HydroMet

User Manual

AquisNet DMO

KISTERS

Empowering decisions of tomorrow

Internet: www.kisters.eu
E-Mail: info@kisters.de

Author: KISTERS
Date of current print: 24/05/2024
Current software version: 4.6.2

Table of contents

1 Introduction

AquisNet DMO (Data Monitoring Office) is the validation tool for the professional data validators in an AquisNet system. Workstation computers integrated into the AquisNet system are used to process the data from the acquisition systems. The DMO serves the user for the professional validation of incoming measurement data and, in addition to their validation and commenting on input data, also allows the subsequent tracking of executed validation steps and, if necessary, their reversal.

The tasks of the DMO include the following areas:

- Display time series data in graphs and tables
- Interactive data validation in graph and table form
- Extension of time series and station information by comments for time ranges.
- Follow-up of validation steps (transactions) and, if necessary, their reversal
- Configuration of error codes and status code groups
- Configuration of plausibility checks for time series

2 The desktop of the data collection center DMO (Data Management Office)

In this chapter you will get to know the user interface of the data acquisition center DMO.

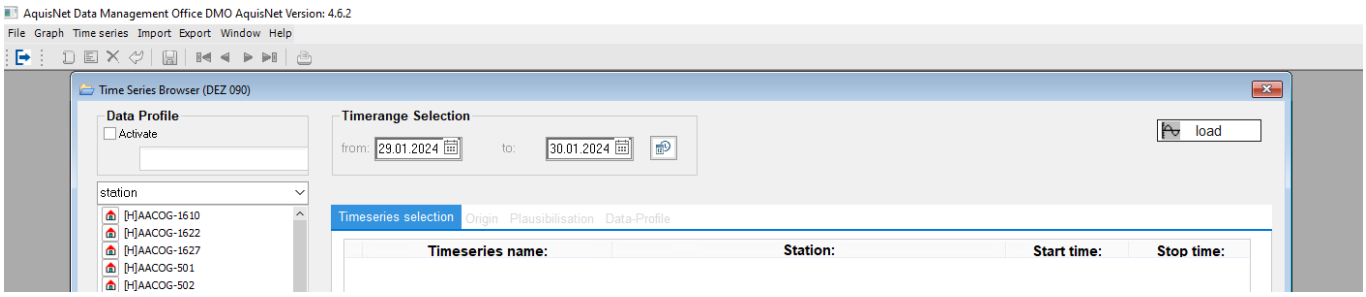
3 Program start

To start the DMO on your workstation, click on the AquisNet DMO icon on your desktop.

Registration

Enter your user name and password in the login window and confirm with the [**LOGIN**] button.

The main menu of the data management office DMO opens with the time series selection open (time series browser 12).



4 Displaying time series: The time series browser of the data collection center DMO

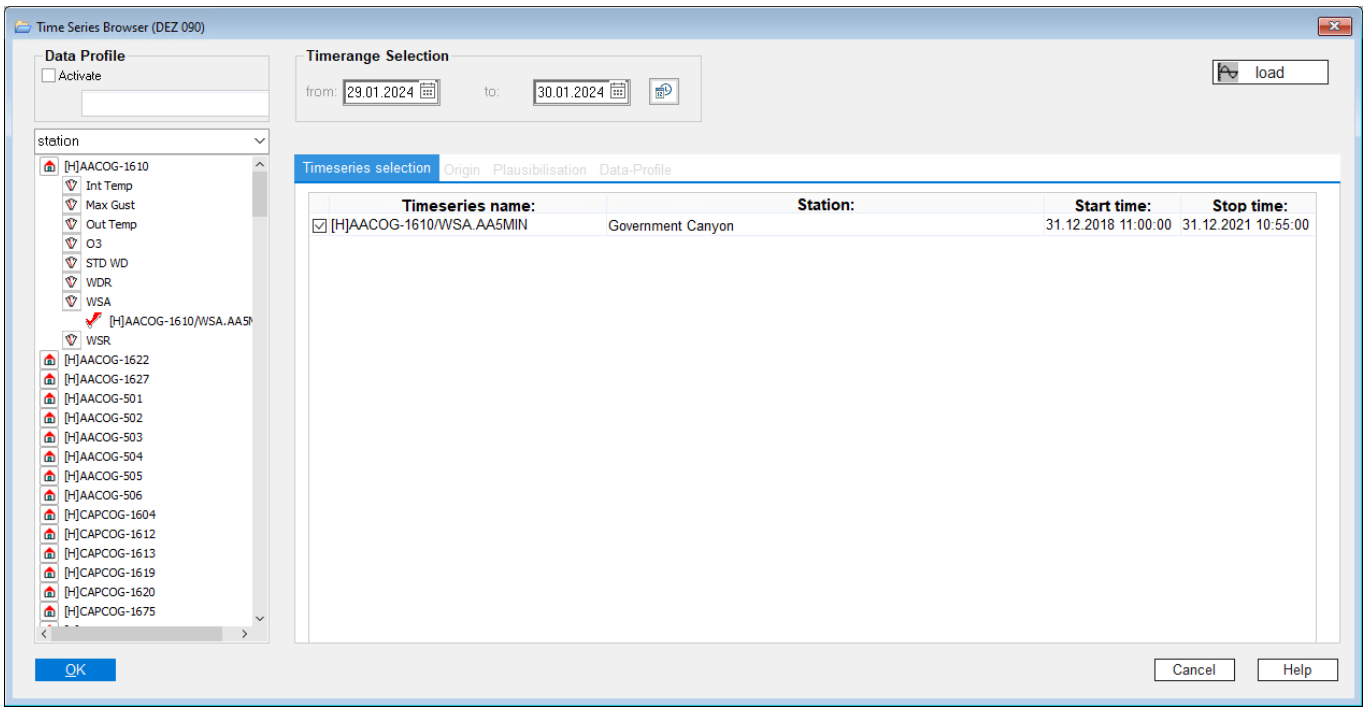
The work options related to the time series browser are divided into the following areas:

- Launch time series browser 11
- The tree view of the time series browser 12

4.1 Launch time series browser

AquisNet DMO opens the time series browser window automatically when the program starts.

If you have closed the window manually, you can open it again via the main menu by selecting **TIME SERIES** > **TIME SERIES BROWSER**.



Use the time series browser to select the time series to be viewed or edited. You also set the desired time range and easily obtain graphical and tabular display and editing options for your measured data.

The browser window is divided into four parts:

- Data profile 11
- Time range selection 12
- Tree view with time series 12
- Time series list 12

4.1.1 Data profile

Here you can select a data profile that supports the selection as a filter for the time series and check its box. Entries not contained in the data profile (stations, time series ...) are thus hidden from the tree view and thus the clarity is increased.

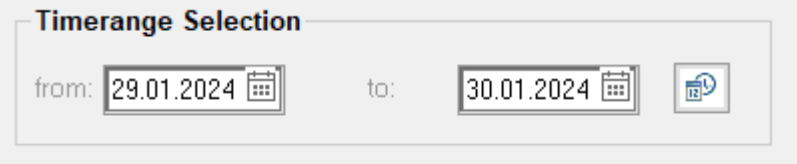
For more information on data profiles, see the Data profiles 18 tutorial.

4.1.2 Time range selection

Set the desired time range here. This time range is used when displaying the measured data in graph and table, but also for time filtering of the tree structure. There are six different modes available between which you can switch with the toggle button on the right.

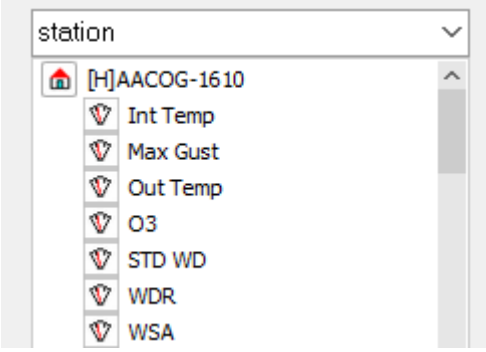
The following modes for selecting a time range are possible:

- Entering a start and end date by input or calendar
- Entering a number of days/weeks/months/years until an end date
- Entering a number of days/weeks/months/years until the current date
- Entering a number of days/weeks/months/years from a start date
- Selection of a calendar year (01.01. - 31.12.)
- Selection of a meteorological year (01.10. - 30.09.)



4.1.3 Tree view with time series

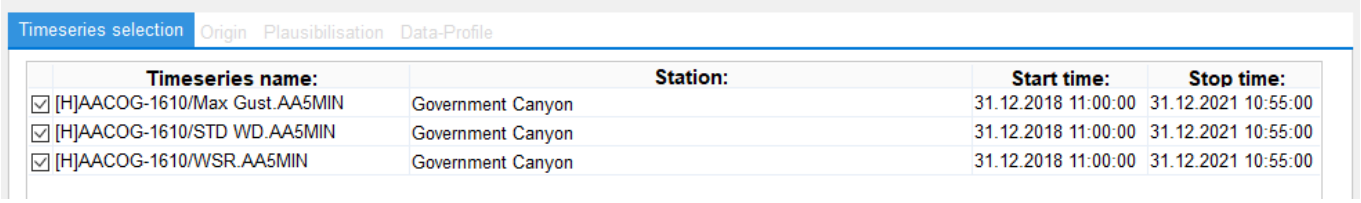
In the left part of the dialog, a tree view is available in which you can select the time series. This tree view can be changed at runtime by different entry points so that the tree is built up starting from a station or a parameter, for example, and the desired time series can thus be selected as quickly and easily as possible. Above the tree view is a selection of these entry points using a drop-down list box.



A detailed description of the different entry points can be found in the chapter about the time series browser tree view 12.

4.1.4 Time series list

The selected time series are listed in the right part of the window. The time series can still be deactivated here via a check box to include in the new display only some of the time series shown in the list.

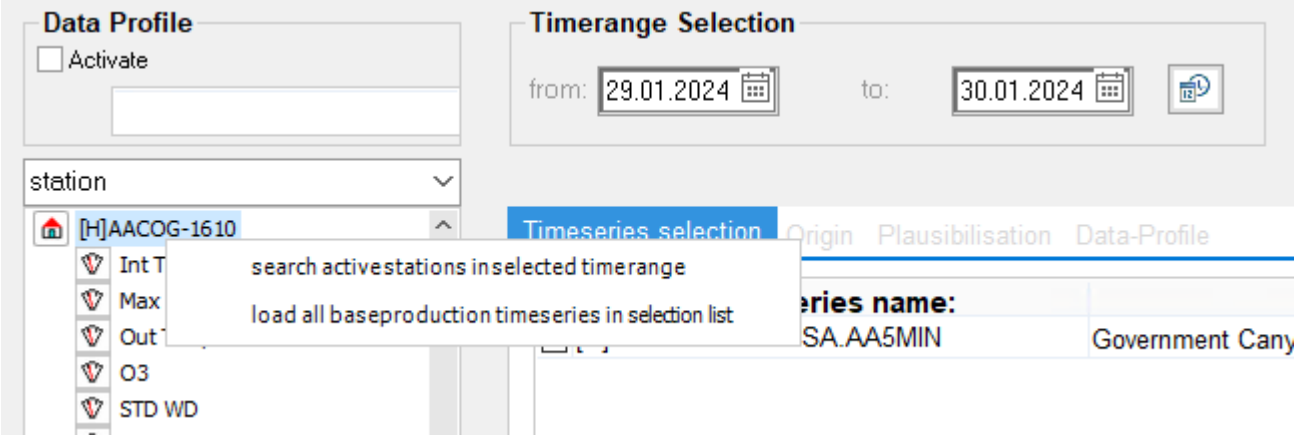


4.2 The tree view of the time series browser

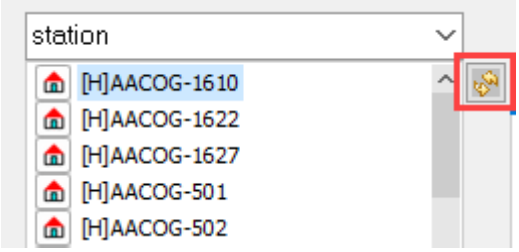
For more comfortable navigation in the time series browser tree, you can select different entry points in the structure of the time series browser tree. With the help of these entry points, it is easier to select related time series from the current evaluation context, e.g. if you want to evaluate several time series of a station or several time series of a parameter. Simply clicking on a time series in the tree selects it and adds it to the selection list on the right, if it is not already included there.

Depending on the selection of the structure or entry point, the elements are arranged in a structured way in the time series browser tree.

In addition, you can apply a temporal filter to the selection tree that uses the set period in the time range selection. Here, for example, only stations are displayed that also have measured data in the selected period. You can reach this filter option via the respective context menu (right mouse button).



If you want to cancel this temporal filtering, you can do so by clicking the [**CANCEL TIME FILTER**] button. This appears as soon as you apply a time filter function to the tree.

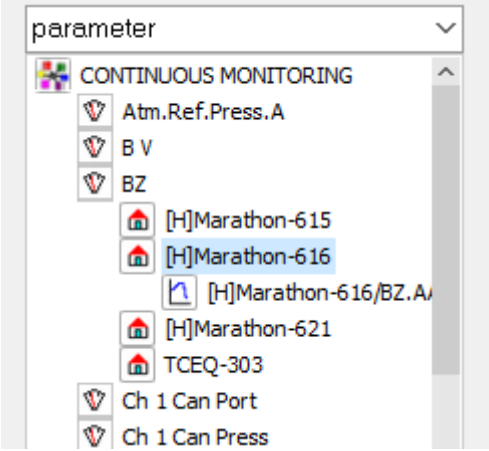


You can also make a direct selection of time series via the context menu. You will find more details in the chapter Entry points of the tree view 13.

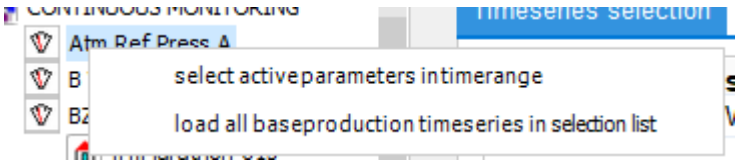
4.2.1 Entry points of the tree view

Entry point parameter

Via the parameters in alphabetical order, you get to the stations that record the preselected parameter and from there to the time series. Here again, the context menu offers the possibility to select time-filtered all basic production time series of the system of these parameters or to display only those parameters that also have data in the selected time range.

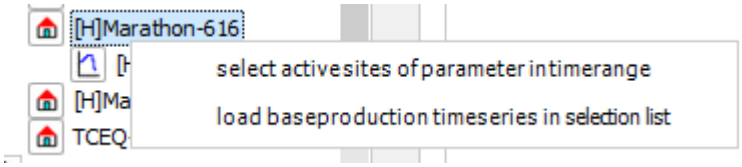


Right-click here on a parameter to find the following context menu:



- Option 1 leaves only parameters in the tree that have data in the selected time range.
- Option 2 selects all basic production time series of the parameter into the selection list on the right that have data in the selected time range.

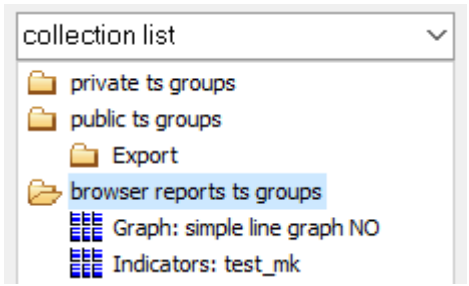
Right-click on a station to get the following context menu:



- Option 1 leaves only stations below the parameter in the tree that have data of the parameter in the selected time range.
- Option 2 selects the basic production time series of the parameter of this station into the selection list on the right, if it has data in the selected time range.

Entry point Collection list

You can access the collection lists and their time series via the collection lists, grouped by private, public and browser groups in a folder structure. Filters in the context menu are not provided here.

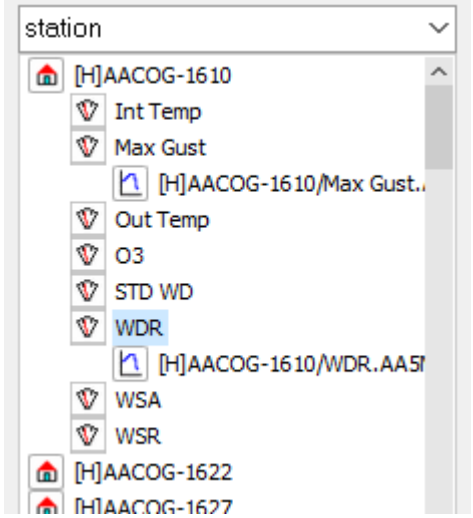


Double-clicking on the collection list loads the time series of the folder directly into the selection on the right.

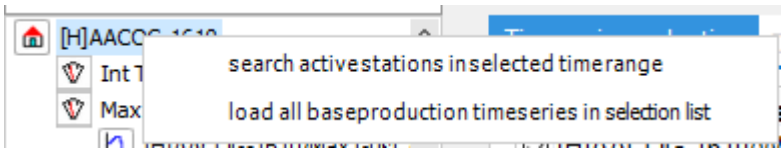
For collections lists, see also the tutorial Collection lists 17.

Entry point Station

Here you get from the stations, sorted alphabetically, via the parameters to the time series that are created in a station. The additional filter and selection options are already described above.

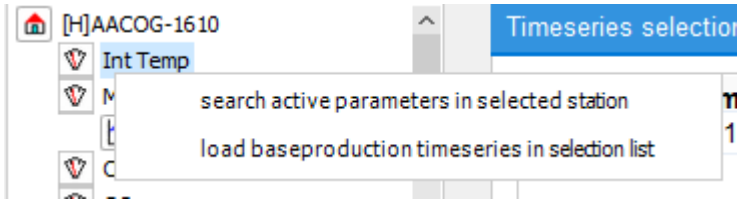


Right-click here on a station to find the following context menu:



- Option 1 leaves only stations in the tree that have data in the selected time range.
- Option 2 selects all basic production time series of the station into the selection list on the right that have data in the selected time range.

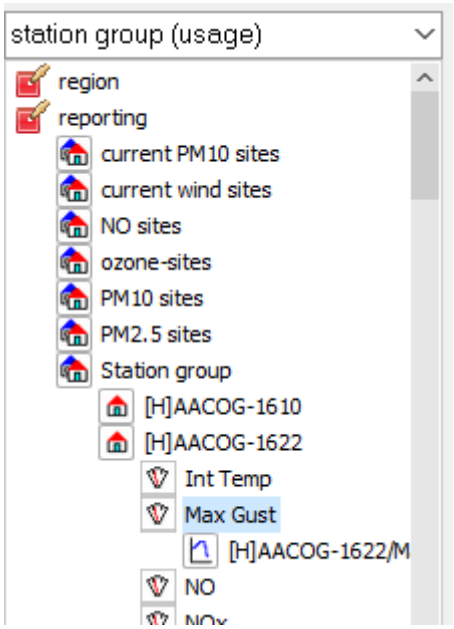
Right-click on the parameter to get the following context menu:



- Option 2 leaves only parameters below the station in the tree that have data of the station in the selected time range.
- Option 2 selects the basic production time series of the parameter of this station into the selection list on the right, if it has data in the selected time range.

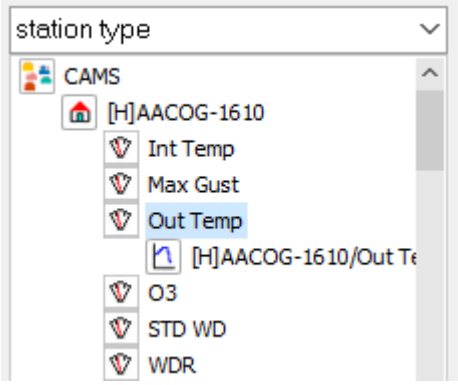
Station group (usage)

Via the station groups sorted by purpose, you can access the stations and their time series. The filters on a station selected here correspond to those of the **STATION** entry point.



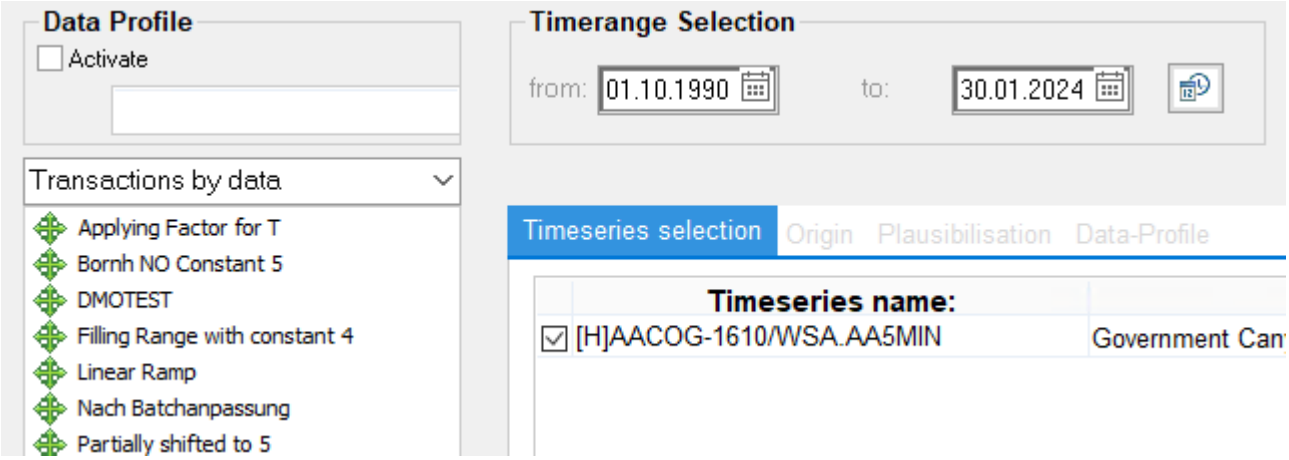
Entry point Station type

If several station types are defined in your system, they will be used as anchors for the tree. The subordinate structure corresponds to that of the **STATIONS** entry point. Filters via the context menu are not provided here.

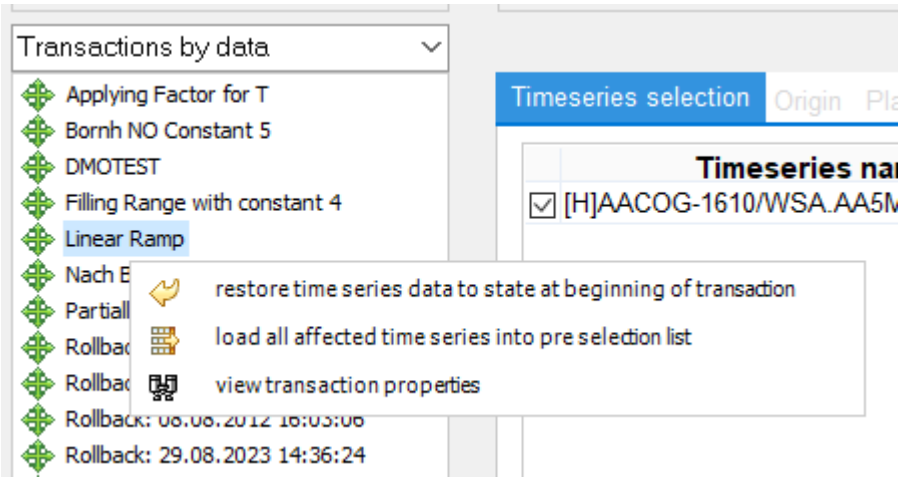


Entry point Transactions by data

This entry point displays the transactions in the selection tree on the left, filtered by time. Transactions that have changed data in the set time range are included in the selection list. This entry point is used to determine data changes in a certain time range (changed data from the last month or similar). In the level below the transactions are the affected time series.

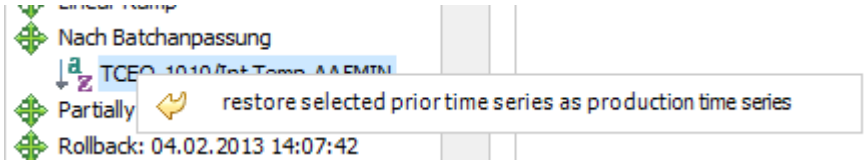


Right-click here on a transaction to find the following context menu:



- Option 1 resets all affected time series to the data state before the selected transaction was executed.
- Option 2 loads all time series involved in the transaction into the selection list.
- Option 3 opens the detailed display of the transaction.

Double-click the transaction to expand the transaction tree by the time series involved in the transaction.



- Option 1 resets all selected time series to the data state before the selected transaction was executed.

Entry point Transactions by execution

This entry point shows the transactions filtered in the selection tree on the left. Transactions that were carried out in the set time range are included in the selection list. This entry point is used to determine data changes in a certain processing period (changes made in the past week or similar). The tree structure and context menus correspond to the description in the previous chapter.

For more information on transactions, see the tutorial Edit time series: Transactions in the time series browser of the data collection center DMO 33.

4.2.2 Collection lists

Collection lists allow you to give a name to a time series selection you have made and store it in a folder structure. This selection can be restored at any time using the collection list function, so that you do not have to make repeated identical selections manually.

- Call up the collection list 17
- Rename existing folder or collection list 17
- Creating new collection list 18
- Deleting a folder or a collection list 18

4.2.2.1 Call up the collection list

Select the item **COLLECTION LIST** as the entry point for the time series tree and select the desired collection list from the folder structure that appears by double-clicking.

This is marked with a red arrow.

The time series of the collection list are loaded directly into the selection window on the right. If you already have selected time series in the list, select whether existing time series selections should be overwritten or extended.

4.2.2.2 Rename existing folder or collection list

To do this, click on the entry in the time series browser tree that you want to change.

Open the context menu of the tree entry by activating the right mouse button.

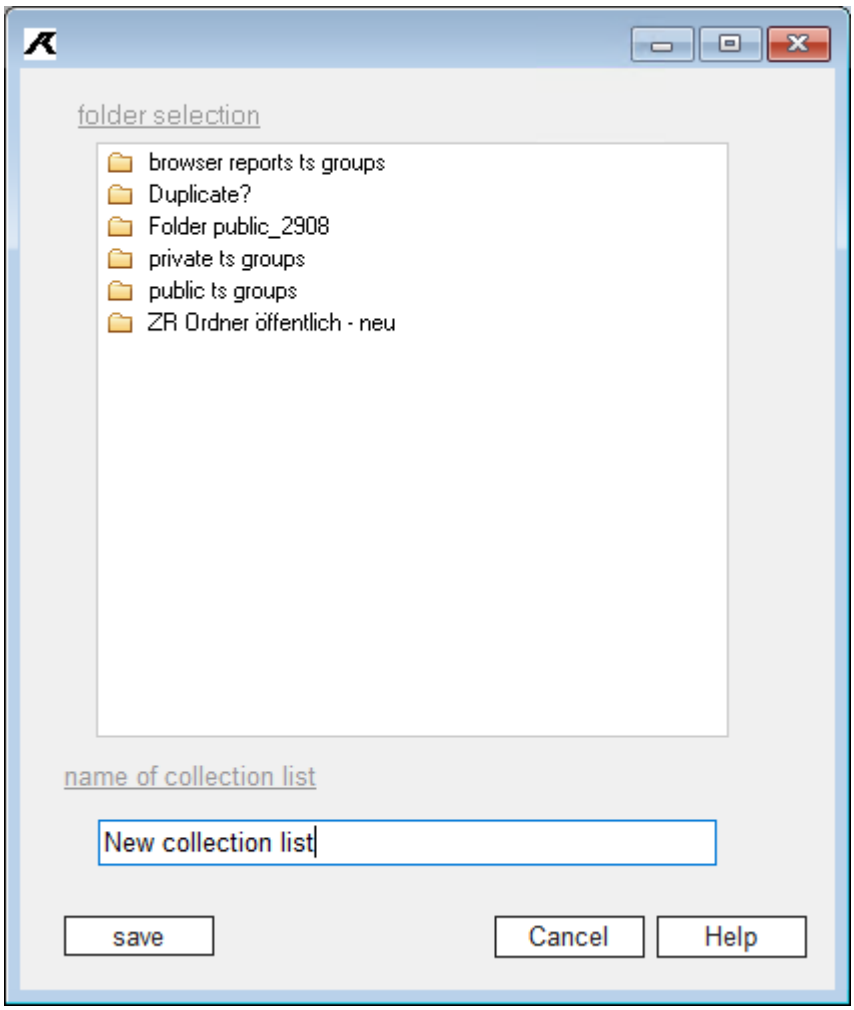
Now select the entry **RENAME FOLDER/COLLECTION LIST**.

Change the entry directly in the name field within the tree structure.

By activating the [**RETURN**] key, you confirm your entry.

4.2.2.3 Creating new collection list

Select the desired time series so that they can be found in the time series selection. In the context menu of the time series selection you will find the entry [**MANAGE TIME SERIES SELECTION AS COLLECTION LIST**]. Select this entry.



In the above dialog, select the destination folder and enter the desired name of the new collection list. The [**SAVE**] button then writes the corresponding selection into the database.

4.2.2.4 Deleting a folder or a collection list

Note: You can only delete a folder if it no longer contains another (sub)folder. Should you nevertheless have made such a selection, you will be informed by an error message.

- Select the desired subfolder and open the context menu with the right mouse button.
- Select the function **DELETE FOLDER/COLLECTION LIST**.
- Now confirm the dialog with [**OK**].

The subfolder is deleted.

4.2.3 Data profiles

The data profiles in the data collection center are a tool to pre-filter the large amount of time series in the system so that evaluations can be carried out more quickly. Thereby you determine the amount of data from which you want to create evaluations via the filter view. With data profiles you can also assign defined visible subsections to users so that individual employees can only view part of the data.

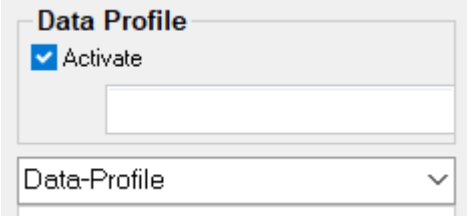
The amount of data filtered by the data profile applies to all selection options (station, station group, parameter, etc.).

To use a data profile, it must have been created in the system beforehand.

Working with data profiles is divided into the following steps:

4.2.3.1 Select existing data profile

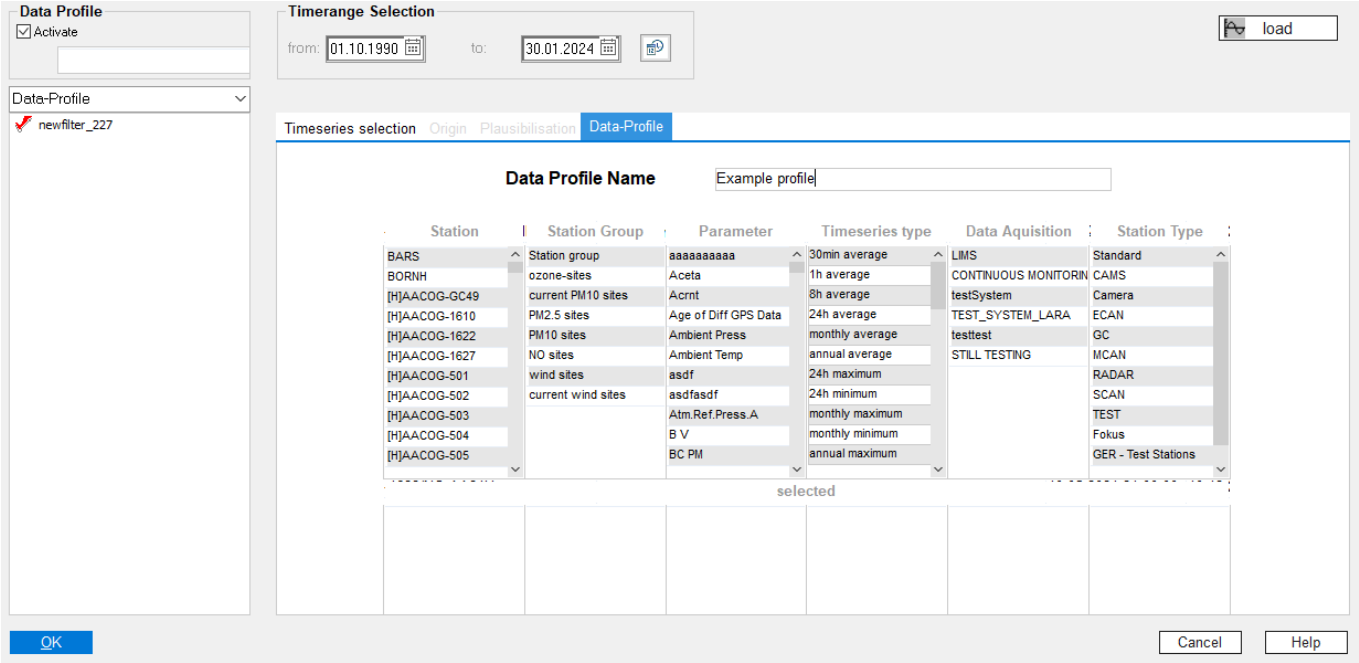
To use a data profile, select the desired profile from the selection list in the browser window.



Then check the box on the left of the selection list. The AquisNet profile is only used after this.

4.2.3.2 View and edit data profiles

To edit a data profile, select the entry point **DATA PROFILES** for the time series tree. The browser tree now shows the existing data profiles:



Call up data profile and view/edit configuration

Double-click to call up the configuration of a data profile for editing, which is then displayed in the main window, see figure above.

- Now transfer the desired objects to the lower selection cells with a mouse click.
  Only these selected objects will be available later when using the filter.
- By clicking on already selected objects, you deselect them.
  Note:
- For example, if you select only **XX** as station, a user with the current time series filter will only have access to time series for station **XX**.
- If you select a station group, you have access to all time series recorded in the stations of this station group with the current time series filter.

