrated is selected in the Probe list in the main toolbar. This must be done, before you start the calibration procedure.

6.6.4 Two-point Velocity Calibration

Probes belonging a family in the Two-point Library Calibrations can be calibrated on the basis of two points only. This is done by means of the Dantec Two-point Calibrator, which is a free jet device. It has two fixed velocity settings: approximately 1.5 m/s and 50 m/s. When the barometric pressure and the temperature in the stagnation chamber are known, the actual velocities are calculated in the MiniCTA software. The family transfer function is then modified, so that it matches the probe voltages acquired at the two velocities. The modified transfer function can then be saved as a calibration event and used for data conversion.

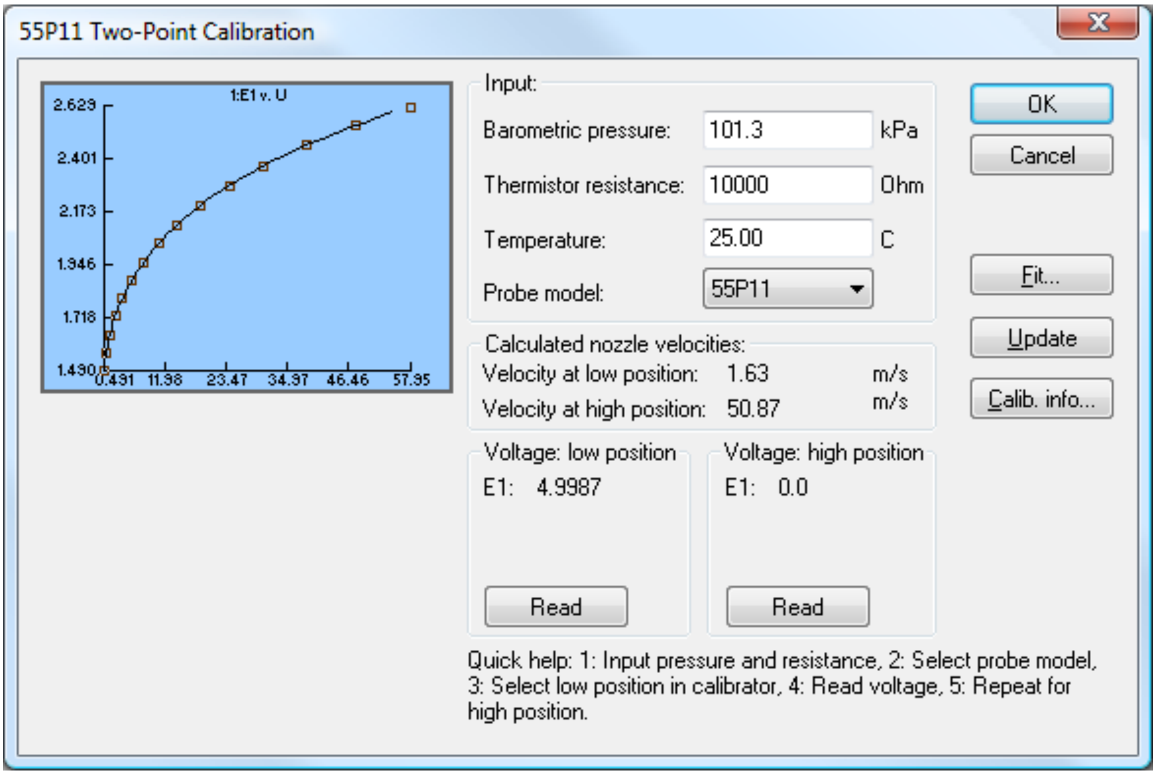
Installing the Two-Point Calibrator

Connect the calibrator to the air supply line, preferably with a proper oil separator and filter in front. Mount the probe with the prongs parallel to the jet axis with the wire flush with the upper surface of the nozzle. Set the handle to low. The stagnation chamber temperature can be measured via the thermistor permanently mounted in the chamber. It is accessible via the BNC connector marked “Temperature”.

To Run the Two-point Calibration 55H

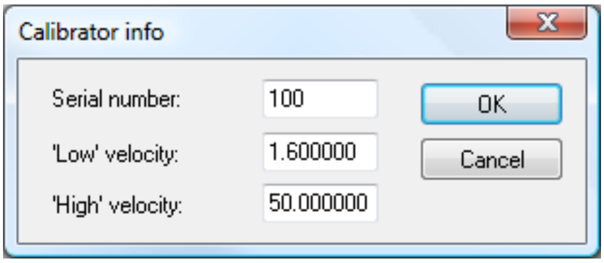
Running the two-point calibrator requires measurement of the air temperature in the stagnation chamber and of the barometric pressure. An ohm-meter is used to measure the resistance of the thermistor in the stagnation chamber, while a barometer is used to measure the ambient pressure, which is very close to the static pressure in the jet.

1. Choose Two-point calibration in the Setup menu.
2. The Two-point calibration dialog box opens.



If it is the first time that the calibrator is used with your version of MiniCTA software, you must enter the Calibrator constants now.

3. Click on the Calibrator button. The Calibrator info. dialog box opens.

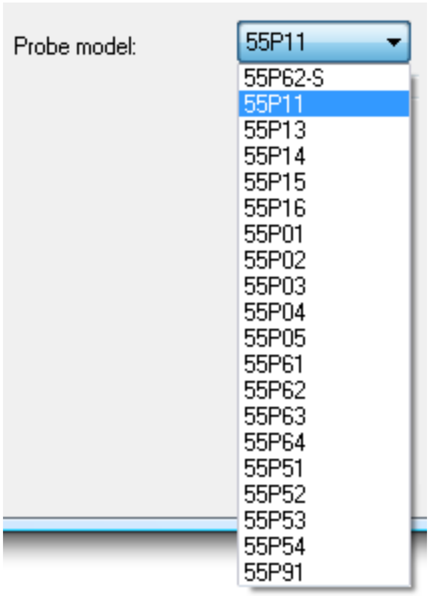


4. Enter the serial number and the values for the low and high velocity from the calibrator documentation.

Note
The calibrator info only needs to be entered once, as the constants will be saved as global variables in MiniCTA software and used for all future calibrations.

Enter probe model

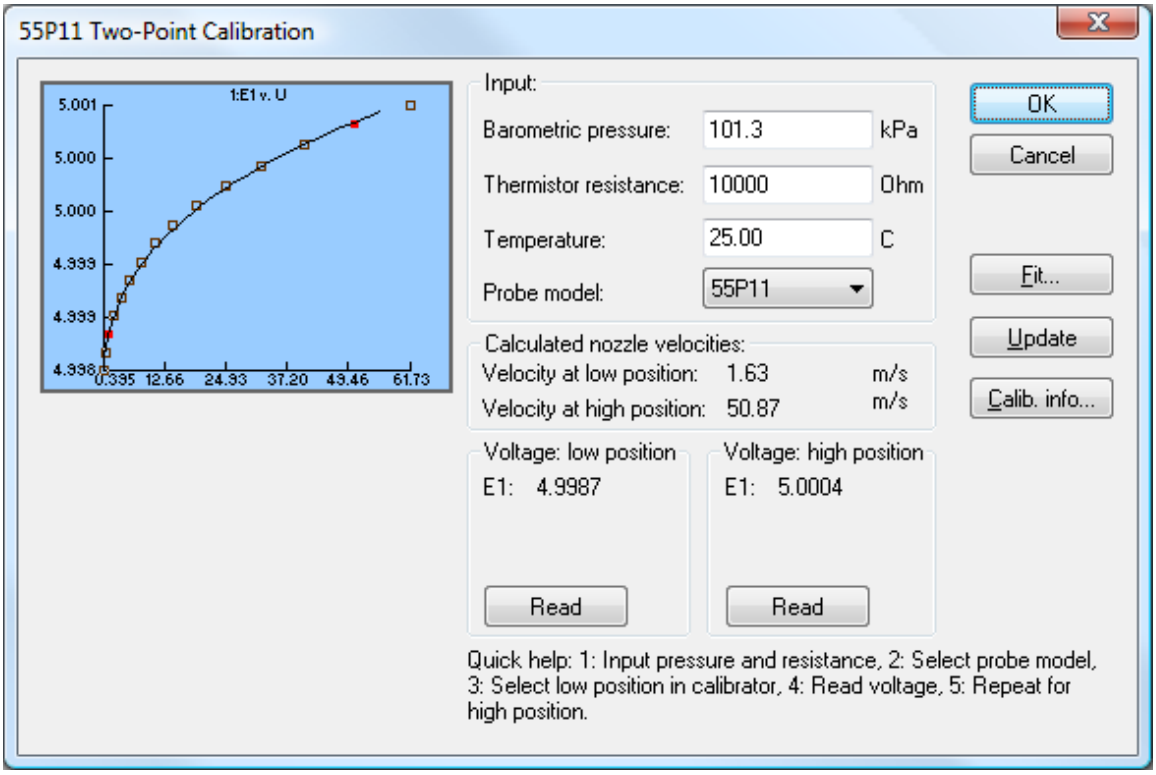
1. Select the probe model to be calibrated. The family calibration points from the Two-point library will now be used as basis for the transfer function.



Enter calibration points

1. Set the handle bar on the calibrator in position ‘LOW’.
2. Click on the Read button in the ‘Voltage, low” field.
3. You are now prompted to wait until the flow has stabilized.

4. Click OK and the probe voltage is acquired and displayed. At the same time the first modification of the family transfer function is performed based on the low point alone and displayed. Note that the 15 family calibration points from the Two-point library is also modified and displayed.
5. Set the bar handle bar in position ‘HIGH’.
6. Click on the Read button in the ‘Voltage, high” field.
7. You are now prompted to wait until the flow has stabilized.
8. Click OK and the probe voltage is acquired and displayed. The final transfer function based on both points is calculated and displayed. Note that the two measured points are indicated with red symbols. The remaining points are all modified family calibration points from the Two-point library.



Error Analysis and Curve Fitting Options

The curve fitting errors, which represents the difference between the
transfer function and the modified family calibration points, may now be analyzed using the Fit button. From here you can follow the outline in See "Curve Fit Setup" on page 111.

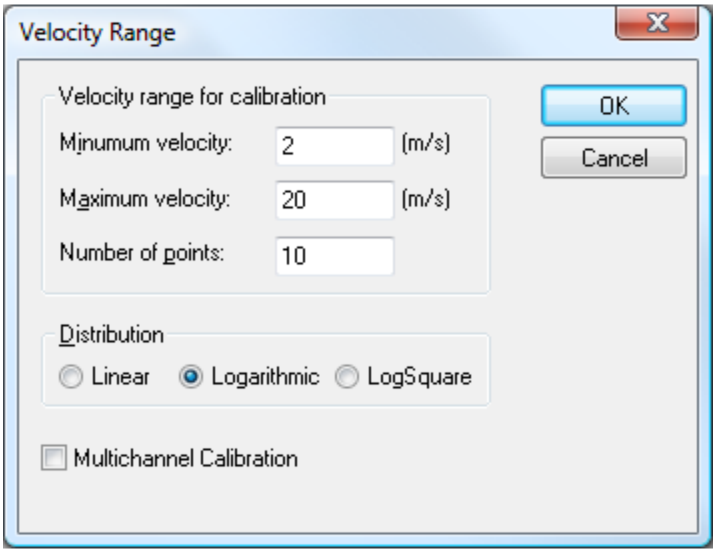
6.6.5 Velocity Calibration

1. Choose the Velocity Calibration command in the Setup menu or click on the Calibrate icon  in the main toolbar.

First thing to do is to enter a set of velocities into the Calibration worksheet. It is important to select both minimum and maximum velocity well outside the expected velocity to be measured in the flow, as the transfer function will not extrapolate if the calibration velocity is exceeded.

Note
X-array and Tri-axial probes should be calibrated down to approximately 10 % of the expected minimum

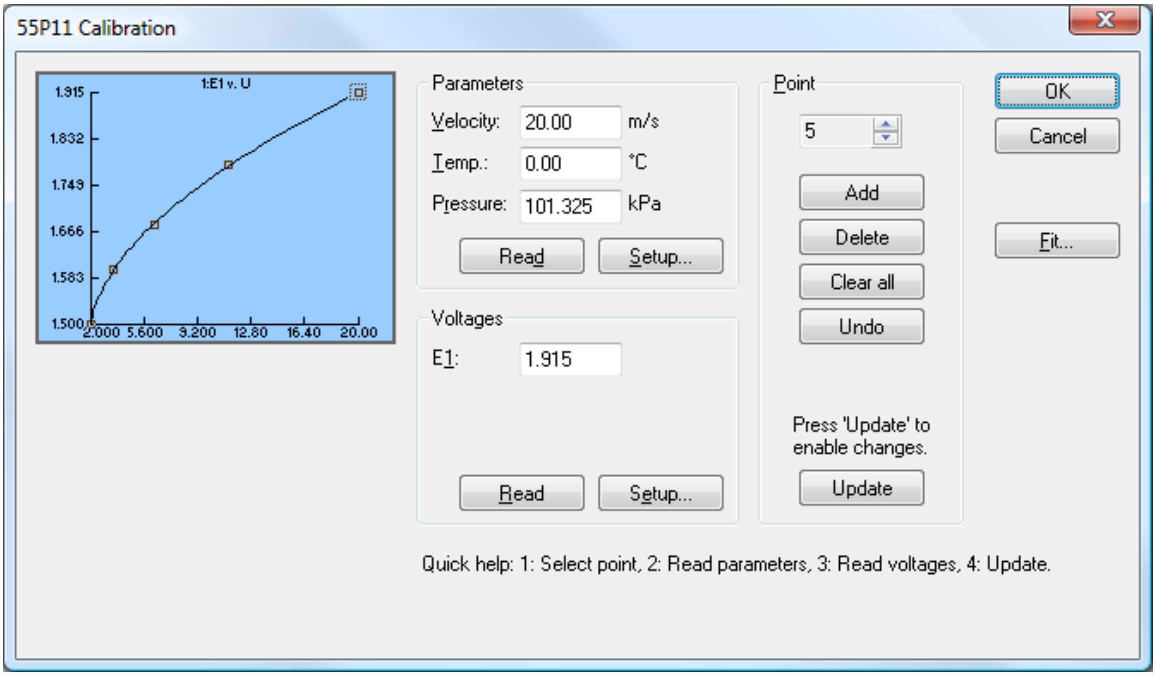
velocity and up to 50% above the expected maximum velocity in order to linearize the two extremes, where the flow attacks parallel with and perpendicular to the sensors, respectively.



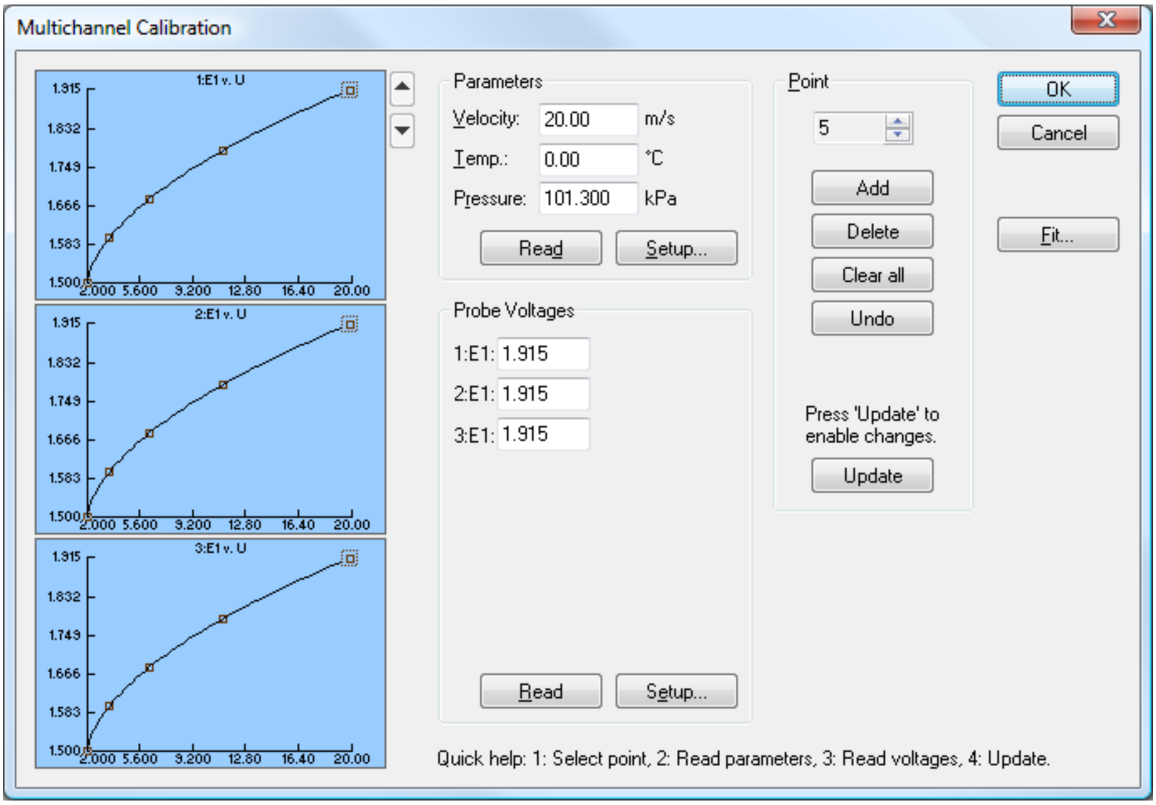
Velocity Range dialog box opens where you define velocity range and number of calibration points.

2. Select Linear or Log distribution in the Distribution field. The Log distribution is recommended, as is provides most points, where the transfer function varies the most.
3. Select Multichannel Calibration, if you want to calibrate more than one probe at a time.
4. Click OK and the Calibration dialog opens.

To Run the Calibration



Dialog box for single probe calibration.



Dialog box for Multichannel calibration (after completed calibration).

Parameters

Calibration parameters are velocity, ambient temperature and pressure.

1. Choose the Setup button in the Parameters field.
   Parameter Input Setup dialog box opens, where you can select the input sources for velocity and temperature:

Select velocity and temperature input:

- Keyboard input:

The parameter is updated from the keyboard.

- Reference probe, velocity:

Choose the Probes button and select probe from the Velocity reference library.

Reference, temperature:

The temperature is acquired from the Reference temperature probe selected during Configuration setup.

- A/D Channel:

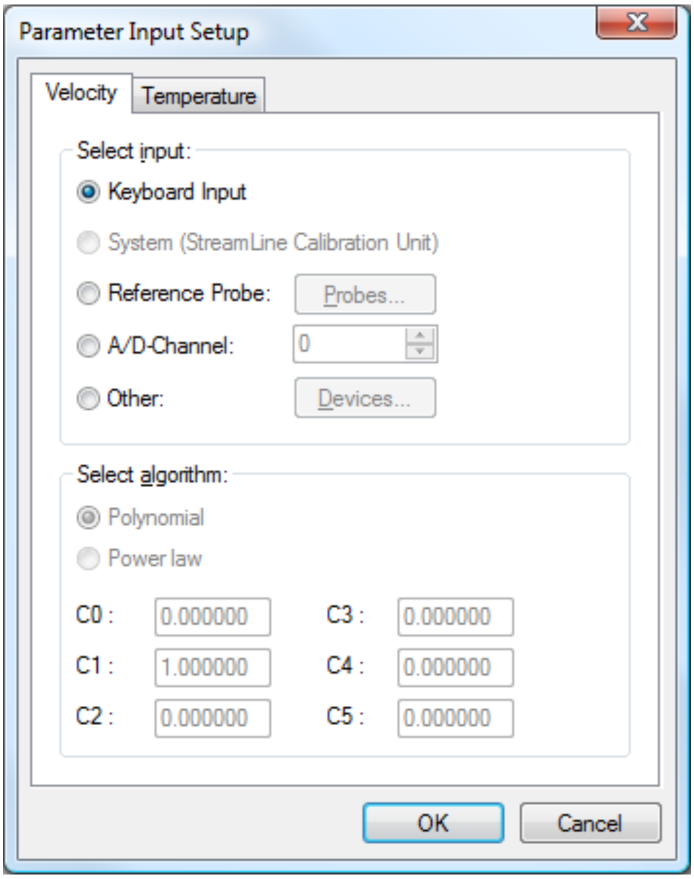
The parameter is acquired via the selected A/D channel.

- Other:

The velocity is acquired via a device defined by a user-defined driver in the External device driver library.
Select algorithm (only when A/D channel input is selected):
Defines the transfer function that converts the acquired voltage to m/s or °C.

- Polynomial or power law:

2. Enter the calibration constants C0 to C5 or A, B and n depending on the choice of algorithm.



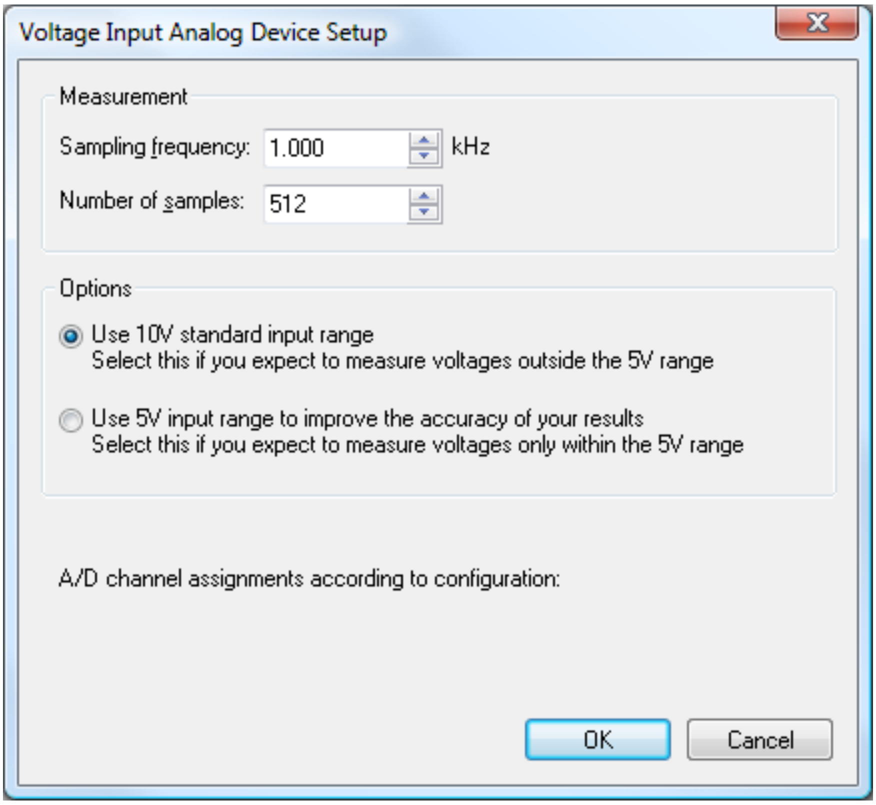
1. Choose OK.
   The Parameter Input Setup dialog box disappears, and you are now back in the Calibration dialog box.