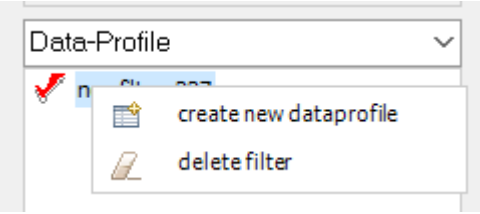
- If you select a parameter, you have access to all time series containing data for this parameter with the current time series filter.
- If you select a station group, the stations currently contained in the selection cell are automatically taken over.
- If you remove a station group from a filter configuration, the station entries are retained. Thus, there is a coupling in the selection, but not in the deselection of station groups!
- Confirm your adjustments with [**SAVE**].

The edits are transferred and taken into account when the filter is selected via the selection list at the top left.

4.2.3.3 Creating a new data profile

To create a new data profile,

- select the **DATA PROFILES** option from the drop-down list below the data profiles selection.
  The browser tree now shows the existing data profiles:

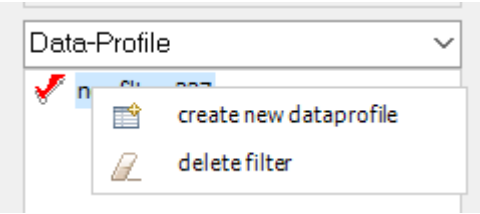


- Select the **CREATING A NEW DATA PROFILE** function in the context menu.
- The empty data profile interface is now called up, ready for the configuration of a new data profile.
- Continue entering a name if necessary.
- The data profile is saved in the system under this name.
- Configure the data profile according to the chapter View and edit data profiles 19.

4.2.3.4 Delete existing data profile

Select the **DATA PROFILES** entry point for the browser tree from data profiles.

- The browser tree now shows the existing data profiles:



- Select an entry and open the context menu.
- Select the **DELETE DATA PROFILE** function.
  If an existing data profile cannot be deleted because it is currently being used, either the **DELETE DATA PROFILE** function is not available for this data profile, or you receive a system message that the deletion process cannot be carried out.
- If you really want to delete the data profile, confirm the dialog with [**OK**]:

The data profile is now deleted and is no longer available.

4.2.4 General functions of the browser tree view

The browser tree view provides the following general functionalities:

- Rename existing tree entry 20
- Delete tree entry 21

4.2.4.1 Rename existing tree entry

The names of user-created entries (e.g. collection lists or station groups) in the tree view can be changed. This is not possible for entries that are used system-wide, such as parameters, station names, etc.

- Select the tree entry you want to rename.
- Open the context menu with the right mouse button.
- Select the function **RENAME FOLDER/COLLECTION LIST**.

- The field of the name in the tree view is activated for a user input.
- Change the entry directly in the **NAME IN THE TREE VIEW** field.
- Confirm your entries with [**RETURN**].

4.2.4.2 Delete tree entry

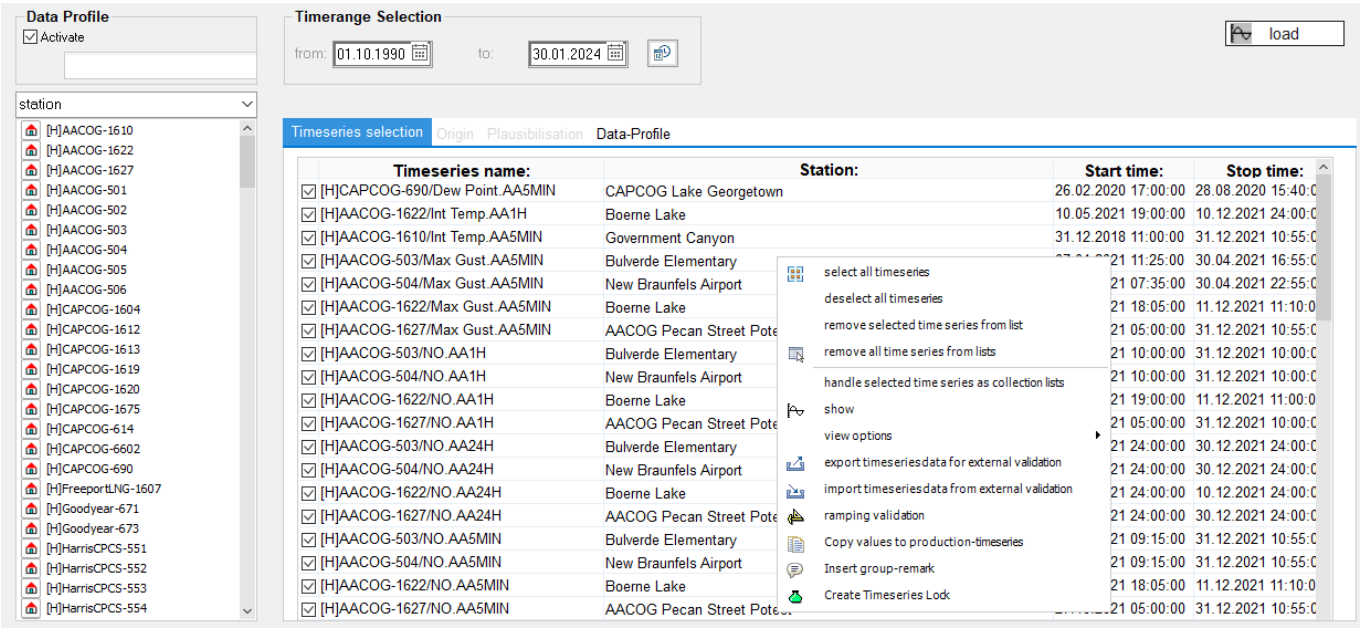
Note: You can only delete folders from the system as long as they do not contain any other folders. In this case, the system issues a notification message.

- Select the (sub)folder you want to delete in the tree view.
- Open the context menu with the right mouse button.
- Select the function **DELETE FOLDER/COLLECTION LIST**.
- Confirm with [**OK**].

The folder or subfolder is now deleted.

4.2.5 The time series selection window

The time series selection window lists the time series that are selected to be displayed in the graph or table, for example. The two columns **VALUES FROM** and **VALUES TO** provide an overview of the existing data in the time series.



In the time series selection, after activating the right mouse button, you get the above-mentioned context menu.

Select all time series

With this option you can select all time series.

Deselect all time series

With this option you can deselect all time series. The time series remain in the selection list.

Remove selected time series from list

With this option you can remove all selected time series from the time series selection.

Remove all time series from list

With this option you remove all time series from the time series selection.

Handle selected time series as collection lists

Please refer to the section Creating new collection list 18.

Default view

Activate this button to open the graph and table display similar to the [**VIEW**] button in the time series browser.

Insert collective remark

This allows you to insert a common remark for all selected time series. Common here means that an identically worded remark is inserted for each of these time series. However, a subsequent change must be made separately for each of these remarks.

4.2.6 Menu options and additional functions

This chapter introduces you to more advanced menu items and functions of DMO:

- Functions for the graph 22
- Display graph and table 25

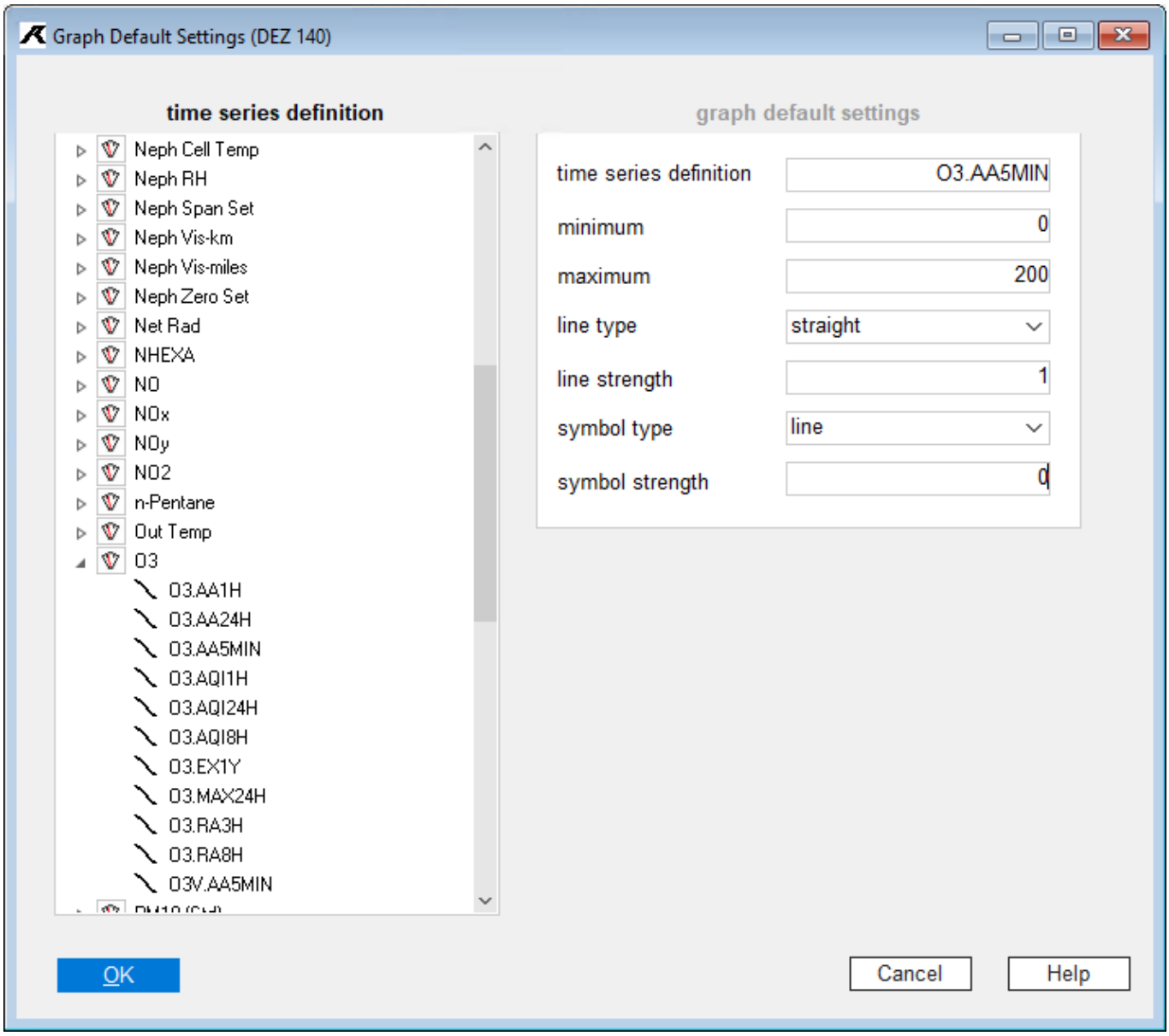
4.2.6.1 Functions for the graph

The main menu item **GRAPH** offers you the possibility to select graph presets and display options and to open the graph display.

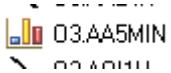
Graph default settings for time series definitions

In order to display time series clearly, you have the option of specifying a format for each time series definition by means of standard settings, which is used for the graph display.

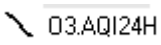
Select the **GRAPH** > **NORMAL VIEW DEFAULT SETTINGS** > **TIME SERIES DEFINITION DEFAULT SETTINGS** menu item in the main menu.



In the selection list on the left, you will find all the parameters created in the system; in the level below, you will find the time series definitions. Time series definitions with an existing configuration are shown as follows:



Time series definitions that have not yet been configured are displayed as follows.



Make default settings for the graph

The following settings determine the format in which a time series is displayed later:

- **MINIMUM**
  Used to set the lower limit of the displayed measured values.
- **MAXIMUM**
  Used to set the upper limit of the displayed measured values.
- **LINE TYPE**
  Selecting a type from the selection list
- **LINE WIDTH**
- **SYMBOL TYPE**
  Display of the measured values by a specific symbol.

- **SYMBOL STRENGTH**
  Setting the symbol in the display.

When you have finished the settings,

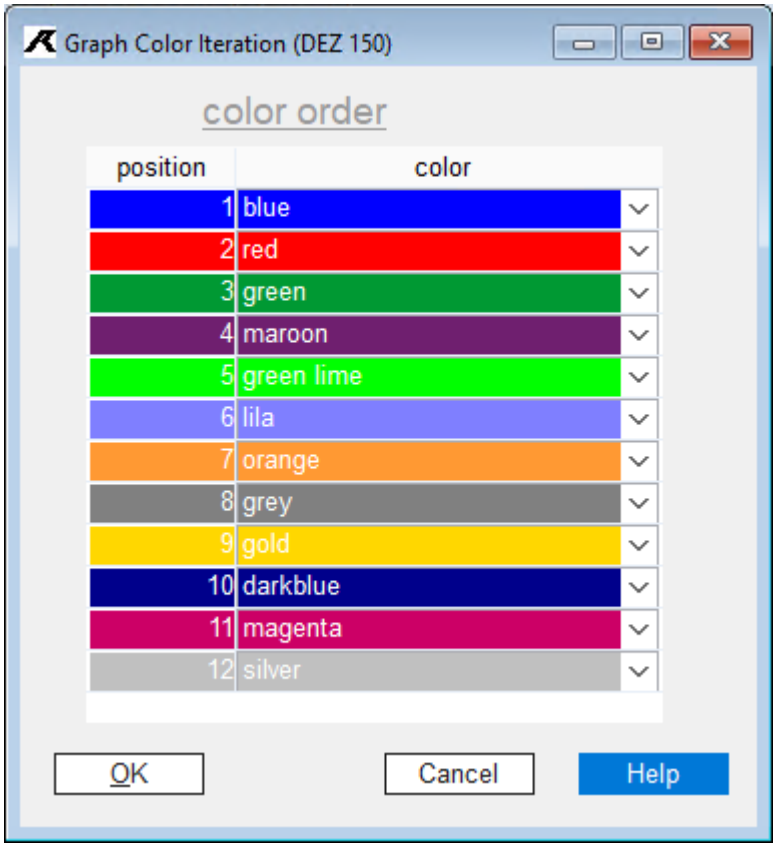
- close the window by clicking the [**OK**] button.
- You return to the time series browser.

Colour order

In addition to the ZRD-dependent settings, colour assignments can be defined for the time series in the time series selection window. Unless otherwise configured for the time series definition, the various plots in a graph display receive the colours in the order defined here. According to the following picture, the line of the first time series will become blue, the line of the second time series red. Etc.

This ensures unique identification of each time series in the graph report.

- In the main menu, select the **GRAPH** > **NORMAL VIEW DEFAULT SETTINGS** > **COLOUR ITERATION** menu item.
  You will receive the following dialog.



Set colour order

You can assign a colour to each of the individual position numbers, which ensures the distinguishability of time series in a graph.

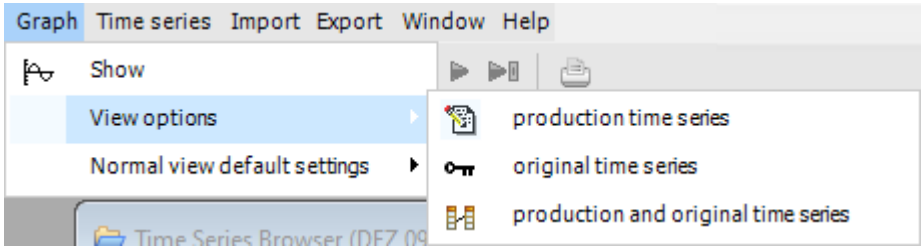
- To do this, select a colour from the colour selection list in the corresponding line.
  If you want to add a new row,
- open the context menu with the right mouse button and
- select the **ADD NEW ROW** function.
  The new last row is added at the end of the list.
- Click on the new row and set its colour.
  If you want to delete the last row of entries,
- select the last entry in the list and
- open the context menu,
- to select the **DELETE LAST ROW** function.
  If you want to close the colour definition dialog,
- click the [**OK**] button.

Confirm the control message to complete the process.

View options

You can display production time series, original time series or both in graph and/or table.

- Click the menu item **GRAPH** > **VIEW OPTIONS** in the main menu.
- Select one of the submenu entries to specify the time series type to be displayed.

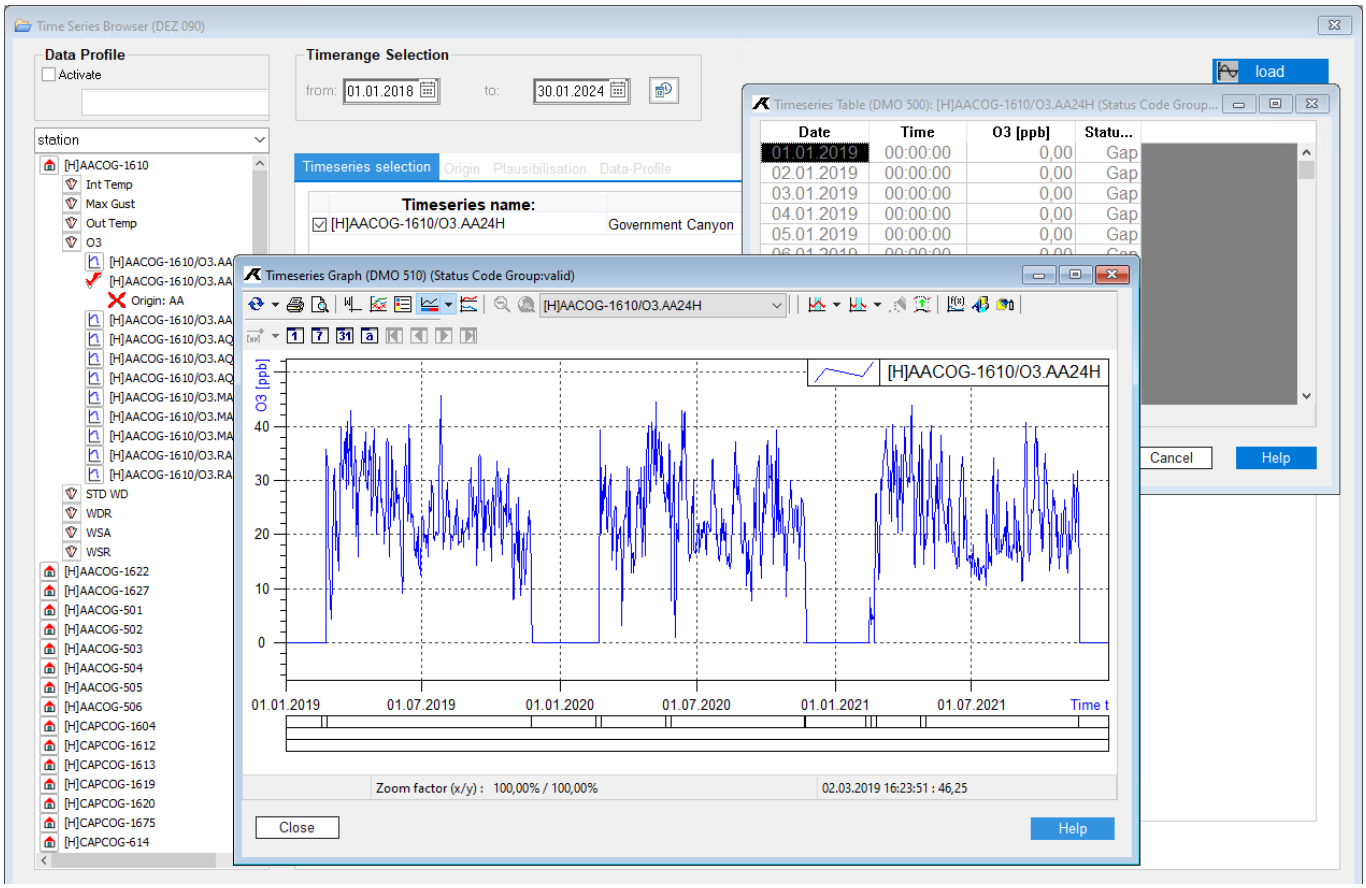


4.2.6.2 Display graph and table

You have now selected one or more time series for viewing and defined a time range for the display.

- Now click the [**VIEW**] button, or
- select the **DEFAULT VIEW** function from the context menu of the time series selection window,
- or the same function from the main menu item **GRAPH**.

A corresponding graph and table is output for the selected time series:



More options

Tables and graphs also offer a range of further convenient functions:

- All functions within the graph can be executed by the buttons in the **GRAPHIC** window or by the context menu (to be opened with the right mouse button).
- You can view and print the data in the graph at an arbitrary resolution using the zoom function. The time series to be represented can be combined as you like and the type of representation can be configured freely.

- You can combine any number of time series you like in one graph and choose different kinds of display for each of these time series.
- You can also reload your current time series into the graph to update the view when the database has been updated with new measured data.
- You can edit the values in the table by editing single cells just as you would a spreadsheet.
- Values of time series that are not being edited yet, always appear in grey at first.
- Non-editable values of time series which are being edited are shown in grey; editable values in black.

Measured values of non-editable time series always appear in grey in the table.

5 Time series: error codes

Error codes and error code groups are used to assess measurement data qualities and their handling. In this chapter you will learn how AquisNet uses the error codes, which settings you can make as a user and how to work with these settings.

AquisNet assigns a quality statement to each individual measured value during import by assigning an error code. Based on this error code, AquisNet decides on the quality and thus the validity of measured values. AquisNet allows it users to create their own error codes. These error codes are listed in a table that can be called up in the main menu under the menu item Time Series > Error code > Administrate status codes. In addition to the numerical value and a short description of the code, this table also contains a textual short representation (display value, also called error code). The error code is always displayed in the value tables of the DMO, so that the user can easily recognise the status of the measured value without having to memorise its numerical values. The text display can be changed by the user in the editing dialog of error codes.

In the user interfaces of AquisNet, the display value (error code) of the error code is always displayed.

Deleting error codes cannot be performed in the AquisNet DMO user interface for consistency reasons.

For base production data, the quality information on measured values is supplied as a "data flag" during import. According to these "Data Flags", AquisNet error codes are automatically assigned to the measured values during import. There is no graphical user interface available for the configuration of "Data Flags" and for mapping with the AquisNet error code system. However, it is possible to make changes in the data management. Note that only trained specialists are allowed to do this. If you are interested, contact KISTERS directly.

To have different quality perspectives when evaluating measurement data, users can create their own error code groups. The membership of an error code in an error code group determines, for example, the validity of a measured value. System error code groups are provided to define these central properties of the system. Predefined groups contain, for example, all valid error codes, all invalid ones, etc. The maintenance of the assignment of error codes to the groups is done by the user, whereby new error codes can be fully integrated into the system at any time. Error code groups can be used as display filters in the DMO, but also as filter criteria for report outputs, so that, for example, only valid or only invalid values can be output.

Content of this chapter:

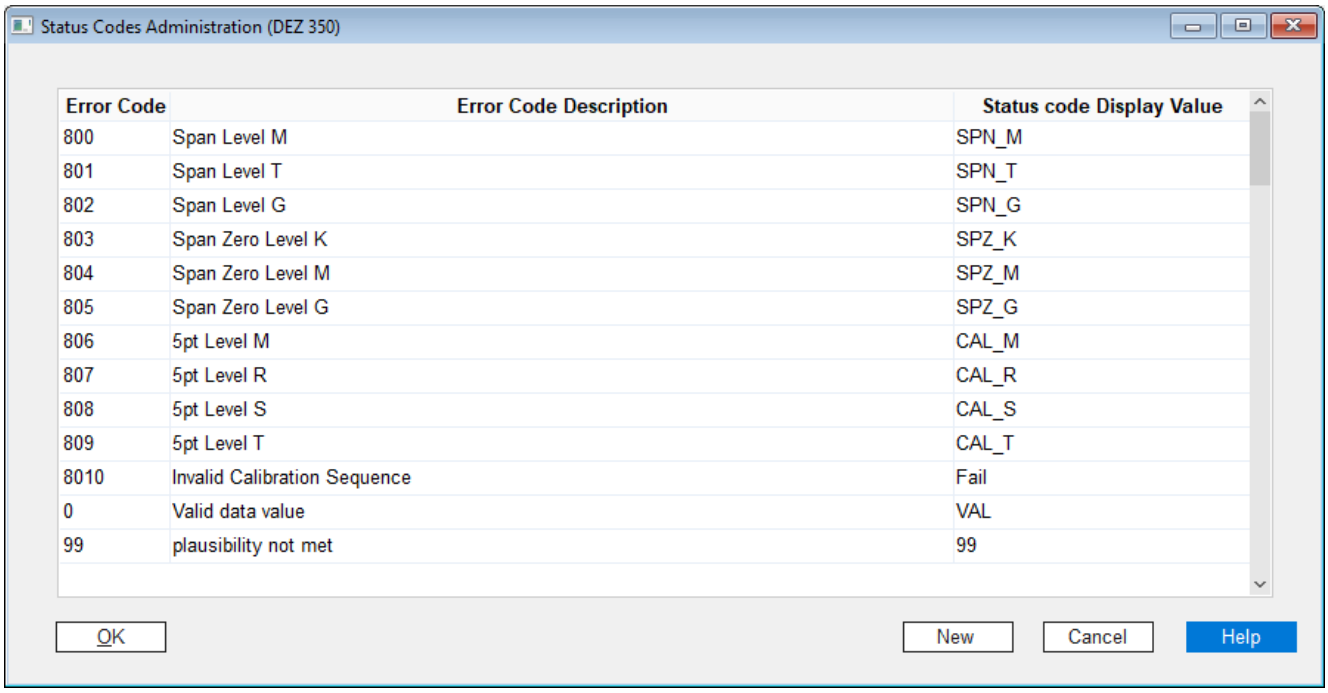
5.1 Manage error codes

AquisNet assigns a quality statement to measurement data during import using different status codes. These status codes are listed in a table that can be opened in the main menu under the **ADMINISTRATE STATUS CODES** menu item. For example, AquisNet assigns the digit 0 as a status code to valid values. Manually changed measured values get the status code 21.

In the user interfaces of AquisNet, users work with status codes. For easy identification of statuses, however, other short names are used here, such as G for valid. This display representation can be adjusted centrally if required.

Changes to status codes, in turn, can be made centrally in the data storage if required. It is therefore not possible to create or delete status codes in the AquisNet DMO user interface.

- Start the **TIME SERIES** > **STATUS CODE** > **ADMINISTRATE STATUS CODES** main menu item.
  The Administrate status codes 81 table opens:



The table lists all existing status codes with their exact description and the associated status code.

- The left column **ERROR CODE** lists the error codes introduced in the system.
- The middle column **ERROR CODE DESCRIPTION** contains a short description of each error code.
- The right column **STATUS CODE DISPLAY VALUE** contains the display value that the user finds in the AquisNet outputs.
- The left and the middle columns are not editable. In the right column, the display values can be edited centrally. Changes in this column are valid system-wide.

5.1.1 Change the display value of error codes

To change an error code entry in the **ERROR CODE (DISPLAY VALUE)** column,

- activate the user input by clicking on the corresponding error code field.
- Now enter the changed error code via the keyboard.
  When you exit the dialog, the system displays a control dialog for confirmation.
- Select the desired option.

5.2 Manage error code groups

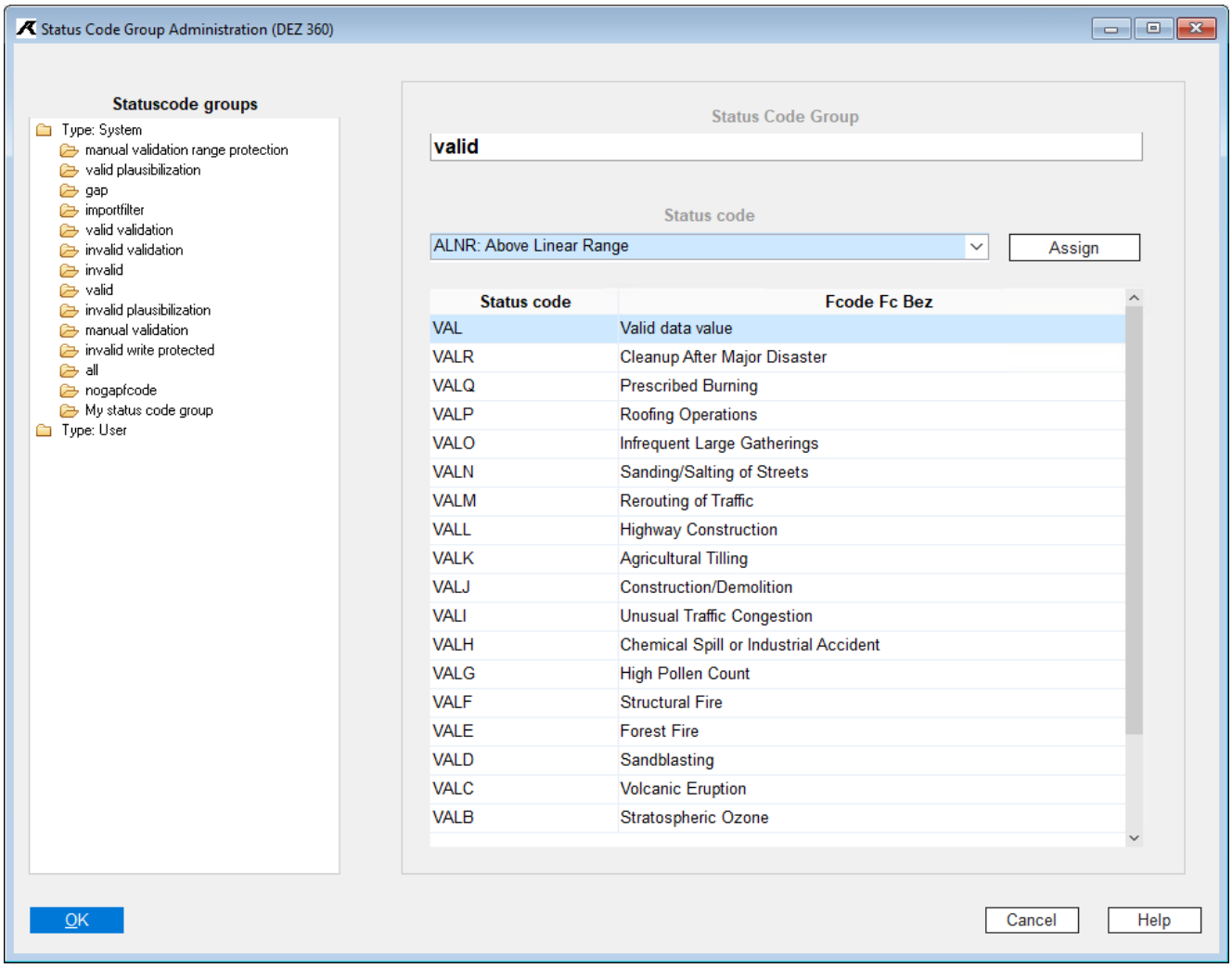
With the concept of error code groups, you have the option of filtering the quality of measurement data using error codes when evaluating or checking data. For this purpose, user error codes can be (meaningfully) grouped together.

Important error code groups are for example:

- the `Valid` group. This group contains all error codes that classify data as valid.
- the `Invalid` group. This group contains all error codes that classify data as invalid.
- the `Manual validation` group. This group contains all error codes that may be used for manual validations. These error codes can be selected in the graph and table in AquisNet DMO for various functions.
  Refer to the Manual validation of measurement data 44 section in ch. Display time series in the data collection center DMO 39.
- AquisNet distinguishes between the categories `System` and `User` for error code groups. For example, groups under System, such as the `Valid` group, cannot be deleted because AquisNet accesses these groups in the course of validation:
- For aggregations, the system accesses source time series and contained measured values with the error code `valid`.
- The table and graph functions of the AquisNet DMO access the error code group `manual validation` to filter measured values.
- Users can access error code groups via the reporting and evaluation functions in AquisNet.
- However, depending on user rights, individual error codes can be added or removed.