Pressure input:
The barometric pressure input can only be done from keyboard.

Probe Voltages

This selection allows you to do the following Define the averaging time for probe signal by combining sample frequency and number of samples:

1. Choose the Setup button in the Voltages field.
   Voltage Setup Input dialog box opens.



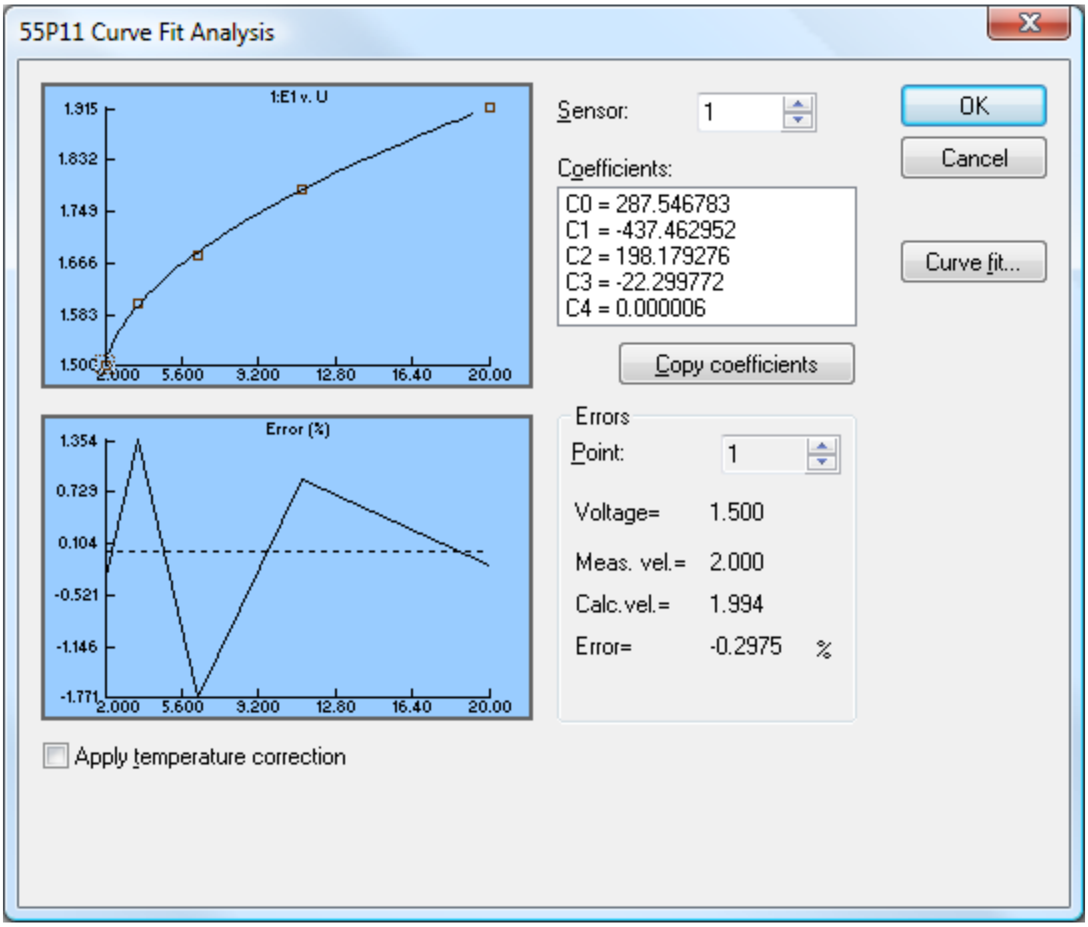
Acquiring Calibration Points

1. Make sure that you are in Point 1.
   Type in the velocity and temperature, if they are not acquired via the A/D device, and press the Read button in the Parameters field.
2. Press the Read button in the Voltages field.
3. Press the Update button. The values for velocity, temperature and voltage are now stored and written into the calibration data sheet. The point is also plotted into the graph.
4. Repeat 2 through 4 until all points are done.
5. Curve fitting

To Analyze the Curve Fit

When all calibration points have been acquired, you can see the result of the curve fit and perform an error analysis of the fit.

1. Choose Fit in the Probe Calibrate dialog box.
   Curve Fit Analysis dialog box opens.



It shows Sensor number, transfer function coefficients, curve fit errors (linearisation errors) and graphs with calibration curve fit and error distribution, respectively.

Linearization Errors

2. Select Sensor in Wire list box.
   Linearization results for this sensor will be shown.
3. Select point in Point list box.
   Voltage, Measured velocity, Calculated velocity and Error in the selected point are shown.

Temperature Correction

Temperature correction is indicated by a Selection box. The reference temperature from the Default setup is used. You can see the impact of the temperature correction on the calibration constants and error distribution by selecting and deselecting the correction.

Copying the Calibration Coefficients to Windows Clipboard

Choose the Copy coefficients command button and they will be written into the clipboard.

4. Choose OK when the curve fit is accepted.

To Select another Curve Fit

You may select another curve fit than the default polynomial fit:

1. Choose the Curve Fit button.

Curve Fit Setup dialog box opens, where you can choose:
Polynomial:
Power law:
Lookup table:
Select the one you want.

2. Choose OK.
   The dialog box disappears.

The new fit is carried out and the new coefficients and new error distribution replace the old ones in the Curve Fit Analysis dialog box.

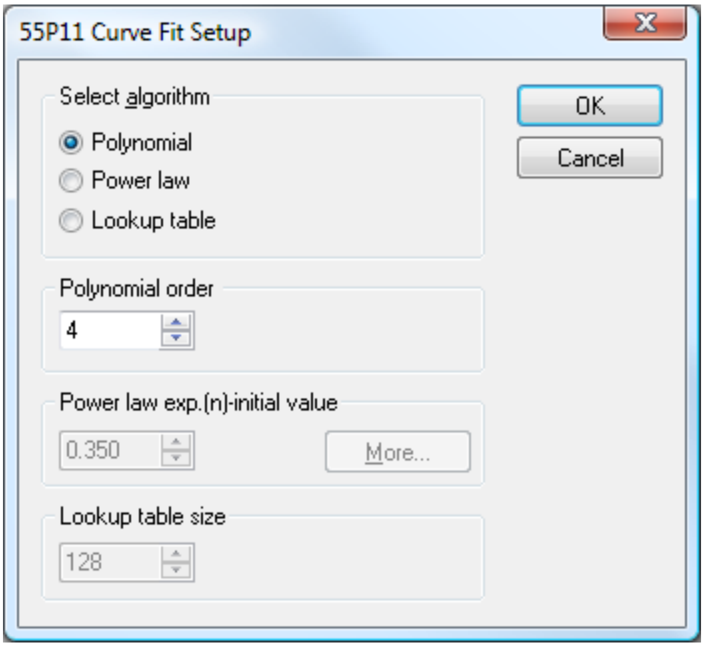
When you are satisfied with the coefficients for one sensor then select the next and continue from 2, until all sensor fits are analyzed.

3. Choose the OK button.
   The dialog box disappears, and you are back in the Calibration dialog box.

Curve Fit Setup

The default curve fit algorithm is a 4^{th} order polynomial. It can be changed to another order or another type.

1. Choose the Fit button in the Calibration or in the Curve Fit Errors dialog box.
   Curve Fit Setup dialog box opens. Here you can choose between following four types of curve fits:



Polynomial

1. Select Polynomial.
   The Polynomial order list box is enabled.
2. Select the wanted order in the Polynomial order list box.
   For an j-order polynomial the curve fit function now becomes:

$$Ei = C0 + C1 \cdot U + C2 \cdot U2 + C3 \cdot U3 + \dots + Cj \cdot Uj$$

where U is velocity, Ei is probe voltage from sensor i and C0 to Cj are the polynomial coefficients.

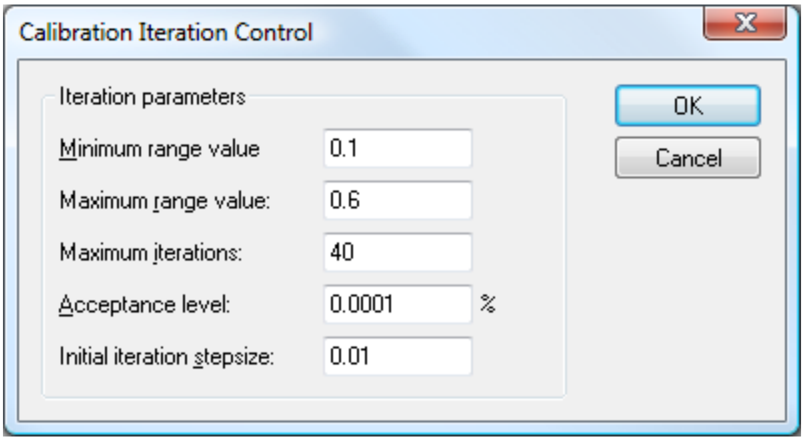
Power Law

1. Select Power law.
   The curve fit function now becomes:

$$Ei2 = A + B \cdot Un \text{(King's law)}$$

where Ei is the probe voltage from sensor i, A and B calibration constants, U velocity and n the selected power.
The More button is enabled.

2. Select start value of n, if you do not want to use the default value.
3. Choose the More button, if you want to redefine the variation range of n and the acceptance level of the iteration.



A Calibration Iteration Control dialog box opens with the following options:
Minimum range value.
Maximum range value.
Maximum iterations.
Acceptance level (defines the maximum accepted squared error).
Initial step size.

4. Choose OK.
   The dialog box closes and you are back in the Curve Fit Setup dialog.,

Lookup Table

1. Select Lookup table.
   Lookup list box is enabled.

Note
A Lookup table requires many more calibration points than polynomials or power law to provide same accuracy, as it is based on linear interpolation between the points.

2. Select size in the Lookup table size list box.

Note
The size of the lookup table should match the resolution of the A/D device, i.e. ideal size is (E_{max}-E_{min})/Resolution.

3. Choose OK.
   The dialog box closes and the Curve fit and errors are updated in accordance with the new selection.

Editing Calibration Points

When all points have been entered, you can edit by adding, updating or deleting points.

To Add a Point

A new point can be added either by means of the Calibration Equipment or simply by entering a new set of calibration values from the keyboard.

1. Select a point.
2. Enter and read the new set of velocity, temperature and voltage values.
3. Choose the Add button.

The new point is plotted in the E-U diagram, and the curve fit is updated.

To delete a Point

1. Select a point in the E-U diagram or by means of the up/down arrows in the Point field.
2. Choose the Delete button.

The point is removed from the E-U diagram.

To Update a Point

1. Select the point in the E-U diagram or by means of the up/down arrows in the Point field.
2. Enter and read the new set of velocity, temperature and voltage values.
3. Choose the Update button.
4. The point is re-plotted in the E-U diagram, and the curve fit is updated.

To Clear All

1. Choose the Clear all button in the Point field.
   You are prompted to accept the Delete all command.

2. Choose the OK button, if you are sure.
   The calibration points are removed from the E-U diagram, and the values in the Parameter and Voltages fields disappear.

Calibration of Other Probes

External probes are transducers and related electronics whose output are acquired by the MiniCTA Software via the A/D device. To a start any customer's probe must as a minimum have a library calibration, which is a simple 1:1 transformation of voltage to physical unit. (C1=1, all other C’s =0 in Library Coefficients)

The calibration is a best curve fit through a set of calibration points filled into the cells of the Calibration data sheet from the keyboard. The voltage signal from the External probe can also be acquired directly, if it is possible to expose the probe to the full range of calibration conditions.

Calibration

1. Select the probe in the Probe list in the top toolbar.
2. Open the Probe Calibration Window and fill in the calibration data in the calibration parameter and probe voltage columns from the keyboard. The calibration parameter can be edited as a data series as described in the section: New Calibration, Definition of calibration velocities.

When all calibration points are filled in, the calibration process is in fact finished. The curve fit is done, when you select the curve fit button  in the Tools toolbar.

Editing points, Error analysis, Curve fit optimization and Graphical presentation

Follow the outlines in section: New Calibration.

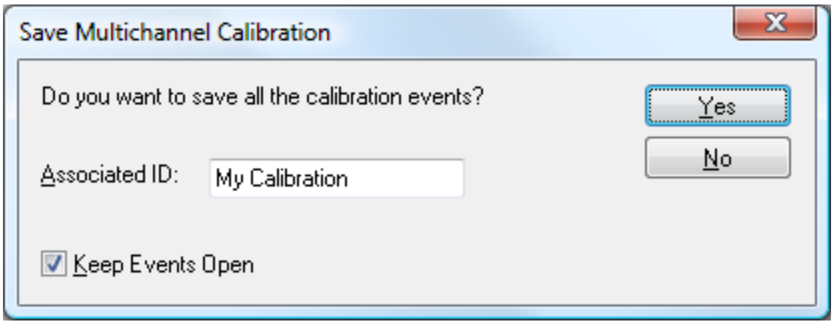
Leaving the calibration

1. Choose OK.
   The dialog box disappears.
2. Leave the Calibration window as described in the section: New Calibration and save the recalibration as its own event.

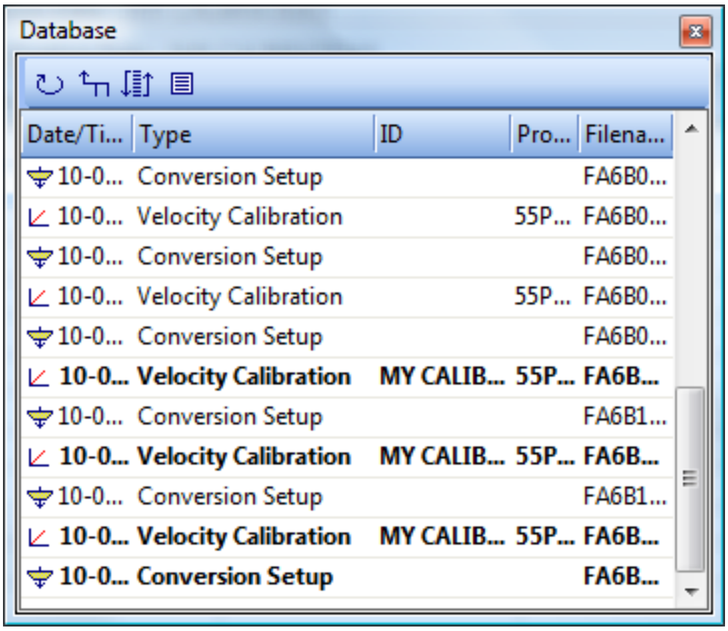
To Leave the Calibration and Save the Calibration Event

1. Click in the Control-menu box of the Calibration Window and select Close or double click in the Control-menu box.
   - or -
   Choose Close from the Window menu.
   Save Event dialog box opens.
2. Type in the Associated Identification (event name)
3. Choose Yes.
   The Save Event dialog box closes. The event is time stamped and added to the event list in the Project Manager.

Before the event is saved you are prompted if you want to make the Calibration event default. It is recommended to answer Yes to this.



Saving Multichannel Calibration



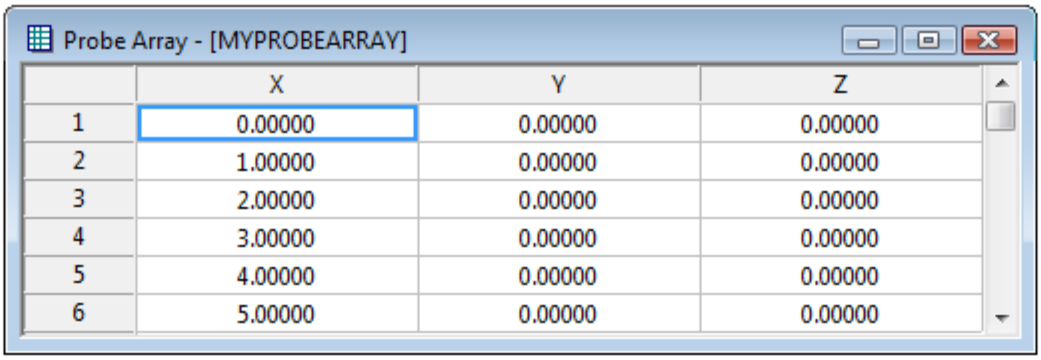
Select “Keep Events Open”, if you want the calibration data sheets open, after the calibration events are saved.

Note

A Conversion Setup is written into the Project Manager after each Velocity Calibration. The last Conversion Setup one is automatically made default and will be used in raw data conversion and reduction. Is is advised to delete all the non-default conversions.

6.6.6 Probe Array

The Probe Array defines the positions (X,Y,Z) of the individual probes in the Traverse system coordinate system. It supports the traverse of probe arrays.



Probe Array data sheet opens with a default array, where all configured probes are aligned along the X-axis at 1-mm intervals.

2. Type in the geometry of the actual array.
3. Close and save the Probe Array.

6.6.7 Traverse Grid

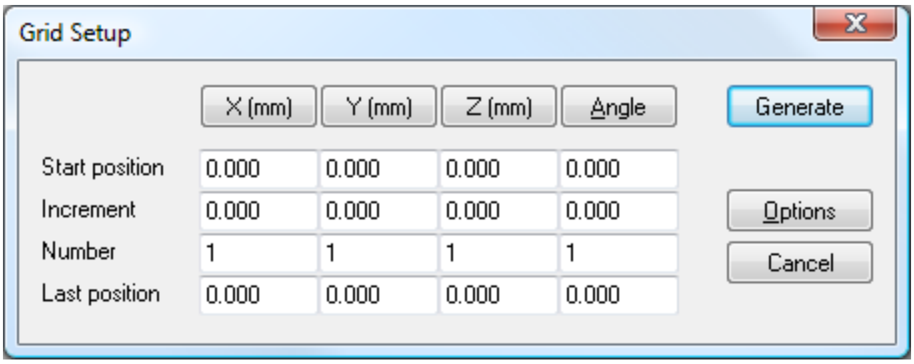
A Traverse Grid is needed in order to traverse the probe. Note that you do not need to have an automatic Traversing System in order to create and use a Traverse Grid.

To Generate the Grid

4. Choose the Traverse Grid command from the Setup menu or click on the Traverse button  in the Project toolbar.

Traverse data sheet opens with Grid Setup dialog box on top of it, where X,Y,Z - positions and probe rotation angle can be defined..

5. Type in Start position, Increment and Number of grid points on each axes. If the probe has to be rotated type in the Angles.



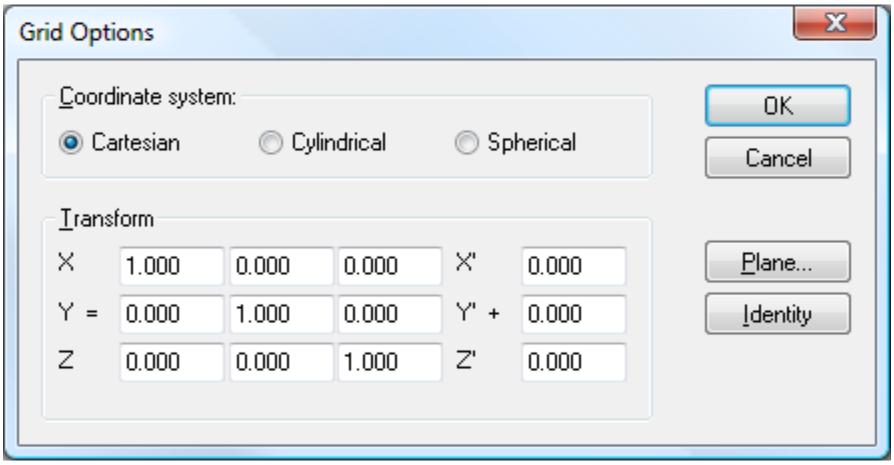
Traverse Grid to be used with a Probe Array

Start position:
Equal to the position of probe no. 1 in the array.
Increment:
Equal to the position of the last probe in the array plus the distance between the probes in the array.

To set Grid Options

Grid coordinate system in relation to laboratory system

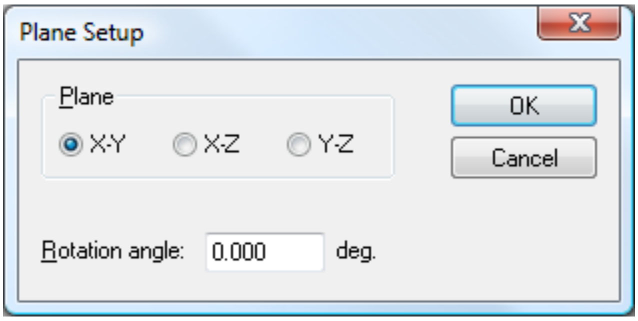
1. Choose the Options button in Grid Setup dialog box.
   Grid Options dialog box opens.



2. Select proper system: Cartesian, Cylindrical or Spherical by means of the radio buttons in the Coordinate system frame.
3. Type in the Coordinate transformation matrix and the grid system Origo offset in the Transform frame.
4. Choose the Identity button if Grid axis are parallel with Laboratory system axis.
   The matrix is filled in accordingly.

Or, if the traverse grid is tilted with respect to the laboratory coordinate system:

1. Choose the Plane button.
   Plane Setup dialog box opens.



2. Select the actual plane and type in the rotation angle with respect to the corresponding plane in the Laboratory system.
3. Choose OK.
   The dialog box closes and the transformation matrix is automatically updated.

4. Choose the Generate button.
   The dialog box closes, and the cells in the Traverse window is filled in with the positions defined in the Grid Setup generator.