To start the management of error code groups,

- select the **TIME SERIES** > **ERROR CODE** > **MANAGE ERROR CODE GROUPS** menu item.
- The Status Code Group Administration 81 dialog opens:



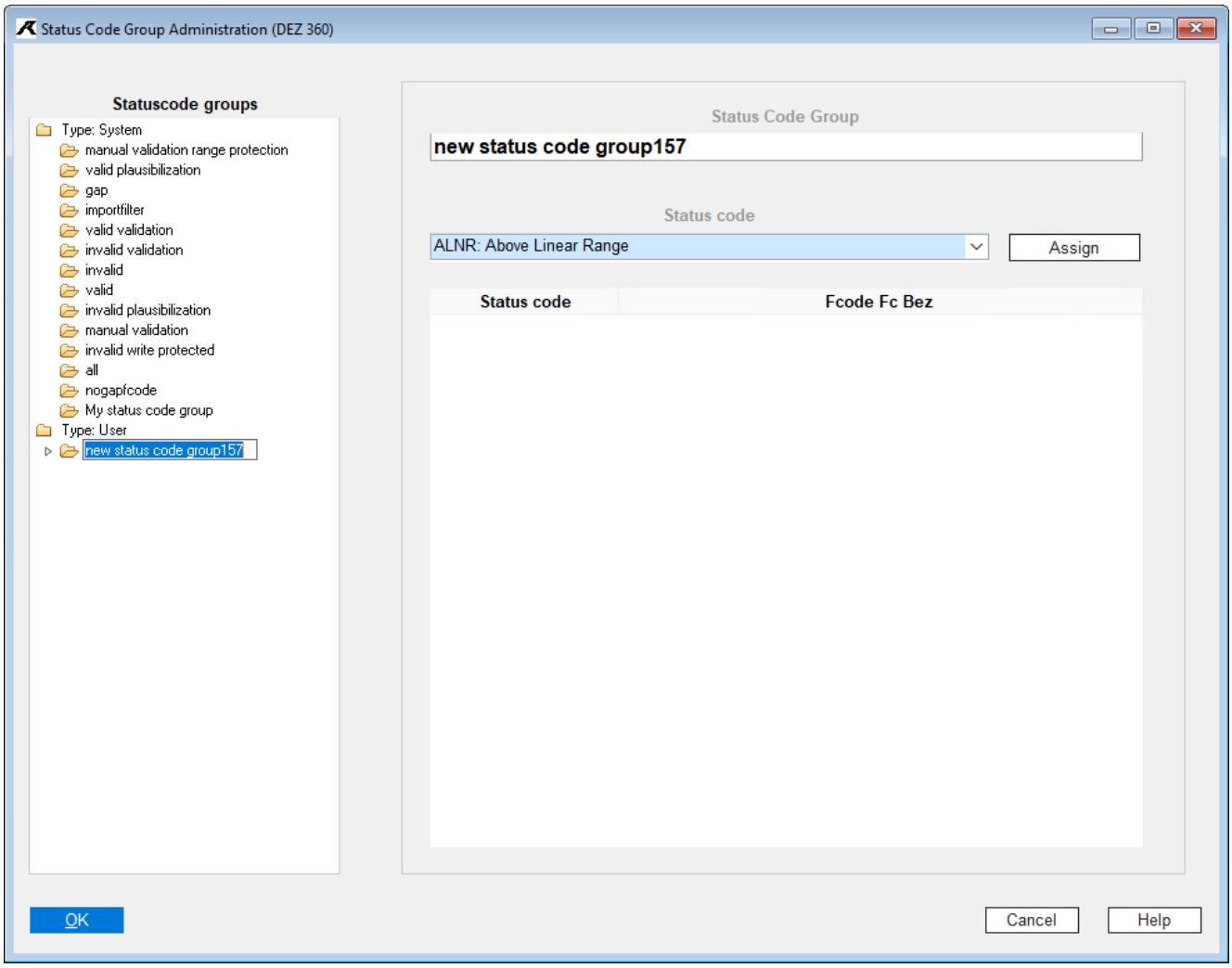
- In the tree view in the left window you can call up the error code groups. These are divided according to system and user. The system folder also contains those error code groups that are not deletable to ensure dependencies within the AquisNet system.
- Under **STATUS CODE GROUPS**, the selected error code group is displayed in the upper part of the right-hand window.
- Below this, all error codes are listed with the description that have already been assigned to the called group.
- Above the list display, further error codes can be selected via a selection list, which are taken over into the existing error code group after pressing the [**ASSIGN**] button.

Content of this chapter:

5.2.1 Create new error code groups

To create a new error code group,

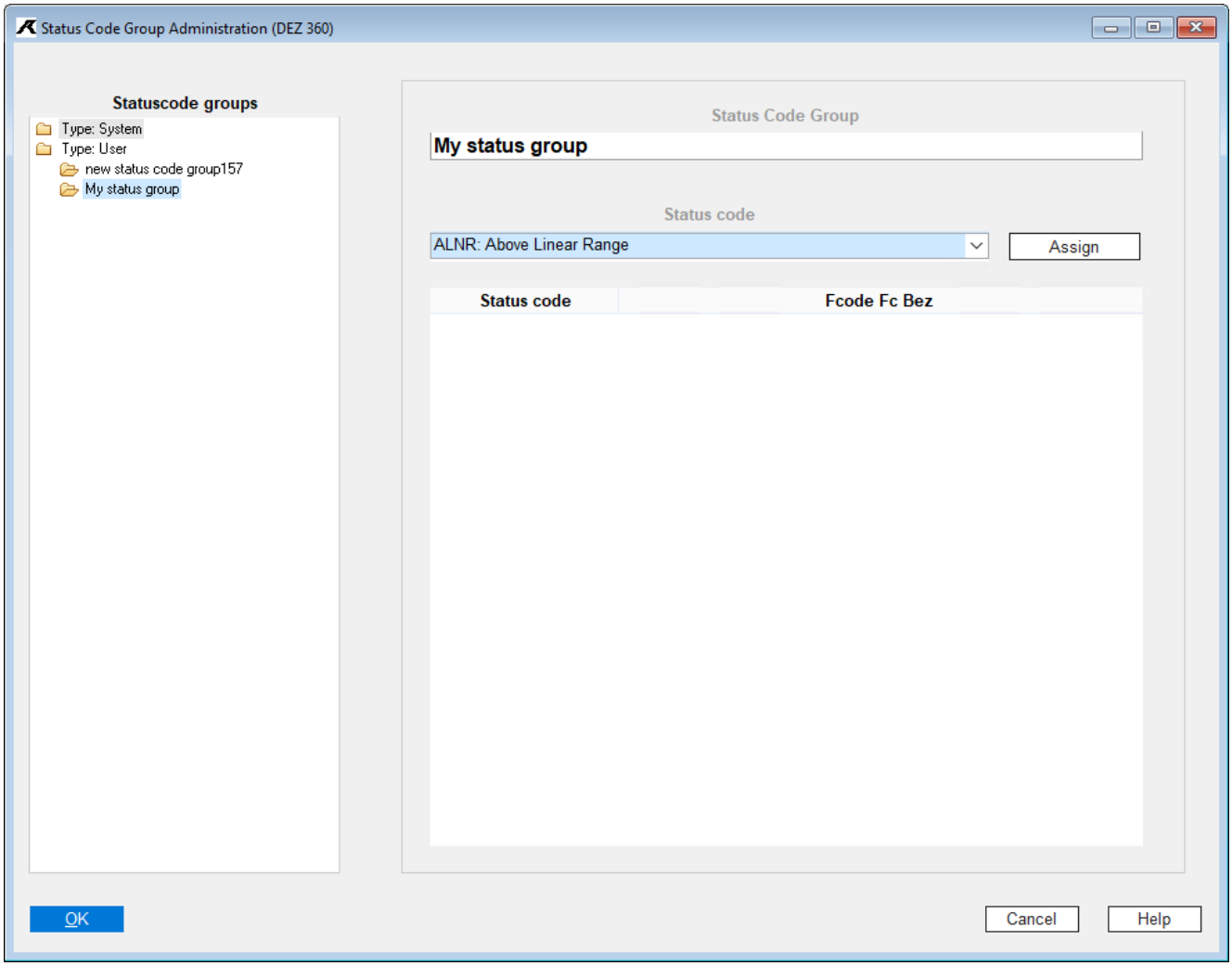
- click any entry in the tree view on the left.
- Open the context menu with the right mouse button.
- Select the **CREATE NEW ERROR CODE GROUP** function in the context menu.
  AquisNet now creates a new error code group and an entry in the tree view. This entry is given a default name.
- You can change the name suggested by the system in the tree view.



- To do this, click it once with the mouse pointer.
  After a short moment, the input field is activated.
- Now enter the new name via keyboard and confirm with [**RETURN**].
  The changed name of the error code group is also shown at the top right of the dialog.
  With the next user action, the system displays a control dialog for saving the changes.
- Confirm it with [**YES**].

5.2.2 New assignment of an error code to an error code group

- First select the error code group in the tree view to which you want to add one or more error codes.
- The error codes of the selected error code group are listed in table form in the right-hand window. The respective error code text and the designation are displayed, but not the internal numerical code.

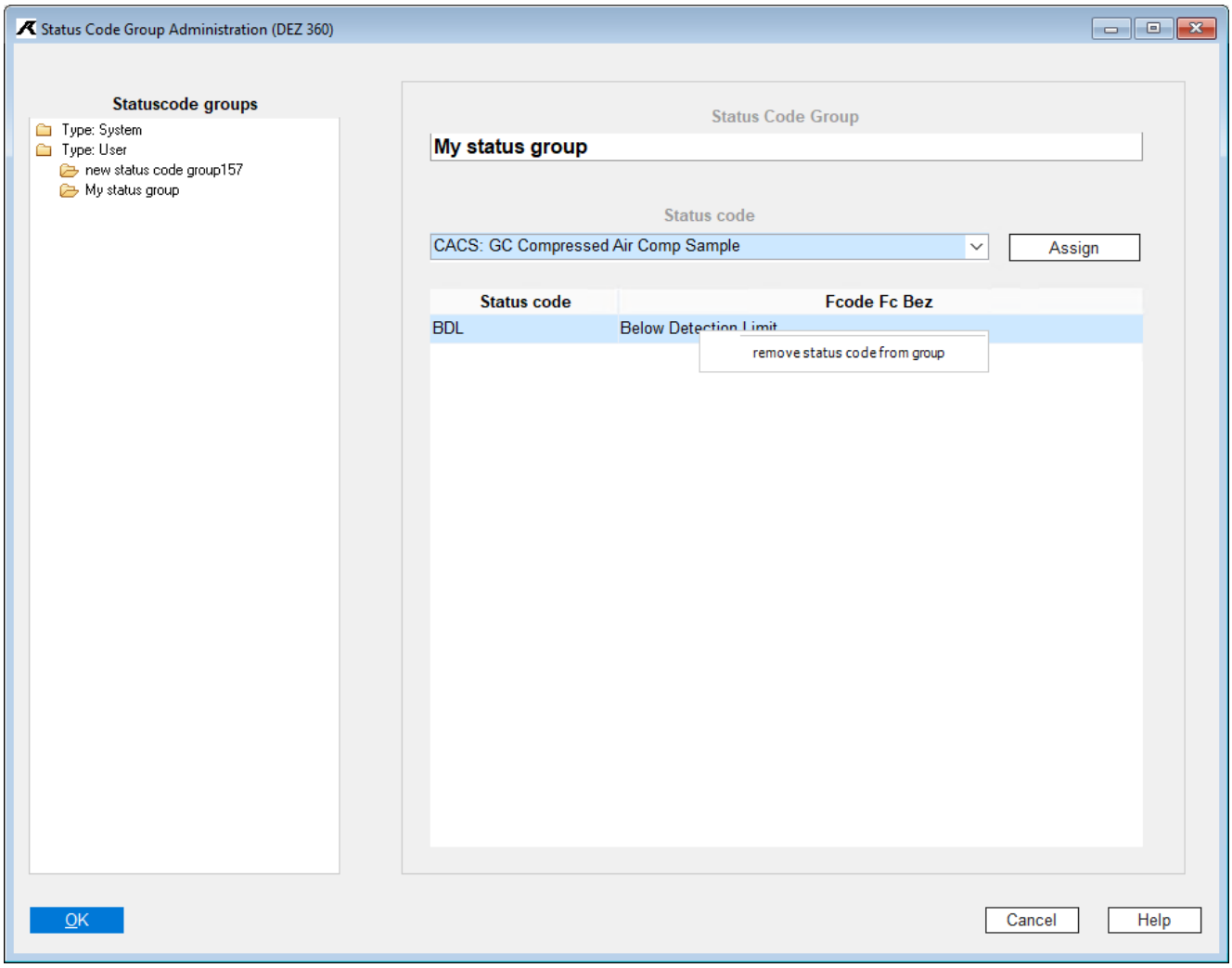


- Now open the drop-down list box above the table display (**STATUS CODE** section).
- You can now select the desired new status code from this list.
- By clicking the [**ASSIGN**] button, the preselected status code is assigned to the status code group and is displayed in the table display.
- To assign one or more additional status codes, repeat the process as necessary.
- When leaving the dialog or selecting another status code group, the system displays a control dialog. Here you can confirm your entries and thus save or discard them.
- Only then the system proceeds to the next action.

5.2.3 Remove an error code from an error code group

To remove an error code from an error code group,

- click the error code line in the table display on the right.
- Open the context menu and select the entry **REMOVE ERROR CODE FROM THE SELECTION**.



The error code is removed from the selection.

- When you exit the dialog or perform another action, the system displays a control dialog.
- Confirm this.
  Your changes will be adopted.

6 Edit time series: Transactions in the time series browser of the data collection center DMO

In AquisNet DMO, you can manually validate time series using the graph and table interfaces. To ensure data consistency, this is only allowed for base production time series. Dependent time series affected by the change are automatically recalculated by the system. AquisNet stores the change history resulting from manual validations in transactions.

If time series values or error codes have been changed, the system asks the user via a control message when exiting the view whether the changes should be saved. If a time series contains at least one changed value or changed error code, AquisNet asks the user for a transaction name and a textual description (comment) of the transaction. Only after the user has entered the information AquisNet saves the before and after status in the transaction so that it is available for later displays or even rollbacks.

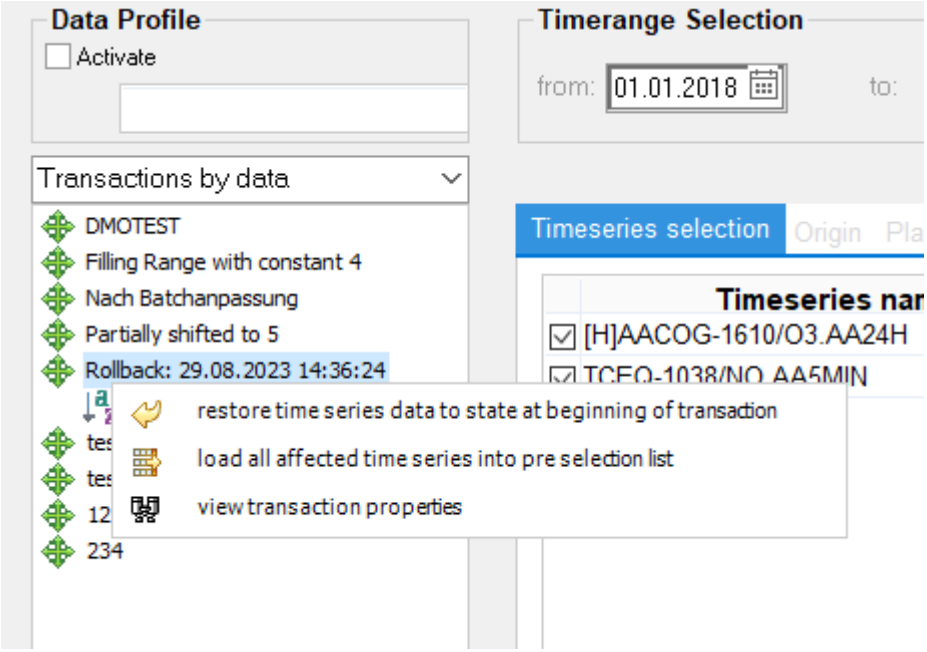
Note:

- Changes are only possible in a base production time series. The base production time series and thus also the time series derived from them always contain the latest and thus best quality information.
- Automatic changes by the import validation or plausibility checker are displayed in the change bar. However, no time series with pre-change information are generated.

You find detailed information on this in the following chapters:

- Working with transactions in the time series browser 33

6.1 Working with transactions in the time series browser



Starting functions in the context menu

The following functions are available via the context menu in the time series browser 33 (tree view for transactions):

- Load all affected time series states in selection list 33
- Display properties of the transaction 35
- Creating new transaction 37
- Restore time series states in production time series (recovery) 37

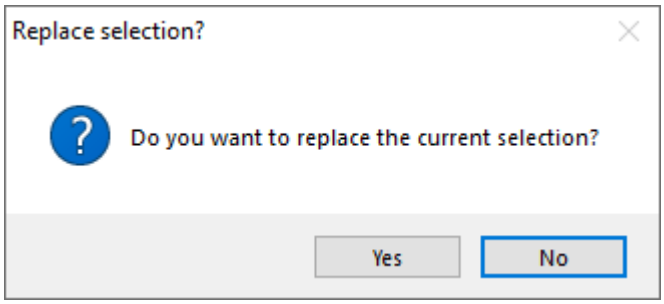
You find detailed information on this below.

6.1.1 Load all affected time series states in selection list

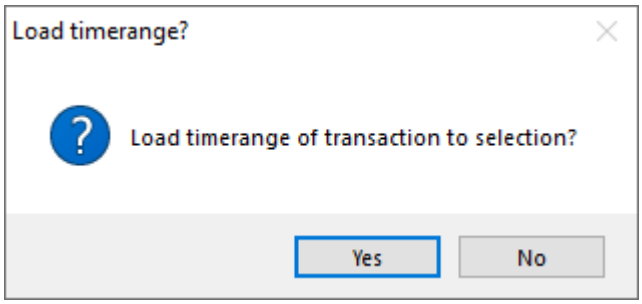
You can transfer all affected base production time series of a transaction into the time series selection window on the right at the same time.

- To do this, click on the desired transaction.

- To do this, open the context menu in the left-hand window and select the **LOAD ALL AFFECTED TIME SERIES IN DROP-DOWN LIST BOX** entry. The following dialog appears:



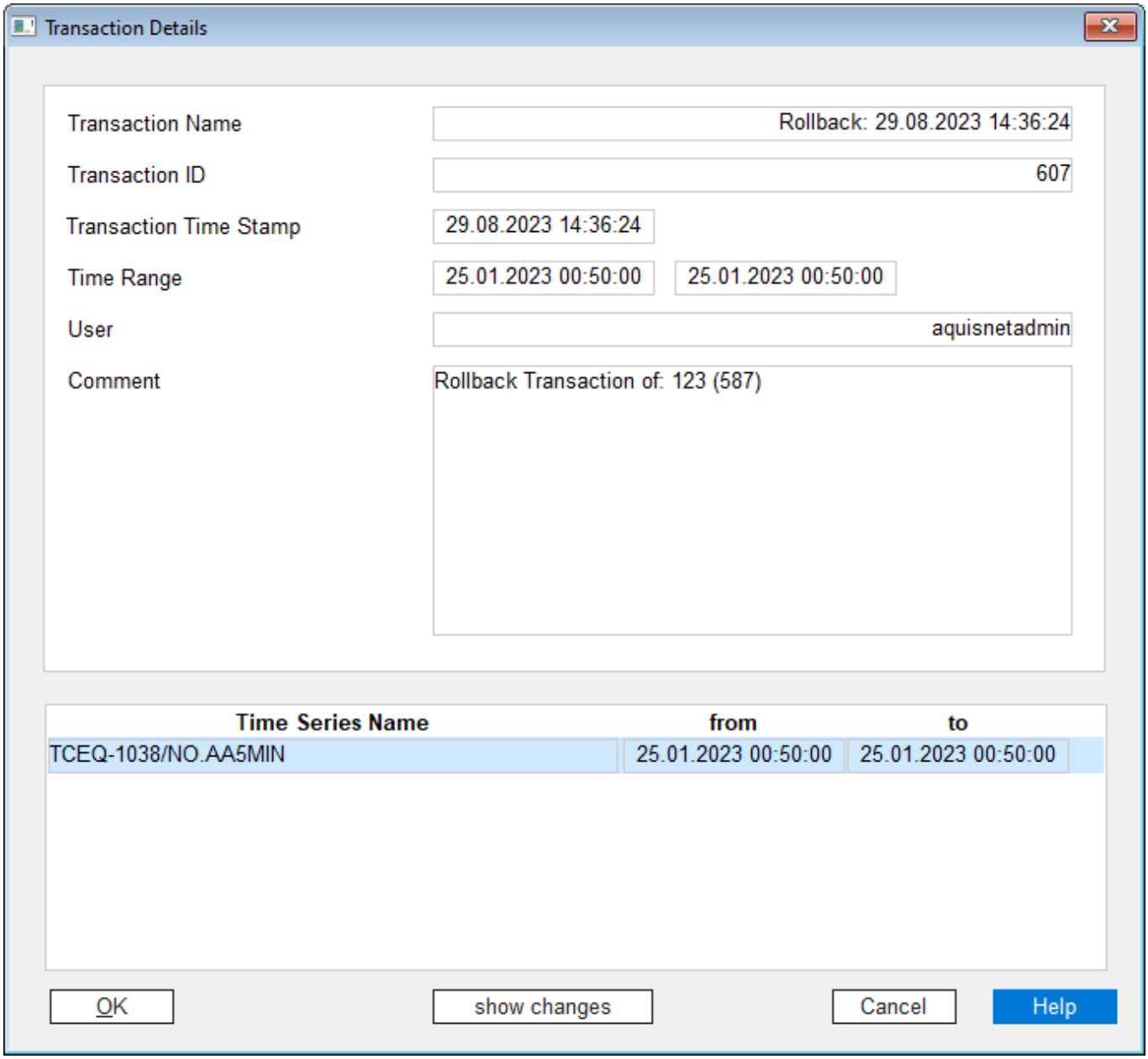
Here you can overwrite (**YES**) or extend (**NO**) the existing time series selection. If you can apply the time range of the transaction to the time selection, select this in the dialog that opens.



The time series are transferred to the time series selection window.

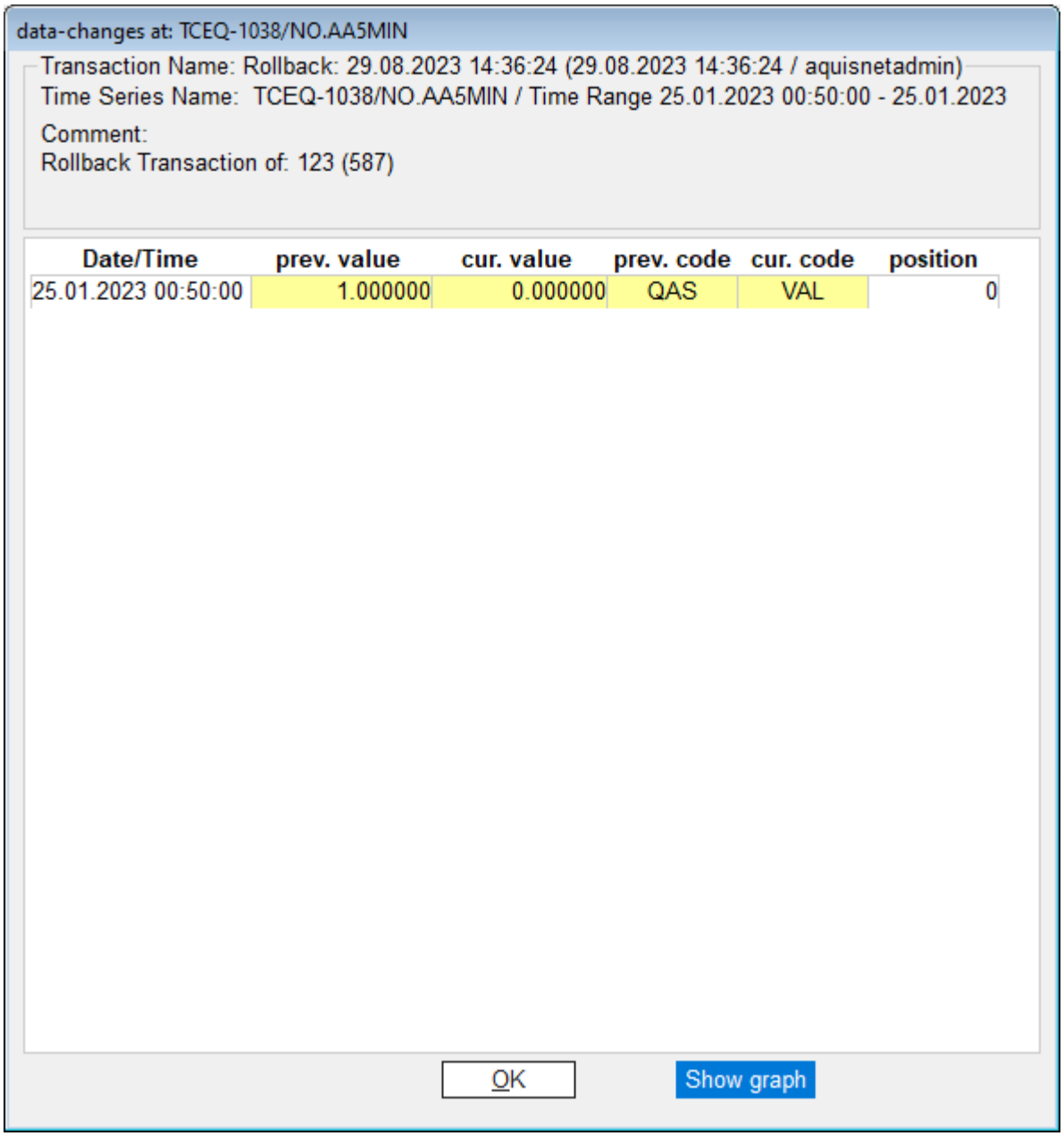
6.1.2 Display properties of the transaction

With the **VIEW TRANSACTION PROPERTIES** function of the Details context menu, you open details of the present transaction. They are displayed in a new window:



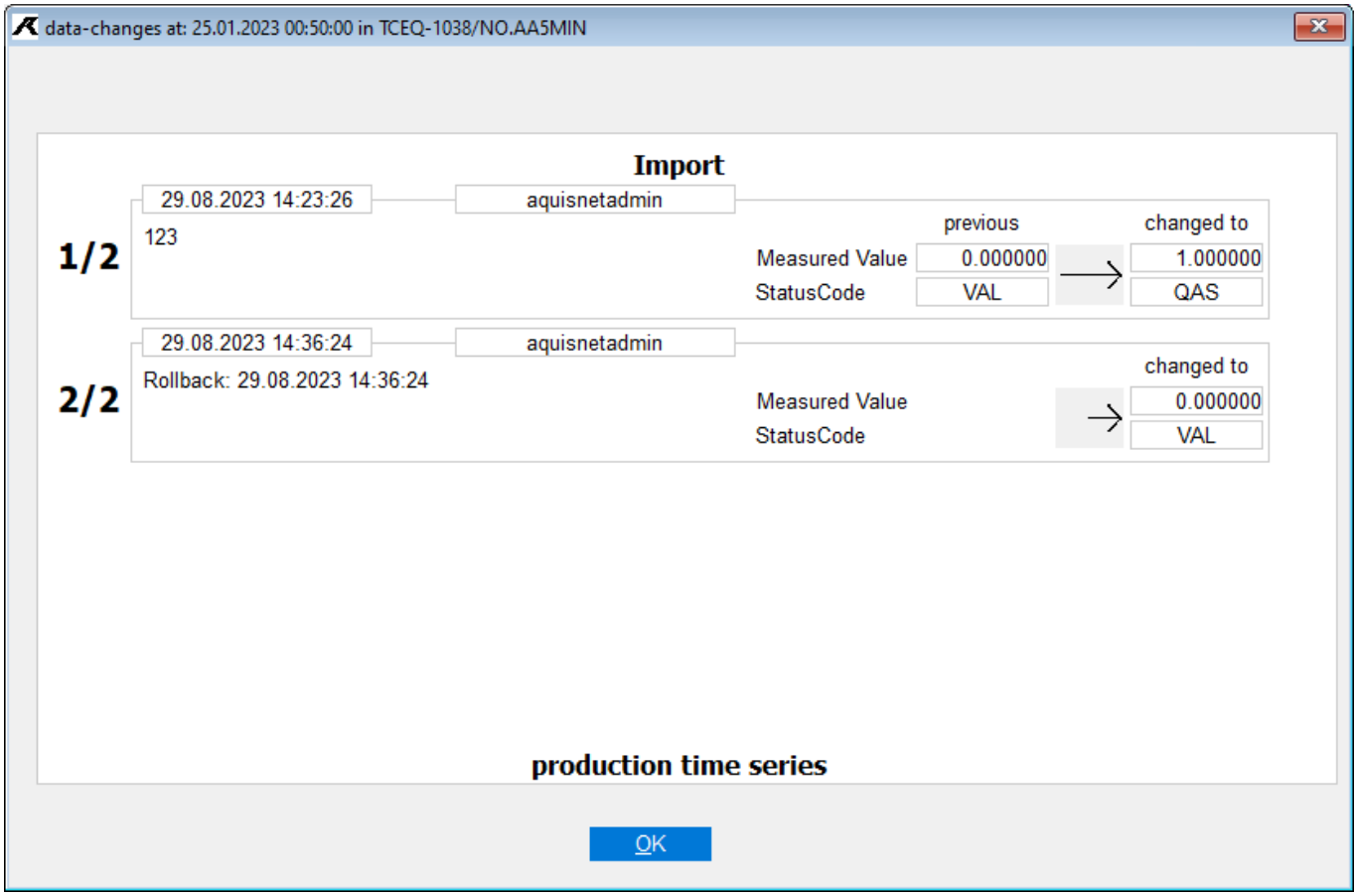
- Transaction name
- Transaction ID
- Transaction time stamp
- Time range stored time range
- User who executed the transaction
- Description of the executed transaction
- List of all affected time series with time series name (left column) and time range of data changes (right column).

After selecting a time series in the list above, you can display the changes in the following dialog:



In addition to the transaction data, you can see the data states of the individual measured values before and after the transaction.

Double-clicking on one of these rows opens the transaction details for the selected measured value and thus an overview of all transactions and states relating to this measured value:



Here you can now see all transactions with their name, time of execution, user and the values before and after the transactions that have made changes to the selected value.

Double-clicking on one of the displayed transactions then opens the transaction detail display (see above).

6.1.3 Creating new transaction

You create new transactions in the Transactions 99 dialog.

- Transactions for the data management office DMO are described in detail under Transaction for saving changes 45. This program function opens when you exit the graphical or tabular time series display after you have manually changed measured values or error codes.

6.1.4 Restore time series states in production time series (recovery)

Recovery means: Restore the previous data status of one or more time series by means of information available in a transaction for these same time series. Data is changed within a certain time range. The transaction used only contains data for this time range.

To ensure data consistency, this action is also saved as a transaction. This writes the changes concerning the specific time range into the base production time series. Information on the previous status is also saved.

You can restore the time series status of individual production time series or all time series of a transaction:

- Recovery for all time series of a transaction 37
- Recovery for individual time series of a transaction 38

6.1.4.1 Recovery for all time series of a transaction

With the **RESTORE THE TIME SERIES STATUS BEFORE THE TRANSACTION OF ALL TIME SERIES** function, you restore the state of base production time series before the change that was saved in the called transaction.

- To do this, select the desired transaction in the tree view.
  All time series below are reset.

This reset is saved as a rollback transaction; you can display its properties 35.

6.1.4.2 Recovery for individual time series of a transaction

With the **RESTORE THE TIME SERIES STATUS BEFORE THE TRANSACTION OF THE SELECTED TIME** function, you restore the state of the selected base production time series before the change that was saved in the called transaction.

- To do this, select an individual time series below the transaction.
  Only this time series is reset.
  This reset is saved as a rollback transaction; you can display its properties 35.

7 Display time series in the data collection center DMO

There are two visualisation elements in the AquisNet data collection center DMO: the time series graph and the tabular representation. The editing of the time series is done interactively.

In this chapter you will learn how to represent time series from within the DMO in tables and graphs and how to format and design graph and table views for your interactive editing.

The chapter will explain how to insert and edit user-defined graphical elements (for example lines, arrows, rectangles, texts).

You can view any number of time series with measured values in one or several graphs or tables (one graph or one table each per time series):

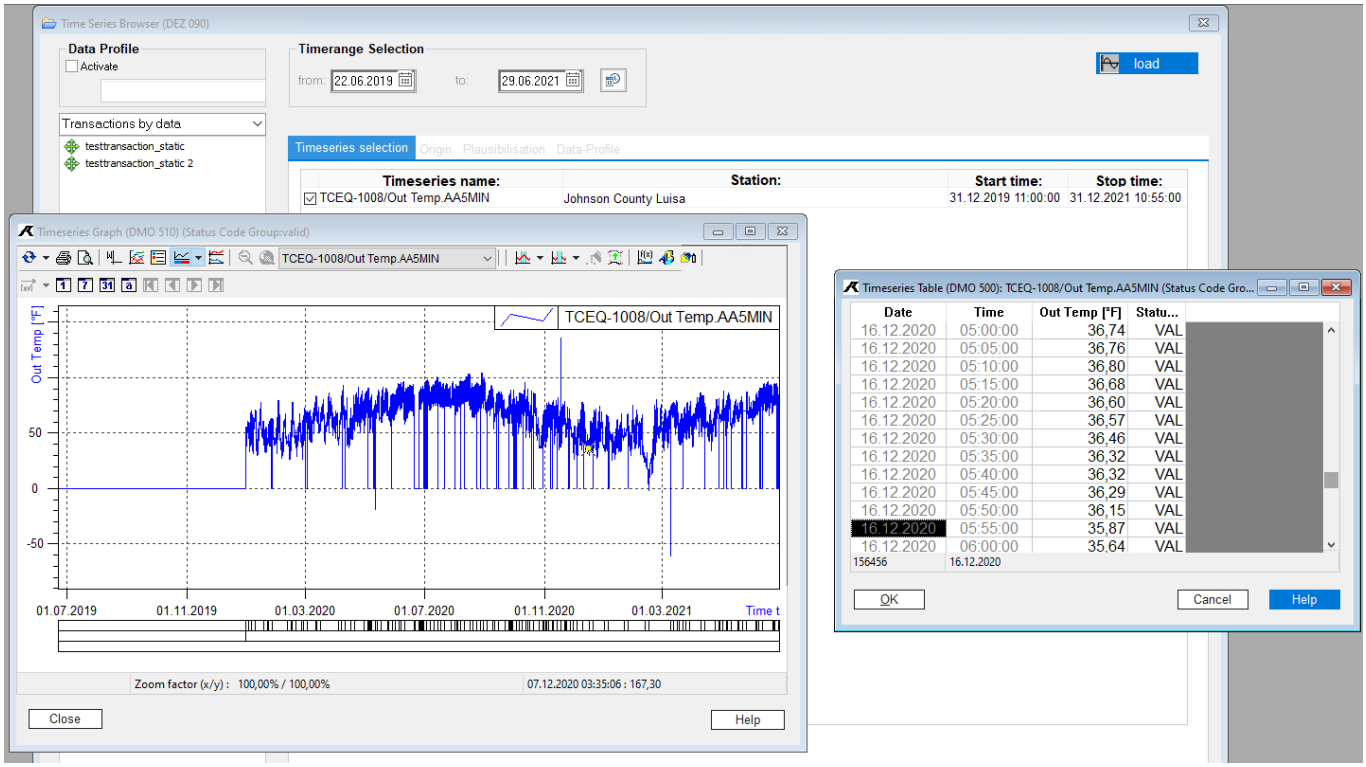
- Open the AquisNet time series browser.
- Select the desired view for the tree view in the drop-down list box of the same name in the upper left corner of the browser,
- and select a time series by double-clicking.
- Set the time range of the data that you want to display in the graph/table.
- Select the **VIEW** function using the [**VIEW**] button, or using the **DEFAULT VIEW** entry in the context menu, which you can open using the right mouse button.

The Graph 88 or Table 96 dialog opens. The measured values of the selected time series within the specified time range are displayed.

You have now selected one or more time series for viewing and defined a time range for the display.

- Now click the [**VIEW**] button or
- select the **DEFAULT VIEW** function from the context menu of the time series selection window,
- or the same function from the main menu item **GRAPH**.

You get the time series browser and the **SHOW GRAPH** and **SHOW TABLE** reports combined on the screen:



- To enlarge the **DISPLAY GRAPH** report, expand the report to screen size.
- To enlarge the **DISPLAY TABLE** report, expand the report to screen size.

The representation of the time series within DMO is explained in the following chapters:

- Working with tables 40
- Working with graphs 41
- Standard functions 43
- Manual validation of measurement data 44
- Transaction for saving changes 45
- Adapting graph display 48
- Using crosshairs 49

7.1 Working with tables

Working with tables comprises these steps:

7.1.1 The context menu in time series tables

You can open processing and view options in the table using the context menu. The context menu always appears by activating the right mouse button.

The functions that you can open using the context menu are described in detail in the following chapters:

7.1.2 Fixing the selection and order of columns

Open in the **TABLE** dialog the context menu using the right mouse button.

Select the **TABLE** > **CONFIGURE** function.

You can use the **FIXING THE SELECTION AND ORDER OF COLUMNS** dialog to specify which columns (and in which order) are shown in a table.

- To do so, draw by the Drag&Drop function the desired column names from the **COLUMNS NOT DISPLAYED** list into the **DISPLAYED COLUMNS**list. Define the column order using the **UPWARDS**/**DOWNWARDS** buttons.

7.1.3 Setting the column width

In the **TABLE** dialog, move the mouse pointer along the column title row until it changes from arrow display to shift display.

- Holding the left mouse button down, drag the vertical line to the desired position.

7.1.4 Viewing table values in the graph:

- Select the desired value in the **TABLE** dialog.
- Click the right mouse button.
- Select the **SYNCHRONISE TABLE AND GRAPH** context menu item.

7.1.5 Searching for table entries

You can search for entries in specific columns inside long tables:

- Set the mouse pointer to any column header in the **TABLE** dialog.
- Activate the context menu with the right mouse button and there the function **COLUMN: SEARCH**.
- In the Search for dialog, specify the name you are looking for.
- In the **CONDITIONS** section you can enter
  - whether only whole words should be searched for according to the entered search text and
  - whether the search should be case-sensitive.
- Define the search direction (upwards or downwards from the selection in the table).
- Start the search for the search term using the [**SEARCH NEXT**] button.

7.1.6 Visualise changes in time series (e.g. transactions)

In the table display, changes are displayed in the **COMMENT** column for each time series individually.

- This lists all changes that have been made to a time series and arranges them in chronological order. In this way, users can record changes made to a time series "at a glance".
- The following information is displayed:
- Transaction ID and transaction name
- Date of the user action
- Responsible user
- Output options:
  - Case 1 - Change of values:
    In this case, the description "Value: X to Y, error code A to B" is set.
    X represents the original value and Y the changed value. A is the original error code and B is the automatically changed error code, in case of changed values.
  - Case 2 - Change of error code:
    In this case, the description "Error code: A to B" is set. A is the original error code and B is the changed error code.
    Example: "171 daily, 21/11/2006, hr, error code: M to G".

AquisNet automatically enumerates the changes, ordered by time stamp and time series.

7.1.7 Copying values to the clipboard

- In the **TABLE** dialog, select the desired values.
- Click the right mouse button.
- Select the **COPY** function.
- These values may be reinserted from the clipboard.

7.1.8 Undoing actions

Actions executed by mistake can be undone in single steps:

- Select the **UNDO** context menu entry in the table window.

7.1.9 Restoring actions

You can restore undone actions, just like undoing actions.

- Select the **REDO** context menu entry in the table window.

7.2 Working with graphs

The Graph object lets you display and edit time series graphically.

In the graph you can

- Edit data
- Display calculated time series
- Create sums and differences from different time series
- Calculate correlations

- Calculate regressions
- Edit related time ranges (stretching, tipping, filling with constants, etc.)

The graph window allows you to edit your time series in an easy and comfortable way. All functions within the graph can be executed using the buttons or using the two context menus of the standard graph display or the working range.

You can view and print the data in the graph at an arbitrary resolution using the zoom function. The time series to be represented can be combined as you like and the type of representation can be configured freely.

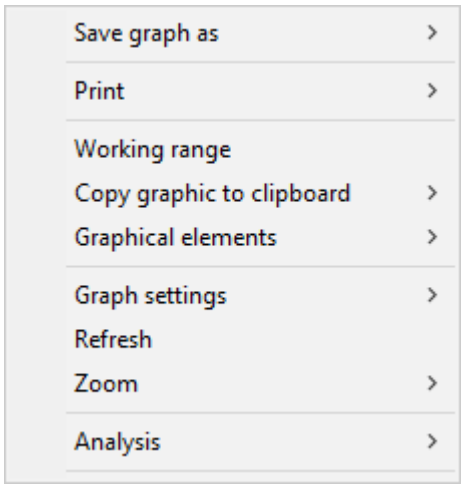
You can combine any number of time series you like in one graph and choose different kinds of display for each of these time series.

You can also reload your current time series into the graph to update the view when the database has been updated with new measured data – e.g. by a poll task, by import (also using import server) or by a third party program.

Working with graphs comprises these steps:

7.2.1 The context menu in time series graphs

You can open processing and view options in the graph not only using the icons in the header, but also using the context menu. The context menu always appears by activating the right mouse button.



The functions that you can open using the context menu are described in detail below.

7.2.2 Displaying measured values in the table

Select in the graph window the desired plot.

- Open the context menu of the table and select the **SYNCHRONISING GRAPHS AND TABLES** function.
- Select with the mouse the desired measured value.

The corresponding measured value is automatically selected in the table.

7.2.3 Copying graphs into the clipboard

You can save the active graph to the clipboard, to e.g. use it in other applications (graphic or text editor software):

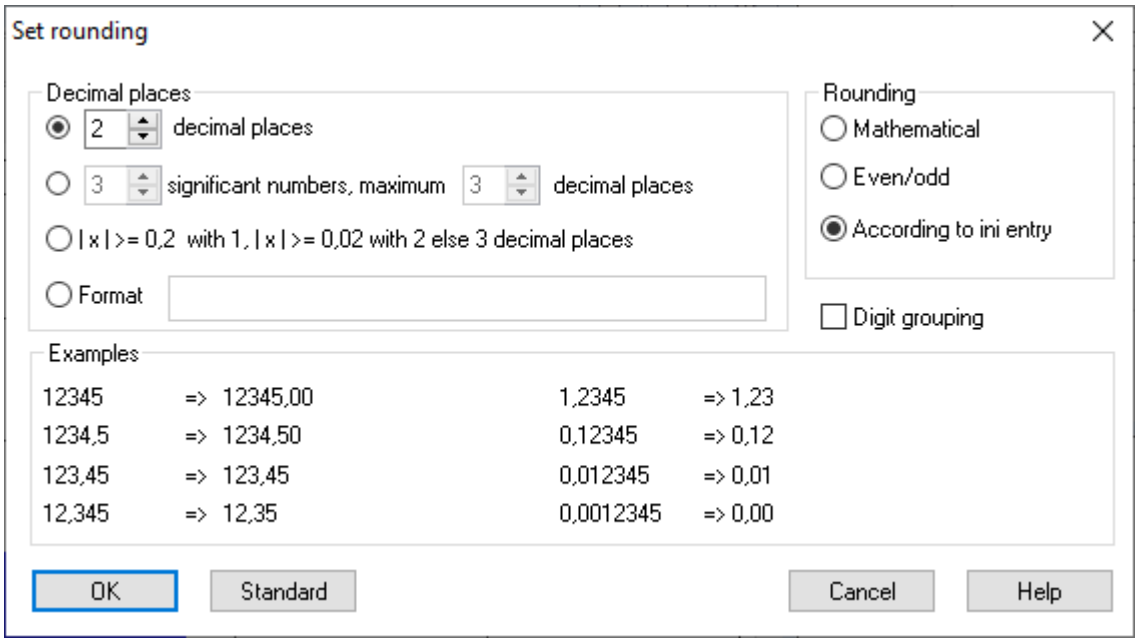
- Open the context menu of the graph with the right mouse button.
- Enable the **COPY GRAPH INTO CLIPBOARD** function.

The graph can be pasted from the clipboard into other programs now.

7.2.4 Setting the rounding method

You can set how to round measured values in the table:

- Open the context menu in the Table dialog using the right mouse button:
- Select the **ROUNDING** function.
  Note: Roundings are valid only for those table columns where you have set the mouse pointer. The function from the context menu can only be selected here. Other value columns, if existing, are not influenced.
- The Set type of rounding 84 dialog opens.



Set roundings

- Specify in the Decimal places section the number of decimal places and the number of significant digits (e.g. three digits), if required; choose an entry from the drop-down list.
- You may also specify that the amount of the measured value is to be taken into consideration (if >= 0.2, one decimal place is shown, if >= 0.02, two, otherwise three decimal places are shown).
- Choose the kind of rounding in the **ROUNDING** section: mathematical (standard rounding method), even/odd (rounded to the even figure, if the last figure is 5) or according to INI entry (rounding as specified in the luqs.ini file).

The kind of rounding is stored in the `luqs.ini` file for each type of time series and is available when opening a table the next time.

7.3 Standard functions

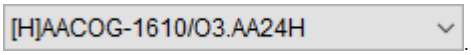
The tutorials about the graphing and table features comprise these topics:

- Selecting plots 43
- Select a time series for the graph 44
- Inserting plots into an open graph 44
- Print preview 44
- Print 44
- Updating Display 44

Follow the links to learn more about graphs and tables in AquisNet.

7.3.1 Selecting plots

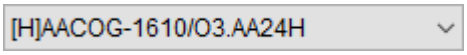
You can represent several time series in the graph window. Editing of a time series applies only to the selected plot:

- In the graph window, select the desired plot from the list 

.

7.3.2 Select a time series for the graph

If several time series are opened in one graph window, you can choose the desired table

- by selecting the corresponding time series in the list 

of the graph window.

7.3.3 Inserting plots into an open graph

To reload plots into the graph,

- additionally select the desired time series in the AquisNet browser (double-click) and
- load the graph again by clicking the [**VIEW**] button.

7.3.4 Print preview

- Select the desired plot.
- In the graph window, select the  button.

7.3.5 Print

- Select the desired plot.
- In the graph window, select the  button.

7.3.6 Updating Display

If an executed action is not shown immediately in the graph window, you can update the whole display:

- In the graph window, select the  button.

7.4 Manual validation of measurement data

You can manually validate measurement data and error codes using the tabular time series display. These validations are displayed using a control bar in the graphical time series display.

Note here that a changed value gets a changed error code. By default, a manually changed measured value receives the error code I for Invalid Data - Manual Validation.

In the two display options, you can filter the displayed time series according to error code groups and the error codes stored in them. This filtering is also possible for time ranges.

The manual validation of measurement data comprises the following options:

- Table view with several time series side by side 44
- Manual validation through measured value change 44
- Manual validation by setting an error code 45
- Manually change error code over a time range 45

7.4.1 Table view with several time series side by side

In the table, all time series are listed side by side and in columns. In this way, you can select several time series (row by row) at the same time and validate them manually.

7.4.2 Manual validation through measured value change

To manually validate a value,

- click in the table cell of the value to be changed in the table display.

The table cell is now activated for user input.

You can now change the value manually.

- Confirm your entry with [**RETURN**].

The value is rounded if necessary and finally saved.

7.4.3 Manual validation by setting an error code

To set an error code manually,

- click in the table cell of the error code to be changed in the table display.
- The system now offers a drop-down list box with error codes.
- These error codes are imported from the error code group manual validation.
- If the required error code is not available in the list, add it to the **MANUAL VALIDATION** group as described under Manually changing the error code via a time range 45 before continuing with the validation.

7.4.4 Manually change error code over a time range

To set an error code manually,

- select one or more rows in the table by moving the mouse pointer over the rows to be selected while holding down the left mouse button, or select several rows by pressing the [**CTRL**] key to select several lines one after the other.
- In the context menu, select the **SET STATUS CODE** > **QUALITY FLAG** entry.

The Set status in the selected area dialog opens.

- Here you can subsequently vary the edit range in the **TIME RANGE** section.
- Now select the desired error code from the drop-down list box in the **STATUS** section.

These error codes are imported from the error code group **MANUAL VALIDATION**.

If the required error code is not available in the list, proceed as follows:

- Close the current dialog and the graph and table windows.
- Select the **TIME SERIES** > **ERROR CODE** > Manage error code groups 81 menu item.
- Assign the desired error code to the error code group manual validation under system.
- Close the dialog and save your entries.
- Now open the time series browser again.
- Click the [**VIEW**] button here again.
- AquisNet reloads the time series in the time series list for table and graph.
- Select the desired time range in the table again.
- In the context menu, select the **SET STATUS CODE** > **QUALITY FLAG** entry.

The **SET STATUS IN THE SELECTED AREA** dialog opens.

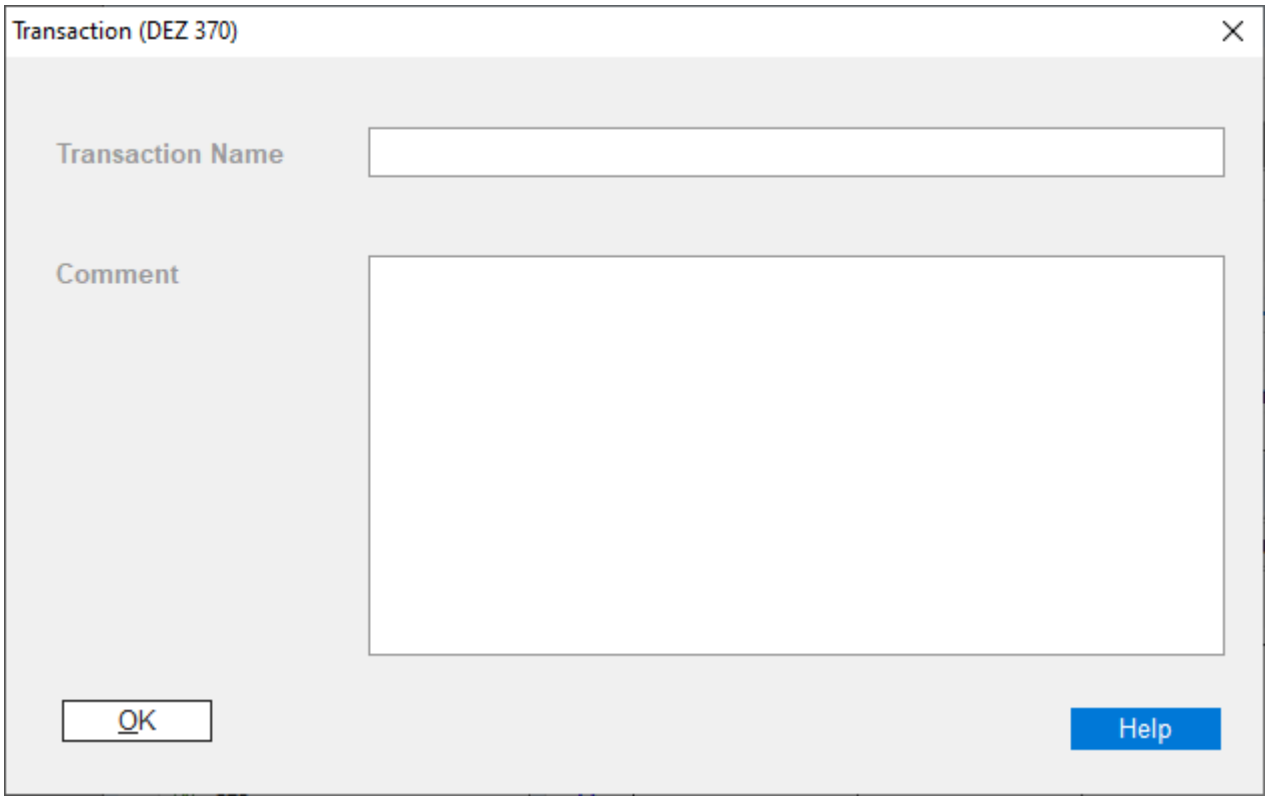
- Here you can subsequently vary the edit range in the **TIME RANGE** section.
- Now select the desired error code from the drop-down list box in the **STATUS** section.
- This list has been extended by the error code just added to the **MANUAL VALIDATION** group.
- Now select the desired error code.
- Confirm your entry with [**RETURN**].

The new error code is now set for all values or row entries in the marked time range.

7.5 Transaction for saving changes

If you change and save values or error codes in one or more time series, AquisNet proceeds as follows.

- Changes are written directly to one or more basic production time series.
- These changes remain persistently present in the time series.
- The data status before the changes is saved per transaction.
- This makes it possible to access the data before the change. The data is used in the event of a possible later restoration of a previous data status.
- To reduce the storage volume and the organisational effort of the transactions, AquisNet only saves the "affected" time range at a time and this - see above - with the data status before the changes.
- In addition to the pure measurement data of all affected time series before the change, AquisNet stores an identifier, a comment and the responsible user in transactions. Furthermore, the system stores the date and a transaction ID.
- When you close the table, the system displays a save dialog.
- If you confirm this with [**YES**], the system generates a transaction 99.



- To do this, enter a transaction name and a comment.
- Save the entries.
  The transaction is transferred to the data storage.
- The transaction can now be opened in the time series browser in the tree view. To do this, select one of the two entries for transactions from the drop-down list box.

The transaction for saving changes comprises these steps:

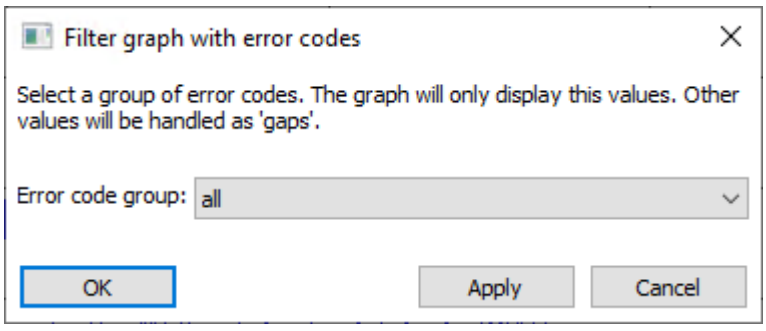
- Filter time series graph using an error code group 46
- Filter time series table using error codes 47

7.5.1 Filter time series graph using an error code group

You can adjust the display of the time series in the graph so that only values with certain error code groups are displayed.

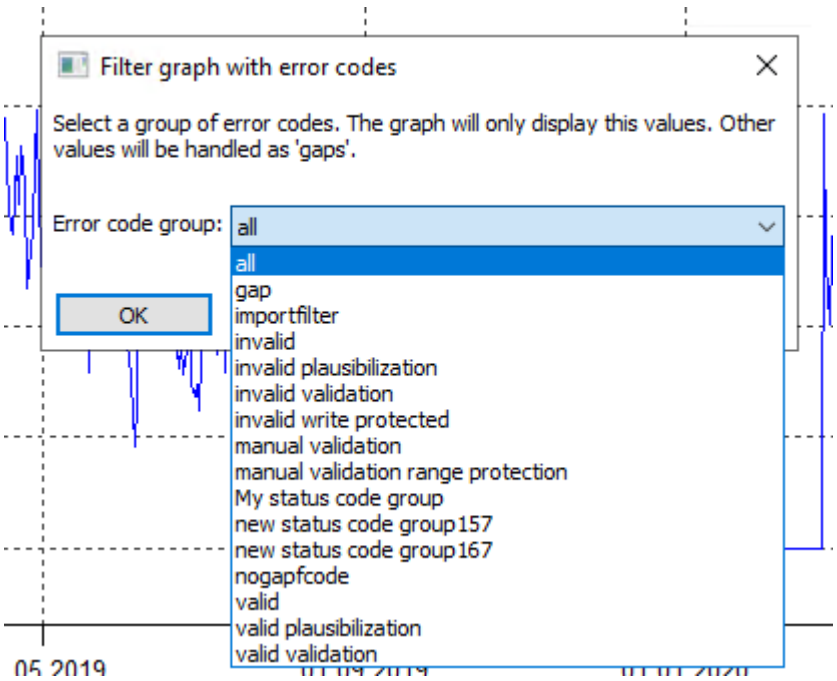
To filter time series by error codes,

- first select the **GRAPH SETTINGS** > **FILTER GRAPH WITH ERROR CODES** entry in the context menu.
- The **FILTER GRAPH WITH ERROR CODES** dialog opens:



This dialog lists all error code groups created in the system using a drop-down list box.

- Select an error code group:



Drop-down list box for the selection

- Confirm with [**ACCEPT**] and with [**OK**].

After confirming the selection, only the values that have an error code according to the preselected error code group are displayed in the graph. The result can be – depending on the error code group – a completely changed graph.

Note: The last selection of an error code group is saved in your user profile. The next time the graph is opened, this group is also used for filtering.

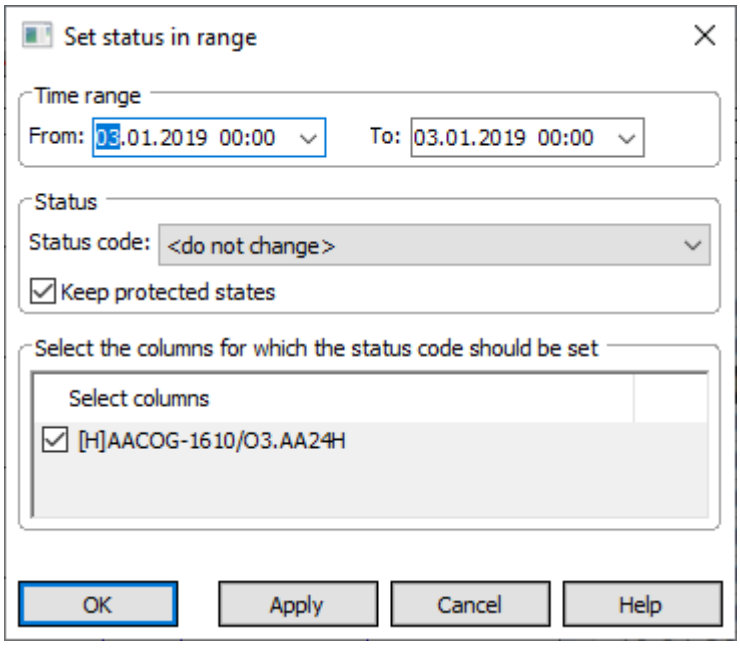
7.5.2 Filter time series table using error codes

You can adjust the time series in the table to show only values with a certain error code.

To filter time series by error codes,

- first select the **SET STATUS CODE** > **ERROR CODE** context menu entry.

The Set status for time range 96 dialog opens.



- This dialog lists all error code groups created in the system using a drop-down list.

- In the Time range section, you can select the time range for which you want to filter the table view. This way you can conveniently set the time range.
- Select an error code group.
- Confirm with [**ACCEPT**] and with [**OK**].

After confirming the selection, only the values that have an error code according to the preselected error code group are displayed in the table. The result can be – depending on the error code group – a completely changed table.

7.6 Adapting graph display

To display the time series in the graphics window, you can change the settings for

- axes 48,
- plots 49 and
- the legend 49

individually.

Additionally, the units of measure for the measured values can be freely configured.

7.6.1 Axis representation

The axis properties can be changed. Apart from the default presettings you can also specify extended presettings for the axis.

- Specifying axis settings 48
- Extended axis settings 48

7.6.1.1 Specifying axis settings

- Select the desired plot in the graph.
- Click the button or double-click on one axis.
- In the Axis properties 82 dialog, specify the type, scaling and visibility of the axis.
- You cannot deactivate the auto scaling function when the graph is in zoom mode!
- Confirm your selection with the [**OK**] button.

7.6.1.2 Extended axis settings

As for a time axis you can set the format and alignment of caption for each axis element (first value, consecutive values, last value). Up to two lines per caption can be defined here:

- Click the [**EXTENDED SETTINGS**] button in the axis dialog.

The Extended axis settings 83 dialog is displayed, where these settings are made.

The check box **ACTIVE** indicates whether the extended axis settings are valid; in that case, the check box is activated. To return to the default axis configuration, uncheck the box.

- Check the box in the frame of the upper/lower line section if you want to make the top line (usually for date) and the bottom line (usually for time) of the axis labelling separately visible.

These settings can be made separately for the upper and the lower caption line as well as individually for each axis element, or they can be applied for all axes.

- To do so, select the respective elements in the **AXIS ELEMENTS** section or select the elements in the preview window by clicking on them.
- Specify the caption format; you can choose between the following format elements in the drop-down list:
  - YY: year (two-digit number)
  - YYYY: year (four-digit number)
  - MM: month (two-digit number)
  - MMM: month (token consisting of three letters)
  - MMMM: month (full name)
  - DD: day (two-digit number)
  - DDD: weekday (token consisting of three letters)
  - DDDD: weekday (full name)
  - hh: hour (two-digit)
  - mm: minute (two-digit)
  - ss: second (two-digit)

These format elements can be supplemented with free text and combined as required (e.g. to include separators).

- Specify the caption alignment (e.g. left aligned or centred); choose an entry from the drop-down list.
- **Note** the difference between
  - centered (refers to the scale mark of the axis) and
  - centered in the interval.
    The alignment centred in the interval appears only, if all axis elements are selected.
- Finally, specify whether the caption is displayed only after a value change or always. In the first case, each value appears only once in the caption. In the second case, equal values may be repeated in the caption.

The advanced axis settings are also included in the layout templates, so that they may also be transferred to other time ranges and other time series of the same type.

Note: The length of the time range is stored in the layout template and may need to be adjusted when loaded for another time range.

7.6.2 Plot display

You can define

- the plot style (e.g. line style or colour),
- the symbols for the display of the single measured values,
- captions of the plots and
- grid settings (colour and style of the current grid)

for the plots displayed in the graph.

When captioning plots you can insert e.g. the date of each measurement or the current stage value in the graph. You can also draw a connecting line between a caption and measuring value.

- Click the  button or choose from the context menu of the graph (to be opened with the right mouse button) the Plot properties item.
- Make the desired settings in the Plot properties 86 dialog.
- To do so, open the desired tab and make your specifications.
- Confirm your selection with the [**ACCEPT**] button (the plot dialog remains open) or with [**OK**] (the window closes).

7.6.3 Legend display

Select the desired plot.

- Click the  button or double-click on one legend.
- Specify the desired position of the legend in the Legend 95 dialog; enter the X and Y coordinates (between 0 and 1000).
- Use the check boxes to control if the legend should be visible.
- If desired, you can save the legend position in order to use it in the next session.
- click the [**SAVE AS DEFAULT**] button.
- You can also move the legend with the mouse.
- Confirm your selection with the [**OK**] button.

7.7 Using crosshairs

A crosshair is available for the graph to facilitate reading the associated values at the axes. In addition, you can activate a search function for particular points within a settable snap radius.

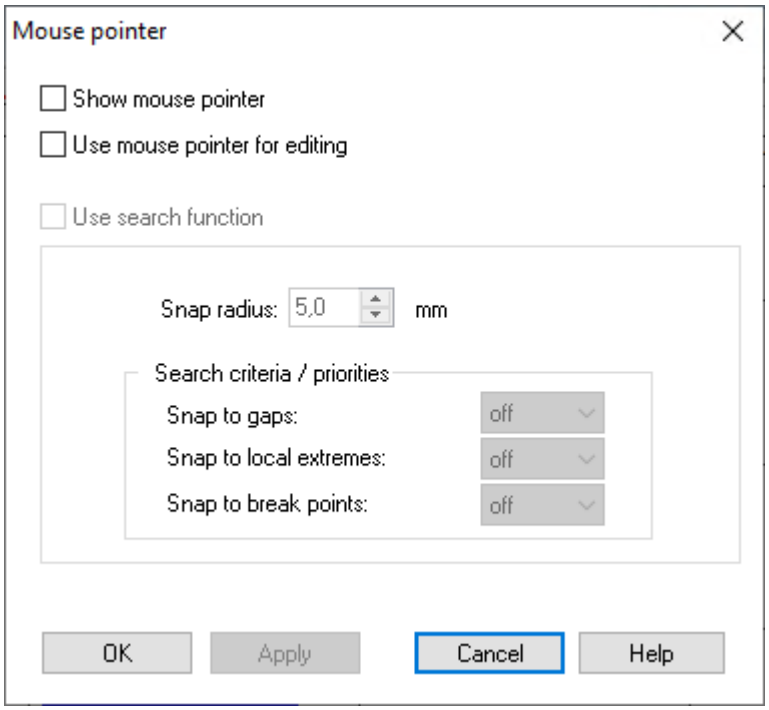
Working with crosshairs comprises these steps:

- Activating the crosshairs 49
- Activating search function/snapping 50
- Deactivating the crosshairs 50

7.7.1 Activating the crosshairs

The crosshairs is activated using the context menu of the graph:

- Start from the context menu of the graph (to be opened with the right mouse button) the **CROSSHAIRS** menu item.
  The Crosshairs 88 dialog opens:



- Check the Show crosshairs box to switch it on.
- You can activate the crosshairs for your current graph either by clicking [**ACCEPT**] (the dialog remains open) or by clicking [**OK**] (the dialog is closed).

7.7.2 Activating search function/snapping

In addition to just activating the crosshairs, a search function can also be activated that allows intelligent snapping to specific unusual values:

- Check the **USE SEARCH FUNCTION** box.
- If desired, set the snap radius in millimetres and
- define in the section **SEARCH CRITERIA/PRIORITIES** whether the cross hair snaps
- to gap starts and ends,
- to local extremes and/or
- inflection points (these are points where the gradient changes; not reversal points in terms of mathematics)

should snap.

You can then allocate to each of these search criteria a priority for the search function (choose from the list):

- **OFF**: deactivated
- **HIGH**: top priority
- **MEDIUM**: mean priority
- **LOW**: low priority

The crosshairs snaps to the points set here within the snap range. The point with the top search priority and with the smallest distance to the mouse position is snapped.

Note:

- The status bar of the graph shows the current mouse pointer position with date and time, including the current value; when snapping to found points, the exact values are read out from the database and displayed.
- When changing into the editing mode, the mouse pointer snaps to the points found.

7.7.3 Deactivating the crosshairs

To switch off the crosshairs pointer,

- select from the context menu of the graph the **CROSSHAIRS** item again and
- uncheck the **DISPLAY CROSSHAIRS** box in the Crosshairs 88 dialog.

7.8 Zooming graphs

For clearer editing of the time series, you can zoom into the display as needed. You can take back the zoom display in individual steps:

Working with the zoom comprises these steps:

- Zoom 51
- Zoom out 51

7.8.1 Zoom

- Select the desired plot.
- Press the [**CTRL**] key and scroll the mouse wheel.

7.8.2 Zoom out

- To zoom out step by step, select the  button.

7.9 Automatically adjusting the value range

When zooming in the time series graph and scrolling then along the time axis using the horizontal scroll bar, the time series is not always visible within the graph window. It depends on the course of the plot.

The following sub-chapters explain how to solve this problem.

- Shifting time axes 51

7.9.1 Shifting time axes

In addition to stepless scrolling using the horizontal scroll bar, you can also shift time series step by step along the time axis.

Only the currently selected time series is shifted. If several time series are displayed, a new axis is created for the selected time series so that you can compare time series at different times.

Shifting the time axes comprises these steps:

- Shifting time axis in fixed steps 51

7.9.1.1 Shifting time axis in fixed steps

- Select the desired plot.
- Select one of the  buttons.
- Select the  or  button to shift the plot by one section to the left or to the right.
- Select the  or  button to shift the plot by one marking section to the left or to the right.

7.10 Specifying axis and plot ranges

There are two visualisation elements in AquisNet: the time series graphics object and the tabular representation. In the graph window, you can display several plots in different curve areas:

- In the Graph window, select the  icon from the toolbar.

A menu is opened now.

- Select the **PLOT AREA** function and
- specify the new plot position for the selected plot.

Occupied plot ranges appear in grey with the time series symbol; the range of the plot selected in the graph window is shown in colour.

- To evenly distribute all existing plot ranges, select the **REARRANGE** function.

The ranges are rearranged with same height and width.

- However, you can also change the height of the ranges manually using the grey separating bar in the graph (mouse pointer becomes a double arrow).
- To bring separated ranges together again, activate the function **ONLY ONE PLOT RANGE**.

7.11 Additional information

You can display additional information in the graph for the measured values of the selected time series:

- Remarks on measured values.

Note the following prerequisites:

- The additional parameters, normal values and the information are displayed for the active time series. A time series (if more than one time series are displayed in the graph) is activated by the drop-down list at the top window border.
- The additional values to be shown must have been entered or calculated previously e.g. in the course of basic data maintenance of stations and parameters.
- The additional information is activated using the context menu of the graph (to be opened with the right mouse button): Select the **ADDITIONAL INFORMATION** menu item there and select the corresponding subfunction.
- If more than one time series is loaded in the graph, most functions will cause a dialog to be opened first where you can specify (by checking the box in front of the time series entry) for which time series you want to activate the additional information.