5. Close the Traverse Window by double clicking on the Menu Control box.
   Save Event dialog box opens, where you can type in an Event Identification. The Grid is now saved as a Traverse Event in the Project database.

6.6.8 Define Data Conversion/Reduction Setup

Definitions

Data conversion and reduction is the process of linearizing data and reducing them into statistical quantities. In this way large number of samples stored during a data acquisition are reduced to a limited number of statistical quantities, like mean values or standard deviations. The data may be either raw data or data converted into physical units referring to a specific coordinate system. The Data conversion/reduction setup also contains information on how the conversion is performed prior to reduction.

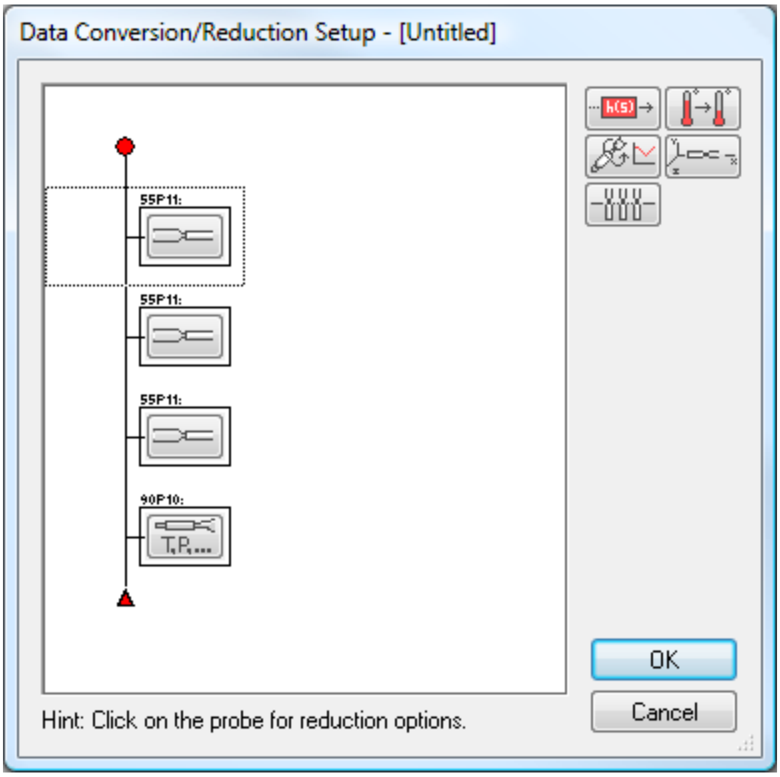
A Data conversion/reduction setup is an event and can be assigned to both Default setup and to a group in an experiment.

Note
The Data conversion/reduction setup only allows you to reduce the raw data into first, second and third order moments or correlations between one, two or three probe signals. For further reduction into e.g. spectra etc. you must use the Extended Processing from the Run menu.

Note
Only raw and reduced data are stored in files. You will have no files with converted data alone. If you want to see complete set of raw data in converted form, you must load the raw data record and choose the wanted conversion in the Option. The conversion will be based on the Data conversion/reduction event assigned to the setup used to acquire the data.

To Open the Data Reduction Setup Menu

1. Choose the Data conversion/reduction setup from the Project menu.
   Data conversion/reduction setup dialog box opens with a map of all probes defined in the project and a data conversion toolbox.

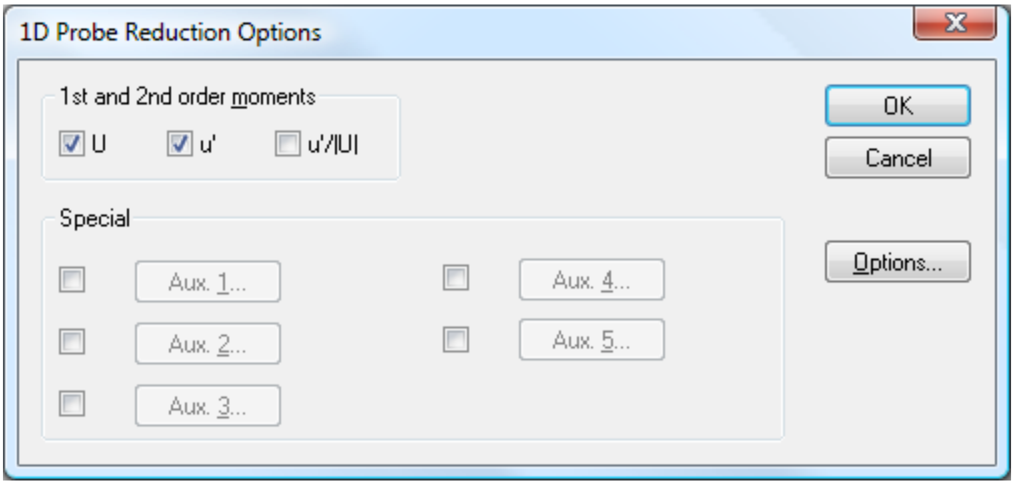


Now add the required conversion processes and define them. They are:

- Temperature correction.
- Linearization.
- Decomposition into wire/probe coordinate system.
- Transformation into laboratory coordinate system.
- Probe array.

To Define Data Reductions

1. Select the wanted probe by choosing the Probe button in the Reduction map.



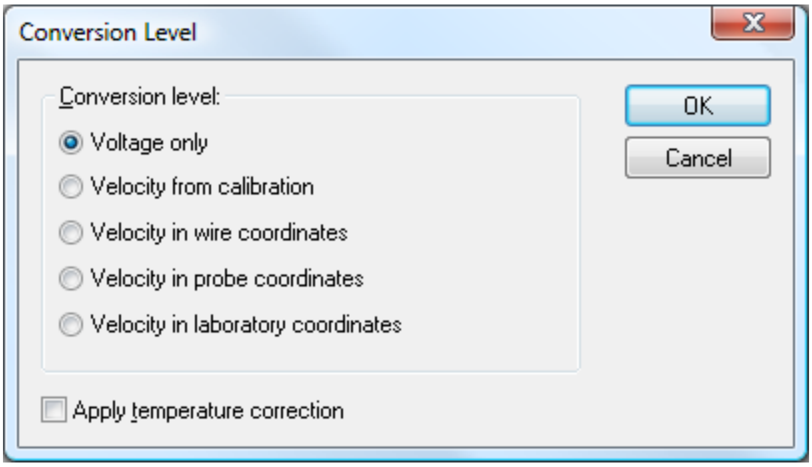
Probe Reduction Options dialog box opens, where you can select the following:

- Data conversion level before reduction.
- Type of data reduction.
- Auxiliary data and custom defined moments.

To Select Conversion Level (Options)

This option allows you to select how the raw data are converted before the data reduction is carried out.

1. Select the Options Button in the Probe conversion dialog box.
   A Conversion Level dialog box opens.



Here you can choose between the following:

Voltage only

The reduction will be done directly on the acquired raw voltages. If temperature compensation is selected, the voltages are corrected before reduction.

Velocity from calibration

The voltages are converted into velocities in accordance with the selected transfer function.

Velocity in Wire coordinate system

The voltages are converted into velocities in the wire oriented coordinate system using yaw-pitch corrections.

Velocity in Probe coordinate system

The voltages are converted into velocities in the probe oriented coordinate system using yaw-pitch corrections.

Velocity in Laboratory coordinate system

The voltages are transformed into velocities in a laboratory defined coordinate system.

Temperature compensation

If selected the probe voltages or velocities will be corrected for temperature variations in the flow.

Note
The voltages are only corrected, when a calibration event with polynomial curve fit or table look-up has

been selected. If power law fit was selected the calibration constants are corrected instead and the corrected velocity is loaded.

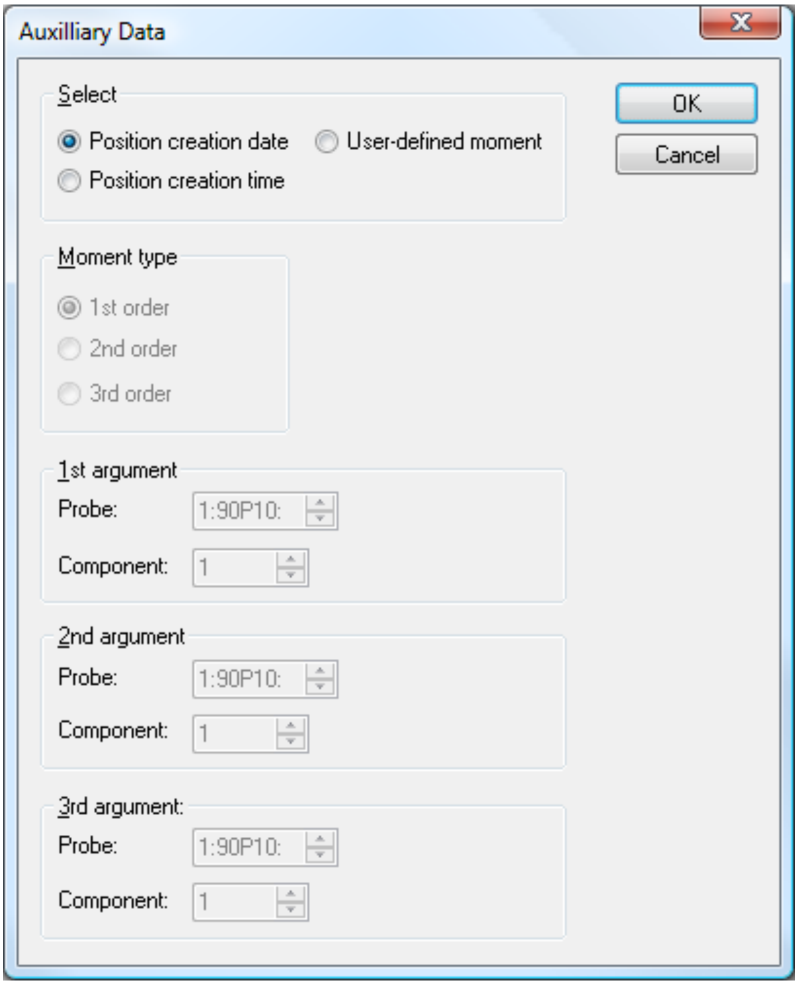
2. Choose Ok.
   The dialog box disappears.
3. Choose OK.
   The Probe Conversion Options dialog box disappears.

You are now ready to add and define the processes corresponding to the selected conversion.

To select moments

Select one or more of the U, u' or u'/U check boxes.
This defines the moments: mean, standard deviation (RMS) and turbulence intensity, that will be calculated from the raw data.

To select auxiliary data and user defined moments



Auxiliary data are data acquired in connection with raw data, like date
and time and assigned local variables. They can also be user defined moments between data in the selected file or between data from the file and data from other probes or sensors.

You can add up to 5 sets of auxiliary data to each data file by the check boxes connected to the Aux.1 to Aux.5 command buttons.

1. Select one or more of the Aux. check box.
   The Aux. command buttons are enabled.
2. Choose the Aux. button.
   Auxiliary data dialog box opens.

Here you can select one of the following data types:

Position creation date

This shows the date when the probe was moved to the position.

Position creation time

This shows the time, when the probe was moved to the position.

User-defined moment

Select the User-defined moments radio button.
The Moment type radio buttons are enabled.

1. Select the wanted Moment type radio button: 1st, 2nd or 3rd order.
   The arguments select lists are enabled.
2. Select the probe and the components that you want to use as arguments in the moment by means of the up/down arrows.

To Define the Transfer Function

The raw data is converted into velocities by means of a transfer function from a calibration event.

1. Choose the Transfer Function button  in the toolbox.
   The button is copied into the data reduction sequence of the selected probe.
2. Click on the button in the sequence map.
   Transfer Function dialog box opens.
3. Select the Use default check box.

   - or, if you want to use another calibration event instead:

   Unselect the Use defaults.
   Event Load button is enabled.
4. Choose the Load button.
   Select Event dialog box opens.
5. Select the wanted calibration event.
6. Choose OK.
   The dialog box disappears.
   The name of the event is displayed in the event box in the Transfer Function dialog box.
7. Choose OK.
   The Transfer function dialog box disappears.

To Define Temperature Compensation

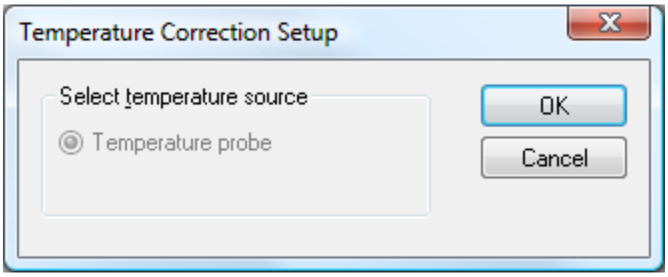
The Temperature compensation is a process that corrects the acquired raw voltages and/or the velocities for the influence of ambient temperature variations. For details on the temperature correction algorithms

see See "Algorithms" on page 143.

Temperature compensation requires that a temperature probe is selected.

As the output from the temperature probe is connected to an A/D input channel, each voltage/velocity sample is corrected with a simultaneously sampled temperature

1. Choose the Temp. comp. button  in the toolbox.
   The button is copied into the probe frame.
2. Click on the button in the probe frame.
   Temperature Correction Setup dialog box opens, which confirms your choice of temperature probe.

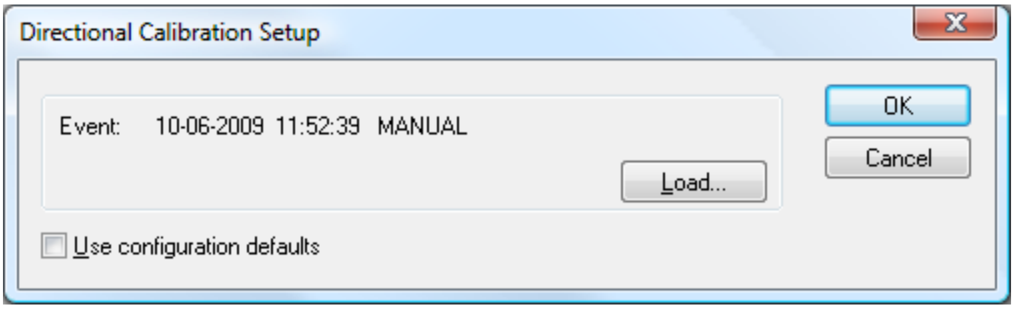


To Define Velocity Decomposition into Wire/Probe Coordinate System

This process decomposes the velocities calculated from the transfer functions into the wire or probe coordinate system using the sensor geometry from the Probe Library and the yaw and pitch corrections either from the Probe Library or from Directional Calibration Events.

CHANGED

1. Choose the Velocity decomposition Button  in the toolbox.
   The button is copied into the frame of the previously selected probe.
2. Click the Decomposition Button in the probe frame.



Yaw/Pitch Correction dialog box opens with the pitch- and yaw-coefficients from the Probe Library. You can enter other values from the keyboard.

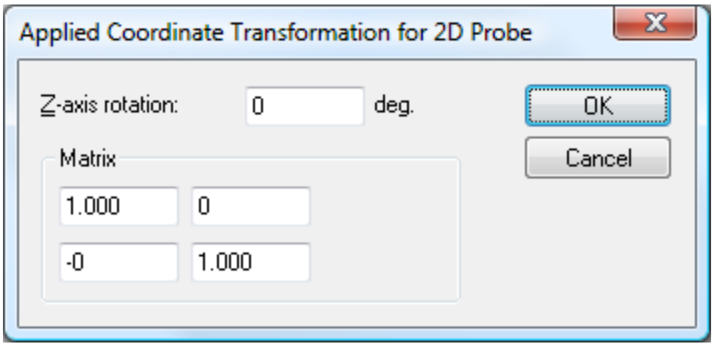
To Define Coordinate Transformation

Coordinate Transformation is the process of transforming the decomposed velocities from the probe coordinate system into a laboratory coordinate system.

1. Choose the Coordinate Transformation Button in the toolbox.
   The button is copied into the frame of the previously selected probe.
2. Click the Coordinate Transformation Button in the probe frame.
   Applied Coordinate Transformation dialog box opens.

2-D probe

1. Enter the z-axis rotation. This is the angle that the laboratory coordinate system is rotated on the z-axis with respect to the probe coordinate system.

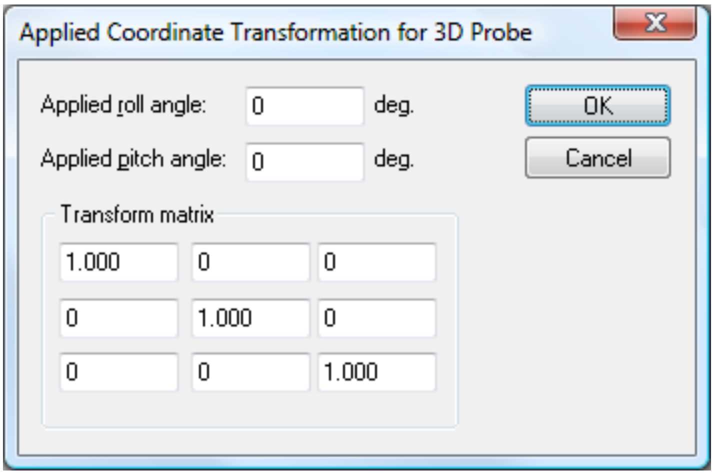


A transformation matrix is filled out with the directional cosines to be used during the transformation.

3-D probe

1. Enter the Applied roll angle. This is the angle that the laboratory coordinate system is rotated on the x-axis with respect to the probe coordinate system.

Two transformation matrices are filled out with the directional cosines to be used during the transformation.



2. Enter the Applied pitch angle. This is the angle that the laboratory coordinate system is rotated on the z-axis with respect to the probe coordinate system.
3. Choose OK. The Applied Coordinate Transformation dialog box closes.

To Add a Probe Array

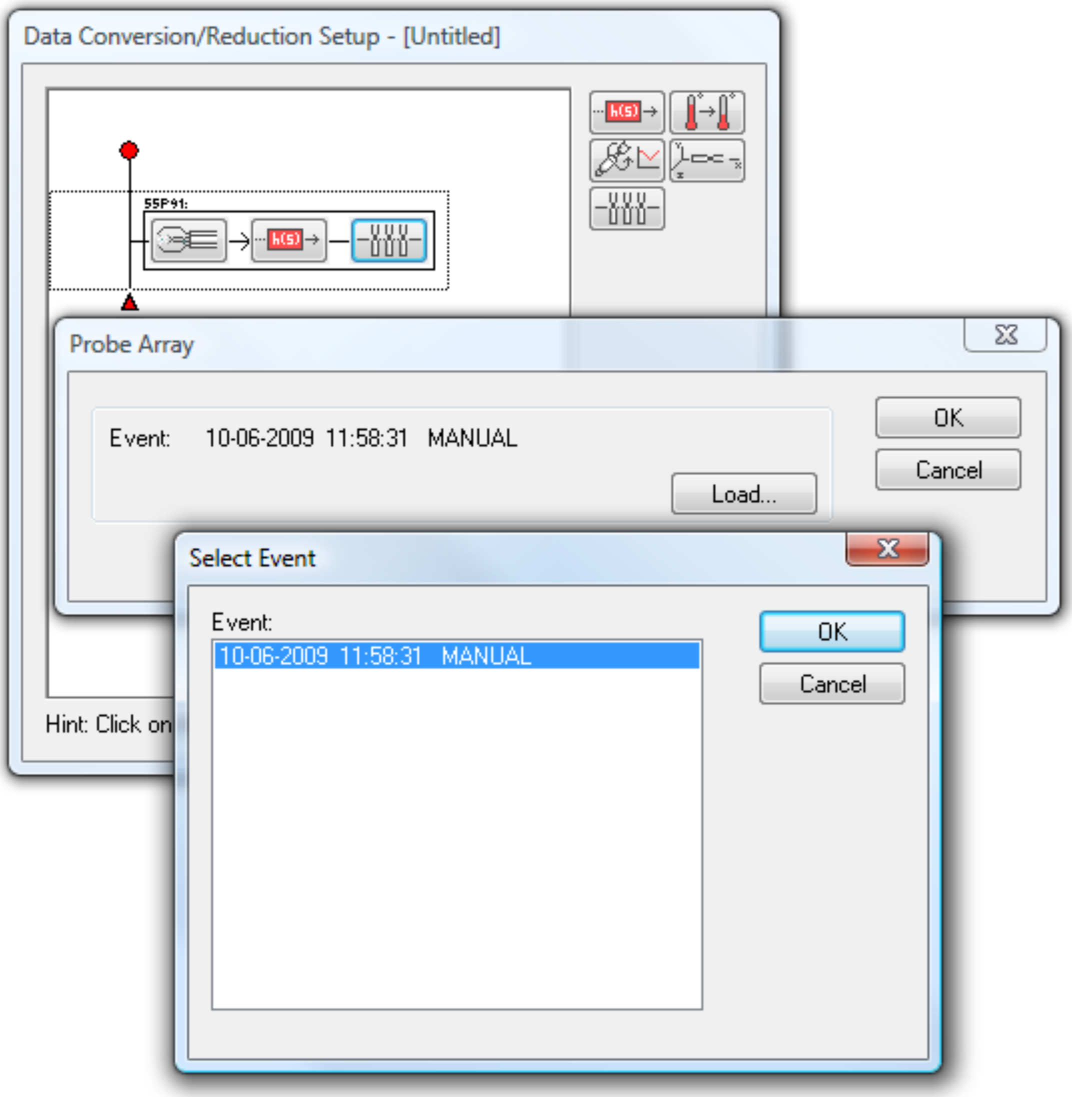
The Probe Array defines the positions of probes in an X,Y,Z-coordinate system oriented in parallel with the Traverse coordinate system.

1. Select a probe.
2. Choose the Probe Array Button  in the toolbox.

The button is added the frame of the selected probe.

3. Click the Probe Array button in the probe frame.

Probe Array dialog box opens, where you can load the array from the Select Event box.