Working with additional information comprises these steps:

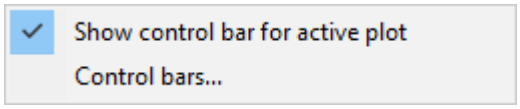
- The control bar 52
- Minimum, maximum and mean values 52

7.11.1 The control bar

Using the control bar, you can display the plausibilised values of an import time series. The bar displayed for this purpose informs you using markings about the time stamp(s) at which values were plausibilised.

- In the Graph window, select the icon from the toolbar.

Use the arrow button next to the icon to open the following drop-down menu:



- Select the **CONTROL BAR** entry.

If more than one time series is loaded in the graph, most functions will cause a dialog to be opened first where you can specify (by checking the box in front of the time series entry) for which time series you want to activate the additional information.

In the graphical representation of the selected time series, you now see the control bar(s) below the x-axis, which will inform you about the plausibilised values of the displayed time series using marks.

- To hide a bar, open the context menu again and select the **CONTROL BAR** entry once more.
- In the **DISPLAY CONTROL BAR SELECTION** dialog, check the box of the time series whose control bar you want to hide and confirm with [**ACCEPT**] and then with [**OK**].

The selected bar is hidden now.

7.11.2 Minimum, maximum and mean values

Using the context menu with which you open the selection dialog for the control bar, you also reach the function or the dialog Show minimum and maximum.

By checking the box in front of a time series entry, you can now determine for which time series the function is to be switched on.

- Confirm your selection with [**ACCEPT**] and then with [**OK**].
- In the graphical representation you will now get a minimum, maximum and mean value line for each selected time series, related to all values of the preselected period. The values for maximum, minimum and mean value are displayed on the corresponding line on the right.

7.12 Insert user defined graphical elements

In order to clarify, mark or comment on your plot representations, you can insert Graphical elements, e.g. lines, arrows, texts, etc., at any position in the graphic. Editing these graphical elements can change their appearance. You can also copy these graphical elements. You can execute all works with graphical elements from an overview.

You can start inserting elements from the toolbar icons or using the graph context menu. The context menu entries exist only if the two icons are available in the toolbar.

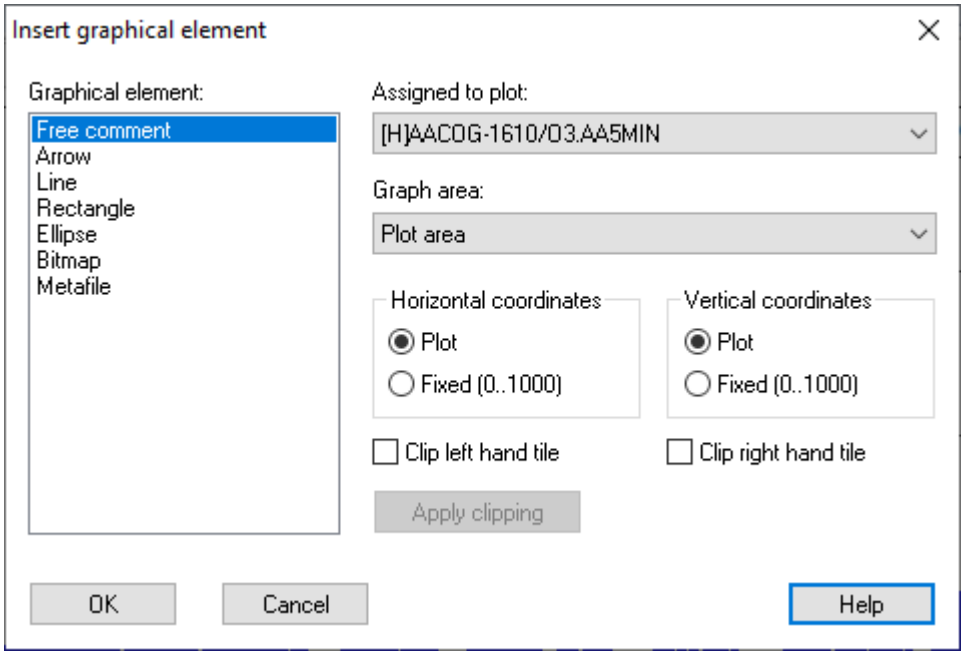
Working with user-defined graphical elements comprises these steps:

- Inserting a graphical element 53
- Editing a graphical element 54
- Graphical elements and their properties 55
- Copying graphical elements 58
- List of all graphical elements 59

7.12.1 Inserting a graphical element

To insert a new graphical element,

- click the button.
- alternatively, you can open the Insert graphical element context menu entry.
  The Insert graphical element 89 dialog opens:



The Plots list box shows all time series (plots) contained in the graph.

- Choose the time series to which the graphical element should be added.
- Select the element to be added to the time series from the Graphical element list box.

Using the radio buttons in the Assigned to section you specify the connection of the graphical element with the time series:

- **AXIS (HORIZONTAL/VERTICAL)**:The graphical element is connected to the plot (the x- as well as the y-coordinates of the graphical element correspond to the plot coordinates), i.e. the graphical element is always exactly at the same position relatively to the plot. This means that the graphical element is moved along the axes when the display is enlarged or reduced or when the plot is moved.
- **AXIS (HORIZONTAL)**: The x-direction of the graphical element depends on the plot whereas the y-direction remains constant (coordinates 0 to 1000). That means, it shifts only in x-direction, not in y-direction.
- **AXIS (VERTICAL)**: The y-direction of the graphical element depends on the plot whereas the x-direction remains constant (coordinates 0 to 1000). That means, it shifts only in y-direction, not in x-direction.
- **NONE**: The graphical element remains constant in both directions, irrespective of the plot (both coordinates between 0 - 1000). It therefore does not change its position even when enlarged or reduced or moved along the axes.
- **HEADER**: The graphical element is inserted into the header. If a header does not yet exist, it will be created. Both coordinates range from 0 to 1000.

- **FOOTER**: The graphical element is inserted into the footer. If a footer does not yet exist, it will be created. Both coordinates range from 0 to 1000.

Note: The user defined graphical elements in the header and footer are assigned to the time series. If a time series is deleted from the graph, the graphical elements belonging to that time series will also be deleted from header and footer!

After having made these specifications,

- confirm with [**OK**].

The dialog closes; the mouse pointer changes to a cross ([+]).

- Click the left mouse button to define the start point of the graphical element within the graph window.
- Hold the left mouse button down and drag the mouse in any direction in order to define the end point of the graphical element.

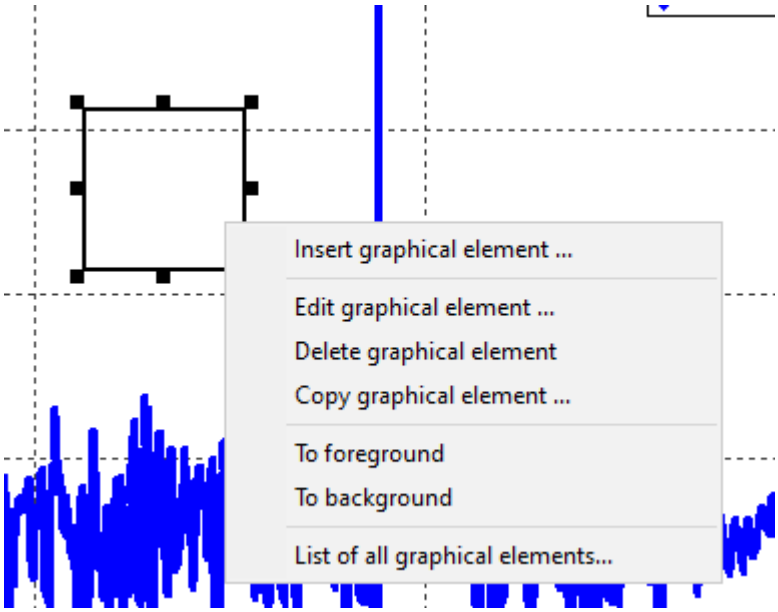
While you are dragging, the graphical element and the surrounding rectangle are already shown.

- When releasing the left mouse button, the end point of the graphical element is defined and the insertion process is completed.
- When inserting a text element, the edit window will be opened immediately on clicking the left mouse button. In this window you enter your text and specify its layout and formatting.

7.12.2 Editing a graphical element

To edit a graphical element

- click the button from the toolbar.
  The button is then active (pressed in). The **GRAPHICAL ELEMENTS** dialog opens, which contains a list of all graphical elements that are located at this position.
- Select the desired graphical element here.
  It will change to a selected element.
- Right-click in the selection rectangle to open the following context menu:



- The **INSERT GRAPHICAL ELEMENTS** menu item has the same functionality as the corresponding icon in the toolbar (or the menu entry of the same name in the context menu of the whole graph).
- Selecting the **EDIT GRAPHICAL ELEMENT** menu item (or double-clicking within the rectangle) shows the tab(s) to edit the properties of the corresponding graphical element (in the Edit <Name of graphical element> 90 dialog).
- The menu item **DELETE GRAPHICAL ELEMENT** (or the Backspace or [**DEL**] key) deletes the selected graphical element; there will be no warning notice.
- The **COPY GRAPHICAL ELEMENT** menu item lets you open the Copy graphical element 91 dialog by which you can copy a graphical element to another plot and with another assignment.
- The **TO FOREGROUND** menu item lets you draw the selected graphical element on top of all other elements of the same time series and the same assignment. To background moves the element to the lowest position.
- The **LIST OF ALL GRAPHICAL ELEMENTS** menu item opens the Graphical elements 91 dialog containing a list of all graph elements; you may select, edit or delete a graphical element from this dialog. You can also open this dialog from the normal graph context menu.
  - As long as you are in the mode “Edit graphical elements”, neither time series dialogs nor axis property dialogs can be called by a double click.

- When clicking next to the rectangle, the selected graphical element will be deselected.
- Should you change to another application using the WINDOWS task bar and return again, the focus is no longer on the graph window, although it seems to be so. To delete the graphical element using your keyboard ([**DEL**] or Backspace key), you must have previously clicked inside the rectangle to return the focus to the window.

7.12.3 Graphical elements and their properties

Each type of graphical element has a separate tab showing the properties of the element. You can change the properties on these tabs.

Note the following characteristic features of the different graphical elements:

- Each graphical element is defined by position and direction coordinates.
- All elements are defined by start coordinates, but only arrows and lines are defined also by end coordinates.
- Texts are defined only by start coordinates.
- Rectangles or ellipses are defined by their height and width (relative coordinates beginning at the starting point).
- The units of the coordinates depend on the assignment of the graphical element.
- As already mentioned, the fixed axes are represented in a value range between 0 and 1000.
- If the x-axis is not fixed, its value range represents the number of seconds beginning at a fixed start date.
- Apart from this, an individual name can be allocated to each graphical element (in the **NAME OF <ELEMENT>** box at the lower window border), so that it is easier to identify it later (for example in the list of graphical elements).

For each graphical element, you can set in the **STYLE** tab

- the line appearance via
  - line style,
  - line colour and
  - line width

  and
- the fill mode via
  - fill colour and
  - fill pattern

.

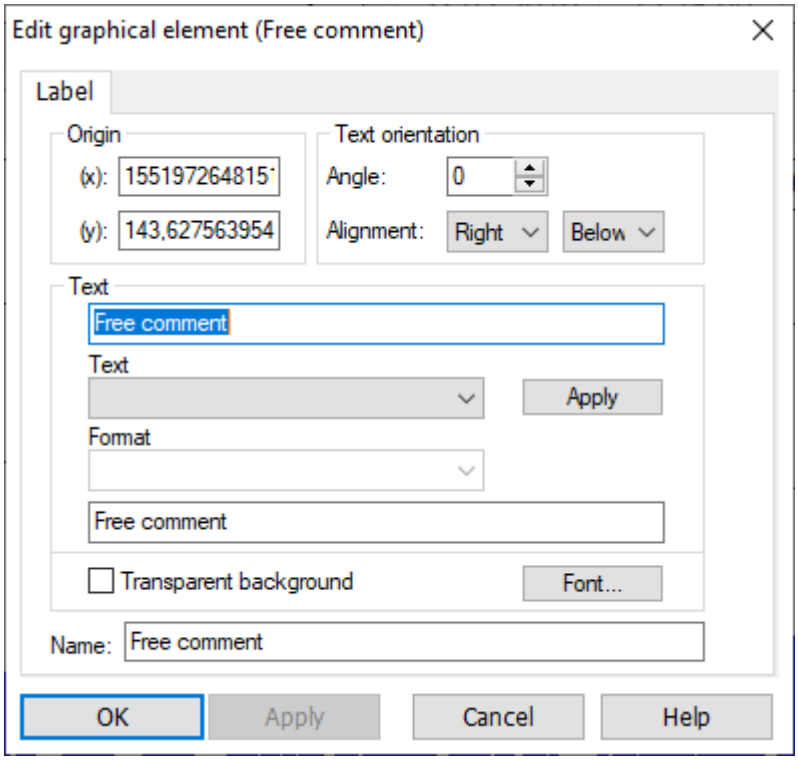
The start date is specified in the `luqs.ini` initialisation file.

Working with graphical elements comprises these steps:

- Inserting/editing a text element 56
- Inserting/editing lines, rectangles, ellipses and arrows 57

7.12.3.1 Inserting/editing a text element

The property dialog appears automatically when you insert a text element; the text (label) tab contains all data that can be used to control the layout of the graph text element.



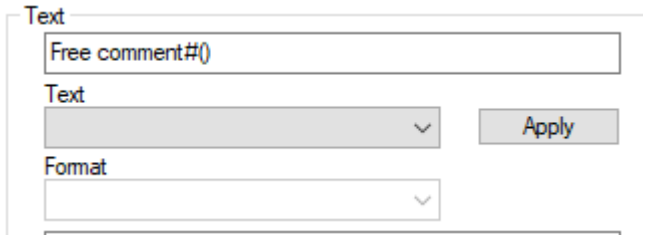
- Origin:
- This section contains the position and original coordinates of the graphical text element. You can enter here other coordinates manually.

The graphical text element will be shifted according to your definition.

- Text orientation (section):
- Angle: Specify the angle (in degrees) of the text in this field. 0 degree corresponds to a horizontal text orientation. The text will be rotated clockwise around its end point by the specified angle (at 180° the text will be upside down).
- Horizontal alignment selection box: Sets the origin point of the text in x-direction. Possible settings are right alignment, centred and left alignment.
- Vertical alignment selection box: Sets the origin point of the text in y-direction. The following entries are possible: above, central and below.

An example: "right above" means that the origin point of the text is in the bottom left corner of the text (i.e. the text is oriented to the top right).

- Text entry (section):



- In the text box of the section you can enter an own text; such a text can also be combined for example with punctuation marks and the variables described below.
- In the **TEXT VARIABLES** list box you will find a lot of time series specific (for example time series name or station name) or preconfigured variables (operator, place of creation) that can be inserted into the text.
- Here you can display for example the Axis time range as text element. To do so, select the variables first time stamp of the axis and last time stamp of the axis.

All variables with time data can be specified even more exactly:

- Select in the now active **FORMATTING** combo box the date format from the list or enter it manually.
  Use the WINDOWS default:
  - T: one-digit date (without leading zero; e.g. 1),
  - TT: two-digit date (with leading zero; e.g. 01),
  - TTT: three-digit abbreviation of the weekday (e.g. Mon),
  - TTTT: long name of weekday (e.g. Monday),
  - M: one-digit month (without leading zero; e.g. 1),
  - MM: two-digit month (with leading zero; e.g. 01),
  - MMM: three-digit abbreviation of the month (e.g. Feb),
  - MMMM: long name of month (e.g. February),
  - YY: two-digit year (with leading zero; e.g. 01),
  - YYYY: four-digit year (e.g. 2001).

  Note: When opening the dialog, no date format is yet selected. When selecting a text variable with date format repeatedly, the format selected last will be offered.
- Select the desired variable from the drop-down list,
- check whether you have defined the text element as desired using the display box (highlighted in grey), displaying the plain text inserted as element into the graph, and
- finally click the [**ACCEPT**] button.

The chosen text variable will be inserted into the text (see Text entry box where the variable is displayed (can be identified by the hash; example: `#(time series name)` or for formats with time specification: `#(start time)`.

Note: You shouldn't make any further entries within the variable; the variable contents shouldn't be changed. The program may not be able to identify them if they have been changed.

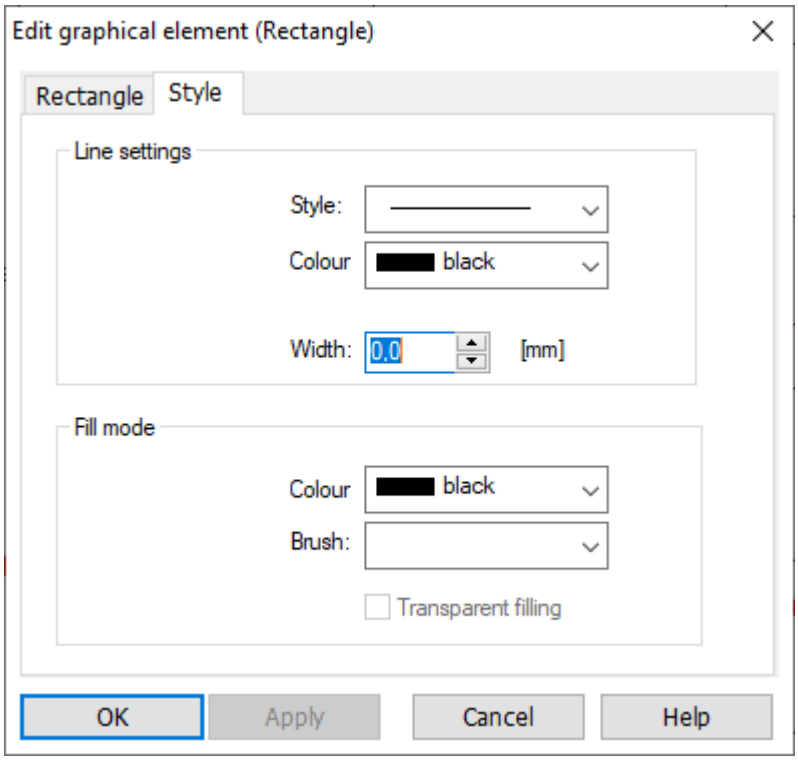
When clicking the [**FONT**] button, you will get the usual dialog for font selection and its properties, as known from WINDOWS programs.

For the Operator and Place of creation entries of the Text variables drop-down list box, the respective variables are read in from the `aquisnet.ini` initialisation file (`[Report_Configuration]` section, `InstAdministration` and `InstLocation` entries).

In addition to the axis time ranges, you can display for example also the time stamp of the first, the last or the current value of the time series as text element.

7.12.3.2 Inserting/editing lines, rectangles, ellipses and arrows

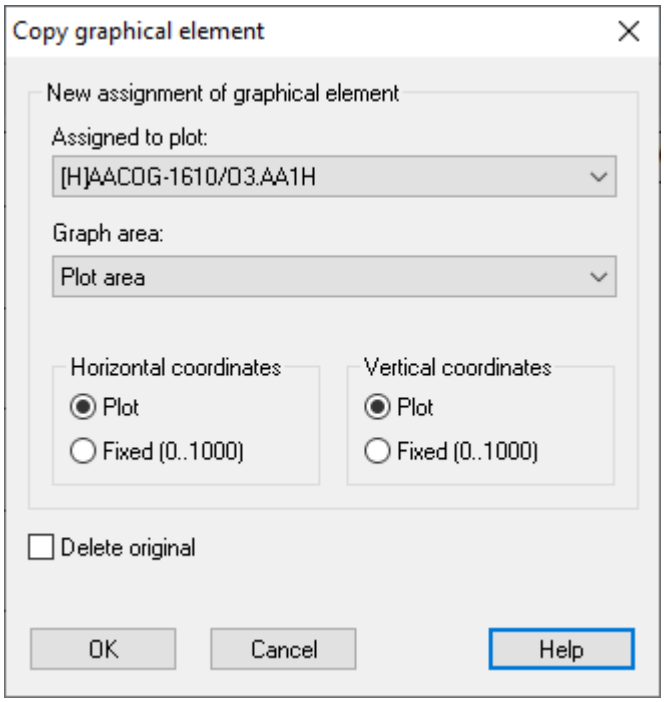
The second tab page **STYLE** in the [Name of graphical element] edit 90 form (not existing for Free Comment) shows the properties of pen (section **LINE STYLE**) and brush (section **FILL MODE**) of line and area (rectangles, ellipses) graphical elements. Only graphical elements with a closed area have brush properties whose fill colour and pattern can be defined (i.e. not for curves/lines).



- Specify the line style:
- The drop-down list box **LINE STYLE** offers you a selection of various line types.
- Select the line colour from the drop-down list.
- Enter the line width in millimeters either manually or choose the width by means of the arrow keys.
- Specify the fill mode:
  - Choose a fill colour from the drop-down list.
  - Choose from the drop-down list box **FILL PATTERN** the desired pattern. If the area is completely filled with a colour or if it remains white, the check box **TRANSPARENT FILLING** is deactivated.
  - When checking this box, the filling becomes transparent.

7.12.4 Copying graphical elements

You can copy individual graphical elements to avoid repeated inserting and drawing.



The Plots drop-down list box shows you all time series of the graph window. The plot to which the graphical element to be copied belongs is selected. The **ASSIGNED TO** section contains the assignments specified by you when inserting the graphical element (see ch. Inserting graphical elements 53).

- Select the plot where the graphical element is copied to.
- Specify the assignment as described in the Inserting graphical elements 53 tutorial.

If the graphical element to be copied is to be (irreversibly) deleted after copying,

- check the **DELETE ORIGINAL** box.
- Start copying by clicking [**OK**];

afterwards, the graphical element is copied to the selected time series with the specified assignment (e.g. “Axis (vertical)”).

The copied graphical element is shown approx. one centimetre on the right under the original and becomes the selected graphical element.

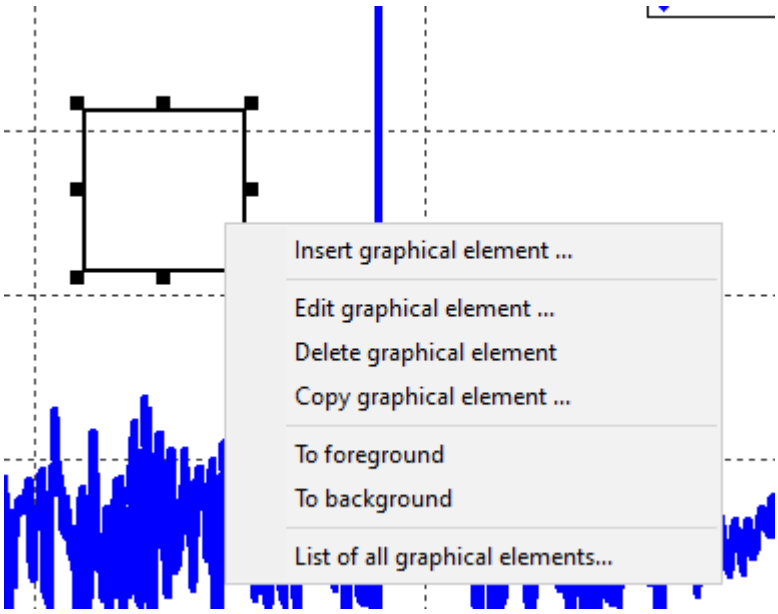
Hint: If you want to copy a graphical element to the same time series with the same assignment, you do not need to open this dialog but

- you can instantly copy a selected element using the keyboard shortcut [**CTRL**]+[**E**].

7.12.5 List of all graphical elements

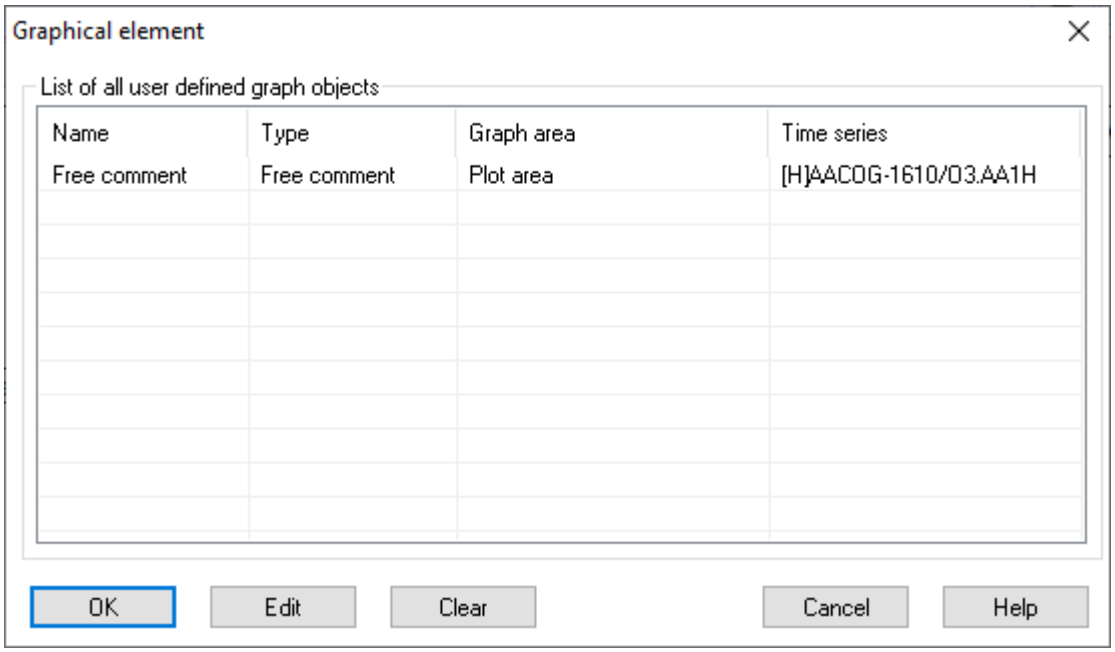
You can have displayed any time a list of all user-defined graphical elements created in the graphical window; from this list you can even edit or delete graphical elements:

- Click the context menu item **LIST OF ALL GRAPHICAL ELEMENTS FROM THE ENTIRE GRAPHIC**.
- Select the desired graphical element (by clicking within the rectangle surrounding the element).
- Right-click in the selection rectangle to open the following context menu:



- Select the context menu item **LIST OF ALL GRAPHICAL ELEMENTS**.

The Graphical element 91 dialog is opened:



When opening the dialog, the graphical element currently selected in the graph is selected in the single boxes of the list. The list entries indicate every graphical element by

- its name,
- its type,
- its assignment and
- the name of the time series which it has been assigned to.

Use this window to select, edit and delete an element:

- Choose the desired element from the list and
- click the [**OK**] button.

The dialog will be closed and the corresponding graphical element will be selected.

- Click the Edit button to edit the element.

The selected graphical element's tab opens.

To delete a graphical element from this list (alternatively to [**DEL**] or Backspace key with selected element in the graph),

- click [**DELETE**].

First, deletion is only temporary: the graphical elements will be inserted in a so-called deletion list.

- If you then exit the window with [**CANCEL**], the deletion list is cleared and none of the graphical elements are removed.
- Clicking the [**OK**] button will irrevocably delete all elements in the deletion list without any inquiry. If you have moved a graphical element to the delete list, then want to change another element with [**EDIT**], you will get a message window asking you whether to delete the elements from the deletion list before editing or whether to cancel the deletion process – i.e. not delete anything.

7.13 Filter time series graph using an error code group

You can adjust the time series display in the graph to show only values with certain error codes. Thus, besides the valid values, graphs can also handle e.g. only invalid values or also other aspects. The error code group to be used for display can be selected in the graph. Based on error codes in a selected error code group, values in the graph are subsequently filtered. Values whose code is not included in the selected group are not displayed (in the graph) or are displayed as – (in the table).

- To filter time series by error code groups,
- first select the **GRAPH SETTINGS** > **FILTER GRAPH WITH ERROR CODES** entry in the context menu.
- The **FILTER GRAPH WITH ERROR CODES** dialog opens.
- This dialog lists all error code groups created in the system using a drop-down list.
- Select an error code group.
- Confirm with [**ACCEPT**] and with [**OK**].

After confirming the selection, only the values that have an error code according to the preselected error code group are displayed in the graph. The result can be – depending on the error code group – a completely changed graph. The last selected error code group is saved for each user. Hereby it is important to pay attention, if e.g. after viewing invalid values the application is closed and at the next opening an initially unexpected graphic is displayed.

7.14 Visualise changes in time series (e.g. transactions)

Stored changes to measured values and error codes are displayed by the change bar or control bar below a graph. This change bar marks all changes that have been made to a time series and orders them sequentially in time. In this way, users can record changes made to a time series "at a glance".

Information accessed by the change bar is comma-separated and structured in the following way, so that the display is standardised accordingly:

- User, transaction name, comment, execution time.

In the graphical display, AquisNet automatically enumerates the changes, ordered by time stamp and time series.

Detailed information on the changes of a transaction can be found in the transaction detail displays (see above).

7.15 Synchronising graphs and tables

Time series are represented in the graph and/or table window. Whereas the graph gives you a survey of the time series course, the table shows you the single values exactly.

You may synchronise both table and graph to make the best use of these two advantages for representations. When you click on a measured value in the graph in this mode, the corresponding value will be highlighted in the table.

This also works vice versa: when clicking on a value in the table, the corresponding value will be highlighted in the graph.

Working with graphs and tables comprises these topics:

7.15.1 Synchronisation mode ON

- Select the desired value in the **TABLE** dialog.
- Click the right mouse button.
- Select the **SYNCHRONISE TABLE AND GRAPH** context menu item.
- Click on the desired value.

The corresponding measured value is selected in the table.

7.15.2 Synchronisation mode OFF

- Select the desired value in the **TABLE** dialog.
- Click the right mouse button.
- Select the **SYNCHRONISE TABLE AND GRAPH** context menu item again.

Synchronisation mode is switched off again.

7.16 Summary

When displaying time series graphically, you can individually customize axes, plots, legends and grids for display and editing purposes.

For your standard work with the time series graphs, the following functions are available:

- Select plot
- Update display
- Undo action
- Restore action
- Print (default printout and special printouts in fixed sizes) and
- Save

.

You can view time series in a graph and in a table, also synchronised (to view values at the same time in the graph and the table). Time series are represented in the graph and/or table window. Whereas the graph gives you a survey of the time series course, the table shows you the single values exactly. Here, for example, the type of measured value rounding can be set.

The zoom feature lets you view a portion of the time series in greater detail.

To have the time series graph always displayed centred in the graph window when you are in zoom mode and scrolling along the time axis, you have to select the value range to be adjusted automatically when zooming.

As well as scrolling along the time axis, you can also shift the time series along the time axis.

You can display one or more graphs in the graph window. To distinguish individual graphs, the plots are shown for example in different colours and the measured values using different symbols. The iteration sequence is used to specify the graph setting for the first, the second, etc. plot.

You can display additional information in the graph for the measured values of the selected time series.

8 Editing time series interactively

In the following, you will learn how to edit individual measured values or plot ranges in tables and graphs. Final release of time series is also described.

Content of this chapter:

8.1 Edit time series interactively in the data collection center DMO

AquisNet offers extensive possibilities to edit single measured values or plot ranges in tables or graph displays. These features let you correct faulty data or fill missing periods.

8.2 Specify remarks

The graph and table view of the measured values allows to define remarks for whole stations, single parameters or time series.

Each remark applies to a specific time period. Internal or external remarks can be assigned (the latter are also displayed in the reports). Some remarks are automatically created by the system (if measurement values have been changed, for example).

The remarks are displayed in a control bar; they can be entered in the table or called using the context menu of the graph.

The control bar also enables the display of the plausibility check messages (highlighted in red); however, these cannot be edited.

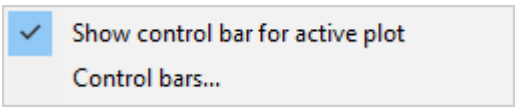
Working with remarks comprises these steps:

8.2.1 Switch off control bar

You can switch the control bar off again using the toolbar.

In the Graph window, select the  icon from the toolbar.

Use the arrow button next to the icon to open the following drop-down menu:



- Select the **SHOW CONTROL BAR FOR ACTIVE PLOT** entry.

8.3 Edit Measured Values in a Table

It is not only possible to edit measured values in the graph display of a time series but you can also edit single values in the related table window and view the graph changes directly using the synchronisation mode. You can edit the values in the table by editing single cells just as you would a spreadsheet.

Working with the table editing of measured values comprises the following options:

8.3.1 Manual entry into the table

Apart from import and remote call operations, you can use manual entry to input measured values into the database. You can also add missing values into the table this way.

This feature is only available for editable time series. Check first to see whether manual entry is possible.

This entry of measured values is a useful aid for stations and transducers without automatic recording and/or transfer of data.

- Edit cell entries by double-clicking them.
- Enter the following for each value:
- the measured value (either as absolute value (e.g. `S [cm]`) or as relative value with reference to a height system (`S [maOD]`),
- the error code {0,7,21,306}, and
- remarks/hints, if required
- Confirm your date selection with [**RETURN**]; the table will automatically switch over to the next useful input position.
  Note possible problems when entering the time manually in the table (24.00 h): entering 24.00 h is not allowed. Thus, the time following the last entry is entered automatically.
- Enter 0.00 h of the next day.
  Entering 24:00 h is not necessary!
- Move the mouse pointer on the table column for measured values (e.g. `S [cm]`).
- Move the mouse pointer to the table column of the error code.
- Double-click on the field that shows the error code.

You can now select **ERROR CODE** in a drop-down list box.

- Remarks or tasks for the single values can be entered in the **REMARKS** text column.