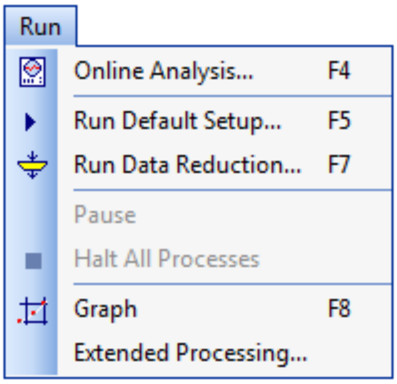
To Save the Data Reduction Setup

1. Choose OK.
   The Data Reduction Setup dialog box disappears.
   A Save Event dialog box opens.
2. Enter a name in the Associated ID text box, if you want to add a further identification to the event.
3. Choose the Yes command button.
   The dialog box closes.

The event is saved and added to the Project Manager.

6.7 Running the System

6.7.1 Definitions



The Run menu gives access to Online analysis, to run the Default setup and to perform Data reduction. It also allows you to introduce a Pause in the measurement sequence or to Halt it. Finally it can be used to make a Graph and to perform Extended processing of raw data.

6.7.2 Run Online Analysis

In Online Analysis you can analyze the output signals from one probe at a time. It is recommended that you run the Online analysis in a known flow after defining the Default Setup including Data conversion/Reduction. In this way you can check the entire measuring chain from probe to A/D device and you can get an impression of how the Data conversion works.

The the CTA-outputs can be displayed either as voltages, velocities from calibration or as velocities in wire- , probe- or laboratory coordinate systems.

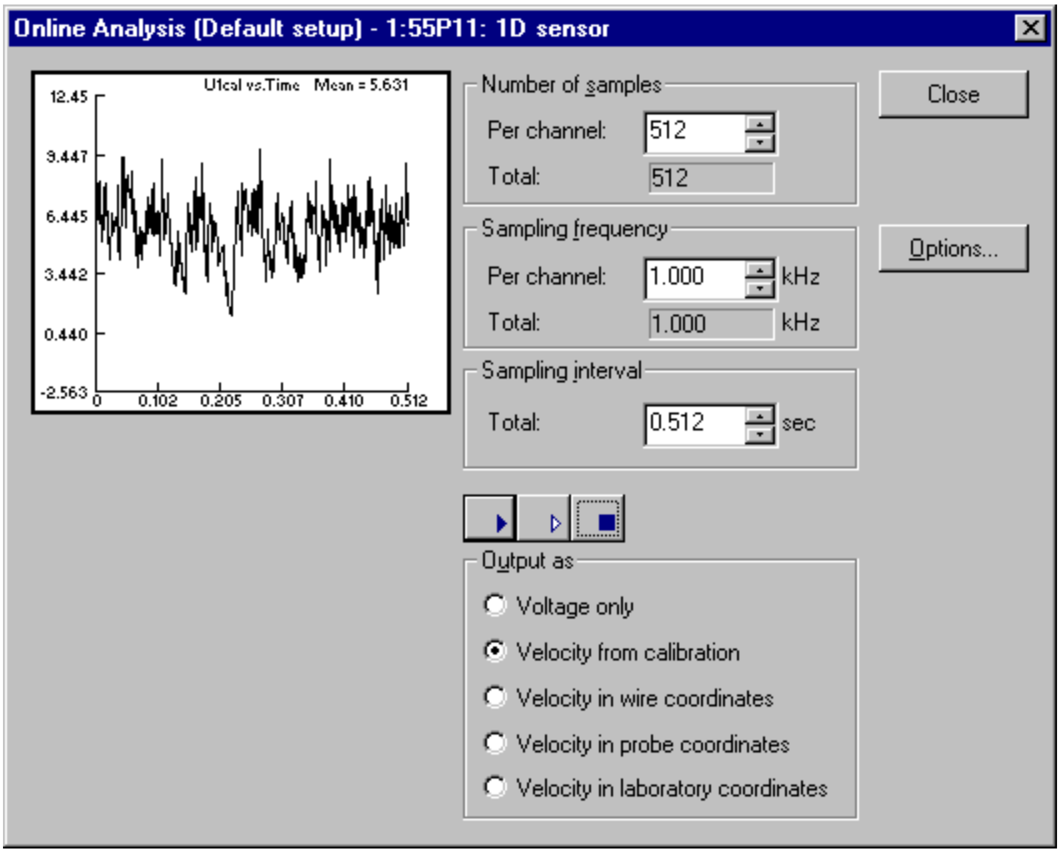
In Online analysis no data are saved. If you want to acquire and save data you must run either the Default setup.

To Open the Online Analysis Display

1.Select the wanted probe in the Probe selection box in the main toolbar.

2. Choose Online analysis in the Run menu. Or click on the Online icon in the main toolbar:

Online dialog box opens where you can select conversion level for the acquired data, define acquisition setup (sample frequency and numbers of samples) and start and stop the acquisition.



To Select Output Format (Conversion Level)

Voltage

Raw data in the form of voltages are displayed.

Velocity from Calibration

The velocity is calculated by inserting the raw voltages directly into the transfer function and displayed.

Velocity in Wire Coordinates

The calibration velocities are decomposed by means of the pitch -and yaw constants defined in the Default Data Conversion event and displayed.

Velocity in Probe Coordinates

The wire velocities are decomposed by means of the wire to probe angles defined in the Probe Library and displayed.

Velocity in Laboratory Coordinates

The probe velocities are transformed into the laboratory fixed coordinate system (as defined in a Grid setup assigned to the Default setup) and displayed. If you have not defined a laboratory coordinate system, the probe- and laboratory systems are by default identical.

To Define the Data Acquisition Setup

The default values for the number of samples, the sampling frequency and the sampling interval are shown. You can select other values:

Number of Samples

1. Select the wanted number per channel in the list box.

The corresponding total number of samples for the channels connected is shown.

Sampling Frequency

2. Select the wanted frequency per channel in the list box.

The corresponding total frequency for all channels is shown.

Interval

Sampling interval will now be displayed in the third list box.

The X-axis on the display is scaled to match sampling interval. You can select another interval if wanted. This will change the sampling frequency accordingly but leaves the number of samples unchanged.

To Run the Online Analysis

Online analysis can be performed continuously or in one shot.

Continuous Analysis

1.Click on the Run button. 

The Input Signal is now acquired, and the Output is displayed in the selected format. The display is updated for each new sampling.

2.Click on the Stop button. 

The sampling stops and the display is left with last sample frozen.

One Shot

1.Click on Step. 

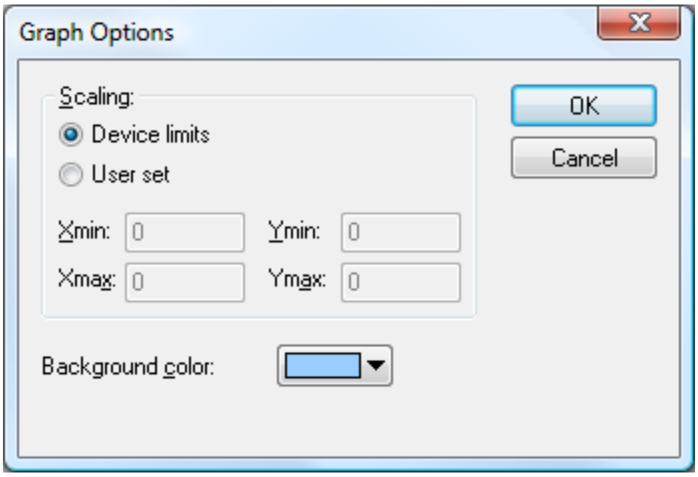
Only one sample is performed and the result is displayed.

Graphical Display Options

You can change scaling of the axis, and you can enlarge the graphical displays by clicking on the graph with the right mouse button. This opens a selection menu with 3 selections.

To Define axis (Options)

1. Click on Options.
   A Graph Options dialog box opens where you can select between Device limits and User set.



Device Limit

If selected, the X-axis is scaled according to the data acquisition setup. The Y-axis is scaled in volts or m/s in accordance with the Output selection. The maximum Y-value corresponds to the A/D channel input range (normally 0 - 10V) or to the maximum velocity from calibration.

User Set

If selected, you can type in minimum and maximum values for X and Y respectively in the X_{min}, X_{max}, Y_{min} and Y_{max} text fields.

Colors

Click on the Color command button, if you want to change the color of the graph. A Color dialog box opens where you can select a new background color.

2. Select OK and the graph is appears in its new form.

Auto Scaling

1. Click on Auto-scale.
   The X-axis is scaled according to the data acquisition setup. The Y-axis is scaled in volts or m/s in accordance with the Output selection. Y_{min} and Y_{max} corresponds to minimum and maximum value acquired.

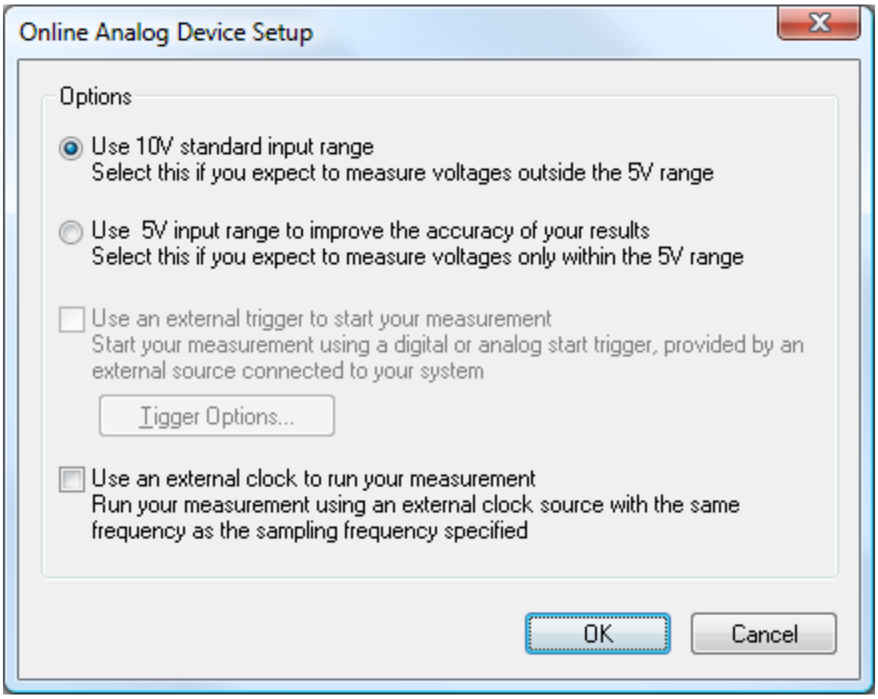
Zoom Out

1. Click on Zoom out.
   An Online zoom dialog box opens with an enlarged graph. From here you can copy the graph to clipboard and paste it into a document in another Windows application.

A/D Device Setup Options

You can temporarily change the basic setup (Range, Clock and Trigger) of the A/D device if you choose the Options command button.

1. Choose the Options button.
   A Online Options dialog box opens, where you can do the following selections:



Range

Select between 0 - 5 Volts and 0 - 10 Volts.

Clock

Select the clock to be used for timing of the data sampling:

1. Select the Internal radio button, if you want to use the clock on the A/D device.
   - or if you want to use an external clock -
2. Select the External radio button.

Trigger

Select the trigger to be used for starting the acquisition by choosing one of the following:

Internal Trigger

Select the Internal radio button, if you want the acquisition to start as defined by the software in the Group scheduling.

External Trigger

Select the External radio button, if you want to use an external voltage signal to start the data acquisition. The More button is enabled.

1. Choose the More button.
   External Trigger Options dialog box opens.
2. Select the trigger level by means of the up/down arrows in the Trigger level select box.

Note
These are the only options active in this dialog. The remaining selections (Trigger channel, Polarity and Sensitivity) are not implemented in the MiniCTA software translation drivers.

To Leave Online Analysis

1. Choose OK or double click on the control menu box.
   The dialog box closes. No settings or data will be saved.

6.7.3 Move Traverse Home

This command is used to move the Traverse (if connected and defined in the Configuration) to its starting position.

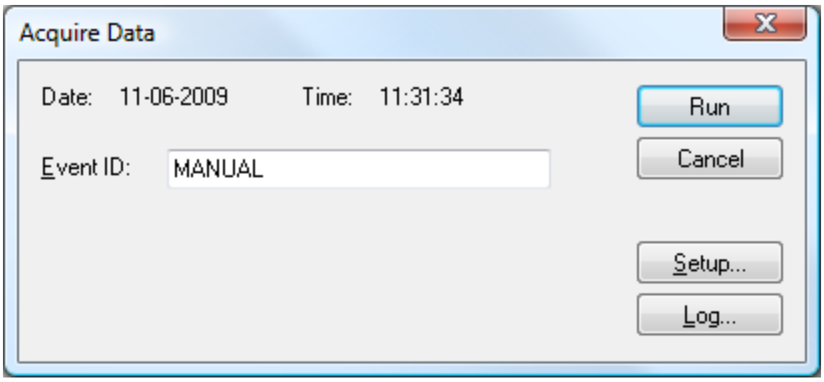
6.7.4 Run Default Setup

Default setup performs a group iteration, which acquires data and saves them as an event. The data acquisition is performed as defined by the default parameters.

If a traverse system is included in the configuration, the probe will be moved in accordance with the selected traverse grid before each acquisition.

To Start Run Default

1. Choose the Default setup command from the Run menu or the Run button  in the Main toolbar.
   Acquire Data to Disk dialog box opens.



To Name the Event

Type in the identification of the raw data to be acquired in the Event ID text box.

To Select Number of Points

The number of positions in the Traverse grid (if defined) are displayed in the No of positions configured text field . If you have no Traverse assigned, only one data acquisition in one point is carried out.

You can select the points in the grid where you want acquisition to performed:

1. Choose the Include all points radio button.
   Data are acquired in all grid points.

- or -

2. Choose the Only points radio button.
   Type in the wanted start and end points in the Only points boxes.

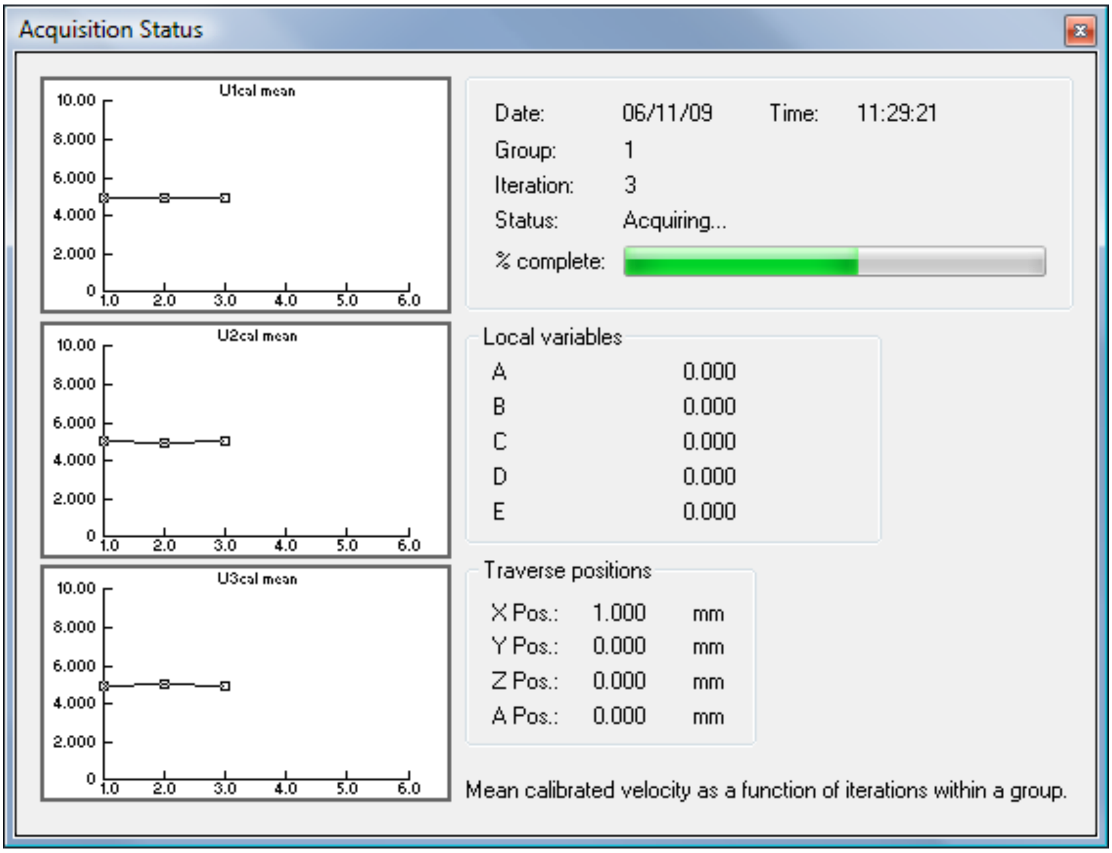
To Fill in the Log

1. Choose the Log button.
   Log dialog box opens with a text field, where text can be entered.
   Write in your comments and:
2. Choose OK.
   The dialog box closes.

To Acquire Data

1. Choose Run.

The Acquisition status window opens, where you can follow the acquisition point by point. The status window also shows the Local variables and the traverse positions. The system starts acquiring data as specified. The Status bar shows Mode: RUNNING.



In the Set Defaults you have selected either Infinite loops, Single Count or Multiple counts.

Infinite Loops

The acquisition continues infinitely. You are, however, prompted for continuation between each loop.

1. Choose the STOP button to stop data acquisition.
   Data are saved as a Raw Data record and added to the event list in the Project Manager.
2. Choose the STEP button to continue with one more loop.
   Data are saved after each loop.

Single Count

The acquisition stops automatically after one loop. This is indicated by a beep.

Data are saved as a Raw Data record, which is added to the event list in the Project Manager.

The Status bar shows Mode: INACTIVE.

Multiple Counts

The acquisition stops automatically, when the last loop is finished. This is indicated by a beep.

Data are saved as a Raw Data record, which is added to the event list in the Project Manager. The

Status bar shows Mode: INACTIVE.

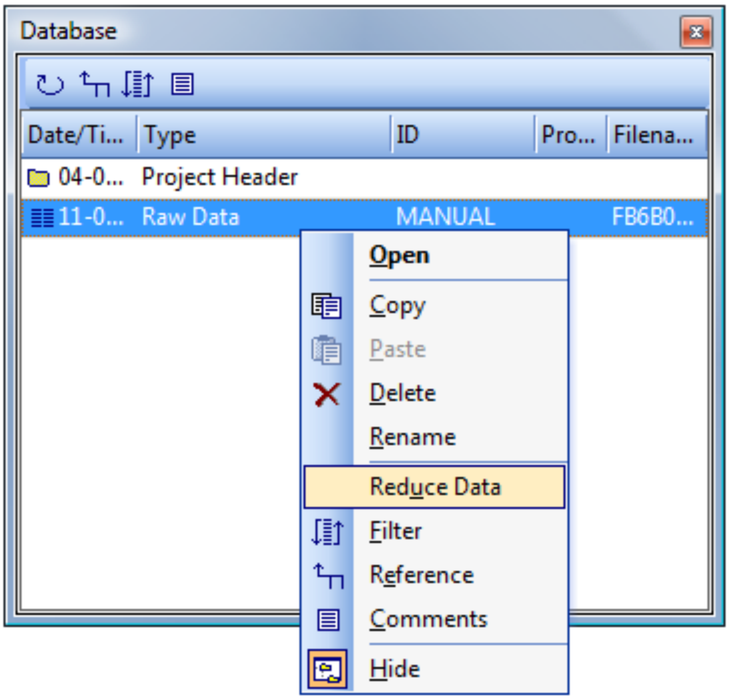
6.7.5 Data Reduction

Data Reduction converts raw data to physical units (velocities) and reduces them into statistical quantities as defined in the default Conversion/Reduction event.

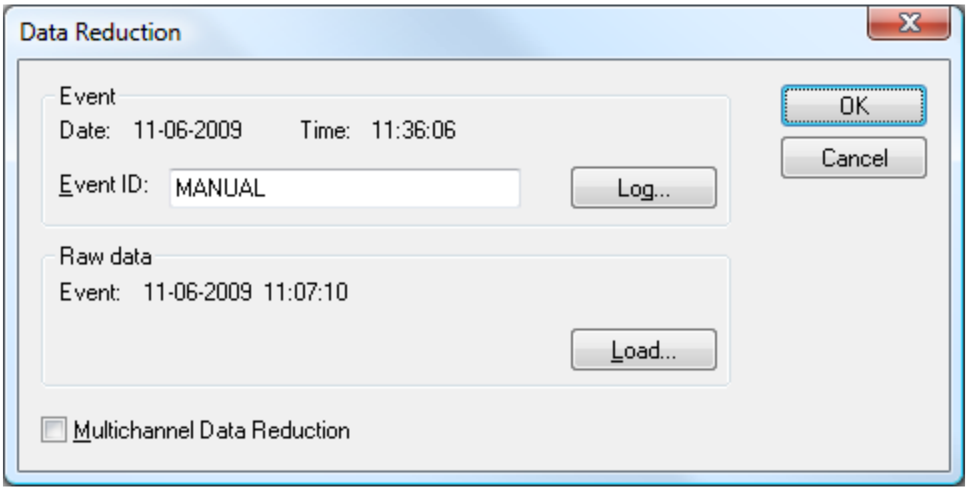
Note
Data Reduction can only be done for one probe at a time.

To Start Data Reduction

1. Select a probe in the Probe list in the upper toolbar.
2. Choose the Data Reduction button in the main toolbar or the Data reduction command from the Run menu. You can also click with the right mouse button on the Raw data that you want to reduce.



A Data Reduction dialog box opens:



To Name the Event

Type in the identification in the Event ID box. After completion the reduced data will be saved as a Reduced Data record, which is added to the Project Manager.

To Select Raw Data Event

If you have not double-clicked on the Raw data record, you must now load the wanted data.

1. Choose the Load button.
   Select Event dialog box opens.
2. Double click on wanted Raw data event in the Event list.
   The dialog box closes, and the selected event is now displayed in the Raw data field.

To Fill in Log

Comments can be written into the project log.

1. Choose the Log button.
   Log dialog box opens with a text field, where text can be entered.
   Write in your comments and:
2. Choose OK.
   The dialog box disappears.

Multichannel Data Reduction (only active with Multichannel CTA Add-On)

When selected, raw data from all the probes in the configuration are reduced in the same run. The reduced data are saved in individual Reduced Data events, one for each probe, and in a Multichannel Reduced data event containing all array and traverse positions.

To Perform the Data Reduction

1. Choose the OK button.

The data reduction is carried out.

The Status bar shows Mode: RUNNING.

When finished it is automatically saved as a Reduced Data record, which is added to the event list in the Project Manager.

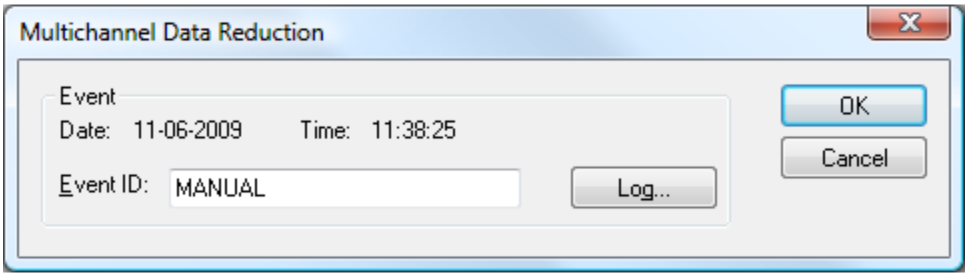
To Leave Data Reduction

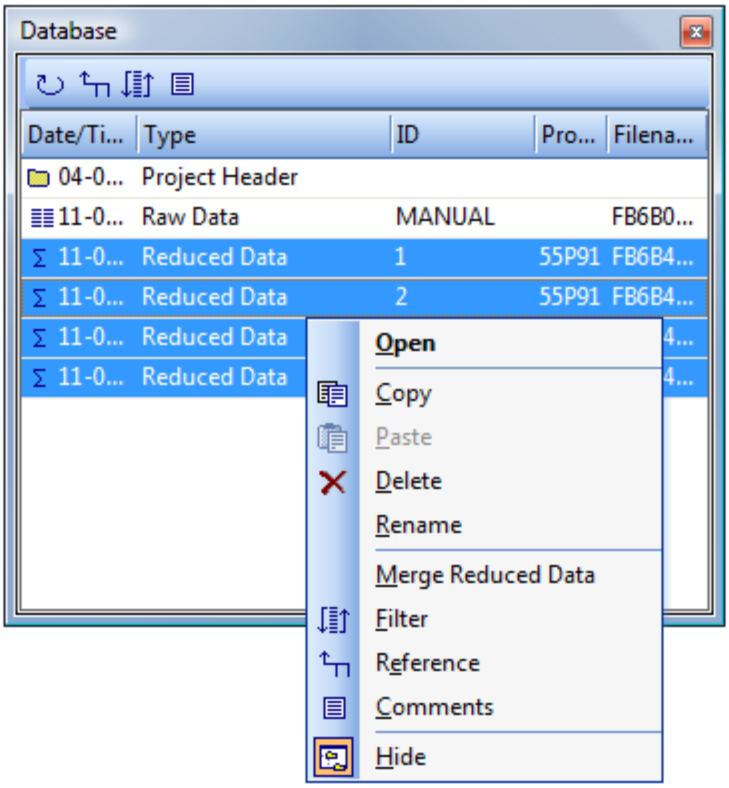
When the reduction is finished, it is recorded as an event. The Run Data reduction sequence is over and needs no separate action to be finished or closed down.

6.7.6 To Merge Reduced Data (only active with Multichannel CTA Add-on)

This function re-arranges reduced data from the individual probes in a probe array into one Multichannel Reduced Data event containing all array and traverse positions.

1. Select the Reduced Data events related to a specific set of Raw data in the Project Manager.
2. Click with the right mouse button and choose Merge reduced data.
3. Multichannel Data Reduction dialog box opens, where you can enter an event identification.
4. Choose OK and the event is saved and added to the Project Manager.





6.7.7 Pause/Resume

These commands to stop and start a program execution temporarily.

To Stop

1. Choose Pause from the Run menu.

All program execution is stopped.

The Task Queue Console shows Task Status: Disabled

The sub menu command Pause changes to Resume.

To Start Again

1. Choose Resume from the Run menu.

The program continues from where it was stopped.

The sub menu command Resume changes to Pause.

6.7.8 Halt All Processes

This command stops a program execution at any point during execution.

To Stop

1. Choose Halt all processes from the Run menu.

The program execution is stopped. The Task Queue Console, if open, shows Task Status: Disabled. The sub menu command Halt all processes are grayed out.

No data are saved after a Halt all processes command. To run the same Default setup or Experiment once more requires that you start from the beginning with a new Run .... selection.

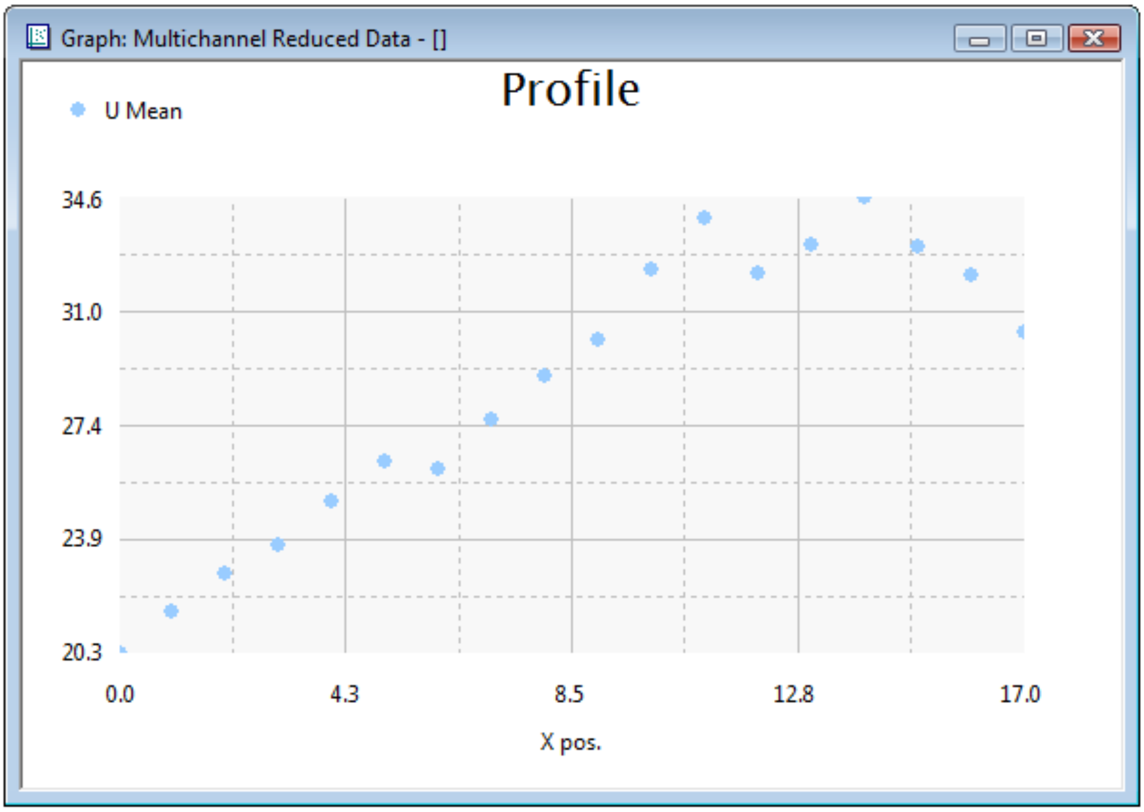
6.7.9 Graph

This command creates a graph on basis of a Data sheet. See "Graphs" on page 73.

To Present a Graph of Multichannel Reduced Data

1. Double-click on the Multichannel Reduced Data Project Manager.
2. A worksheet with the data opens. The positions, e.g. X-positions, represent the positions of each probe in turn. For a traverse with N positions, the first N points contains positions and reduced data from probe 1, the next N positions represent probe 2 and so forth.
3. Click on the Graph icon in the upper toolbar. The Select Data dialog box opens.
4. Select the data. See "Graphs" on page 73.

Note
Choose “No line” in Style in order to avoid lines between “non-neighboring” points.



Multichannel Reduced Data - []

	X pos.	Y pos.	Z pos.	A pos.	U Mean
1	0.000	0.000	0.000	0.000	20.279
2	6.000	0.000	0.000	0.000	26.002
3	12.000	0.000	0.000	0.000	32.158
4	1.000	0.000	0.000	0.000	21.562
5	7.000	0.000	0.000	0.000	27.589
6	13.000	0.000	0.000	0.000	33.068
7	2.000	0.000	0.000	0.000	22.751
8	8.000	0.000	0.000	0.000	28.963
9	14.000	0.000	0.000	0.000	34.574
10	3.000	0.000	0.000	0.000	23.659
11	9.000	0.000	0.000	0.000	30.057
12	15.000	0.000	0.000	0.000	33.025
13	4.000	0.000	0.000	0.000	24.985
14	10.000	0.000	0.000	0.000	32.268
15	16.000	0.000	0.000	0.000	32.123
16	5.000	0.000	0.000	0.000	26.238
17	11.000	0.000	0.000	0.000	33.875
18	17.000	0.000	0.000	0.000	30.297

Graph of velocity profile measured by traversing a probe array of 6 probes in 3 positions.

6.7.10 Extended Processing

You can use command to perform data reduction in a software applications placed outside MiniCTA. They are called via function drivers in the Function Library.

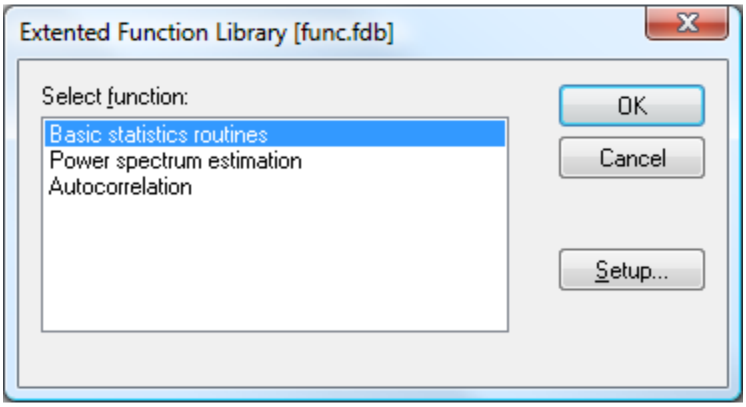
Before you can process data they must be selected from a Data sheet.

To Open and Select Data

1. Open the Raw data event by double-clicking with the left mouse on the event in the Project Manager and select the traverse point (if applied).
   A Data sheet with time stamped data appears.
2. Select the wanted data from the Data sheet.

To Run Extended Processing

1. Choose the Extended processing in the Run menu.
   An Extended Processing dialog box opens which shows the content of the Function Library.



2. Double click on the wanted function.
   A Data sheet with the processed data appears.

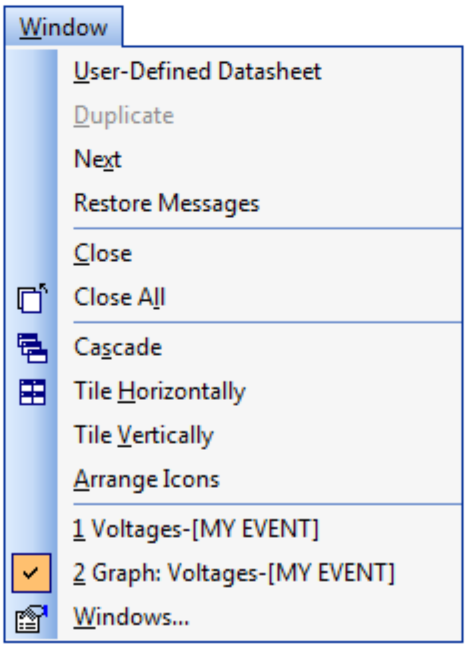
The Extended Processing dialog box contains a Setup command. This is not yet implemented.

To Save Processed Data

1. Double-click in the Menu-control box in the left top corner.
   A Save event dialog box opens, where you can type in an event identification and make comments in an event log.
2. Choose OK.
   The processed data are saved as a record and listed in the Project Manager.

6.8 Windows

6.8.1 Definitions



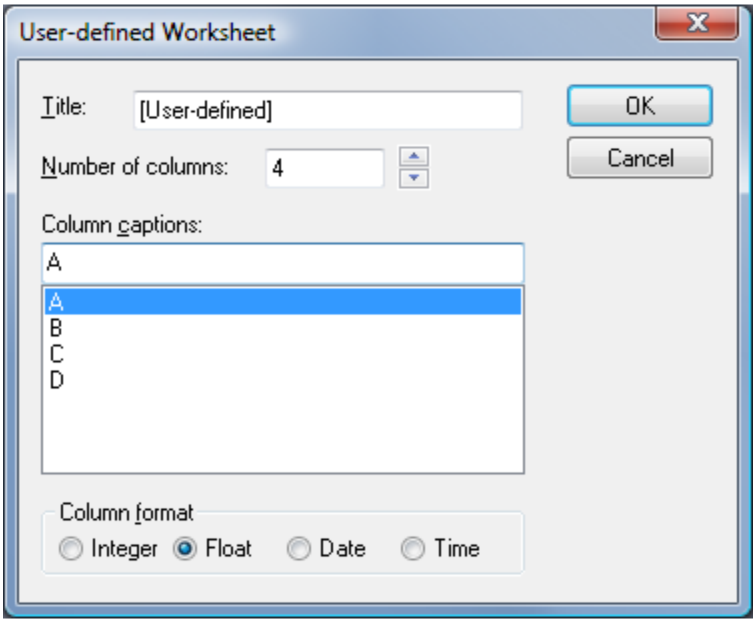
Use this command to open a new window. When you open a new window, it is displayed on top of all other windows. The total number of windows that can be opened at a time is limited by the memory size. Normally about 30 windows can be opened before you get memory problems.

When you open several windows at a time, they begin to overlap in such a way that some of the windows are hidden beneath other windows. You can rearrange them by means of the Arrange All, Tile or Cascade commands. When you open a new window you will have to repeat the Tile or Cascade command in order to have it arranged into the group.

6.8.2 User Defined Worksheet

Use this command to open a User-defined Worksheet.

1. Choose User defined in the Options menu.
   A User-defined Worksheet dialog box opens, where you can define your own worksheet. It has the following options:



Title

Enter the title of the worksheet here.

Number of Columns

Selects the number of columns in the worksheet.

Column Captions

Enter the caption of the column selected in the column list.
Column format:

Integer

Defines integer format for the column selected in the column list.

Float

Defines floating point format for the column selected in the column list.

Date

Defines present date to be written in the column selected in the column list.

Time

Defines present time to be written into the column selected in the column list.

2. Choose OK.
   The dialog box and opens a User-defined window with a worksheet with the number of columns, the captions and pre-formatted cells as defined.

6.8.3 Window Commands

These are normal Window commands.

Duplicate

Opens a new window identical with the active one and displays it on top of that. Data are not transferred into to the new window. You can copy data from the previous window and paste it into the new one.

Next

Rearranges the windows, so that the previous window is displayed on top of the others.

Arrange All

Lines application Icons up across the bottom of the desktop.

Tile

Divides the available space in the MiniCTA software workspace into smaller windows of similar sizes for each group of windows that you have open.

Cascade

The open group windows are re sized and layered within the workspace so that each title bar is visible.

Close

Closes the active window in the group of open windows. If you have made changes in the window, for example entered or changed data , you are asked if you want to save the window as an event.

Close All

Closes the all windows. If you have made changes in a window, for example entered or changed data, you are prompted to save the window as an event.

7 Software Reference

This Software Reference contains definitions of algorithms and probe coordinate systems algorithms that are used for correcting and converting probe signals into velocities and decompose them into wire or probe coordinate systems.

7.1 Algorithms

This section describes the algorithms that are used in the MiniCTA application software for creating Hardware Setup, acquiring and correcting CTA voltages, converting them into velocity components and reducing these into statistical quantities.

7.1.1 Hardware Setup

Hardware setup is the setup of a CTA module. It includes adjustment of the overheat ratio.

Sensor Cold Resistance

The total probe resistance is measured and the sensor cold resistance is calculated by subtracting probe leads, support and cable resistance's:

$R_{s,cold} = R_{tot} - (R_{leads} + R_{support} + R_{cable})$

Overheat Ratio

The overheat ratio is defined as:

$a = (R_{s,hot} - R_{s,cold})/R_{s,cold}$

It is related to the sensor temperature by the relation:

$a = a_{ref} \cdot (T_w - T_{ref})$

where $a_{ref} = a_{20}/(1+a_{20} \cdot (T_{ref} - T_{20}))$

Sensor Hot Resistance

The hot resistance of the sensor is defined as:

$R_{s,hot} = (1 + a) \cdot R_{s,cold}$
or:
$R_{s,hot} = (1 + a_{ref} \cdot (T_w - T_0)) \cdot R_{s,cold}$

Overheat Resistance

The overheat resistance is the value that is maintained constant by the servo-loop in the CTA:

$R_{overh.} = R_{s,hot} + (R_{leads} + R_{support} + R_{cable})$

Decade Resistance

The decade resistance is the value that is established in the bridge and determines the sensor over-temperature:
$R_{dec} = BR \cdot R_{overh.}$

Nomenclature

$R_{s,cold}$ = Sensor cold resistance

R_{tot} = Probe total resistance

$R_{support}$ = Support resistance

R_{cable}= Cable resistance

T_{over} = Over temperature = (T_w-T_0)

T_w = Sensor hot temperature (average)

T_0 = Ambient reference temperature

a = Overheat ratio

a_0 = Sensor temperature coefficient at ambient reference temperature

$R_{s,hot}$ = Sensor hot resistance at Tsensor

$R_{s,cold}$ = Sensor cold resistance at ambient temperature

BR = bridge ratio (normally 20)

R_{overh} = Overheat resistance

R_{dec} = Decade resistance

7.1.2 Probe Calibration

Probe calibration creates a set of related values of CTA output voltage (either bridge voltage directly or Signal Conditioner output). The raw calibration data are stored in a Data sheet and form basis for the curve fitting.

Curve Fitting

Creates relation between CTA output signal and fluid velocity for a specific CTA Hardware setup and probe orientation. The curve fitting is based on voltages that are corrected to the Hardware setup reference temperature.

In MiniCTA you can select between 4 different curve fitting algorithms.

Polynomium

The polynomial curve fit defines velocity as function of voltage:

$$U = C_0 + C_1 \cdot E + C_2 \cdot E^2 + C_3 \cdot E^3 + C_4 \cdot E^4 + C_5 \cdot E^5 + C_6 \cdot E^6$$

where:
U = Velocity

E = CTA output volts

Coefficients C_0 to C_6 are calculated from a standard least square fit. (Polynomial order can be selected from 1 to 6).

Power Law (Kings Law)

The power law defines the voltage as function of velocity:

$E^2 = A + B \cdot U^n$

where:
U = Velocity

E = CTA output volts

A, B and n = Calibration constants

A, B and n are calculated from a standard least square fit.

Lookup Table

The Look-up table linearizes the bridge voltage by means of a table in E and U (max size 120 k). Table values are created through linear interpolation between calibration points. Note that the lookup table requires many calibration points (normally between 20 and 40 points) and that the size should match the resolution of the A/D converter (size = (E_{max}-E_{min})/Resolution)).

7.1.3 Data Conversion

Data Conversion is a number of actions in the Data Reduction process during which the raw CTA output voltages are rescaled, temperature corrected, linearized and decomposed into velocity components:

- Temperature Correction
  If polynomial fit or table look-up is selected: E_{bridge} -> E_{corr}

  If power law is selected: E_{bridge} is not corrected but A, B and n are corrected to match acquisition temperature.

- Linearization
  If polynomial fit or table look-up is selected: E_{corr} -> U_{eff}

  If power law is selected: E_{bridge} -> U_{eff}

- Velocity Decomposition (U_{eff} -> >$U_{wire\ coord.}$ -> > $U_{probe\ coord.}$)

- Velocity Transformation ($U_{probe\ coord.}$ -> $U_{lab.\ coord.}$)

Temperature Correction

Temperature correction is done in two different ways depending on the transfer function (curve fitting algorithm) selected in the Calibration event assigned to the actual Conversion/reduction.

CTA output voltages acquired at one ambient temperature are corrected to the reference temperature by means of the formula:

$E_{corr} = ((T_w - T_0)/(T_w - T_1))^{0.5} \cdot E_1$

Nomenclature

E_{corr} = Voltage corrected to reference temperature

E_1 = Voltage acquired at ambient temperature T_1

T_w = Sensor hot temperature

T_0 = Reference temperature (from Hardware setup)

T_1 = Ambient temperature during data acquisition

Note
The influence from variations in heat conductivity, Prandtl number, density and dynamic viscosity is not corrected for.

Linearization

Linearization is the process of converting a CTA voltage to velocity by means of the transfer function selected in the Calibration Event that is assigned to the Conversion/reduction event. If temperature correction is selected, the voltage or the calibration constants are corrected for temperature variations. The result of the linearization is the effective cooling velocity U_{eff}.

Transfer Functions

CTA Probes (wire, fibre, film)

Polynomium

Transfer function (= curve fit):

$$U = C_0 + C_1 \cdot E + C_2 \cdot E^2 + C_3 \cdot E^3 + C_4 \cdot E^4 + C_5 \cdot E^5 + C_6 \cdot E^6$$

U = Calibration velocity

E = CTA output volts (rescaled)

C_0 to C_6 = Coefficients

The degree is in accordance with the selected Calibration Event.

Power Law (Kings Law)

Transfer function:

$$U = ((E^2 - A)/B)^{1/n}$$

U = Calibration velocity

E = CTA output volts

A, B and n = Calibration constants

Look-up Table

U from table with E input. The table is created on the basis of linear interpolation between the calibration points.

Temperature Probes (Misc.)

Temperature probes (e.g. 55P32 and 90P10) for use with the MiniCTA and Multichannel CTA have uni-curve thermistor sensors and are operated by amplifiers (e.g. 54T40) with a voltage output.

Logarithmic Polynomial

$T = C(0) + C(1)\cdot \ln(E)\ ) + C(2)\cdot \ln^2(E) + C(3)\cdot \ln^3(E) \qquad [°C]$

T = temperature in °C

E = amplifier output volts

The linearisation constants are contained in the probe library.