8.3.2 Synchronisation mode ON

- Select the desired value in the **TABLE** dialog.
- Click the right mouse button.
- Select the **SYNCHRONISE TABLE AND GRAPH** context menu item.
- Select the desired value.
- Edit the desired value. You can edit the values in the table by editing single cells just as you would a spreadsheet.

8.4 Selecting Measured Values

To edit measured values, first select the desired measured value range and switch to edit mode.

Then select the measured values that you want to change.

You may select one single measured value, several adjoining measured values or several non-adjoining ones:

- Selecting Single Values 64
- Selecting Neighbouring Values 64
- Selecting Non-adjoining Values 65

8.4.1 Selecting Single Values

- Select the desired plot.
- Switch to editing mode for the measured values to be changed.
- Select the desired measured value with the mouse.

8.4.2 Selecting Neighbouring Values

- Select the desired plot.
- Switch to editing mode for the measured values to be changed.
- Select the first measured value at the beginning of the range with the mouse and hold down [**SHIFT**] and click on the last measured value at the end of the range.

8.4.3 Selecting Non-adjoining Values

- Select the desired plot.
- Switch to editing mode for the measured values to be changed.
- Select the first desired measured value by using the mouse and using [**CTRL**] + click on further desired values.

8.5 Changing Measured Values

After you have selected measured values in editing mode, you can change the measured values with the mouse.

You can move or tip measured values vertically or proportionally.

Changing measured values comprises the following options:

- Shifting Measured Values Vertically 65
- Shifting Measured Values Proportionally 65
- Tipping Measured Values 65

8.5.1 Shifting Measured Values Vertically

- Select the desired plot.
- Switch to editing mode for the measured values to be changed.
- Select the desired measured values.
- Move the selected value with the mouse.

8.5.2 Shifting Measured Values Proportionally

- Select the desired plot.
- Switch to editing mode for the measured values to be changed.
- Select the desired measured values.
- Move the selected measured values proportionally with the mouse while holding down the [**CTRL**] key.

8.5.3 Tipping Measured Values

- Select the desired plot.
- Switch to editing mode for the measured values to be changed.
- Select the desired measured values.
- Move the selected measured values proportionally with the mouse while holding down the [**CTRL**] and [**SHIFT**] keys.

8.6 Editing plot ranges

The editing mode allows you to edit selected measured values. It is also possible to change a whole range in one go, for example to fill it with a constant or to fit another time series into that range:

- Change plot ranges 65
- Inserting or fitting plots 66

8.6.1 Change plot ranges

- Select the desired plot.
- Click the  button.
- Use the mouse to draw a limiting rectangle over the area you want to edit.
- Click the  icon or - if the mouse pointer is set within the working range - open the context menu of the range with the right mouse button.
- Select the desired function.
- The following functions are available:
  - Fill with a constant (*)
  - Shift vertically by the constant (*)
  - Flip range horizontally (*)
  - Fill gaps with a constant (*)
  - Fill gaps with last value before gap
  - Fill using linear interpolation

  - Fill gaps by linear interpolation
  - Fill gaps by spline interpolation
  - Undo selection
  - Set gap range
- For the functions marked with (*) you have to specify the desired constant in the following dialog and confirm your selection with [**OK**].

8.6.2 Inserting or fitting plots

You can insert or fit time series when two time series at least are opened in the **GRAPH** form.

- Select the plot into which you want to insert another plot.
- Click the  button.
- Use the mouse to draw a limiting rectangle over the area you want to edit.
- Click the  button.
- Specify in the Insertion/fitting dialog the source time series and the insertion/fitting mode.
  The following functions are available:
  - Copy
  - Fit left
  - Fit right
  - Fit at both sides
  - Fit proportionally left
  - Fit proportionally right
  - Proportional fitting at both ends
- Select a proportional fit to proportionally adjust the course of the plot in the edit range to the transition points.
- Confirm your selection with the [**OK**] button.

8.7 Summary

AquisNet saves data in relational databases and has a powerful time series management. This time series management prevents currently edited time series from being opened and edited elsewhere.

Each single measured value of a time series is displayed in the graph window by a special graph symbol.

To edit measured values of a time series, switch with single measured values, several neighbouring or several non-neighbouring measured values into the editing mode. The values currently in editing mode are represented by a special symbol: a cross with arrows at its ends.

After you have selected measured values in editing mode, you can change the measured values with the mouse. You may shift or tip measured values vertically or proportionally, delete or insert them. It is also possible to change a whole range in one go, for example to fill it with a constant or to fit another time series into that range.

The graph offers you the possibility of calculating time series from formulas. You can work with these temporary time series "as usual"; however, you cannot save them in the database. You can also calculate integrals from the graph.

9 Time series browser: Plausibility check

A plausibility check is an automatic check of the values of imported time series using preconfigured check criteria that can be changed by you as the user. It is used exclusively for base production time series. If the values do not correspond to the set plausibility criteria, the values of the time series are marked accordingly and considered invalid for use in further calculations.

The plausibility check comprises the following options:

9.1 Plausibility check types

The following plausibility check types with different parameters are available.

- Slope

A sequence of base production values is compared by this plausibility check type with the respective temporal predecessors.

The amount of change in a value compared to the previous value and the maximum allowed gradient within a likewise parametrisable time period can be parameterised:

- maximum allowed percentage delta
- maximum allowed gradient
- Time range in minutes
- Completeness

The imported values of the import time series are checked with this plausibility type for completeness within the time range to be checked for plausibility. This makes it particularly easy to find gaps in graphs by means of the associated plausibility comment.

- Outliers

This plausibility check type checks a time series for outliers or extreme values.

The check is carried out against an upper limit. If a value exceeds this limit, this value is marked with the status code "99".

- Interpolator

Imported values are checked for gaps with this plausibility check type.

"Gap" in this context means a sequence of values with status code that are included in the invalid status code group, and are therefore invalid.

Gaps are only interpolated if a gap is exactly one value wide. Otherwise there is no interpolation. This can be used when you want to fill in individual gaps caused by calibrations with valid values.

If interpolation was done, the interpolated value gets the status code "6" ("valid after interpolation").

To interpolate the missing value, the system calculates the arithmetic mean from the two neighbouring values of the gap

- 2-way Outlier

Imported values are checked for outliers upwards or downwards.

You can specify a maximum allowed delta or accept the value suggested by the system.

AquisNet identifies two consecutive measured values whose delta is greater than the permitted one as outliers.

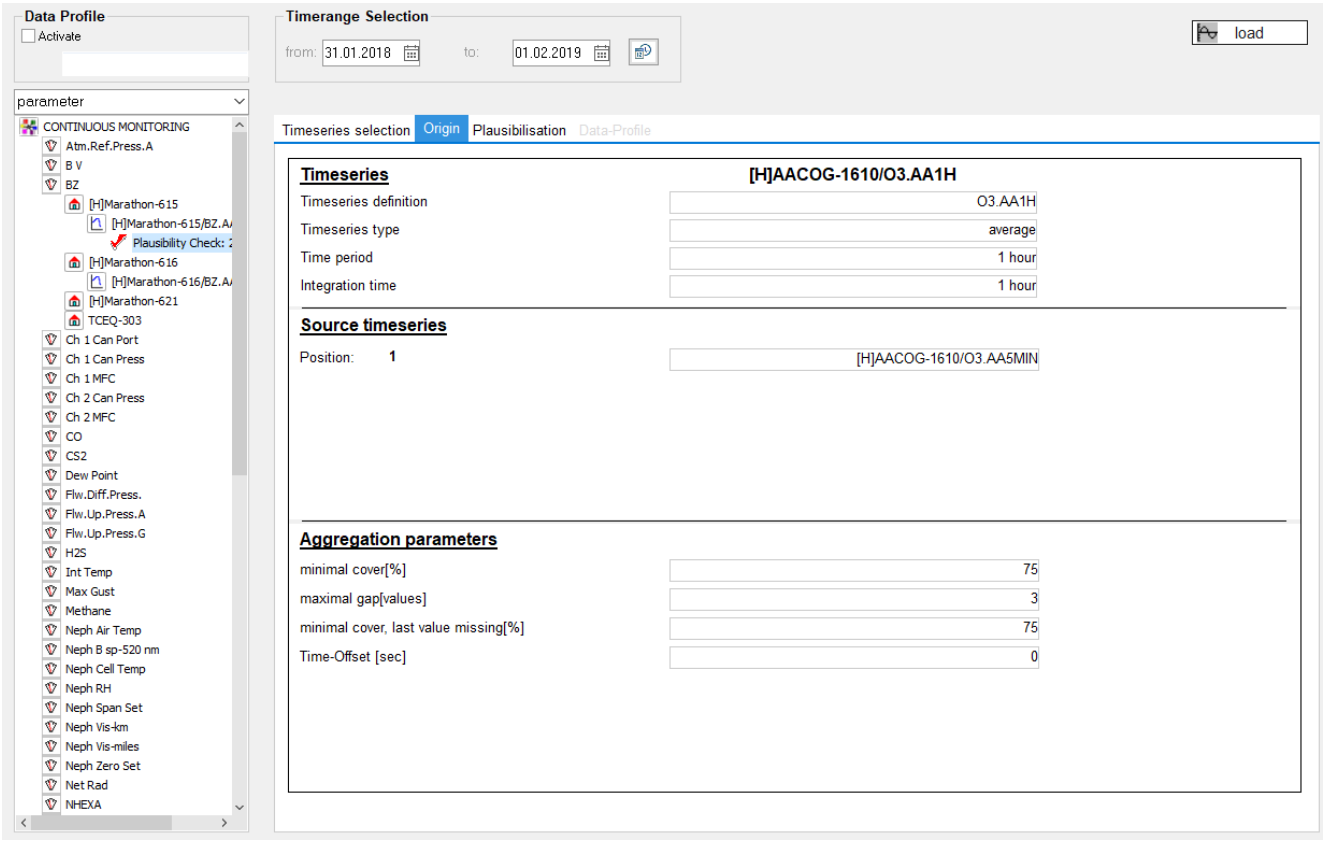
9.2 Call up plausibility information on time series

Details on the plausibility check of a time series as well as the call-up of a plausibility checker for the post-processing of time series measured values can be called up via the context menu for import time series in the browser tree. This function is not available for time series generated by origins.

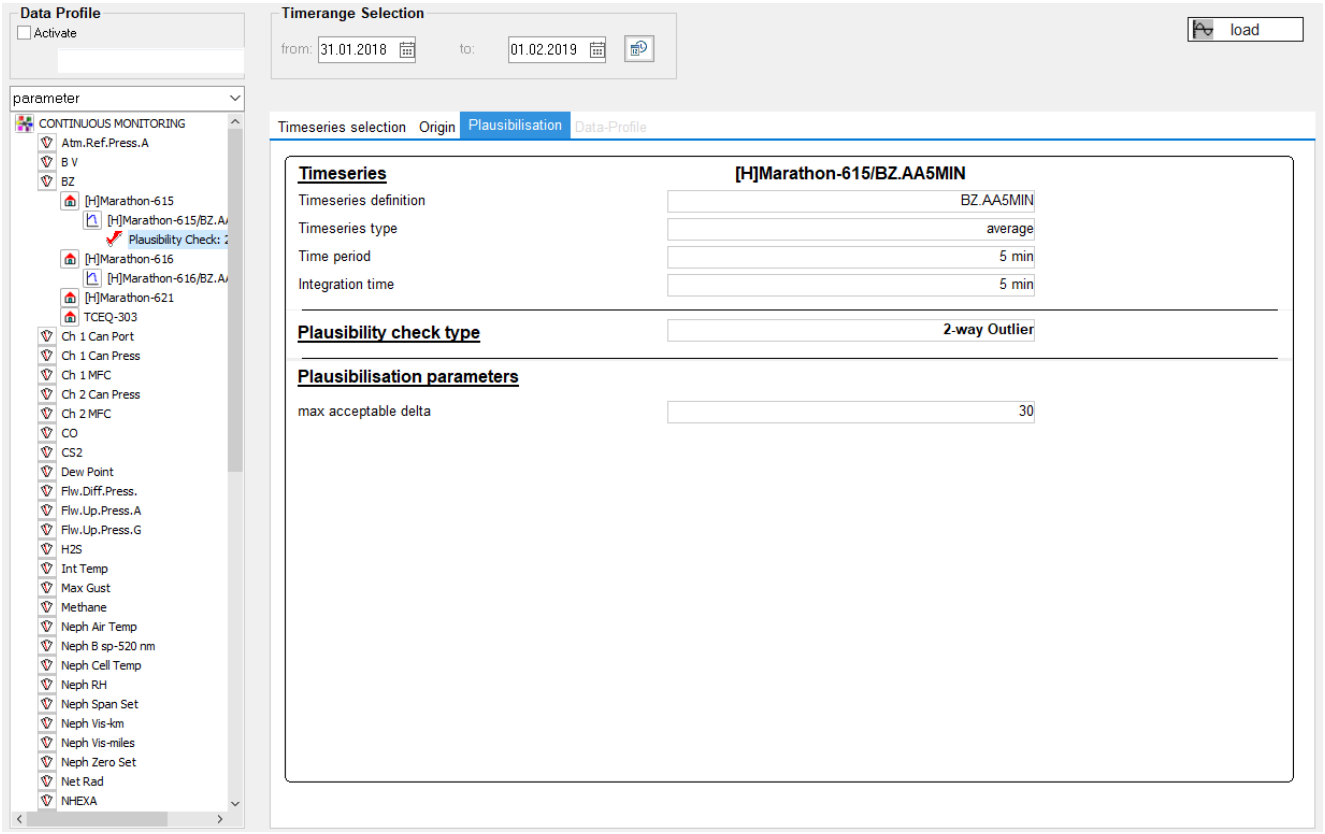
To call plausibility information on time series,

- expand the tree view (independent of the browser tree configuration) to the time series level.
- double-click the desired time series:
- If the time series cannot be extended with regard to an origin entry or a plausibility checker, it is an import time series for which no plausibility checker has been created.

- The Origin entry tells you that it is a time series generated by an origin. The plausibility check function is not available for time series generated by emergence:



- If the time series can be extended with regard to a plausibility check entry, the tree entry contains the type of plausibility check used.
  If you find several (but no more than five) plausibility types under the time series, a red arrow points to the type whose parameters are displayed in the right-hand part of the dialog.
- A mouse click on a plausibility type displays the stored details of the time series in the main window.
  In the plausibility parameters section (in the right part of the dialog), the parameters of a type are displayed.
- By double-clicking on the plausibility check entry, the stored details are displayed in the main window:

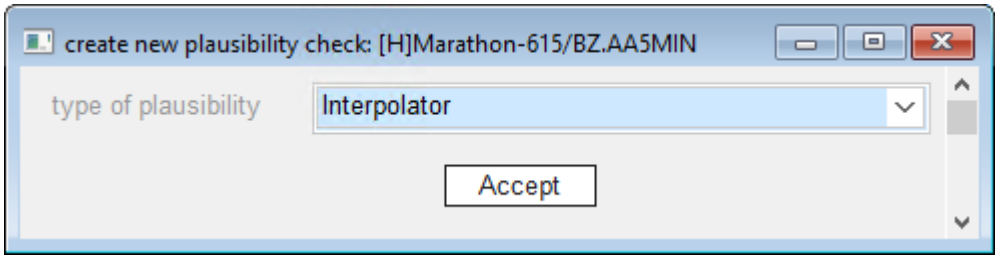


9.3 Create new plausibility check

Since you can only assign each plausibility type to an import time series once, the maximum possible number of plausibility checks in a time series is equal to the number of plausibility types. This function is not available for time series generated by origins.

To assign a new or further plausibility check to an import time series,

- select the corresponding time series by mouse click.
- Open the context menu with the right mouse button and
- Select the **CREATE NEW PLAUSIBILITY CHECK** function.
  The **NEW PLAUSIBILITY CHECK** dialog opens:



- Select a new plausibility check from the drop-down list box.
  A plausibility type that has already been defined is no longer available for selection in the list.
- Now confirm your selection by clicking the [**ACCEPT**] button to further determine the plausibility type.
- The possible parameters are explained at the beginning.
- Confirm your entries with [**OK**].

The new plausibility type has been created in the system and you now get back to the time series browser.

9.4 Delete plausibility check

- In the tree view, select the plausibility check type you want to delete by mouse click.
- Open the context menu with the right mouse button and
- select the **DELETE PLAUSIBILITY CHECK** function.

The plausibility check type is removed without a control dialog.

However, you can undo the deletion process by re-creating a plausibility type.

9.5 Plausibility check results in graphs and tables

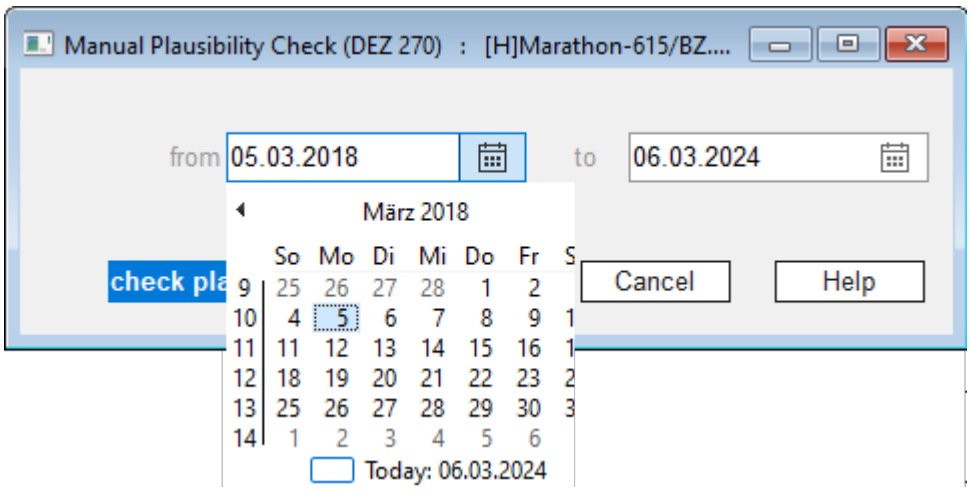
Plausibilised values are displayed in the graphical display by means of a control bar (see ch. The control bar 52). Marks on this bar indicate the time stamp of the plausibilised value.

In the tabular representation, the column **REMARKS** informs about a plausibility check. The plausibility check type used by the system for the corresponding value of the import time series is listed.

Manual plausibility check

AquisNet allows you to check plausibility manually. For example, you can subsequently check old data that entered the AquisNet system without plausibility checks for implausibility.

- Specify a time range by selecting a lower time limit (from) and an upper time limit (to) using the calendar functions.



- Confirm with [**CALCULATE**].

The calculation is performed and communicated via system message.

10 Time series browser: Origins

This chapter describes the Origins module. This module has been integrated into the time series browser and enables centralised work parallel to evaluation functions and the plausibility module.

"Persistent origins" are calculation rules (formation rules) for time series. "Persistent" stands for time series that remain in the system permanently, as opposed to temporarily created time series that are automatically deleted on a regular basis.

Such a formation rule specifies how defined source time series are to be calculated with each other according to a formula in order to obtain the measured values of the target time series.

You can define up to ten source time series depending on the origin type.

When you create a new time series, this means creating an origin for this new time series so that it can be calculated from a source time series.

Example:

The time series "daily mean values", for example, can be calculated from half-hourly mean values.

In this process, a value of the new time series is formed from the mean values of one day. The average value of the next day results in the next entry.

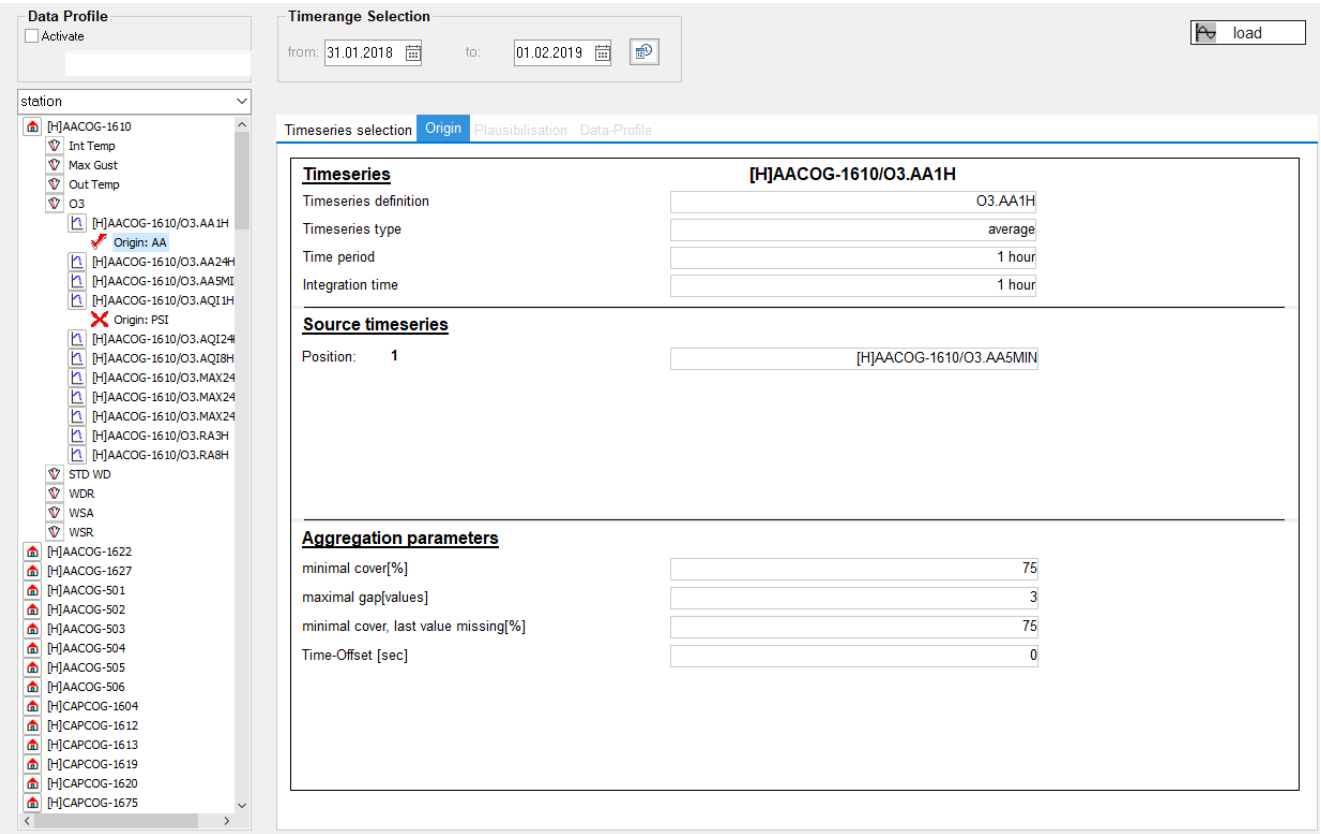
The origins comprise the following options:

10.1 Call up origins

The dialog for origins is integrated into the time series browser. You can open details on the origin of a time series by clicking on the time series or the origin. You can open further functions for the creation of time series via the context menu.

To open origins for time series,

- expand the tree view (independent of the browser tree configuration) to the time series level.
- double-click the desired time series:
- The **ORIGIN** entry tells you that it is a time series generated by an origin.



10.2 Structure of the dialog

In the right part of the dialog, three sections inform about the attributes of the time series selected in the tree. You also define the properties of a new time series to be created in these three sections.

The entries below indicate the properties of the displayed time series:

- Time series section 72
- Source time series section 72
- Origin specific parameters section 72

10.2.1 Time series section

The name of the time series, consisting of the station short name and the time series definition, can be freely selected when a new time series is created if the time series is not imported into the

AquisNet system.

- Time series definition
- Time series type
- Time period
- Integration time

10.2.2 Source time series section

- Display of the time series that serve as source time series for the calculation of the time series.
- The time period of the source time series is smaller than the time period of the target time series when it is created, and is also divisible by it.
- When using an origin of the free formula type, the time periods of the source time series and target time series must be identical.

10.2.3 Origin specific parameters section

The origin-specific parameters are different for different types of time series. Below you will find the list of origin-specific parameters and explanations of their respective meaning:

- Minimum cover [%]

The percentage indicates what percentage of the measured values must be present in the source time series in order to obtain a value of the target time series that is valid.

- Maximum gap [values]

This value specifies the maximum number of consecutive time stamps in the source time series that can have invalid values for the target value to be valid.

- Minimum coverage if last value is missing [%]

If the last value of the time series is not available, this additional setting of the minimum coverage serves as an additional check of the validity of the time series.

- Percentile

A K percentile value defines the value of a measurement series below which K percent of all measured values lie.

- Time-Hour-Start

Time with which the start of a time range within a day can be defined at which the calculation begins.

- Calm condition

The Calm condition (in m/s) represents an adjustable limit value with which you can define up to which wind speed calm is assumed. The usual limit value is 0.5 m/s.

For values equal to or less than this wind speed, no assignment of the wind direction is made in the display of the concentration rose

- Limit value/Threshold

The limit value specifies a value against which all measured values of a time series with time series type "Exceedance frequency" are to be checked for exceeding or falling below.

- Time offset [sec]

With a time offset, you specify the shift between the time stamp and the end of the integration time. The integration time shifts by the specified offset seconds with a positive sign clockwise backwards or with a negative sign counterclockwise forwards.

With the time offset parameter you can create or calculate time slices (time raster and integration time equal to 24 hours) and time intervals (time raster and integration time unequal).

- Parameter correction factor

This factor enables the subsequent correction of measured values based on manual input. This way, laboratory readings can be corrected later if laboratory conditions were not optimal. It is also conceivable that an incorrect calibration of a measuring device can be corrected in this way.

The correction is made by multiplying the measured values by the manually entered value. The correction factor is therefore dimensionless.

10.3 Calculation formulas and Boolean formulas

You can use the following formulas:

- Calculation formulas 73
- Boolean formulas 74

10.3.1 Calculation formulas

Formulas are formed according to the known rules with constants, functions, operators and brackets, whereby the usual object hierarchy – potentiation before multiplication/division before addition/subtraction – applies. When entering constants, please pay attention that the system distinguishes float and integer constants. Thus, the expression 1/2 as division of two int values (integer division) yields the result 0.0, while 1.0/2.0 as division of two float values yields the result 0.5. If the data type of an argument is not correct, a type conversion from **FLOAT** to **INT** or from **INT** to **FLOAT** is automatically carried out. Conversion from **FLOAT** to **INT** is done by separating the decimal places.

The calculation formulas comprise the following options:

- Standard operators 73
- Standard functions 73
- Default constants 74

10.3.1.1 Standard operators

- Addition: `float1+float2`
- Subtraction: `float1-float2`
- Multiplication: `float1* float2`
- Division: `float1/float2`
- Division for integer values with integer result: `int1 div int2`
- Potentiation: `float1^float2`
- Comparison operators, result is of type int (0 or 1): `float1 <, <=, =, >, >= float2`

10.3.1.2 Standard functions

- Absolute value of the argument: `float abs(float)`
- Sign of the argument: `float sign(float)` (-1, 0 or 1, if arg < 0, == 0 or > 0, respectively)
- Sine of the argument (in radiant): `float sin(float)`
- Cosine of the argument (in radian): `float cos(float)`
- Tangent of the argument (in radian): `float tan(float)`
- Arc tangent of the argument (in radian): `float arctan(float)`
- Power with Euler's constant e as the base and the argument as the exponent: `float exp(float)`
- Decadic logarithm of the argument: `float log(float)`
- Natural logarithm of the argument: `float ln(float)`
- Square root of the argument: `float sqrt(float)`
- Integer part of the argument: `float int(float)`
- Decimal part of the argument: `float frac(float)`
- Pseudo-random number between 0 and 1: `float rnd()`

10.3.1.3 Default constants

- `pi`: approximation to the circular number
- `e`: Approximation to Euler's constant e

10.3.2 Boolean formulas

Boolean formulas allow you to specify conditions that control the calculation of a formula, for example, calculating a target time series depending on the key curve of a source time series above a certain value.

Example:

The expression `if { $a<99 } { expr $a+$b } else { expr $a-$b }` means:

Only if the value (`$a`) at a time stamp of the first source time series is smaller than 99 g/cm³, an addition with the corresponding value (`$b`) of the second source time series takes place. If this is not the case, a subtraction of (`$a-$b`) takes place.

The following standard operators and standard functions represent the most important notation types:

- Standard operators (Boolean formulas) 74
- Standard functions (Boolean formulas) 74

10.3.2.1 Standard operators (Boolean formulas)

- binary AND operator, result is of type `int`: `int1 AND int2`
- binary OR operator, result is of type `int`: `int1 OR int 2`
- binary negation, result is of type `int`: `NOT int`

10.3.2.2 Standard functions (Boolean formulas)

- `<float1>` if `<intBool>!=0`
- `<Float2>` if `<intBool>==0`
- `float if (intBool, float1, float2)`

11 Functions and dialogs

The reference section of the user manual shows you with an overview of the program options and their functions as well as all dialogs of the DMO segment, which are AquisNet-specific.

In this chapter you will learn

- which program options the main menu contains
- which functions these individual main menu items contain
- which dialogs in AquisNet are available and
- how to work in these dialogs.

The main menu items and their functions are each summarised in a single chapter.

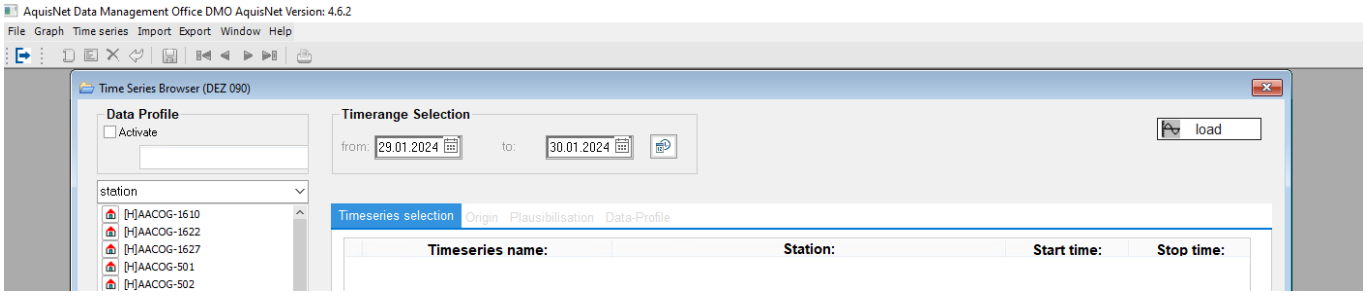
Each chapter describing a dialog explains in up to three sections

- the contents and the general purpose of the dialog as well as the most important controls (“Explanation of the dialog and its controls”)
- how to work in the dialog (e.g. how to navigate and similar; “Features in This Dialog”) and
- further (professional and textual) information in connection with that dialog box (“Further Hints”)

Depending on the complexity of the described functionality, the working options are not explained separately but are contained in “Explanation of the dialog and its controls”. Likewise, the further hints may be missing.

The reference section of the user manual is divided into the following sections:

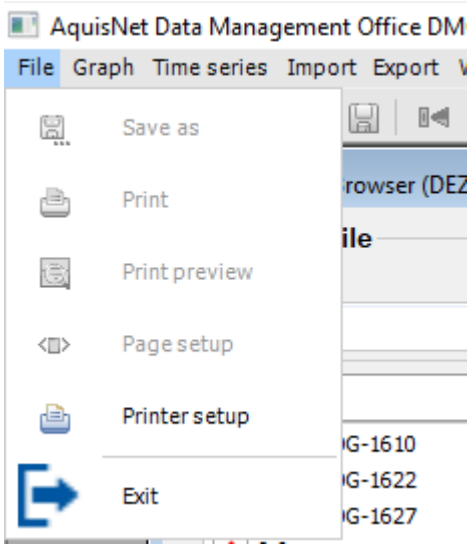
11.1 The main menu of the data collection center DMO



The main menu comprises the following items:

The task bar 78 below the main menu items groups some higher-level functions that are used in several windows.

11.1.1 The main menu item "File"



Main menu: File menu item

In the **FILE** menu you open functions to manage files and data:

Save as

- is used to export data to specific file formats. Alternatively, click the icon.

Print

- opens the WINDOWS default dialog for printing data. Alternative: [**CTRL**]-[**P**] or the icon.

Preview

- shows a preview of the data to be printed.

Page setup

- allows you to set the paper size and orientation for printing.

Printer setup

- opens the WINDOWS default dialog for setting up the desired printer. Alternative: [**CTRL**]-[**SHIFT**]-[**P**].

Exit

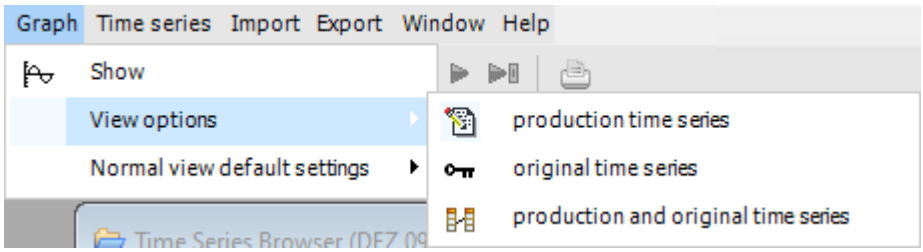
- closes the program system. Alternative: [**ALT**]-[**F4**] or the icon.

11.1.2 The main menu item "Graph"

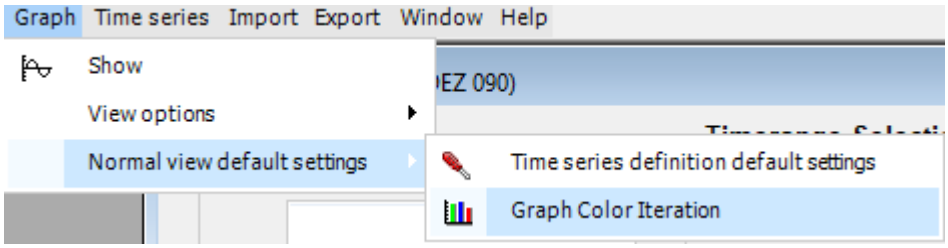
Select in the main menu the **GRAPH** entry.