Reference Velocity Probes

The reference velocity probe 54T29 is linearized on the basis of an individual velocity-voltage calibration in 20 points performed at Dantec Dynamics.

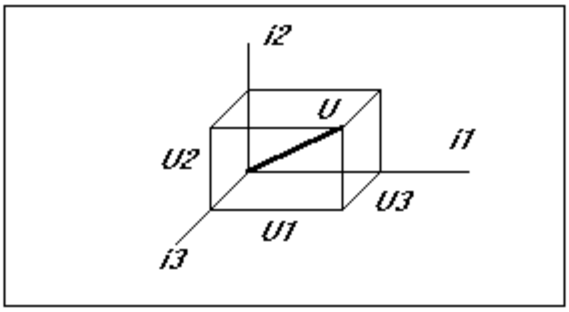
Log-square

The velocity range is divided into 19 sub-ranges each with its own transfer function. The transfer function relates the probe output voltage to the square of the logarithmic velocity with linearisation constants valid for the specific range. The constants are contained in a calibration file delivered together with the reference probe.

7.1.4 Velocity Decomposition

Velocity decomposition is the process of converting effective cooling velocities into velocity components the wire coordinate system and into in the probe coordinate system.

Wire Coordinate System



Definitions

The wire coordinate system is a right-hand ruled Cartesian Coordinate System defined relative to the axis of the sensor(s). The hot wires defines the axis of the coordinate system. Sensor 1 is aligned with wire coordinate axis 1, sensor 2 is aligned with axis 2 and sensor 3 is aligned with axis 3.

General Velocity Decomposition in Wire Coordinate System

A fluid vector U decomposed into wire coordinates is described by:

$U = U_1 i1 + U_2 i2 + U_3 i3$

where U_1, U_2 and U_3 are components of U in the directions i1, i2 and i3 respectively.

The general equations between the effective cooling velocity U_{eff} and the velocity components U_1, U_2 and U_3 in the wire system which takes tangential cooling and prong interference into account yields:

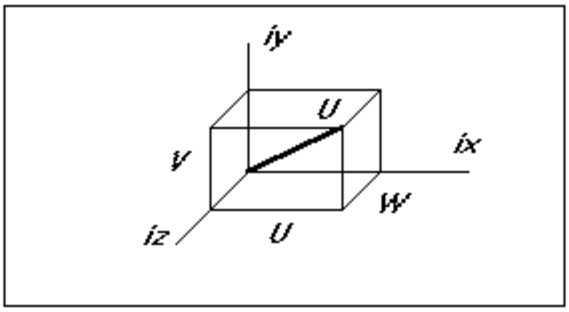
$U_{1eff}^2 = k_1 \cdot U_1^2 + U_2^2 + k_2 \cdot U_3^2$

$U_{2eff}^2 = k_2 \cdot U_1^2 + k_1 \cdot U_2^2 + U_3^2$

$U_{3eff}^2 = U_1^2 + k_2 \cdot U_2^2 + k_1 \cdot U_3^2$

Where k_1 and k_2 are the yaw and pitch factors, respectively.

Probe Coordinate System



Definition

The probe coordinate system is a right-hand ruled Cartesian Coordinate System defined by the orthogonal unit vectors (ix,iy,iz).

Each Dantec standard probe has a fixed orientation between probe coordinate system and wire coordinate system given by the orientation of sensor(s), sensor planes and probe stem.

These orientations are fixed for each probe type and can only be changed, if you overwrite the default sensor/probe angles in the Probe Library.

General Velocity Decomposition in Probe Coordinate System

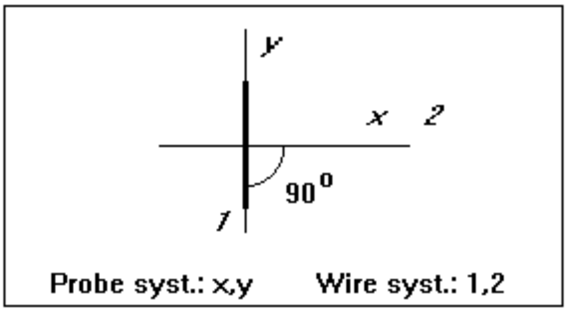
A fluid vector U decomposed into probe coordinates is described by:

U = Uix + Viy + Wiz

where U, V and W are components of U in the directions ix, iy and iz, respectively.

It is recommended to place the probe, so that the probe coordinate system coincides with the laboratory coordinate system.

Single-sensor Probes



Single sensor probes have the x-axis of the probe coordinate system coinciding with axis 2 of the wire coordinate system.

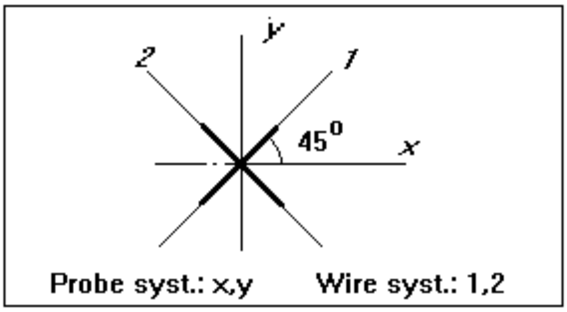
Velocity Decomposition

For single-sensor probes in a 1-dimensional flow, where the probe x-axis is in the direction of the flow vector U we obtain:

$U_{eff}^2 = (U_1^2) + U_2^2 + (U_3^2)$ or $U = U_2 = U_{eff}$

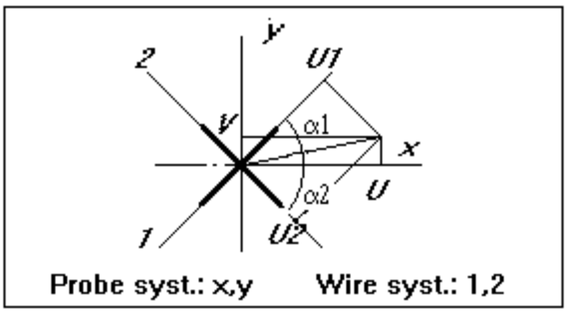
where U_{eff} is the effective cooling velocity obtained from linearization of the anemometer output voltage.

Dual-sensor (X-array) Probes



Dual sensor probes (X-probes) have the sensor plane in the xy-plane of the probe coordinate system. The angle between the x-axis and wire 1 is denoted α(x/1) and between the x-axis α(x/2).

Velocity Decomposition



The following equations are used for calculation the velocity components (W is zero or can be neglected) on basis of the calibration velocities and the yaw-factors k_1, k_2 for the wires. The following steps are carried out:

Calibration velocity (U_{1cal},U_{2cal}) to Wire coordinate system (U_1,U_2):

$U_{1cal}^{2}\cdot(1 + k_1^2)\cdot\cos(90 - \alpha(x/1)) \quad = k_1^{2}\cdot U_1^2 + U_2^2$

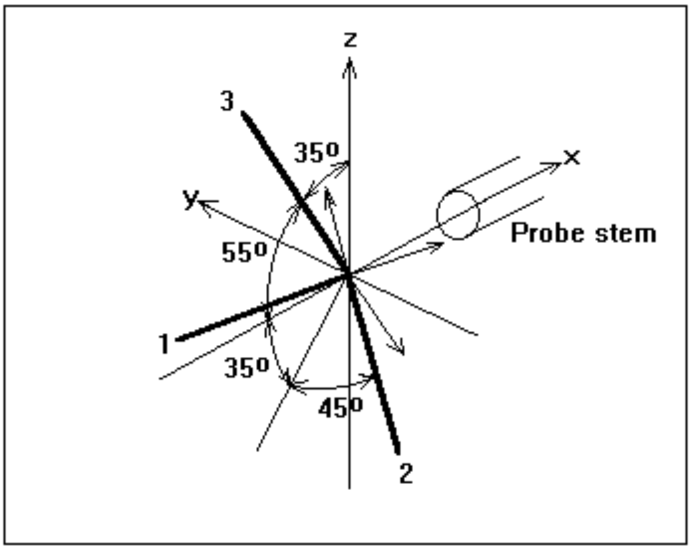
$U_{2cal}^{2}\cdot(1 + k_2^2)\cdot\cos(90 - \alpha(x/2))^2 = U_1^2 + k_2^{2}\cdot U_2^2$

These two equations are solved with respect to U_1 and U_2 and inserted in the Wire (U1,U2) to Probe (U,V) coordinate equations:

$U = U_1\cdot\cos\alpha_1 + U_2\cdot\cos\alpha_2$

$V = U\cdot_1\sin\alpha_1) - U\cdot\sin\alpha_2$

Tri-axial Probes



The probe is placed in a three-dimensional flow with the probe axis aligned with the main flow vector.

The velocity components U, V and W in the probe coordinate system (x,y,z) are given by:

$U = U_1\cdot\cos54.736° + U_2\cdot\cos54.°736° + U_3\cdot\cos54.736°$

$V = -U_1\cdot\cos45° - U_2\cdot\cos135° + U_3\cdot\cos90°$

$W = -U_1\cdot\cos114.094° - U_2\cdot\cos114.094° - U_3\cdot\cos35.264°$

where U_1, U_2 and U_3 are calculated from the general set of equations:

$U_{1eff}^{\ 2} = k_1^{2}\cdot U_1^2 + U_2^2 + h_1^{2}\cdot U_3^2$

$U_{2eff}^{\ 2} = h_2^{2}\cdot U_1^2 + k_2^{2}\cdot U_2^2 + U_3^2$

$U_{3eff}{}^2 = U_1{}^2 + h_3{}^2 \cdot U_2{}^2 + k_3{}^2 \cdot U_3{}^2$

where U_{1eff}, U_{2eff} and U_{3eff} are the effective cooling velocities acting on the three sensors and k_i and h_i are the yaw and pitch factors, respectively.

As Tri-axial probes are calibrated with the velocity in direction of the probe axis Ueff is replaced by $Ucal \cdot (1+k_i 2+h_i 2)^{0.5} \cdot \cos 54.736°$, where 54.736° is the angle between the velocity and the normal to the wires:

$U_{1cal}{}^2 \cdot (1+k_1{}^2+h_1{}^2) \cdot \cos^2 54.736° = k_1{}^2 \cdot U_1{}^2 + U_2{}^2 + h_1{}^2 \cdot U_3{}^2$

$U_{2cal}{}^2 \cdot (1+k_2{}^2+h_2{}^2) \cdot \cos^2 54.736° = h_2{}^2 \cdot U_1{}^2 + k_2{}^2 \cdot U_2{}^2 + U_3{}^2$

$U_{3cal}{}^2 \cdot (1+k_3{}^2+h_3{}^2) \cdot \cos^2 54.736° = U_1{}^2 + h_3{}^2 \cdot U_2{}^2 + k_3{}^2 \cdot U_3{}^2$

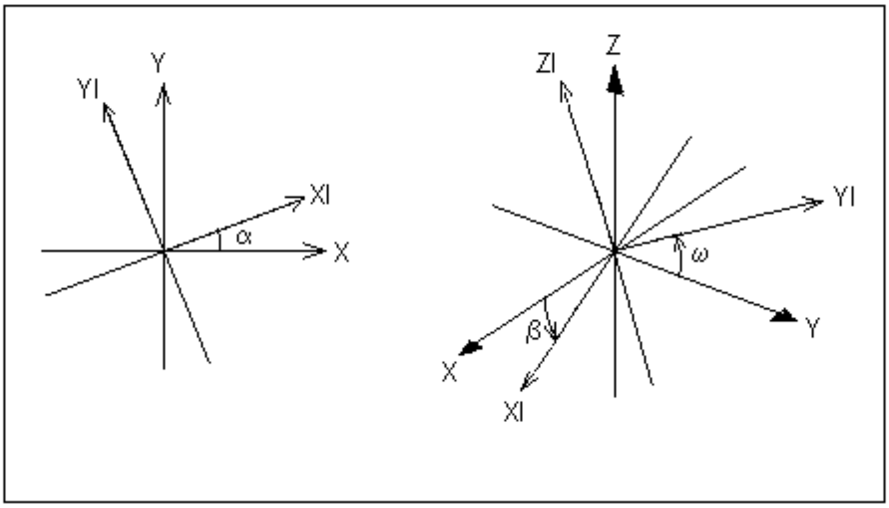
With the default values $k^2 = 0.0225$ and $h^2 = 1.04$ the solution for U_1, U_2 and U_3 becomes:

$U_1=(-0.3477 \cdot U_{1cal}{}^2+0.3544 \cdot U_{2cal}{}^2+0.3266 \cdot U_{3cal}{}^2)^{0.5}$

$U_2=(0.3266 \cdot U_{2cal}{}^2-0.3477 \cdot U_{2cal}{}^2+0.3544 \cdot U_{3cal}{}^2)^{0.5}$

$U_3=(0.3544 \cdot U_{1cal}{}^2+0.3266 \cdot U_{2cal}{}^2-0.3477 \cdot U_{3cal}{}^2)^{0.5}$

Transformation into Laboratory Coordinate System



Coordinate transformation can only be done for 2D and 3D probes.

2-D Transformation

The velocity components U and V in the probe coordinate system X,Y are transformed into UI,VI in the laboratory coordinate system XI,YI. The transformation is a rotation of the XY-plane in itself.

The transformed velocities then becomes:
(UI,VI) = M(U,V)
where the rotation matrix M is defined as:

$\cos\alpha$	$-\sin\alpha$
$\sin\alpha$	$\cos\alpha$

3-D Transformation

Transforms the velocity components (U,V,W) in the probe coordinate system (X,Y,Z) into (Ul,Vl,Wl) in the laboratory coordinate system. The transformation represents an inclination of the XY plane (rotated around the Y-axis) with the angle β and α following rotation around the Xl-axis, roll ω
The transformed velocities then becomes:

(Ul,Vl,Wl) = C(U,V,W)
where the matrix C = PxR is defined by the pitch matrix P and the roll matrix R:

Pitch matrix P			Roll matrix R		
$\cos\beta$	0	$-\sin\beta$	1	0	0
0	1	0	0	$\cos\omega$	$\sin\omega$
$\sin\beta$	0	$\cos\beta$	0	$-\sin\omega$	$\cos\omega$

7.1.5 Data Reduction

Data reduction can be performed on any set of data, both in raw and converted form. In the following it is assumed that the reduction is done on velocity components in the Probe coordinate system.

Moments

1st. order

Average

$Umean = 1/n\Sigma Ui$

2 nd. order

Standard deviation

$Ustd = (\Sigma(Ui - Umean)^2/(n-1))^{0.5}$

Cross moments

$uv = 1/n\Sigma(Ui - Umean)(Vi - Vmean)$

$uw = 1/n\Sigma(Ui - Umean)(Wi - Wmean)$

$vw= 1/n\Sigma(Vi -Vmean)(Wi - Wmean)$

3rd. order

$uvw = 1/n\Sigma(Ui - Umean)(Vi - Vmean)(Wi - Wmean)$

$uvv = 1/n\Sigma(Ui - Umean)(Vi - Vmean)^2$

$uww = 1/n\Sigma(Ui - Umean)(Wi - Wmean)^2$

$uuv = 1/n\Sigma(Ui - Umean)^2(Vi - Vmean)$

$uuw = 1/n\Sigma(Ui - Umean)^2(Wi - Wmean)$

$uuu = 1/n\Sigma(Ui - Umean)^3$

$vvv = 1/n\Sigma(Vi - Vmean)^3$

$vvw = 1/n\Sigma(Vi - Vmean)^2(Wi - Wmean)$

$vww = 1/n\Sigma(Vi - Vmean)(Wi - Wmean)^2$

$www = 1/n\Sigma(Wi - Wmean)^3$

7.1.6 Extended Processing

Basic Statistics

Average = $\Sigma Xi/n$

Avg. Deviation = $\Sigma(Xi - Xavg)/n$

Standard Deviation $\sigma = (\Sigma(Xi - Xavg)^2/(n - 1))^{0.5}$

Standard Variance $\sigma^2 = \Sigma(Xi - Xavg)^2/(n - 1)$

Skewness = $\Sigma(Xi - Xavg)^3/n\sigma^3$

Kurtosis = $\Sigma(Xi - Xavg)^4/n\sigma^4$

Auto-Correlation

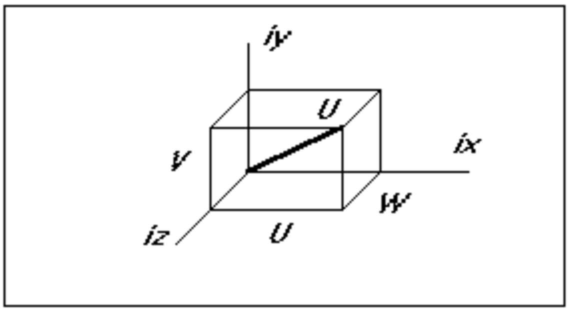
Calculated using Fourier-Stieltjes integrals.

Power Spectrum

Calculated using Fourier-Stieltjes integrals.

7.2 Probe Coordinates

7.2.1 Probe Coordinate Systems



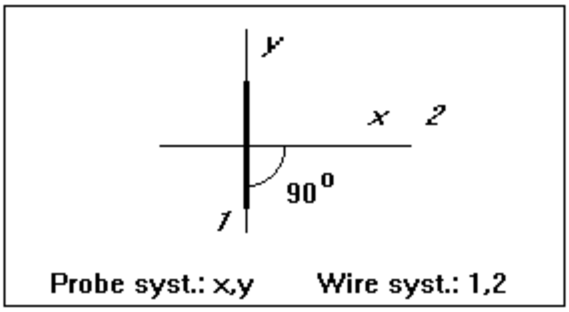
Definition

The probe coordinate system is a right-hand ruled Cartesian Coordinate System defined by the orthogonal unit vectors (ix,iy,iz).

Each Dantec standard probe has a fixed orientation between probe coordinate system and wire coordinate system given by the orientation of sensor(s), sensor planes and probe stem.

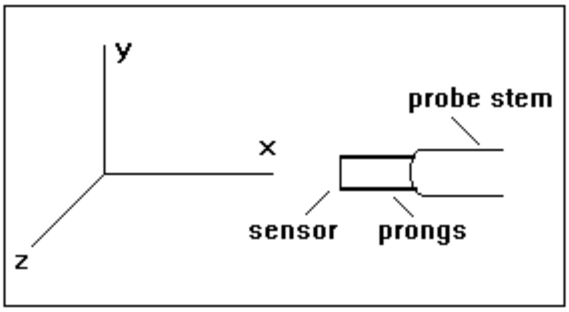
These orientations are fixed for each probe type and can only be changed by the user by overwriting the default values in the Probe Data Base.

7.2.2 Single-Sensor Probes, Coordinate Systems



Single sensor probes have the y-axis of the probe coordinate system coinciding with the 1st axis of the wire coordinate system.

Straight and Boundary-layer Types



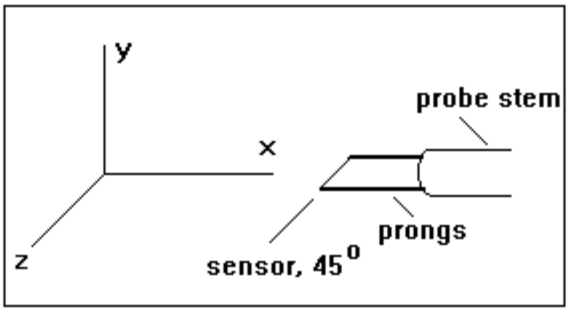
Single sensor probes have the probe stem aligned with x-axis of the probe coordinate system, while the sensor is parallel with the y-axis. The plane of the prongs is in the xy-plane.

This orientation is valid for the following probe types:
55A52, 52A53 Subminiature wire probes
55A75 High-temperature probe
55P01, 55P11 Gold-plated/miniature wire probe
55P05, 55P15 Gold-plated/miniature boundary-layer probes
55P76 Temperature-comp. gold-plated wire probe for 1:5 bridge
55P81 Temperature-comp. miniature wire probe for 1:1 bridge
55P86 Temperature-comp. miniature wire probe for 1:5 bridge
55R01, 55R11 Fiber probe, air/water applications
55R05, 55R15 Fiber probe, air/water appl., boundary-layer probes

55R31, 55R32 Wedge-shaped film probe, air/water appl.
55R76, 55R86 Temperature-comp. fiber probe, air/water applications

Single-sensor Probes, 45° Slanting-sensor Types

Probe stem is aligned with x-axis of probe coordinate system with the sensor in the xy-plane. The long prong is in the 1st quadrant so that the sensor forms an angle of -45° with the x-axis.

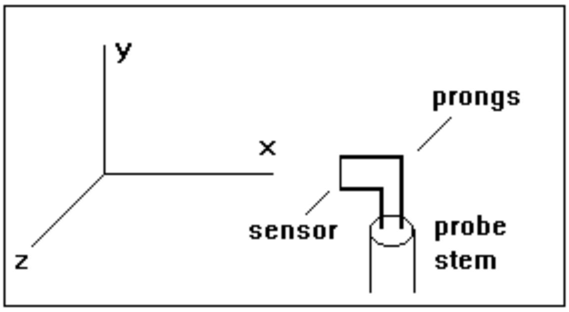


This orientation is valid for the following probe types:

55A54, Subminiature wire probe
55P02, 55P12 Gold-plated/miniature wire probe
55R02, 55R12 Fiber probe, air/water applications

Single-sensor Probes, 90°-bent Types, Sensor Parallel to Probe System

Probe stem is aligned with y-axis of probe coordinate system with the sensor-prong plane in the xy-plane. The sensor is perpendicular to the x-axis.

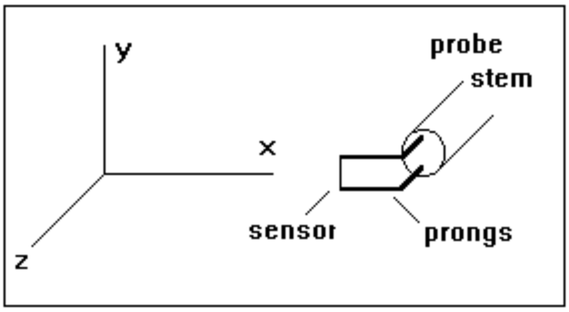


This orientation is valid for the following probe types:

55A55, Subminiature wire probe
55P03, 55P13 Gold-plated/miniature wire probe
55R03, 55R13 Fiber probe, air/water applications
55R33, 55R34 Wedge-shaped film probe, air/water appl.

Single-sensor Probes, 90°-bent Types, Sensor Perpendicular to Probe System

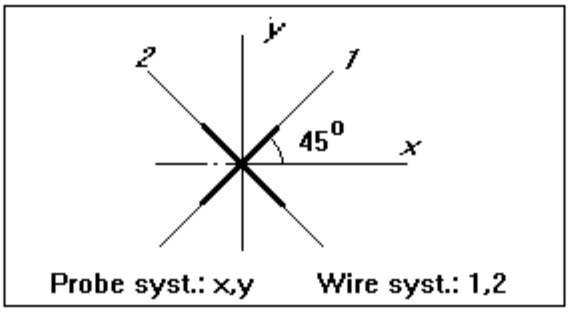
Probe stem is aligned with z-axis of probe coordinate system with the sensor-prong plane in the xy-plane. The sensor is perpendicular to the x-axis.



This orientation is valid for the following probe types:

55P04, 55P14 Gold-plated/miniature wire probe
55R04, 55R14 Fiber probe, air/water applications
55R35, 55R36 Wedge-shaped film probe, air/water appl.

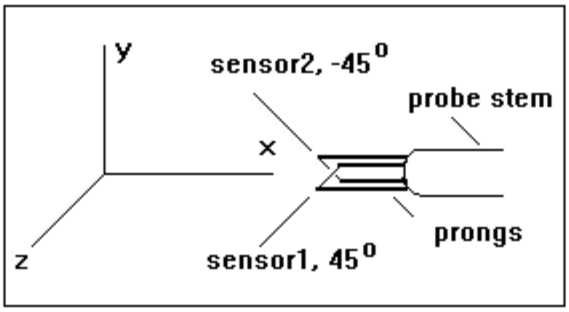
7.2.3 Dual-sensor Probes, Probe Coordinates



Dual sensor probes (X-probes) have the sensor plane in the xy-plane of the probe coordinate system. The angle between the x-axis and wire 1 is denoted (x/1) and between the x-axis (x/2). For the ideal probe they are both 45°.

Dual-sensor Probes, Straight Types

The probe stem is aligned with x-axis of the probe coordinate system, while the sensors are parallel with the xy-plane. Sensor 1 forms +45° and sensor 2 forms -45° with the x-axis.The planes of the prongs are in the xy-plane.

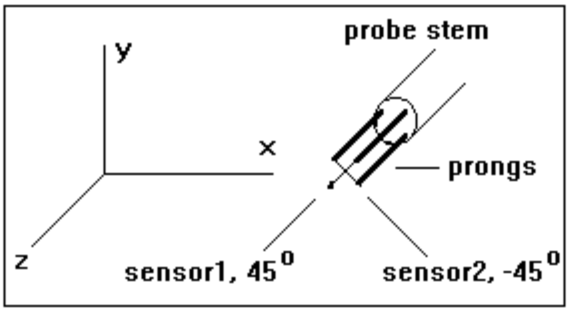


This orientation is valid for the following probe types:

55P51, 55P61 Gold-plated/miniature X-wire probe
55R51, 55R61 Fiber probe, air/water applications

Dual-sensor Probes, Cross-flow Types

The probe stem is aligned with z-axis of the probe coordinate system.The sensors are in the xy-plane. Sensor 1 forms +45° and sensor 2 forms -45° with the x-axis.

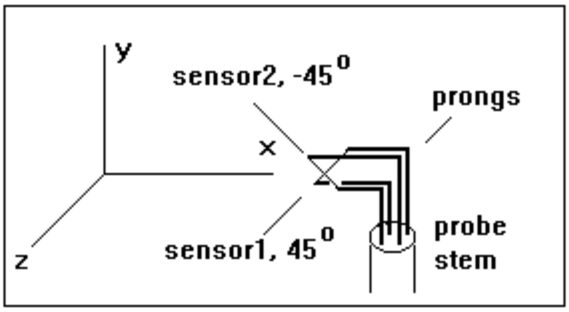


This orientation is valid for the following probe types:

55P52, 55P62 Gold-plated/miniature X-wire probe
55R52, 55R62 Fiber probe, air/water applications

Dual-sensor Probes, 90° Sensor Plane Parallel to Probe Axis

The probe stem is aligned with y-axis of the probe coordinate system. The sensors are in the xy-plane. Sensor 1 forms +45° and sensor 2 forms -45° with the x-axis.

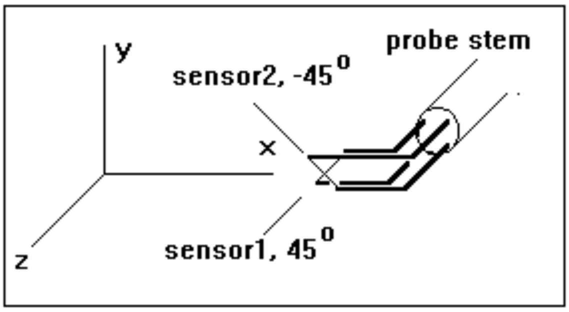


This orientation is valid for the following probe types:

55P53, 55P63 Gold-plated/miniature X-wire probe
55R53, 55R63 Fiber probe, air/water applications

Dual-sensor Probes, 90° Sensor Plane Parallel to Probe Axis

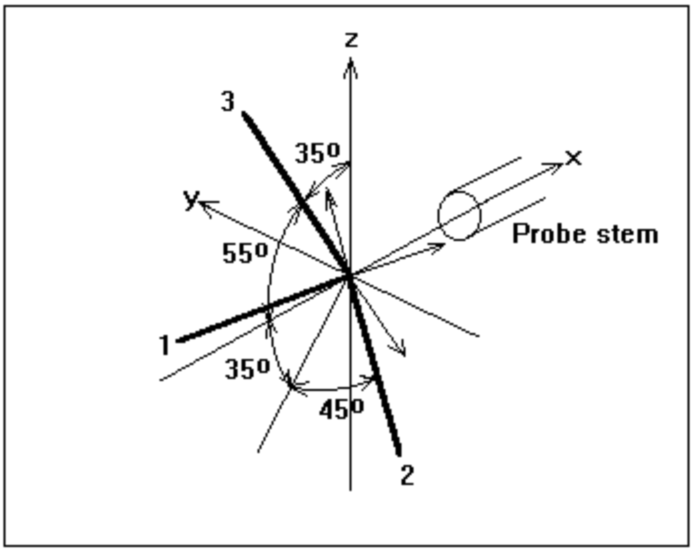
The probe stem is aligned with z-axis of the probe coordinate system. The sensors are in the xy-plane. Sensor 1 forms +45° and sensor 2 forms -45° with the x-axis.



This orientation is valid for the following probe types:

55P54, 55P64 Gold-plated/miniature X-wire probe
55R54, 55R64 Fiber probe, air/water applications

7.2.4 Triple-sensor Probes, Probe Coordinates



The probe stem is aligned with the x-axis and sensor 3 is in the xz-plane in the z-direction.

Triple-sensor Probes, Straight Types

The angles between the wire coordinate system (1,2,3) and probe coordinate system (x,y,z):

1/x = 54.736°___2/x = 54.736°___3/x = 54.736°

1/y = 135°___2/y = 45°___3/y = 90°

1/z = 65.906°___2/z = 65.906°___3/z = 35.264°

Note that the angles in the Probe Library are defined differently:

1/x = 54.736°___2/x = 54.736°___3/x = 54.736°

1/y = 45°___2/y = 135°___3/y = 90°

1/z = 114.094___2/z = 114.094___3/z = 35.264°
This difference in definitions, however, is not reflected in the decomposition of the velocity components.

These angles are valid for:

55P91 Triple wire probe
55P91 Triple fiber probe, air applications

7.3 A/D Devices

The MiniCTA software have drivers for the most commonly used A/D devices from National Instruments and United Electronics. For a complete list see the Device library in your version of the MiniCTA software. As new drivers regularly are added to the software, your list may not be complete. An updated list is available from the download section on Dantec Dynamics homepage. Here you can also download new drivers.

The Software signal simulator may be used as a dummy input, when you want to test MiniCTA Software without having an A/D device installed on your PC. It generates a sine wave as input, when you run data acquisitions.

7.3.1 General Consideration

Communication Between MiniCTA Software and A/D Devices

MiniCTA Software communicates through to an A/D device in the following fashion: A manufacturer (third-party) supplied Dynamic Link Library (DLL) controls all the essential operations of the A/D device. The calls made by this library are rerouted by a translation driver, supplied by
Dantec or other party (with an .ADV extension), that converts the MiniCTA Software hardware calls to the third-party driver.

MiniCTA Software -> Translation driver -> Manufacturer supplied driver

Installation of A/D Devices

You are advised to install the board and its driver as recommended by the A/D device manufacturer. Note that the DLL from the A/D driver should be installed in the Windows directory. Configuration of the board is done according to the type of measurement. It is recommended, however, to select Single-ended referenced, Uni-polar if possible.

Note
For MiniCTA check the wiring of the connector box if it is single-ended referenced or differential and configure the board accordingly. The Multichannel CTA is always wired single-ended referenced. Installation of the A/D driver should be done before MiniCTA Software is opened.

National Instruments DAQmx Devices

The software supports the National Instruments DAQmx Device Driver standard. In principle this means that all current and future devices supported by DAQmx is supported by the software. The software is however limited to only support devices holding AD input channels, and require at least v7.3 of DAQmx driver installed (DAQmx Base is not supported).

Installing a DAQmx Device

When installing a DAQmx device it is recommended to have the latest DAQmx driver installed (v7.3 or later is required). Please refer to the www.ni.com homepage and the manual for the specific device for more information, and for the latest version of DAQmx. The software and device CD will contain a compatible version of the DAQmx driver.

Test Without the Device

The DAQmx device driver standard enables you to run most devices as a “Simulated” (non physical) device. This means, that you can test a device in the software without have the physical device. The data will be simulated and most features will be available though the software, except external hardware clock and triggering.

Test the Device

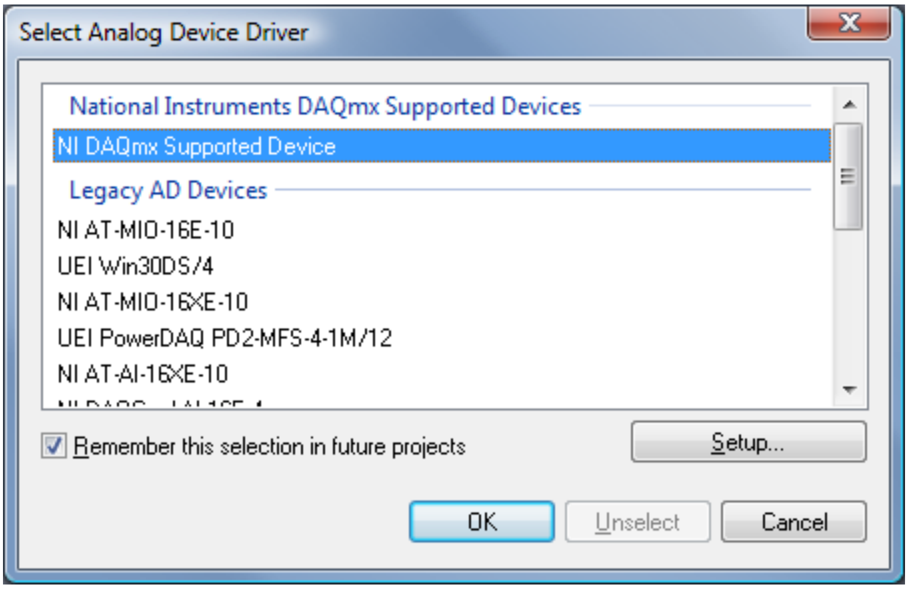
The recommended order of installation is:

1. Install DAQmx v7.3 or later
2. Turn off the PC
3. Install internal bus device or connect the device using USB etc.
4. Turn on the PC and follow the plug-and-play instructions

If something goes wrong during the plug and play installation it is recommended to look in the Windows Device Manager for information and troubleshooting, and please refer to the device manual.
When the device is properly installed, it is displayed in the National Instruments Measurement & Automation Explorer (MAX) under the NI-DAQmx Devices. Please note that other devices can occur in the NI-DAQmx Devices list, and also in the Traditional DAQ section. A device can be tested and verified from within MAX using the Test Panels.

Using a DAQmx Device in MiniCTA

Since DAQmx is a standard platform all DAQmx devices are run using the same driver. In the software this driver is named “NI DAQmx Supported Devices”, and should be used for all DAQmx supported devices. If you during program start up choose to always use DAQmx, this will be selected as the default option in the Select Analog Device Driver dialog.

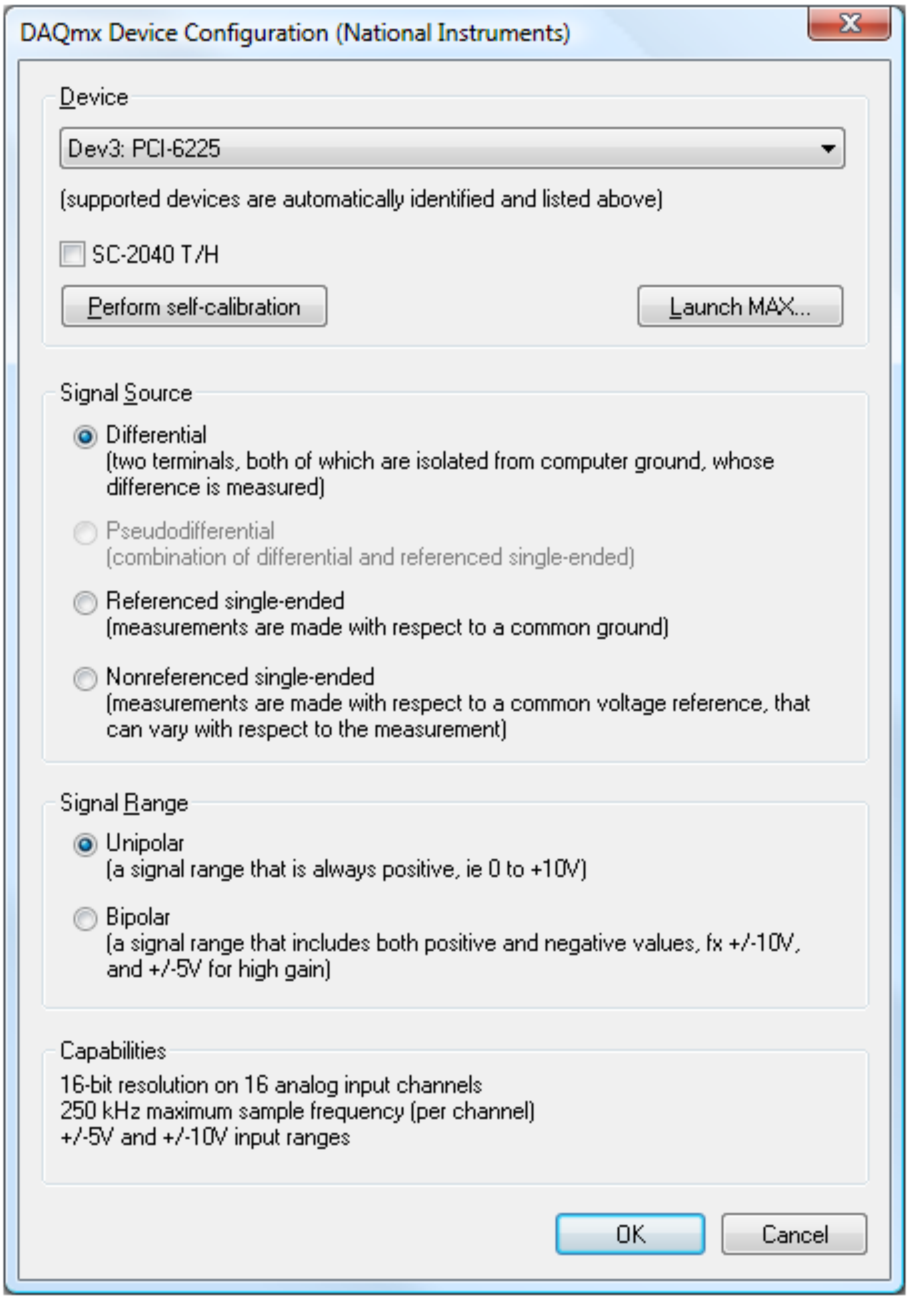


A number of legacy AD drivers are maintained for backward compatibility including United Electronics Power DAQ systems and National Instruments Traditional DAQ systems. Please note that these drivers will not be developed.

DAQmx Device Configuration

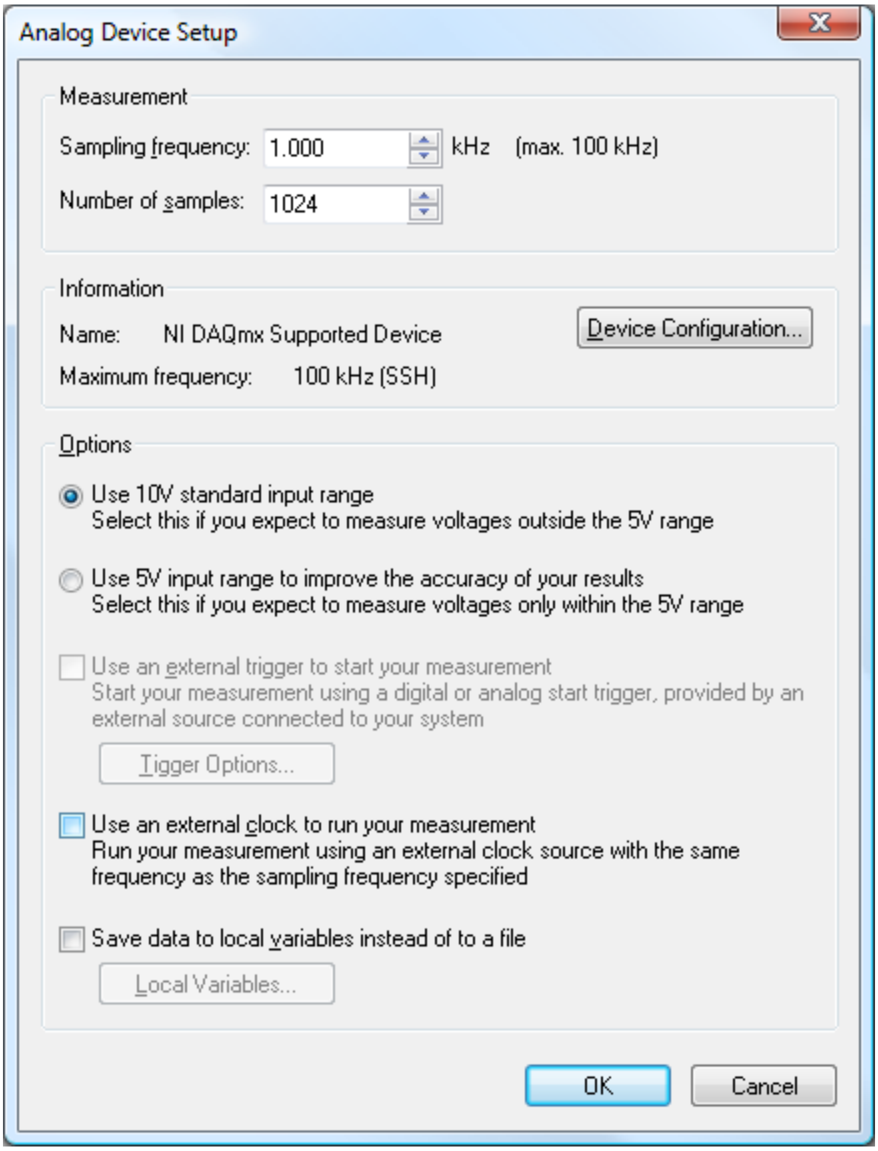
The DAQmx Device Configuration let you select between the installed DAQmx devices (previously found and identified by MAX). By selecting between the different devices the device capabilities are automatically detected and shown.

The primary capabilities used in the software are the Signal Source, and the Signal Range. The descriptions of the different modes can be seen directly in the dialog. Normally you would select between differential or referenced single ended sources. However some devices only supports differential as source and the wirering must be made accordingly using the correct connector box. The signal range defines direction of the input signal. You get the best results if you match the range with the range of you actual signal. Most CTA probes works in the 0 - 5 Volts or 0 - 10 Volts positive range, but some special and user defined probes will require both positive and negative polarity.



As additional options you can launch MAX from within this dialog along with performing an internal calibration of the DAQmx device. For legacy SC-2040 T/H owners and option is included to support this device. However it is recommended to use a native simultaneous sample and hold DAQmx device compared to the costly and slow external track and hold solution.

The selected device configuration; device, signal source and range will be used in you current and future databases and projects when specifying the setup of you measurement. The Analog Device Setup dialog will use the configuration selected and restrict the setup to be within the selected device capabilities.



Based on the DAQmx device, sampling mode and number of used analog input channels the maximum available sampling frequency will be determined. The gain can be selected to match your input signal, ensuring the best possible resolution of your data.

External Trigger

Dependent on the device capabilities both analog and digital external triggering is available. The analog trigger can be defined using a trigger analog threshold voltage, and the slope direction of the signal. For digital triggers the direction of the edge can be defined. Both analog and digital trigger uses the PFI0 port as trigger source by default (see advanced setting for specifying an alternative port).

External Clock

If capable the DAQmx device can be set to use an external sample clock, instead of using the build-in. The sample frequency specified in the dialog must correspond to the maximum expected external clock frequency. The external clock signal must use the PFI2 port by default (see advanced setting for specifying an alternative port).