The main menu item offers you the possibility to select graph presets and display options and to open the graph display.

- **DEFAULT VIEW** opens the display of the selected time series in graphical and tabular form.
- **VIEW OPTIONS** allows you to select whether production time series, original time series or both are to be displayed.



- **GRAPH PRESETTINGS** enables the configuration of parameter-dependent default settings for display colours and colour sequences.

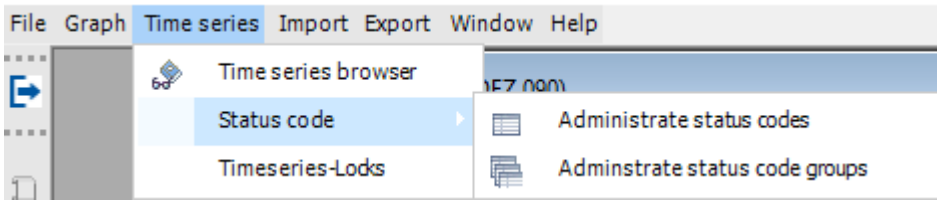


11.1.3 The main menu item "Time series"

Select in the main menu the **TIME SERIES** entry.

This menu offers you the possibility to open the time series selection again and to manage error codes or their groups.

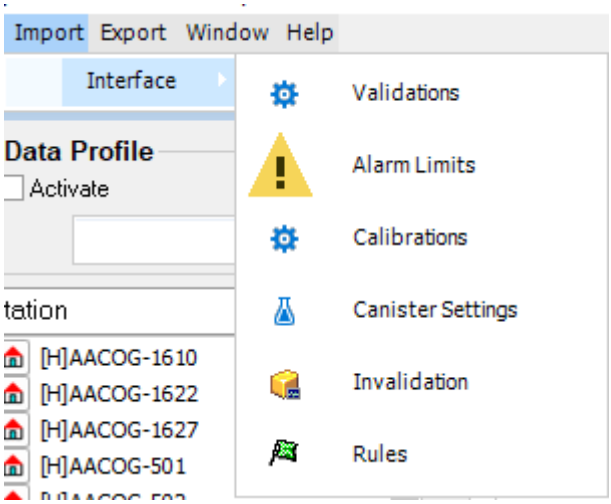
- **TIME SERIES BROWSER** opens the time series selection window if you have previously closed it.
- The **ERROR CODE** menu allows you to open the administration dialogs for error codes and their groups.



11.1.4 The main menu item “Import”

Select in the main menu the **IMPORT** entry.

This menu offers you the possibility to configure customised settings for the import, if this is included in your installation.



11.1.5 The main menu item "Window"

This menu item allows you to organise the display of multiple windows on your screen.

- Select in the main menu the **WINDOW** entry.

This menu offers you the possibility to arrange several windows

- overlapping,
- horizontal,
- vertical, or
- on top of each other

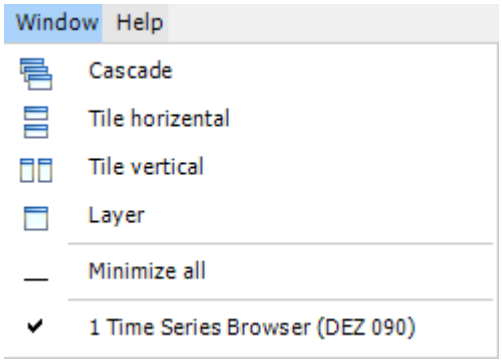
.

An overlapping arrangement of several windows is automatically preselected.

Furthermore you can

- minimise all windows,
- or undo the setting "minimise all windows"

.



11.1.6 The main menu item "Help"

Select in the main menu the **HELP** entry.

Using this menu you can

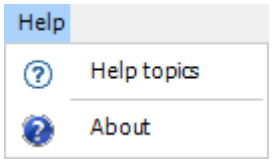
- open help topics for the data collection center DMO module and
- get brief information about the program and KISTERS.
- In the Help menu, you start the WINDOWS help and can display information about the status of the entire system.

Help topics

- In this menu you can open help topics.

Info

- Get brief information about AquisNet and KISTERS.



11.1.7 The task bar

In the task bar shown above, some general functions are grouped that simplify the handling of many data and records. These functions can always be found under different main menu items.

- The tooltip that is opened names the function of the button that you are currently hovering the mouse cursor over.
- The following functions are arranged in order from left to right:
- Exit program.
- Add record
- Edit record
- Delete record
- Undoing delete data record
- Save
- First record
- Previous record
- next page or next record
- last page or last record
- Print

11.2 The buttons in the data collection center DMO

Exit

- exits the program system.

Save

- applies the changes made in an editing window to the database.

Save as

- exports the current record table as ASCII file.

Edit record

- opens the selected record for editing.

Add record

- adds a new, empty record.

Delete record

- deletes the selected data record.

Undo record

- resets the record change.

First record

- scrolls to the first record.

Previous record

- scrolls to the previous record.

Next record

- scrolls to the next record.

Last record

- scrolls to the last record.

Print

- prints the current record table.

11.3 Buttons in the graph

With the graph buttons you can edit AquisNet graphs.

Refresh

- refreshs the plot display.

Print

- prints the graph/table.

Print preview

- opens the WINDOWS default dialog to setup the printer and view the graphics in a print preview.

Format axes

- opens the Axis properties 82 dialog.

Format plots

- opens the Plot properties 86 dialog.

Format legend

- opens the Legend 95 dialog.

Physical units

- ▾ opens a drop-down list where you can choose another physical unit for the graphical display.

Divided axis ranges and/or plot areas

- opens a dialog to represent several plots of one graphics window in several ranges.

Zooming back

- switches to the zoom back mode that enlarges displayed areas of a graph to the last defined display scale.

Value range to be adjusted automatically when zooming

- When you scroll in this mode with the horizontal scroll bar, the displayed graph section follows the plot automatically. The [**VALUE RANGE TO BE ADJUSTED AUTOMATICALLY WHEN ZOOMING**] button is active when the plot is in zoomed state.

Insert mode

- switches to insertion mode.
- You can insert values into the current time series plot.
- Equidistant time series:
- Add values by mouse click, if the time series has gaps at the respective position.
- Non-equidistant time series:
- Add values by mouse click, if the time series has no gaps at the respective position.
- If you want to add values in a gap, delete the beginning of the gap and the end of the gap first.

Shift two units to the left

- shifts the plot one section to the left.

Shift one unit to the left

- shifts the plot one marking section to the left.

Shift one unit to the right

- shifts the plot one marking section to the right.

Shift two units to the right

- shifts the plot one section to the right.

Formulas

- opens the Formula calculation 94 dialog.

11.4 Dialogs in the error codes of time series section

Dialogs in the error codes of time series section comprise:

- Manage error codes 81
- Manage error code groups 81

11.4.1 Manage error codes

The **MANAGE ERROR CODES** dialog is used to manage error codes that can be used in evaluations or when checking data to filter the quality of measurement data.

The Manage error codes tutorial contains detailed information about this dialog.

11.4.2 Manage error code groups

The **MANAGE ERROR CODE GROUPS** dialog is used to bundle error codes in groups to filter the quality of measurement data by error codes when evaluating or checking data.

The Manage error code groups tutorial contains detailed information about this dialog.

11.5 Dialogs in the Graph and Table section

The dialogs in the graph and table section comprise:

- Axis properties 82
- Extended axis settings 83
- Set type of rounding 84
- Influence 84
- Insert remark/duty 85
- Remarks/Duties 85
- Remark/Duty configuration 85
- Set range for editing 86
- Print 86
- Plot properties 86
- Plot properties (style) 87
- Plot properties (symbol) 87
- Plot properties (labelling) 87

- Plot properties (grid) 87
- Insertion/Fitting of time series data 88
- Crosshairs 88
- Graph 88
- Insert graphical elements 89
- Edit [graphical element] 90
- Copy graphical element 91
- Graphical elements 91
- Graph configuration for time series 92
- Save graph 93
- Integration 93
- Iteration sequence 94
- Formula calculation 94
- Extend load range 94
- Save layout 95
- Legend 95
- Create new LIMS project 95
- LIMS filter data input dialog 95
- New time range 96
- Sort 96
- Fixing the selection and order of columns 96
- Search for 96
- Table 96
- Transactions 99
- Transaction details 99
- Time axis shifting 99
- Time series browser 99
- Plausibility check 106
- New plausibility check 107
- Manual plausibility check 107
- Time series versions 108
- [Additional displays] on/off 108
- Create new time series 108

11.5.1 Axis properties

With the **AXIS PROPERTIES** dialog you scale graph axes.

You can scale axes automatically and individually determine the exact step size.

Using [**ADVANCED SETTINGS**] you can specify more detailed axis properties and save them in the dialog Advanced axis settings 83.

The **ACTIVE** check box indicates whether the advanced axis settings or the default settings from the present dialog apply (in the latter case, the box is unchecked).

Related Links:

- Working with graphs
- Legend 95
- Plot properties 86
- Graph 88
- Graph configuration for time series 92

Features in This Dialog:

Select axis type

The following scale types are available:

- for the time axis the settings linear or time axis
- for the measured value axes the settings linear or logarithmic

.

Set the axis position

Choose the axis position (e.g. **BELOW**) from the drop-down list box.

Set the position of the axis title

Choose the axis position (e.g. **RIGHT**) from the drop-down list box.

Set the font of axes labels

Click the [**FONT**] button to set the font for the axis title.

Automatic scaling: yes/no

To individually scale the axis.

- deactivate the autoscaling option. This option cannot be deselected when the graph is in zoom mode.
- Then, in the **FROM** and **TO** specify lower and upper limits of your axis division and
- the increment.

Reflect the axis

Check the **REFLECT AXIS** option to reverse the direction of an axis.

11.5.2 Extended axis settings

In the dialog **ADVANCED AXIS SETTINGS** you can set format and alignment of the caption for each axis element of a time axis (first value, consecutive values, last value). Up to two lines per caption can be defined here. The preview window (upper dialog half) graphically displays the current settings.

Features in This Dialog:

Making the caption visible

You can make the upper line (usually for date) and the lower line (usually for time) of the axis caption separately visible by checking the box in the Upper/Lower line section.

Selecting the captions to be set

These settings can be made separately for the upper and the lower caption line as well as individually for each axis element, or they can be applied for all axes.

- To do so, select the respective elements in the Axis elements section or select the elements in the preview window by clicking on them with the mouse.

Formats for date/time

Choose from this drop-down list box between the following format elements:

- **YY**: year (two-digit number)
- **YYYY**: year (four-digit number)
- **MM**: month (two-digit number)
- **MMM**: month (token consisting of three letters)
- **MMMM**: month (full name)
- **DD**: day (two-digit number)
- **DDD**: weekday (token consisting of three letters)
- **DDDD**: weekday (full name)
- **HH**: hour (two-digit)
- **MM**: minute (two-digit)
- **SS**: second (two-digit)

These format elements can be supplemented with free text and combined as required (e.g. to include separators).

Specifying alignment of the caption

Choose an entry from the drop-down list box to specify the caption alignment.

- left
- right
- centred (with reference to the axis scale mark)

- centred in the interval.

The alignment “centred in the interval” appears only, if all axis elements are selected.

Showing the axis caption: always/only in the case of value change

For both lines of the axis label you can set whether the label is to be displayed only after a possible value change or always. In the first case, each value appears only once in the caption. In the second case, equal values may be repeated in the caption.

Further Hints

The advanced axis settings are also included in the layout templates, so that they may also be transferred to other time ranges and other time series of the same type.

Note: The length of the time range is stored in the layout template and may need to be adjusted when loaded for another time range.

11.5.3 Set type of rounding

The **SET TYPE OF ROUNDING** dialog is used to specify the rounding method for the displayed measured values in the table display of measured values.

Related Links:

- Setting the rounding method 43
- Table 96

Features in This Dialog:

Decimal places section

- Select the first option if you want to specify the number of decimal places on your own; choose an entry from the drop-down list.
- Select the second option field if you want to specify the number of significant decimal places and the maximum number of decimal places.
- Select the third option, if you want the rounding to consider the amount of the measured value: if it is >= 0.2, one decimal place is shown; if it is >= 0.02, two, otherwise three decimal places are shown. If the value is smaller than 0.01, the table contains the measured value expression “<0.01”.
- If you select the Standard radio button, the number of decimal places is taken from the settings for the time series; it is rounded mathematically by default.

Rounding section

By selecting the appropriate radio button, you can choose between three kinds of rounding:

- Mathematical: Default rounding
- Even/odd: rounded up to an even figure if the last digit is a 5. Example: 1.25 => 1.2, but 1.35 => 1.4
- According to ini entry: Rounding as specified in the `aquisnet.ini` file

Examples section

At the bottom of the window an example is shown of how the chosen kind of rounding will look like.

Further Hints

The rounding method is determined from the `aquisnet.ini` file; see the [**EXTRAS**] section, entry `sectionRoundMathematical`. If this entry is 1, the rounding is mathematical, if it is 0, rounding will be carried out according to the even/odd method.

11.5.4 Influence

With the **INFLUENCE** dialog, you can define factors of influence for measured values in the table. You can specify different ice conditions and define influences by weed growth. You can also specify that the current value has been additionally influenced (by any factors) or that it has been supplemented or calculated.

Related Links:

- Table 96
- Manual entry into the table 64
- Working with tables

Features in This Dialog:

Assign factors

- Select the desired value in the table.
- Double-click in the Influence table column.
- Check the box of the option that is to be considered influencing for the current measured value.

11.5.5 Insert remark/duty

You can add remarks or tasks to the time series in the control bar of the graph window using the **INSERT REMARK/DUTY** dialog. Each remark and duty applies to a specific time period. Internal or external remarks can be assigned (the latter are also displayed in the reports). Some remarks are automatically created by the system (if measurement values have been changed, for example).

Related Links:

- Remark/Duty Configuration 85
- Remarks/Duties 85
- Working With Graphs
- Graph 88

11.5.6 Remarks/Duties

The **REMARKS/DUTIES** dialog contains all remarks and duties specified up to now for stations, parameters or time series of the current time stamp as well as existing plausibility check messages.

Open it by double clicking on the remarks column of the table.

From this window you can

- create new remarks or tasks,
- edit existing ones or
- delete them, and
- configure which remarks will be displayed in the graph and table.

Related Links:

- Remark/Duty Configuration 85
- Insert remark/duty 85
- Table 96

11.5.7 Remark/Duty configuration

You can use this dialog to specify which remarks, duties and plausibility check messages are to be listed in the Comments table column.

Check the special boxes to display the various messages (internal, external remarks, duties, plausibility check messages, remarks on station, parameter or time series level):

Related Links:

- Insert remark/duty 85
- Remarks/Duties 85
- Working with tables

11.5.8 Set range for editing

As an alternative to the range specification, you can specify in the **SET RANGE FOR EDITING** dialog another time range in the graph (draw it with the mouse) within which the values are to be changed (corrected, added, deleted) in the graph or the table.

- Either use the calendar or
- enter the date in the `dd.mm.yyyy hh:mm` format by hand or
- select the day, month and year data and browse with the arrow keys of your keyboard through the dates.

Related Links:

- Working with graphs
- Editing plot ranges
- Inserting or fitting plots 66
- Editing Measured Values Graphically
- Edit Measured Values in a Table
- Working with tables
- Graph 88

11.5.9 Print

This dialog is used to print the current (two) graph(s) in the height and width of your choice.

Related Links:

- Print 44
- Print 86
- Graph 88

Features in This Dialog:

Height Section

- Enter here the height of the desired printout in centimetres.
- If you have set divided axis ranges (using the Divided axis and/or plot ranges button), enter the two height values separately (for top and bottom graph).

Width Section

- Enter the width of the printout here. If you have set divided axis ranges, this data applies to both graphs.

Further Hints

The two graphs are printed one above the other. They will thus have the same width, but the heights may be configured separately.

11.5.10 Plot properties

With the **PLOT PROPERTIES** dialog you specify the display type and the symbol that represents a time series in the graph.

The specific data of each plot is managed in the

- Plot properties (style) 87,
- Plot properties (symbol) 87,
- Plot properties (labelling) 87 and
- Plot properties (grid) 87

tabs.

Related Links:

- Working with graphs
- Graph 88
- Legend 95

- Axis properties 82
- Graph configuration for time series 92

11.5.11 Plot properties (style)

In the **STYLE** tab you can specify the following for the selected plot

- Style type (e.g. line, level, steps),
- Line style (e.g. dashed or solid),
- Line colour and
- Line width.

Related Links:

- Working with graphs
- Graph 88
- Legend 95
- Axis Properties 82
- Graph Configuration for Time Series 92

11.5.12 Plot properties (symbol)

In the **SYMBOL** tab you can specify the following for the selected plot

- Symbol type (to indicate the measured values in the graph; e.g. none, point, cross, plus, star),
- Symbol size and
- Frame width of the symbol.

Related Links:

- Working with graphs
- Graph 88
- Legend 95
- Axis properties 82
- Graph configuration for time series 92

11.5.13 Plot properties (labelling)

In the **LABEL** tab you can specify the following for the selected plot

- Alignment of the caption text,
- Labelling type and
- Connecting line between plot and label.

Related Links:

- Plot properties 86
- Working with graphs
- Legend 95
- Axis properties 82
- Graph 88
- Graph configuration for time series 92

11.5.14 Plot properties (grid)

In the **GRID** tab you can define

- Colour,
- Line style, and
- Visibility

of the grid.

You also set the grid properties automatically (check the box).

Related Links:

- Working with graphs
- Graph 88
- Legend 95
- Axis properties 82
- Graph configuration for time series 92

11.5.15 Insertion/Fitting of time series data

With the **INSERTION/FITTING OF TIME SERIES DATA** dialog you insert a selected area of a plot (source) into another plot (target).

The following functions are available:

- **COPY**
- **FIT LEFT**
- **FIT RIGHT**
- **FIT AT BOTH ENDS**
- **FIT PROPORTIONALLY LEFT**
- **FIT PROPORTIONALLY RIGHT**
- **PROPORTIONAL FITTING AT BOTH ENDS**

Select a proportional fit to proportionally adjust the course of the plot in the edit range to the transition points.

Related Links:

- Inserting or fitting plots 66
- Working with graphs
- Legend 95
- Plot properties 86
- Axis properties 82
- Graph 88
- Graph configuration for time series 92

11.5.16 Crosshairs

With this dialog you can set a crosshairs; it facilitates reading of the special values at the axes. You can also activate a search function for particular points within a configurable snap radius.

- Check the Show crosshairs box to switch it on.
- Check the Use search function box.
- If desired, set the snap radius in millimetres and
- define in the Search criteria/priorities section, whether the crosshairs be set to
- gap beginnings and gap ends,
- to local extremes and/or
- inflection points (these are points where the gradient changes; not reversal points in terms of mathematics)
- should snap.
- You can then allocate to each of these search criteria a priority for the search function (choose from the list).
- You can activate the crosshairs for your current graph either by [**ACCEPT**] (the dialog remains open) or by [**OK**] (the dialog is closed).

11.5.17 Graph

The central **GRAPH** window lets you display and edit time series graphically.

In the Graph you can

- Edit data
- Display calculated time series
- Create sums and differences from different time series
- Calculate correlations
- Calculate regressions
- Edit related time ranges (stretching, tipping, filling with constants)
- Visualise changes in time series

Related Links:

- Working with graphs
- Working with tables
- Legend 95
- Plot properties 86
- Axis properties 82
- Graph 88
- Graph configuration for time series 92
- Table 96

How to Work in This Dialog

Using the buttons and context menus

The graph window allows you to edit your time series in an easy and comfortable way. All functions within the graph can be executed using the buttons or the context menu. The function of the button over which you move the mouse pointer is displayed in the bottom left of the status area of the graph window.

Zooming

You can view and print the data in the graph at an arbitrary resolution using the zoom function.

Freely adjustable display range

The time series to be represented can be combined as you like and the type of representation can be configured freely.

11.5.18 Insert graphical elements

In this window you can choose graphical elements to insert them in the current time series layout.

Related Links:

- Insert user-defined graphical elements
- Working with graphs
- Edit [graphical element] 90
- Copy graphical element 91
- Graph 88

Features in This Dialog:

- The Plots list box shows all time series (plots) contained in the graph.
- Choose the time series to which the graph element should be added.
- Select the element to be added to the time series from the Graphical element list box.
- Using the radio buttons in the Assigned to section you specify the connection of the graphical element with the time series:
  - **PLOT**: The graphical element is connected to the plot (the x- as well as the y-coordinates of the graph element correspond to the plot coordinates), i.e. the graph element is always exactly at the same position relatively to the plot.
  - **AXIS (HORIZONTAL)**: The x-direction of the graphical element depends on the plot whereas the y-direction remains constant (coordinates 0 to 1000).
  - **AXIS (VERTICAL)**: The y-direction of the graphical element depends on the plot whereas the x-direction remains constant (coordinates 0 to 1000).
  - **NONE**: The graphical element remains constant in both directions, irrespective of the plot (both coordinates between 0 to 1000).
  - **HEADER**: The graphical element is inserted into the header. If a header does not yet exist, it will be created. Both coordinates range from 0 to 1000.
  - **FOOTER**: The graphical element is inserted into the footer. If a footer does not yet exist, it will be created. Both coordinates range from 0 to 1000.
- After you have finished making these specifications, confirm with [**OK**].

11.5.19 Edit [graphical element]

In this window you can edit a graphical element selected before that has been inserted into the time series layout.

Related Links:

- Edit [graphical element] 90
- Working with graphs
- Insert graphical elements 89
- Copy graphical element 91
- Graph 88

How to Work in This Dialog

Specifying origin

- In the Origin section you can enter coordinates manually. The graphical text element will be shifted according to your definition.

Specifying text orientation

- Specify the angle of the text (in degrees) in the Angle field. 0 degree corresponds to a horizontal text orientation. The text will be rotated clockwise around its end point by the specified angle (at 180° the text will be upside down).
- Horizontal alignment text box: Sets the origin point of the text in x-direction. The following settings are possible: right, centred and left alignment.
- Vertical alignment text box: Sets the origin point of the text in y-direction. The following settings are possible: above, centred and below.

Specifying the text variable

A drop-down list box of time specific variables which can be inserted in the text are available in the Text ID list box (e.g. time series name, station name, operator, place of creation, etc.).

- Here you can display for example the Axis time range as text element. To do so, select the variables first time stamp of the axis and last time stamp of the axis.
- All variables with time data can be specified even more exactly: Select in the currently active combo box the date format from the list or enter it manually.
- Select the desired variable from the drop-down list and
- click the [**ACCEPT**] button.
- The selected text variable will be inserted into the text.
- When clicking the [**FONT**] button, you get the usual dialog for font selection and its properties, as known from Windows programs.

Editing a Graphical Element

The second Style tab in the Edit <graphical element> 90 dialog (not existing for text elements, but e.g. for “arrow”) shows the properties of pen (Line style section) and brush (Fill mode section) of line and area (rectangles, ellipses) graphical elements. Only graphical elements with a closed area have brush properties whose fill colour and pattern can be defined (i.e. not for curves/lines).

Specifying pen properties

- The Line type list box offers you a selection of various line types. If the line width (see below) exceeds 0.4 mm, only the solid line is available for use (set by MS WINDOWS).
- Select the line colour from the drop-down list.
- Enter the line width in millimetres either manually or choose the width with the arrow keys.

Specifying brush properties

Specify the fill mode:

- Choose a fill colour from the drop-down list.
- In the Fill pattern list box, select the desired one from the selection of fill patterns. If the area is completely filled with a colour or remains white, the **TRANSPARENT FILLING** box will be unchecked.
- Checking this box makes the filling transparent.

11.5.20 Copy graphical element

In this window you can duplicate a graphical element selected before that has been inserted into the time series layout.

Related Links:

- Copying graphical elements 58
- Working with graphs
- Insert graphical elements 89
- Edit [graphical element] 90
- Graph 88

How to Work in This Dialog

Selecting graph elements

Note:

The graph element to be copied must be selected (clicked) before this dialog is opened.

Selecting time series

- Select from the list the time series to which the copied graph element is to be assigned.

Specifying affiliation

- Using the radio buttons in the Assigned to section you specify the connection of the graphical element with the time series:
  - **PLOT**: The graphical element is connected to the plot (the x- as well as the y-coordinates of the graph element correspond to the plot coordinates), i.e. the graph element is always exactly at the same position relatively to the plot.
  - **AXIS (HORIZONTAL)**: The x-direction of the graphical element depends on the plot whereas the y-direction remains constant (coordinates 0 to 1000).
  - **AXIS (VERTICAL)**: The y-direction of the graphical element depends on the plot whereas the x-direction remains constant (coordinates 0 to 1000).
  - **NONE**: The graphical element remains constant in both directions, irrespective of the plot (both coordinates between 0 to 1000).
  - **HEADER**: The graphical element is inserted into the header. If a header does not yet exist, it will be created. Both coordinates range from 0 to 1000.
  - **FOOTER**: The graphical element is inserted into the footer. If a footer does not yet exist, it will be created. Both coordinates range from 0 to 1000.

Deleting original graph elements

If the graph element you used as the copy template should be deleted afterwards, check the Delete original box.

Completing copying

After you have finished making these specifications, confirm with [**OK**].

11.5.21 Graphical elements

This window lists all graph elements which have been added to the current time series display.

Related Links:

- List of all graphical elements 59
- Working with graphs
- Insert graphical elements 89
- Edit [graphical element] 90
- Graph 88

How to Work in This Dialog

Use this window to select, edit and delete an element:

- Choose the desired element from the list and
- click the [**OK**] button.
- The dialog will be closed and the corresponding graphical element will be selected.
- Click the Edit button to edit the element.
- The selected graphical element's tab opens.
- To delete a graphical element from this list (alternatively to [**DEL**] or Backspace key with selected element in the graph),
- click [**DELETE**].
- First, deletion is only temporary: the graphical elements will be inserted in a so-called deletion list.
- If you then exit the window with [**CANCEL**], the deletion list is cleared and none of the graphical elements are removed.
- Clicking the [**OK**] button will irrevocably delete all elements in the deletion list without any inquiry. If you have moved a graphical element to the delete list, then want to change another element with [**EDIT**], you will get a message window asking you whether to delete the elements from the deletion list before editing or whether to cancel the deletion process – i.e. not delete anything.

11.5.22 Graph configuration for time series

You can use the **GRAPH CONFIGURATION FOR TIME SERIES** dialog to configure graph styles for time series.

The following entries can be made:

- Set line type 92
- Set line width in [mm] 92
- Set symbol type 92
- Set symbol width in [mm] 92
- Set Y-axis minimum 92
- Set Y-axis maximum 92

Related Links:

- Iteration sequence 94
- Working with graphs
- Working with tables
- Legend 95
- Plot properties 86
- Axis properties 82
- Graph 88

How to Work in This Dialog

Set line type

- Set different line types for time series for better differentiation.

Set line width in [mm]

- Distinguish between equal line types by their line width.

Set symbol type

- Set different symbols for measured values of different time series in the graph.

Set symbol width in [mm]

- Use this function to differentiate between identical symbols using the display width

Set Y-axis minimum

- Set the lower limit of the y-axis

Set Y-axis maximum

- Set the upper limit of the y-axis.

11.5.23 Save graph

In the **SAVE GRAPH** dialog you can save graph settings of single time series as standard graphs.

Related Links:

Graph 88
Legend 95
Plot properties 86
Axis properties 82
Graph configuration for time series 92

Features in This Dialog:

To save a graph, proceed as follows:

- Enter a name for the graph to be saved (mandatory).
- Select the file location from the drop-down list box in the upper part of the dialog,
- In case you find graphs here that are not required, you can select them and delete them from the database with [**DELETE**].
- Select the size,
- the format of the graph to be saved and
- the unit of the size in [**CM**], [**INCH**] or [**PIXEL**].
- Confirm your entries with [**SAVE**].

Note:

- You can also copy a graph to the clipboard with the corresponding function of the graph context menu
- and from there open, edit or save it in another program.

Further Hints

You can reload these saved graphs at any time:

- Open the Explorer.
- Select the **STANDARD GRAPHS** view.
- The folder in which the available standard graphs are stored are listed in a separate window.
- Open the context menu with the right mouse button and select the **OPEN** or **SHOW** entry.
- The selected graph opens in the right-hand part of the Explorer (NOT in a separate graph window).

11.5.24 Integration

In the **INTEGRATION** dialog you specify the settings for the surface calculation below/between curves.

- Set the time range of data for which the integration is to be carried out.
- Use the calendar for this.
- Choose in the **TIME SERIES SELECTION** section the desired rime series for which you want to determine integrals.
- If you want to calculate an area between two plots, select here also a second time series.
- Specify then the base flow separation:
  - **NONE**: no further specifications required;
  - **CONSTANT**: enter then the value of the base flow separation or whether it starts at the left or the right edge;
  - **LINEAR**: Enter in addition the edge points or the connection point of the edge points of your range if required.
- When no base flow separation was selected for two plots, you can specify in the **MODE** section
  - the areas between plots or
  - the ratio of areas.
- In the Integrate over drop-down list you can specify the time reference for the integration.
- Then click [**CALCULATE**].
- The selected integral will be calculated.
- To view the results of the integral determination, click [**RESULTS**].

Related Links:

- Calculate integrals
- Graph 88

11.5.25 Iteration sequence

In addition to the ZRD-dependent settings, colour assignments can be defined for the time series in the time series selection window.

This ensures unique identification of each time series in the graph report.

You can assign a colour to each of the individual position numbers, which ensures the recognition of a time series in a graph among other time series.

When you have finished, confirm your entries with [**OK**].

Features in This Dialog:

Adding/editing a new row

- Open the context menu with the right mouse button.
- Select the **ADD NEW ROW** function.
  The new last row is added at the end of the list.
- Select a colour for the new position number.

Remove last row

- Select the last entry in the list.
- Open the context menu and select the Remove last row function.

11.5.26 Formula calculation

The graph display offers a feature for calculating time series from formulas, e.g. from previously calculated regression formulas. You can work with these temporary time series "as usual"; however, you cannot save them in the database.

Features in This Dialog:

Enter formula

- In the context menu of the graph, select the Formula function.
- Define the desired formula in the Calculate time series dialog.
- To do so, apply the desired selections from the source time series, operators and functions lists to the **FORMULA** box using the arrow buttons.
- In the Axis box you specify which value axis is used to represent the calculated time series.
- This is especially important when representing several time series with different parameters.
- With the Paste function of the context menu you can insert formulae copied before.
- Click [**OK**].
- The temporary time series is displayed in the graph and can be edited like a normal time series and be used for example for insertion /fitting.

11.5.27 Extend load range

With the **EXTEND LOAD RANGE** dialog you define for which period you want to add data to the time series to display them in the graph.

you can either reload data prior to the already displayed time range or after this time range.

Related Links:

- Working with graphs
- Legend 95
- Plot properties 86
- Axis properties 82
- Graph 88
- Graph configuration for time series 92

11.5.28 Save layout

The **SAVE LAYOUT** dialog is used to save graph settings in template form as graph layouts for particular time series types.

Related Links:

- Save layout 95
- Graph 88
- Legend 95
- Plot properties 86
- Axis properties 82
- Graph configuration for time series 92

Features in This Dialog:

To save layouts in the database, proceed as follows:

- Open the Graph > Save as layout main menu item.
- Enter a name for the layout (mandatory).
- In case you find layouts here that are not required, you can select them and delete them from the database with [**DELETE**].
- Confirm your entries with [**OK**].
- Then use the wizard to specify which time series can use the layout.
- The following criteria can be entered:
- Time series with the same parameter as the time series from which you save the layout,
- time series with the same value distance and
- time series with the same value type.

11.5.29 Legend

With the **LEGEND** dialog you determine the position and visibility of the legend in the graph.

11.5.30 Create new LIMS project

Create new LIMS project

When you add an acquisition location to a LIMS project, a filter time series is created in AquisNet with reference to the selected acquisition location.

To add a sample to a LIMS project,

- first select a station
- Now specify a sample parameter.

You can distinguish between filters and canisters as sample parameters. The new filter time series is created in the system and assigned to the desired LIMS project in the tree view.

To enter data, open the LIMS filter data input form 95 dialog.

11.5.31 LIMS filter data input dialog

Enter or change new filter data

To specify the data of a filter sample,

- enter the required data in the input boxes.
- Confirm the control dialog when you exit the dialog.

After configuring the dialog, data can be imported into the new time series. These can be evaluated and edited in table or graph.

Related Links:

- Configuring legend display 49
- Working with graphs

- Plot properties 86
- Axis properties 82
- Graph 88
- Graph configuration for time series 92

11.5.32 New time range

Der Dialog Neuer Zeitbereich

With the **NEW TIME RANGE** dialog you define another time range in the graph for which you want to view values in the graph or table.

Related Links:

- Working with graphs
- Working with tables
- Legend 95
- Plot properties 86
- Axis properties 82
- Graph 88
- Graph configuration for time series 92

11.5.33 Sort

You can specify the column-by-column sort order in tables using the **SORT** dialog.

Drag the desired column name from the Sortable columns list in the Sort list by the Drag&Drop function. Using the Ascending option you can specify whether the columns are sorted in ascending or descending order.

11.5.34 Fixing the selection and order of columns

With the **FIXING THE SELECTION AND ORDER OF COLUMNS** dialog you specify which columns are shown in which order in the table.

Related Links:

- Working with tables

Features in This Dialog:

Selecting columns for display

Drag the desired column name from the Columns not displayed list in the Displayed columns list by the Drag&Drop function.