Advanced Settings

The DAQmx driver will by default use the most common settings recommended by National Instruments. However in some situations it will be necessary to change the default setting for some parameters to suite the device to a specific CTA application.
The following settings can be changed in the Windows Registry using the "regedit.exe" tool in the following location: "HKEY_CURRENT_USER\Software\Dantec Dynamics\CTA\IniSettings\NI-DAQmx"

External trigger source port PFI0 is the default external trigger source port specified by National Instruments and is used as label on e.g. timer boxes etc.	TrigPortEx: TrigPort = PFI1 (default = PFI0)
External clock source port PFI2 is the default external clock source port specified by National Instruments and is used as label on e.g. timer boxes etc.	ExtClkPortEx: ExtClkPort = PFI3 (default = PFI2)
Digital output port The digital output port is used for triggering of other equipment by setting the signal high during a CTA measurement. The default port will be the first available on the device.	DigLineEx: DigLine = port1/line0 (default = first available digital output port)
Task If you use MAX to create a NI-DAQmx Task and name it "CTATask" by default, the software will find this. It will run the Task instead of a specific DAQmx device. Please refer to the MAX documentation on how to create and use NI-DAQmx Tasks.	DAQmxTaskEx: DAQmxTask = MyTask (default = CTATask)

7.4 Global Export

7.4.1 Data Extraction from Binary Data Files

Binary Export Format

The structure of the binary export file depends of the selections made during the export. The total length of the file depends by the number of samples and number of selected positions in the current group. Also the data conversion format can change the size of the file: it will be smaller when selecting raw integer formats, and larger when selecting converted float formats. Finally the file length depends on the number of selected probes. The probe information is added to the file in order to allow the user to perform additional calculations on the data if necessary.

The structure of the file can be illustrated as follows:

Figure 1
The file and user headers are placed in the beginning of the file, followed by one or more probe information headers. Before each data block representing one position there is a data header describing the size and format of the data.

Again, the number of probe information headers and the number of positions will depend on the selections made during the export.

Because of the flexible structure of the export file the offsets to the different data values will change. The table below explains how to extract the different data values and the data itself from the file.

Data Extraction, Table 1

Name	Offset	Length	Description
File Header			
Version	0	2 (unsigned short)	Version number of the file header; starting at 1000.
NumProbes	2	1 (unsigned char)	Number of probes included in the export.
NumTravPositions	4	4 (unsigned long)	Number of traverse positions.
GroupIndex	8	2 (unsigned short)	Group index.
DatabaseName	10	128 (128 x char)	Name of the StreamWare data-base and the project.
ProjectName	138	16 (16 x char)	

RawIdDate	154	20 (20 x char)	Date, time, and name of the raw data event.
RawIdTime	174	20 (20 x char)	
RawIdName	194	50 (50 x char)	
SizeFileHeader	244	2 (unsigned short)	Size in bytes of the file header 372, user header 290, probe information header 550 and data header 78.
SizeUserHeader	246	2 (unsigned short)	
SizeProbeInfoHeader	248	2 (unsigned short)	
SizeDataHeader	250	2 (unsigned short)	
DeviceDescription	252	40 (40 x char)	Name of the A/D device device used for the data acquisition.
LocalVarNameA	292	10 (10 x char)	Name and unit declaration for the five extra user definable local variables.
LocalVarNameB	302	10 (10 x char)	
LocalVarNameC	312	10 (10 x char)	
LocalVarNameD	322	10 (10 x char)	
LocalVarNameE	332	10 (10 x char)	
LocalVarUnitA	342	6 (6 x char)	
LocalVarUnitB	348	6 (6 x char)	
LocalVarUnitC	354	6 (6 x char)	
LocalVarUnitD	360	6 (6 x char)	
LocalVarUnitE	366	6 (6 x char)	
User Header			
User	372	30 (30 x char)	(Optional) Name of the user or creator; default name “StreamWare (tm)”.
AccountNumber	402	4 (unsigned long)	(Optional) Project id number.
Information	406	256 (256 x char)	(Optional) Extra user comments.
Probe Information Header			
Version	662*	2 (unsigned short)	Version number of the probe information header; starting at 1000.

ProbeName	664	10 (10 x char)	Name of the probe.
ProbeType	674	2 (unsigned short)	Type of the probe. 1, 2 and 3 = Wire/Fiber probe, 4 and 5 = Film probe, 7 = Miscellaneous probe, or 8 = Temperature probe.
NumChannels	676	1 (unsigned char)	Number of channels used by the probe.
BridgeConnected#1	677	1 (unsigned char)	Flag indicating if the bridges are connected. 0 = No, or 1 = Yes.
BridgeConnected#2	678	1 (unsigned char)	
BridgeConnected#3	679	1 (unsigned char)	
Shape	680	1 (unsigned char)	Shape of the probe. 0 = Film, or 1 = Wire.
BridgeRatio	681	1 (unsigned char)	Bridge ratio. 0 = 1:20 (20 Ohms), 1 = 1:20 (10 Ohms), or 2 = 1:1.
CTAFilter	682	1 (unsigned char)	Filter
CTAGain	683	1 (unsigned char)	Gain
CableComp	684	1 (unsigned char)	Cable compensation
OverHeat#1	686	4 (float)	Overheat ratio.
OverHeat#2	690	4 (float)	
OverHeat#3	694	4 (float)	
ProbeResistance#1	698	4 (float)	Probe resistance in Ohms.
ProbeResistance#2	702	4 (float)	
ProbeResistance#3	706	4 (float)	
CableResistance#1	710	4 (float)	Cable + support resistance in Ohms.
CableResistance#2	714	4 (float)	
CableResistance#3	718	4 (float)	
SensorColdResistance#1	722	4 (float)	Sensor cold resistance in Ohms.

SensorColdResistance#2	726	4 (float)	
SensorColdResistance#3	730	4 (float)	
DecadeResistance#1	734	4 (float)	Decade resistance in Ohms.
DecadeResistance#2	738	4 (float)	
DecadeResistance#3	742	4 (float)	
CableLength	746	4 (float)	Length of the cable.
RefTempR#1	750	4 (float)	Reference temperature in °C when probe resistance was measured.
RefTempR#2	754	4 (float)	
RefTempR#3	758	4 (float)	
BridgeVolt#1	762	4 (float)	Bridge top voltage.
BridgeVolt#2	766	4 (float)	
BridgeVolt#3	770	4 (float)	
RefTempV#1	774	4 (float)	Reference temperature in °C when bridge top voltage was measured.
RefTempV#2	778	4 (float)	
RefTempV#3	782	4 (float)	
SensorTempCoeff#1	786	4 (float)	Corrected sensor temperature coefficients.
SensorTempCoeff#2	790	4 (float)	
SensorTempCoeff#3	794	4 (float)	
ConversionLevel	798	1 (unsigned char)	Conversion level description. 0 = Raw voltage (2 bytes short), 1 = Raw voltage (4 bytes float), 2 = Velocity/temp. from calibration, 3 = Velocity in wire coordinates, 4 = Velocity in probe coordinates, or 5 = Velocity in laboratory coordinates.
TempCorrection	800	2 (unsigned short)	Flag indication if temperature correction was used. 0 = No, or 1 = Yes.

Algorithm	802	1 (unsigned char)	0 = Polynomium 1 = Power Law 2 = Lookup Table 3 = Logarithmic-Square 4 = Steinhart-Hart 5 = Logarithmic-Polynomial
Case: Algorithm = 0			Polynomial: T=C0 + C1*E + C2*E^2 + C3*E^3 + C4*E^4 + C5*E^5
Order#1	804	2 (short)	Order of the polynomial.
MinU#1	806	4 (float)	Minimum, maximum, and next maximum.
MinE#1	810	4 (float)	
MaxU#1	814	4 (float)	
MaxE#1	818	4 (float)	
Max_1U#1	822	4 (float)	
Max_1E#1	826	4 (float)	
C0#1	830	4 (float)	
C1#1	834	4 (float)	
C2#1	838	4 (float)	
C3#1	842	4 (float)	
C4#1	846	4 (float)	
C5#1	850	4 (float)	
Order#2	854	2 (short)	
MinU#2	856	4 (float)	
MinE#2	860	4 (float)	
MaxU#2	864	4 (float)	
MaxE#2	868	4 (float)	
Max_1U#2	872	4 (float)	
Max_1E#2	876	4 (float)	
C0#2	880	4 (float)	
C1#2	884	4 (float)	
C2#2	888	4 (float)	

C3#2	892	4 (float)	
C4#2	896	4 (float)	
C5#2	900	4 (float)	
Order#3	904	2 (short)	
MinU#3	906	4 (float)	
MinE#3	910	4 (float)	
MaxU#3	914	4 (float)	
MaxE#3	918	4 (float)	
Max_1U#3	922	4 (float)	
Max_1E#3	926	4 (float)	
C0#3	930	4 (float)	
C1#3	934	4 (float)	
C2#3	938	4 (float)	
C3#3	942	4 (float)	
C4#3	946	4 (float)	
C5#3	950	4 (float)	
Order#4	954	2 (short)	Not used.
MinU#4	956	4 (float)	
MinE#4	960	4 (float)	
MaxU#4	964	4 (float)	
MaxE#4	968	4 (float)	
Max_1U#4	972	4 (float)	
Max_1E#4	976	4 (float)	
C0#4	980	4 (float)	
C1#4	984	4 (float)	
C2#4	988	4 (float)	
C3#4	992	4 (float)	
C4#4	996	4 (float)	
C5#4	1000	4 (float)	
Case: Algorithm = 1			Power law:

			T=(E^2 - A)/B)^(1/n)
A#1	804	4 (float)	
B#1	808	4 (float)	
N#1	812	4 (float)	
MinU#1	816	4 (float)	Minimum and maximum cal-ibration points.
MinE#1	820	4 (float)	
MaxU#1	824	4 (float)	
MaxE#1	828	4 (float)	
A#2	832	4 (float)	
B#2	836	4 (float)	
N#2	840	4 (float)	
MinU#2	844	4 (float)	
MinE#2	848	4 (float)	
MaxU#2	852	4 (float)	
MaxE#2	856	4 (float)	
A#3	860	4 (float)	
B#3	864	4 (float)	
N#3	868	4 (float)	
MinU#3	872	4 (float)	
MinE#3	876	4 (float)	
MaxU#3	880	4 (float)	
MaxE#3	884	4 (float)	
A#4	888	4 (float)	Not used
B#4	892	4 (float)	
N#4	896	4 (float)	
MinU#4	900	4 (float)	
MinE#4	904	4 (float)	
MaxU#4	908	4 (float)	
MaxE#4	912	4 (float)	

Case: Algorithm = 2			Lookup table.
Size#1	804	2 (short)	Size of lookup table (largest 4096 elements - 12 bit ADC)
MinU#1	806	4 (float)	Minimum and maximum cal-ibration points (entries in table)
MinE#1	810	4 (float)	
MaxU#1	814	4 (float)	
MaxE#1	818	4 (float)	
Delta#1	822	4 (float)	Step between consecutive entries (equals (MaxE - MinE) / Size)
Size#2	826	2 (short)	
MinU#2	828	4 (float)	
MinE#2	832	4 (float)	
MaxU#2	836	4 (float)	
MaxE#2	840	4 (float)	
Delta#2	844	4 (float)	
Size#3	848	2 (short)	
MinU#3	850	4 (float)	
MinE#3	854	4 (float)	
MaxU#3	858	4 (float)	
MaxE#3	862	4 (float)	
Delta#3	866	4 (float)	
Size#4	870	2 (short)	
MinU#4	872	4 (float)	Not used
MinE#4	876	4 (float)	
MaxU#4	880	4 (float)	
MaxE#4	884	4 (float)	
Delta#4	888	4 (float)	
Case: Algorithm = 3			Logarithmic-Square: T = (exp(sqrt(A + B*E)) - 1)/G
A#1	804	4 (float)	
B#1	808	4 (float)	

G#1	812	4 (float)	
A#2	816	4 (float)	
B#2	820	4 (float)	
G#2	824	4 (float)	
A#3	828	4 (float)	
B#3	832	4 (float)	
G#3	836	4 (float)	
A#4	840	4 (float)	Not used
B#4	844	4 (float)	
G#4	848	4 (float)	
Case: Algorithm = 4			Steinhart-Hart: T = 1/(Ash + Bsh*log(E) + Csh*log^3(E)) - 273.15
Ash#1	804	4 (float)	
Bsh#1	808	4 (float)	
Csh#1	812	4 (float)	
Ash#2	816	4 (float)	
Bsh#2	820	4 (float)	
Csh#2	824	4 (float)	
Ash#3	828	4 (float)	
Bsh#3	832	4 (float)	
Csh#3	836	4 (float)	
Ash#4	840	4 (float)	Not used
Bsh#5	844	4 (float)	
Csh#5	848	4 (float)	
Case: Algorithm = 5			Logarithmic-polynomial: T = C0 + C1*log(E) + C2*log^2(E) + C3*log^3(E)
C0#1	804	4 (float)	
C1#1	808	4 (float)	
C2#1	812	4 (float)	
C3#1	816	4 (float)	

C0#2	820	4 (float)	
C1#2	824	4 (float)	
C2#2	828	4 (float)	
C3#2	832	4 (float)	
C0#3	836	4 (float)	
C1#3	840	4 (float)	
C2#3	844	4 (float)	
C3#3	848	4 (float)	
C0#4	852	4 (float)	Not used
C1#4	856	4 (float)	
C2#4	860	4 (float)	
C3#4	864	4 (float)	
Probe Information Header (continued)			
CalTemp	1004	4 (float)	
AngCalAK#1	1008	4 (float)	Angular calibration coefficient’s.Used for constructing decomposition matrix. Depending on whether the probe is 2D or 3D these coefficients might denote pitch or yaw factors.
AngCalAK#2	1012	4 (float)	
AngCalAK#3	1016	4 (float)	
AngCalBK#1	1020	4 (float)	
AngCalBK#2	1024	4 (float)	
AngCalBK#3	1028	4 (float)	
CoordTransform11	1032	4 (float)	Coordinate matrix.Transformation matrix, from wire to probe coordinates.
CoordTransform12	1036	4 (float)	
CoordTransform13	1040	4 (float)	
CoordTransform14	1044	4 (float)	
CoordTransform21	1048	4 (float)	

CoordTransform22	1052	4 (float)	
CoordTransform23	1056	4 (float)	
CoordTransform24	1060	4 (float)	
CoordTransform31	1064	4 (float)	
CoordTransform32	1068	4 (float)	
CoordTransform33	1072	4 (float)	
CoordTransform34	1076	4 (float)	
CoordTransform41	1080	4 (float)	
CoordTransform42	1084	4 (float)	
CoordTransform43	1088	4 (float)	
CoordTransform44	1092	4 (float)	
ProbeTransformRoll	1096	4 (float)	
ProbeTransformPitch	1100	4 (float)	
ProbeTransformR11	1104	4 (float)	Roll and pitch coef-ficients.Applied roll and pitch angles and matrices formed by them. For conversion from probe to laboratory coordinates.
ProbeTransformR12	1108	4 (float)	
ProbeTransformR13	1112	4 (float)	
ProbeTransformR21	1116	4 (float)	
ProbeTransformR22	1120	4 (float)	
ProbeTransformR23	1124	4 (float)	
ProbeTransformR31	1128	4 (float)	
ProbeTransformR32	1132	4 (float)	
ProbeTransformR33	1136	4 (float)	
ProbeTransformP11	1140	4 (float)	
ProbeTransformP12	1144	4 (float)	
ProbeTransformP13	1148	4 (float)	
ProbeTransformP21	1152	4 (float)	
ProbeTransformP22	1156	4 (float)	
ProbeTransformP23	1160	4 (float)	

ProbeTransformP31	1164	4 (float)	
ProbeTransformP32	1168	4 (float)	
ProbeTransformP33	1172	4 (float)	
SignalCondGain#1	1176	4 (float)	Amplification of the input signal.
SignalCondGain#2	1180	4 (float)	
SignalCondGain#3	1184	4 (float)	
SignalCondOffset#1	1188	4 (float)	Offset of the input signal.
SignalCondOffset#2	1192	4 (float)	
SignalCondOffset#3	1196	4 (float)	
SignalCondHighPass#1	1200	1 (unsigned char)	Bandwidth of high-pass filter. 0 = bypassed, 1 = 10 Hz, or 2 = 100 Hz.
SignalCondHighPass#2	1201	1 (unsigned char)	
SignalCondHighPass#3	1202	1 (unsigned char)	
SignalCondLowPass#1	1203	1 (unsigned char)	Bandwidth of low-pass filter. 0 = 300 Hz, 1 = 1 kHz, 2 = 3 kHz, 3 = 10 kHz, 4 = 30 kHz, 5 = 100 kHz, or 6 = 300 kHz.
SignalCondLowPass#2	1204	1 (unsigned char)	
SignalCondLowPass#3	1205	1 (unsigned char)	
ACorDC#1	1206	2 (unsigned short)	true (1) = AC, or false (0) = DC.
ACorDC#2	1208	2 (unsigned short)	
ACorDC#3	1210	2 (unsigned short)	
Data Header			
Version	1212**	2 (unsigned short)	Version number of the data header; starting at 1000.
Date	1214	10 (10 x char)	Date when the data was acquired.
Time	1224	10 (10 x char)	Time when the data was acquired.

SampleFrequency	1234	4 (float)	Sample frequency used when the data was acquired.
NumSamples	1238	4 (unsigned long)	Number of data samples for each column.
NumColumns	1242	2 (unsigned short)	Number of data columns corresponding to number of active channels.
PositionIndex	1244	4 (unsigned long)	Traverse position index.
BlockIndex	1248	2 (unsigned short)	Data block index.
PositionX	1250	4 (float)	Traverse position in three dimensions and angle.
PositionY	1254	4 (float)	
PositionZ	1258	4 (float)	
PositionA	1262	4 (float)	
LocalVarA	1266	4 (float)	Values of user variables.
LocalVarB	1270	4 (float)	
LocalVarC	1274	4 (float)	
LocalVarD	1278	4 (float)	
LocalVarE	1282	4 (float)	
DataFormat	1286	1 (unsigned char)	0 = integer format (2 bytes short), or1 = floating point format (4 bytes float)
DataType	1287	1 (unsigned char)	0 = application raw data type (default), or1 = auxiliary data type (not yet supported)
DataStatus	1288	1 (unsigned char)	0 = no data errors, or >0 = error

* Offset for one probe.
** Offset for first data header for one probe .

The offset positions in this table only apply to one probe header and the first data header. When more that one probe is included in the export the offset values in the probe header can be described as: 662 bytes + NumProbes*550 bytes. E.g. if two probes are included the start offset value for the first data block will be 1762 bytes instead of 1212 bytes, see next section.

7.4.2 Accessing the Data

Before accessing the data it is necessary to read the information in the corresponding data header. From the data header we find the number of samples, the number of columns of data, and the format in which the data is stored. Data is stored in either short 2 bytes or float 4 bytes format dependent on the conversion level selected.

In the following examples we uses five variables for reading data:

NumProbes	Number of probes selected in the export equal to the number of probe information headers in the file.
NumTravPositions	Number of traverse positions saved in the export file, this equals the number of data blocks in the export file.
NumSamples	Number of data samples for each position.
NumColumns	Number of data columns for each position.
DataFormat	The format in which the data is saved.

Using the defined variables the total size of each data block can be determined by the equation:

Eq. 1

If DataFormat equals 0 (short integer format):
NumSamples*NumColumns*2 bytes,
or if DataFormat equals 1 (floating point format):
NumSamples*NumColumns*4 bytes

For accessing data we must bypass the file and user headers and the probe information header(s). The number of probes is read in the NumProbes variable in the file header, and the total size of each probe information header is 550 bytes; therefor the first data header begins at position:

Eq. 2a

662 bytes + NumProbes*550 bytes

The size of each data header is 77 bytes, therefor the first actual data block can be accessed at the position:

Eq. 2b

662 bytes + 77 bytes + NumProbes*550 bytes

The number of consecutive data blocks is equal to the variable NumTravPositions. Each data header must be read and the size of the corresponding data block determined. The next data immediately follows the data header and the data block.

7.4.3 Examples

Example 1

This example describes how to find the two important variables; the number of probes NumProbes, and the number of positions NumTravPositions. Both these variables are placed in the file header in the beginning of the export file. From See "Global Export" on page 164 we find, that the number of probes can be read as an “unsigned char” at the offset position 2 bytes from the beginning of the file, and the number of positions saved in the export file can be read as a unsigned long at the offset position 4 bytes from the beginning of the file.

Pseudo code:

```
unsigned char numProbes;
unsigned long numTravPositions;
file.Open("exportfile.bin");
file.Seek(2,from_beginning);
file.Read(&numProbes);
file.Seek(4,from_beginning);
file.Read(&numTravPositions);
file.Close();
```

Example 2

This example describes how to access data in the data blocks. Each data block begins with a data header describing the data. Using the information about the number of traverse positions, see example 1, we know the number of data blocks. To read down the data blocks each data header must be read. Reading the number of samples, number of columns and the data format for the first data header is described by this pseudo code example:

Pseudo code:

```
unsigned char numProbes;
file.Open("exportfile.bin");
file.Seek(2,from_beginning);
file.Read(&numProbes);
file.Seek(662+numProbes*550,from_beginning);

see next*

see next**

file.Close();
```

The start position of the first data header by bypassing the file and user headers plus one or more probe information header, see Eq. 2a. The values in the data header can then be found as:

Pseudo code*:

```
unsigned long numSamples;
unsigned short numColumns;
unsigned char dataFormat;
file.Seek(662+numProbes*550+26,from_beginning);
file.Read(&numSamples);
file.Seek(662+numProbes*550+30,from_beginning);
file.Read(&numColumns);
file.Seek(662+numProbes*550+74,from_beginning);
file.Read(&dataFormat);
```

Now we can determine the size dataSize of the first data block as in Eq. 1. Then we need to read through all data blocks if numTravPositions is larger than one.

Pseudo code:**

```
unsigned long numSamples;
unsigned short numColumns;
unsigned char dataFormat;
file.Seek(662+numProbes*550,from_beginning);
for (travPos=1 to numTravPositions)
do
// find dataSize
file.Seek(77,from_lastpos);
// read data
file.Seek(dataSize[travPos],from_lastpos);
end
```

Remember when data is saved in integer format (raw velocities in integers) no time stamp is included, which means, that the first column is the first channel from the first probe in contrast to all the float formats where the first column always is the time stamps.

Sometimes the access to the data blocks can be simplified by assuming that the number of data columns, number of data samples, and data format are the same for all data blocks. This means, that only the first data header needs to be accessed.