Defining the order of columns

Define the column order using the [**UPWARDS**]/[**DOWNWARDS**] buttons.

11.5.35 Search for

With the **SEARCH FOR** dialog you can search for desired entries in the table.

Related Links:

- Searching for table entries 41

11.5.36 Table

You can view and edit time series using the **TABLE** dialog.

Values of time series that are not being edited yet, always appear in grey at first.

Non-editable values of time series which are being edited are shown in grey; editable values in black.

Measured values of non-editable time series always appear in grey in the table.

Related Links:

- Working with graphs
- Working with tables
- Legend 95
- Axis properties 82
- Graph 88

Features in the table display of measured values

Filter time series table using an error code group

You can adjust the time series in the table to show only values with a certain error code. The selection of an error code or an error code set is done using status code groups. Based on error codes in a selected error code group, values in the table are subsequently filtered. Depending on the need, it is necessary to create a new error code group, if necessary also with only one error code.

To filter time series by error codes,

- first select the **SET STATUS CODE** > **ERROR CODE** context menu entry.
- The Set status for time range dialog opens.
- This dialog lists all error code groups created in the system using a drop-down list.
- In the Time range section, you can select the time range for which you want to filter the table view. This way you can conveniently set the time range.
- Select an error code group.
- Confirm with [**ACCEPT**] and with [**OK**].

After confirming the selection, only the values that have an error code according to the preselected error code group are displayed in the table. The result can be – depending on the error code group – a completely changed table.

Edit measured values by manual validation

Notethat the error code of a manually validated value after a user input is set to **INVALID**.

To manually validate a value,

- click in the table cell of the value to be changed in the table display.
- The table cell is now activated for user input.
- You can now validate the value manually.
- Confirm your entry with [**RETURN**].
- The value is rounded if necessary and finally saved.

Setting an error code through manual validation

To set an error code manually,

- click in the table cell of the error code to be changed in the table display.
- The system now offers a drop-down list box with error codes.
- These error codes are imported from the error code group **MANUAL VALIDATION**.
- If the required error code is not available in the list, proceed as follows:
- Close the windows of the graph and table.
- Select the **TIME SERIES** > **ERROR CODE** > **MANAGE ERROR CODE GROUPS** menu item.
- Assign the desired error code to the error code group manual validation under system.
- Close the dialog and save your entries.
- Now open the time series browser again.
- Click here the [**VIEW**] button anew.
- AquisNet reloads the time series in the time series list for table and graph.
- Now click again in the table cell of the error code to be changed in the table display.
- The system now offers a drop-down list box with error codes. This list has been extended by the error code just added to the manual validation group.
- Select the desired error code.
- Confirm your entry with [**RETURN**].
- The new error code is now set.

Manually change error code over a time range

To set an error code manually,

- select one or more rows in the table by moving the mouse pointer over the rows to be selected while holding down the left mouse button, or select several rows by pressing the [**CTRL**] key to select several lines one after the other.
- In the context menu, select the **SET STATUS CODE** > **QUALITY FLAG** entry.
- The **SET STATUS IN THE SELECTED AREA** dialog opens.
- Here you can subsequently vary the edit range in the **TIME RANGE** section.
- Now select the desired error code from the drop-down list box in the **STATUS** section.
- These error codes are imported from the error code group manual validation.
- If the required error code is not available in the list, proceed as follows:
- Close the current dialog and the graph and table windows.
- Select the **TIME SERIES** > **ERROR CODE** > **MANAGE ERROR CODE GROUPS** menu item.
- Assign the desired error code to the error code group manual validation under system (New assignment of an error code to an error code group 31).
- Close the dialog and save your entries.
- Now open the time series browser again.
- Click here the [**VIEW**] button anew.
- AquisNet reloads the time series in the time series list for table and graph.
- Select the desired time range in the table again.
- In the context menu, select the **SET STATUS CODE** > **QUALITY FLAG** entry.
- The Set status in the selected area dialog opens.
- Here you can subsequently vary the edit range in the Time range section.
- Now select the desired error code from the drop-down list box in the **STATUS** section.
- This list has been extended by the error code just added to the manual validation group.
- Now select the desired error code.
- Confirm your entry with [**RETURN**].
- The new error code is now set for all values or row entries in the marked time range.

Saving manual changes during exit (transaction)

If you change and save values or error codes in one or more time series, AquisNet proceeds as follows.

The database records before and after the change are recorded in a transaction. This makes it possible that the data before the change can be accessed again later. The data is used in the event of a possible later restoration of a previous data status.

To reduce the storage volume and the organisational effort of the transactions, AquisNet only saves the "affected" time range at a time and this - see above - with the data status before the changes.

- In addition to the pure measurement data of all affected time series before the change, AquisNet stores an identifier, a comment and the responsible user in transactions. Furthermore, the system stores the date and a transaction ID.
- When you close the table, the system displays a save dialog.
- If you confirm it with [**YES**], the system generates a transaction.

In the table display, changes are displayed in the Comment column for each time series individually.

This lists all changes that have been made to a time series and arranges them in chronological order. In this way, users can record changes made to a time series "at a glance".

The following information is displayed:

- Transaction ID and transaction name
- Date of the user action
- Responsible user

Output options:

AquisNet automatically enumerates the changes, ordered by time stamp and time series.

- If a graph with the same measured values is open, the units in the axis labels for the values will be adapted automatically.

Specify remarks

- Remarks for the individual values can be entered in the Comments column.
- Double-click in the remarks column and select further steps.

11.5.37 Transactions

- The data status of time series before the changes is saved per transaction.
- This makes it possible to access the data before the change. The data is used in the event of a possible later restoration of a previous data status.
- To reduce the storage volume and the organisational effort of the transactions, AquisNet only saves the "affected" time range at a time and this with the data status before the changes.
- In addition to the pure measurement data of all affected time series before the change, AquisNet stores an identifier, a comment and the responsible user in transactions. Furthermore, the system stores the date and a transaction ID.

Features in This Dialog:

Create new transaction / Save manual changes

The transaction list is used to bundle time series with the information status before a change and only for the affected time range.

- Enter a transaction name and
- a comment as short description for other users.
- AquisNet also stores the creating user, the date and a transaction ID.

11.5.38 Transaction details

How to Work in This Dialog

Manage existing transactions

The Transaction details dialog shows the information that is stored in a transaction:

- Transaction name
- Unique transaction ID
- Transaction time stamp
- Time range
- User
- Comment
- In the tabular display in the lower part, the time series for which time series information was saved before a change are listed in the Time series name column.
- The Prior state number column shows how many other information states (each of which existed before a change) were previously saved per transaction.

11.5.39 Time axis shifting

With the **TIME AXIS SHIFTING** dialog you shift the time axis of the graph relatively by a desired time range.

Related Links:

- Working with graphs
- Working with tables
- Legend 95
- Plot properties 86
- Axis properties 82
- Graph 88

11.5.40 Time series browser

Based on this dialog, you have the possibility to display any number of time series with measured values in one graph or table or several ones (one graph or table per time series). To configure these views, you have various tools. In addition, there are other "organisational" possibilities.

Below you will find these features:

- Selecting the structure of the time series browser tree 100
- Search active stations in selected time range 100

How to Work in This Dialog

Selecting the structure of the time series browser tree

You can determine the structure of the time series browser tree by selecting the following anchors in the drop-down list box above the time series browser:

- Parameters
- Collection list
- Station
- Station group (usage)
- Station type
- Transaction
- Filter

You can make this selection in the drop-down list box above the time series browser tree.

Depending on the selection of the structure of the time series browser tree, the elements are arranged in the time series browser tree.

Search active stations in selected time range

A special feature of this time series browser tree configuration is the function "Search active stations in selected time range". This function is available to quickly record the stations that are or were active in a selected period.

- In the upper part of the dialog, use the two calendar functions to select date limits for an entry range.
- Now select the "Search active stations in selected time range" function in the context menu of the browser tree.
- The system marks the stations (or the time series contained) that do not contain data for the specified date range with a red cross.
- No time series are available in this station in the selected time range.

Time Series Selection

- If you want to select time series,
- double-click successively on the desired time series in the time series browser tree in the left-hand section of the time series browser.

Time series selected in this way are taken over in the time series selection window (in the right-hand section of the time series browser).

Entering the time range

Entering the time range to view the selected time series can be done in different ways.

You can

- select the date directly in the date field using the calendar function,
- or set it efficiently in the **MOST RECENT N DAYS FROM TODAY** box as a relative time range without editing the date boxes,
- or double-click on a time series in the time series browser selection window to select the time range for which measured values of the selected time series are available in the database.

Creating a new folder in the user time series collection list

With this function you can add as many additional collection lists as you like to the existing ones.

- To create a new folder, click on the parent folder in the tree structure into which the new folder is to be integrated.
- Open the context menu with the right mouse button to select the **NEW FOLDER** function.
- Select an identifier for the new folder and confirm with [**OK**].

Rename existing folder or collection list

- To do this, click on the entry in the time series browser tree that you want to change.
- Open the context menu of the tree entry by activating the right mouse button.
- Now select the **RENAME FOLDER/COLLECTION LIST** entry.
- Change the entry directly in the name field within the tree structure.
- By activating the [Return] key, you confirm your entry.

Deleting a folder or a collection list

Note:

You can only delete a folder if it no longer contains another (sub)folder.

Should you nevertheless have made such a selection, you will be informed by an error message.

- Select the desired subfolder and open the context menu with the right mouse button.
- Select the **DELETE FOLDER/COLLECTION LIST** function.
- Now confirm the dialog with [**OK**].

Manage time series selection as collection list

Time series that you have loaded into the time series selection dialog can be combined in a collection list:

- To do this, mark a time series of the selection in the time series selection dialog.
- This includes all existing time series.
- Now open the context menu with the right mouse button, or the Graph main menu item, and select the Manage time series selection as collection list function.

New Time Series Collection List Dialog

Now select a folder in the tree view in the Folder selection box and mark it.

Now enter a new name for the new folder, or

select a name from the drop-down list box.

- Click the [**SAVE**] button to confirm.

The new folder is created in the tree view and is ready for use.

Time series selection window

The time series selection dialog lists the time series that are selected to be displayed in the graph or table, for example.

If you do not want to display certain time series from this selection, you can remove these time series from the selection:

- To do this, open the context menu (right mouse button).
- Either select the Remove all time series from lists function, or
- select the Remove selected time series from list function.

View options

For each measured value, there is a read-only "original time series" and a "production time series" in which the measured values can be validated. (As long as no measured values are changed, both time series are identical. With a validation, the production time series is changed in relation to the original time series.).

- Specify the selection of a time series when loading the graph or table:
- Mark the desired option in the **VIEW OPTIONS** box.

Context menu options

Remove selected time series from the list

This option allows you to remove one or more selected time series from the time series list. Select several time series by activating the [**CTRL**] key and clicking with the mouse pointer

Remove all time series from the list

With this option you remove all time series from the time series list.

Combine selected time series into a collection list

Read section Combine selected time series into a collection list for further information.

Load predefined layouts

Preconfigured layouts by the user can be opened for graphical representations using the context menu.

- If you have taken over one or more time series into the time series dialog, now open the context menu with the right mouse button.
- Select the Load predefined layout entry.
- You can now activate the desired layout in an additional menu.
- Using this layout, AquisNet now activates the graph display.

The configuration of a layout can be opened using the main menu (**GRAPH** item)

Display graph and table

You have now selected one or more time series for viewing and defined a time range for the display.

- Now click the [**VIEW**] button, or
- select the Default view function from the context menu of the time series selection window,
- or the same function from the main menu item Graph.

You get the time series browser and the Show graph and Show table reports combined on the screen.

General functions of the browser tree view

Rename existing tree entry

The names of entries created by users (e.g. collection lists or station groups) in the tree view can be changed. This is not possible for entries that are used system-wide, such as parameters, station names, etc.

- Select the tree entry you want to rename.
- Open the context menu with the right mouse button.
- Select the **RENAME FOLDER/COLLECTION LIST** function.
- The field of the name in the tree view is activated for a user input.
- Change the entry directly in the Name in the tree view field.
- Confirm your entries with [**RETURN**].

Delete tree entry

Note:

You can only delete folders from the system as long as they do not contain any other folders. In this case, the system issues a notification message.

- Select the (sub)folder you want to delete in the tree view.
- Open the context menu with the right mouse button.
- Select the Delete folder/collection list function.
- Confirm your entries with [**OK**].
- The (sub-) folder is now deleted.

Restore time series states in production time series (recovery)

Recovery means: Restore the previous data status of one or more time series by means of information available in a transaction for these same time series. Data is changed within a certain time range. The transaction used only contains data for this time range.

To ensure data consistency, this action is also saved as a transaction. This writes the changes concerning the specific time range into the base production time series. Information on the previous state is again saved ("Save recovery as another transaction" 99).

Recovery for all time series of a transaction

With the Restore all affected time series states in their production time series function, you restore the state of basic production time series before the change that was saved in the opened transaction.

Recovery for individual time series of a transaction

With the Restore selected time series states in their production time series function, you restore the state of basic production time series before the change that was saved in the opened transaction.

Save recovery as another transaction

To ensure data consistency, a recovery is also saved as a transaction. This writes the changes concerning the specific time range into the base production time series. Information on the previous status is also saved.

- The transaction is transferred to the data storage.
- The transaction can now be opened in the time series browser in the tree view. To do this, select one of the transaction-related entry points from the drop-down list box.

Filter

Background

- The time series filter in the data collection center is an important tool to pre-filter the large amount of time series in the system so that evaluations can be carried out more quickly. Thereby you determine the amount of data from which you want to create evaluations via the filter view.
- The reduced amount of data applies to all selection options (station, station group, parameter, etc.).
- To use a time series filter, it must have been created in the system beforehand.

Select existing filter

To use a time series filter,

- select the desired filter from the drop-down list box.
- The filter selection is located at the top left of the interface.
- Then check the box on the left of the selection list. Only then the filter is used by AquisNet.
- **Note**: You can only select filters that have already been created in the system.

Opening and editing time series filters

To edit a time series filter,

- select the Filter option from the drop-down list box below the filter selection.
- The browser tree now shows the existing time series filter.
- Double-click to call up the configuration of a filter for editing, which is then displayed in the main window, see figure above.
- Now transfer the desired objects to the lower selection cells with a mouse click.
- Only these selected objects will be available later when using the filter.
- By clicking on already selected objects, you deselect them.
  Note:
- For example, if you select only Bree as station, a user with the current time series filter will only have access to time series for station Bree.
- If you select a station group, you have access to all time series recorded in the stations of this station group with the current time series filter.
- If you select a parameter, you have access to all time series containing data for this parameter with the current time series filter.
- If you select a station group, the stations contained in the selection cell are automatically taken over to the Station entity.
- If you remove a station group from a filter configuration, the station entries are retained. Thus, there is a coupling in the selection, but not in the deselection of station groups!
- Confirm your adjustments with [**SAVE**].

The edits are transferred and taken into account when the filter is selected via the selection list at the top left.

Origins

Background

"Persistent origins" are calculation rules (formation rules) for time series. "Persistent" stands for time series that remain in the system permanently, in contrast to time series that are created temporarily.

Such a formation rule specifies how defined source time series are to be calculated with each other according to a formula in order to obtain the measured values of the target time series.

You can define up to ten source time series depending on the origin type.

When you create a new time series, this means creating an origin for this new time series so that it can be calculated from a source time series.

Example:

The time series "daily mean values", for example, can be calculated from half-hourly mean values.

In this process, a value of the new time series is formed from the mean values of one day. The average value of the next day results in the next entry.

Call up origins

The dialog for origins is integrated into the time series browser. You can open details on the origin of a time series by clicking on the time series or the origin. You can open further functions for the creation of time series via the context menu.

To open origins for time series,

- expand the tree view (independent of the browser tree configuration) to the time series level.
- double-click the desired time series:
- The **ORIGIN** entry indicates that this is a time series generated by an origin. The plausibility check function is not available for time series generated by emergence:
- If the time series can be extended with regard to plausibility check entries, the tree entry contains the type of plausibility check(s) used.
- If the time series cannot be extended with regard to an origin entry or a plausibility checker, it is an import time series for which no plausibility checker has been created.
- By clicking on an origin entry in the plausibility parameters section (in the right part of the dialog), the parameters of a type are displayed:

Structure of the Dialogue

In the right part of the dialog, three sections inform about the attributes of the time series selected in the tree. You also define the properties of a new time series to be created in these three sections.

The entries below indicate the properties of the displayed time series:

- **TIME SERIES** section
- **TIME SERIES**: The name of the time series, consisting of station abbreviation and time series definition, can be freely selected when creating a new time series if the time series is not imported into the AquisNet system.
- Time series definition
- Time series type
- Time period
- Integration time
- Source time series section
- Display of the time series that serve as source time series for the calculation of the time series.
- The time period of the source time series is smaller than the time period of the target time series when it is created, and is also divisible by it.
- When using an origin of the free formula type, the time periods of the source time series and target time series must be identical.
- Origin specific parameters section
- The origin-specific parameters are different for different types of time series. The table provides information on these correlations. Below you will find the list of origin-specific parameters and explanations of their respective meaning:
- minimum coverage [%]: The percentage indicates what percentage of the possible measured values must be present in the source time series in order to obtain a value of the target time series that is valid.
- maximum gap [values]: This value specifies the maximum number of consecutive time stamps in the source time series that can have invalid values during the integration time of a target value for the target value to be valid.
- minimum coverage if last value is missing [%]: If the last value of the source time series is not present in the integration time, this additional setting of the minimum coverage serves as an additional check of the validity of the time series.
- Percentile: A K-percentile value defines the value of a measured value time series below which K percent of all measured values lie.
- Time-Hour-Start: Time with which the start of a period within a day can be defined at which the calculation starts.
- Calm condition: The Calm condition (in m/s) represents an adjustable limit value with which you can define up to which wind speed calm is assumed. The usual limit value is 0.5 m/s.
- For values equal to or less than this wind speed, no assignment of the wind direction is made in the display of the concentration rose
- Limit value/threshold: The limit value defines a value against which all measured values of a time series with time series type "Exceedance frequency" are to be checked for exceeding or falling below.
- Time offset [sec]: With a time offset, you specify the shift between the time stamp and the integration time. The integration time shifts backwards (clockwise with a positive sign) or forwards (counterclockwise with a negative sign) by the specified offset seconds.
- With the time offset parameter you can create or calculate time slices (time raster and integration time equal to 24 hours) and time intervals (time raster and integration time unequal).
- Parameter correction factor: This factor enables the subsequent correction of measured values based on manual input. This way, laboratory readings can be corrected later if laboratory conditions were not optimal. It is also conceivable that an incorrect calibration of a measuring device can be corrected in this way.

The correction is made by multiplying the measured values by the manually entered value. The correction factor is therefore dimensionless.

Time series type, time series definition, origins & origin specific parameters

The terms time series type, time series definition, origins and origin-specific parameters and their functions are closely linked using the two dialogs Origins/Create new time series and Configure time series definitions.

The functional context is therefore explained on the basis of the following points. These points should further anticipate some important information for creating a new time series:

In the Origins/Create new time series dialog, the selection of the time series type is done using the Data type/Time series definition type (TSD type). The time series type is a main component of the time series definition, and thus of the generation of time series.

A specific time series type is therefore created in the system for each time series.

If you want to create a new time series, but the desired time series definition is not listed in the time series definition drop-down list box, you have the option to create the desired time series definition in the Configure time series definitions dialog.

The selected time series type (arithmetic, percentile value, etc.) is decisive for the time series definition and thus ultimately also for the calculation of a new time series: This also determines which origination-specific parameters can be configured in the Origins dialog when creating a new time series.

Therefore, at this point of the "Create a new time series" workflow, the two dialogs Origins/Create new time series and Configure time series definitions are involved.

In the table below the relation of "Time series type" – "Number of source time series" – "Origin-specific parameters" is listed.

Origin templates for time series types

Time series type	Number of source time series	Origin-specific parameters
arithmetic Mean value	a source time series	1. Minimum coverage 2. Maximum allowed gap 3. Minimum coverage if last value is missing
Minimum value	a source time series	1. Minimum coverage 2. Maximum allowed gap 3. Minimum coverage if last value is missing
Maximum value	a source time series	1. Minimum coverage 2. Maximum allowed gap 3. Minimum coverage if last value is missing
Percentile value	a source time series	1. Percentile 2. Minimum coverage 3. Maximum allowed gap 4. Minimum coverage if last value is missing
Total value	a source time series	1. Minimum coverage 2. Maximum allowed gap 3. Minimum coverage if last value is missing 4. Time-Hour-Start
Vectorial wind direction	1. Source time series Direction 2. Source time series Value	1. Minimum coverage 2. Maximum allowed gap 3. Minimum coverage if last value is missing 4. Calm condition
Vectorial wind speed	1. Source time series Direction 2. Source time series Value	1. Minimum coverage 2. Maximum allowed gap 3. Minimum coverage if last value is missing 4. Calm condition
Scalar wind direction	1. Source time series Direction 2. Source time series Value	1. Minimum coverage 2. Maximum allowed gap 3. Minimum coverage if last value is missing 4. Calm condition
Scalar wind speed	1. Source time series Direction	1. Minimum coverage 2. Maximum allowed gap 3. Minimum coverage if last value is missing 4. Calm condition
Exceedance frequency	a source time series	1. Limit value/Threshold 2. Minimum coverage 3. Maximum allowed gap 4. Minimum coverage if last value is missing
Free formula	01. "Source time series $a" 02. "Source time series $b"	1. Free formula

	03. "Source time series $c" 04. "Source time series $d" 05. "Source time series $e" 06. "Source time series $f" 07. "Source time series $g" 08. "Source time series $h" 09. "Source time series $i" 10. "Source time series $j"	
AOT 40 (daily)	1 Source time series	1. Limit value/Threshold

Create new time series

To create a new origin,

- Open the context menu in the tree view.
- Select the **CREATE NEW TIME SERIES** entry.

The **CREATE NEW TIME SERIES** dialog opens.

Delete time series

With this program function you can remove a time series you have created from the system again. Import time series can never be deleted.

To delete a time series,

- first select the time series to be deleted in the tree view.
- If you have marked the time series with a mouse click, open the context menu with the right mouse button.
- Select the **DELETE TIME SERIES** function.
  A confirmation dialog opens.
- Now confirm with [**YES**] to continue the process, otherwise cancel with [**NO**].
- By confirming, the system checks whether it can delete the time series without deleting other dependent time series.
- If there are dependencies between the time series to be deleted and other time series, another dialog opens.
- It displays how many dependent time series use the selected time series as a source time series.
- Confirm the dialog with [**YES**] to continue the process, otherwise cancel with [**NO**].
- If the time series has been removed from the system, you will get a message about this, which you should confirm by pressing the enter key.

This completes the deletion process.

11.5.41 Plausibility check

By means of a plausibility check, you can have the system check values of imported time series using preconfigured check criteria. However, this only applies to the values of imported time series, but not to those that have been created.

The tree view in the left-hand window shows you all the stations.

The parameters that are recorded in these stations are arranged below. In the next lower level you will find the time series for each parameter.

By clicking on a plausibility type, the display of the corresponding parameters of a type appears under the Plausibility parameters section (in the right part of the dialog). If you find several (but no more than four) plausibility check types under the time series, a red arrow points to the type whose parameters are displayed in the right-hand part of the dialog.

Since you can only assign each plausibility check type to a time series once, the maximum number of plausibility types for a time series is four.

Related Links:

- Plausibility check types 67
- Create new plausibility check 69
- Delete plausibility check 69
- The control bar 52

Features in This Dialog:

Create new plausibility check

To assign a new, additional plausibility check to a time series,

- select the corresponding time series by mouse click.
- Open the context menu with the right mouse button and
- Select the **CREATE NEW PLAUSIBILITY CHECK** function.
- Now select the new plausibility check from the **TYPE OF PLAUSIBILITY CHECK** drop-down list box in the New plausibility check 107 dialog.
- You can no longer select a type of plausibility check that has already been set.
- Now confirm your selection by clicking the [**ACCEPT**] button to further determine the plausibility type (see Related links).
- Confirm your entries with [**OK**].
- The new plausibility type is created in the system.

Delete plausibility check

- In the tree view, select the plausibility check type you want to delete by mouse click.
- Open the context menu with the right mouse button and
- select the **DELETE PLAUSIBILITY CHECK** function.

The plausibility check type is removed without a control dialog. However, you can easily undo the deletion process by creating it again.

11.5.42 New plausibility check

This dialog is a part of the new creation process of a plausibility check for an import time series.

As a dialog box for the selection of a new plausibility types, you will find in the drop-down list box the plausibility types that have not yet been assigned to the plausibility of the time series data and can therefore be selected. Related topics:

- Plausibility check (DEZ180) 106
- Plausibility checks
- Create new plausibility check 69

Features in This Dialog:

Selecting a type of plausibility check

- Select a new plausibility check from the drop-down list box.
- A plausibility type that has already been defined is no longer available for selection in the list.
- Now confirm your selection by clicking the [**ACCEPT**] button to further determine the plausibility type.
- The possible parameters are explained in more detail under Plausibility check types 67.
- Confirm your entries with [**OK**].

The new plausibility type has been created in the system and you now get back to the Plausibility check (DEZ180) 106 dialog.

11.5.43 Manual plausibility check

AquisNet allows you to check plausibility manually.

For example, you can subsequently check old data that entered the AquisNet system without plausibility checks for implausibility.

Features in This Dialog:

Manual plausibility check

- First enter a start date using the calendar function.
- Then enter an end date using the calendar function.
- Confirm your entries by clicking the [**EXECUTE MANUAL PLAUSIBILITY CHECK**] button.
- AquisNet carries out the plausibility check of the data automatically.

11.5.44 Time series versions

This dialog serves to save editing versions of your time series; this allows you to access a former editing status any time.

In addition, the dialogue contains a list of all existing versions.

- It is necessary to enter a name for the new version!
- If required, enter a comment.
- In case you find here versions that are not required, you can select them and delete them from the database with [**DELETE**].
- Confirm your entries with [**OK**].

Further Hints

Versioning your time series allows you to e.g. avoid changes to your measured values or layout/formats from being permanently saved to the database. For example, if you find a mistake when editing your data, you can revert to the last saved version.

11.5.45 [Additional displays] on/off

Use the [**ADDITIONAL INFORMATION**] dialog to specify for which plots in the graph window the additional information being specified before (e.g. control bar/MinMax/etc.) are supposed to be made visible.

Click the corresponding check box in front of the time series.

11.5.46 Create new time series

You can maintain or create new time series in this dialog. Various setting options allow many configurations.

Related Links:

- Persistent origins
- Create new origin
- Time series type, time series definition, origins & origin specific parameters
- Create new origin with "free formula"
- Calculation formulas and Boolean formulas
- Delete time series

Features in This Dialog:

Create new time series

- From the Station and Time Series drop-down list boxes in the Time series header section, select the entries that you want to base the new time series on.
- If the desired time series definition is not listed in the time series drop-down list box, you have the option of creating the desired time series definition online using main templates.
- The time series type is automatically taken from the time series specification.
- A name for the time series is formed from these two entries in the **TIME SERIES NAME** entry.
- Then click the [**ACCEPT**] button.
- In the Source time series section, select a time series for each item listed from the respective drop-down list box.
- The time grid of the source time series must be smaller than the time grid of the "target" time series, and also divisible by it.
- In the Specific parameters section, the yellow highlighted input box for minimum overcoverage (see above) requires a mandatory entry.
- You decide about entries in the other two input boxes.
- Finally, confirm your entries with [**OK**].
- The new time series is created in the system.

Time series head section

- From the Station and Time series definition drop-down list boxes select the station and the time series definition that you want to base the new time series on.

- If the desired time series definition is not listed in the time series definition drop-down list box, you have the option of creating the desired time series definition online using main templates.
- For time series that are not imported into AquisNet, you can change the time series name.
- Click with the mouse in the name box and change the entry according to your wish.
- Then click the [**ACCEPT**] button.
- You can now further define the new time series in the two sections below:

Source time series section

- For each item listed, select a time series from the respective drop-down list box to create the new time series with the selected source time series.
- Only the time series belonging to the station you selected in the Time series head section are offered.
- Select the time grid of the source time series so that it is smaller than the time grid of the "target" time series, and also divisible by it.
- For an origin of the "Free formula" type, the time grids of the source time series and the target time series must be identical.

Specific parameters section

- Yellow highlighted input boxes require mandatory input. You decide about entries in input fields that are not highlighted with colour.
- By selecting the time series definition and thus the time series type, the system determines the list of origin-specific parameters.
- After completing your entries, confirm them with [**OK**].
- The new time series is now created in the system.

Create new time series with free formula

This tool gives you the possibility to specify formulas that serve as the creation rule (origin) of a time series from up to ten other time series.

- Select the desired station in the Station drop-down list box.
- From the Time series definition drop-down list box, enter an entry of the **FREE FORMULA (TYPE)** type.
- If the desired time series definition does not yet exist, you can design a new time series definition online using main templates according to your wish to define a formula later.
- In the Time series name dialog you can now select a suitable name.
- Click the [**ACCEPT**] button.
- Select a time series for each of the up to ten items listed from the respective drop-down list box to form the new time series with the selected source time series.
- Now enter a formula in the "Specific parameters" section:
- Start by entering the expression "expr" for a calculation formula or the expression "if" for a Boolean formula.
- Now insert – separated by a space – the calculation rule behind it with which the target value of the new time series is to be calculated from the individual values of the selected source time series.
- Note the format of the formula entries: The expression "$x" (with x for "a" to "i") represents the value of a time series at a time stamp. This value is calculated against the other value(s) of one or more other time series with the same time stamp according to the calculation rule.
- The notation types for calculation formulas, as well as for Boolean formulas, can be found in the Calculation formulas and Boolean formulas section
- Now confirm your entries with [**OK**].
- The newly calculated time series is created in the system and is ready for use.

Calculation formulas

Formulas are formed according to the known rules with constants, functions, operators and brackets, whereby the usual object hierarchy – potentiation before multiplication/division before addition/subtraction – applies. When entering constants, please pay attention that the system distinguishes float and integer constants. Thus, the expression 1/2 as division of two int values (integer division) yields the result 0.0, while 1.0/2.0 as division of two float values yields the result 0.5. If the data type of an argument is not correct, a type conversion from float to int or vice versa is automatically carried out. The conversion from float to int is done by separating the decimal places.

- Standard operators
  - Addition: float1+float2
  - Subtraction: float1-float2
  - Multiplication: float1* float2
  - Division: float1/float2
  - Division for integer values with integer result: int1 div int2
  - Potentiation: float1^float2
  - Comparison operators, result is of type int (0 or 1): float1 <, <=, =, >, >= float2
- Standard functions
  - Absolute value of the argument: float abs(float)
  - Sign of the argument: float sign(float) (-1, 0 or 1, if arg < 0, == 0 or > 0, respectively)

  - Sine of the argument (in radiant): float sin(float)
  - Cosine of the argument (in radian): float cos(float)
  - Tangent of the argument (in radian): float tan(float)
  - Arc tangent of the argument (in radian): float arctan(float)
  - Power with Euler's constant e as the base and the argument as the exponent: float exp(float)
  - Decadic logarithm of the argument: float log(float)
  - Natural logarithm of the argument: float ln(float)
  - Square root of the argument: float sqrt(float)
  - Integer part of the argument: float int(float)
  - Decimal part of the argument: float frac(float)
  - Pseudo-random number between 0 and 1: float rnd()
- Default constants
  - pi: approximation to the circular number
  - e: Approximation to Euler's constant e

Boolean formulas

Boolean formulas allow you to specify conditions that control the calculation of a formula, for example, calculating a target time series depending on the key curve of a source time series above a certain value.

Example:

The expression “if { $a<99 } { expr $a+$b } else { expr $a-$b }” means:

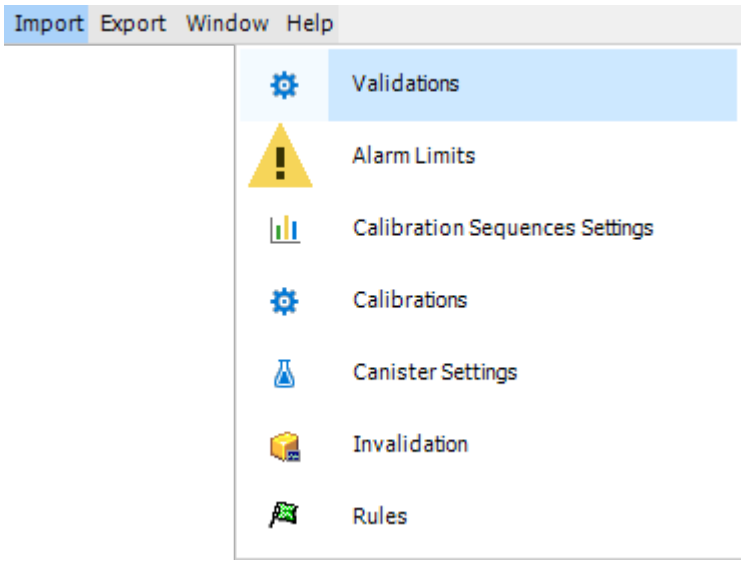
Only if the value ($a) at a time stamp of the 1st source time series is smaller than 99 g/cm³, an addition with the corresponding value ($b) of the 2nd source time series takes place. If this is not the case, a subtraction of ($a-$b) takes place.

The following standard operators and standard functions represent the most important notation types:

- Standard operators
  - binary AND operator, result is of type int: int1 AND int2
  - binary OR operator, result is of type int: int1 OR int 2
  - binary negation, result is of type int: NOT int
- Standard functions
  - <float1> if <intBool>!=0
  - <float2> if <intBool>==0
  - float if(intBool, float1, float2)

12 Settings for Interface Checks

TCEQ uses the parameter-related settings for different interface checks explained here.



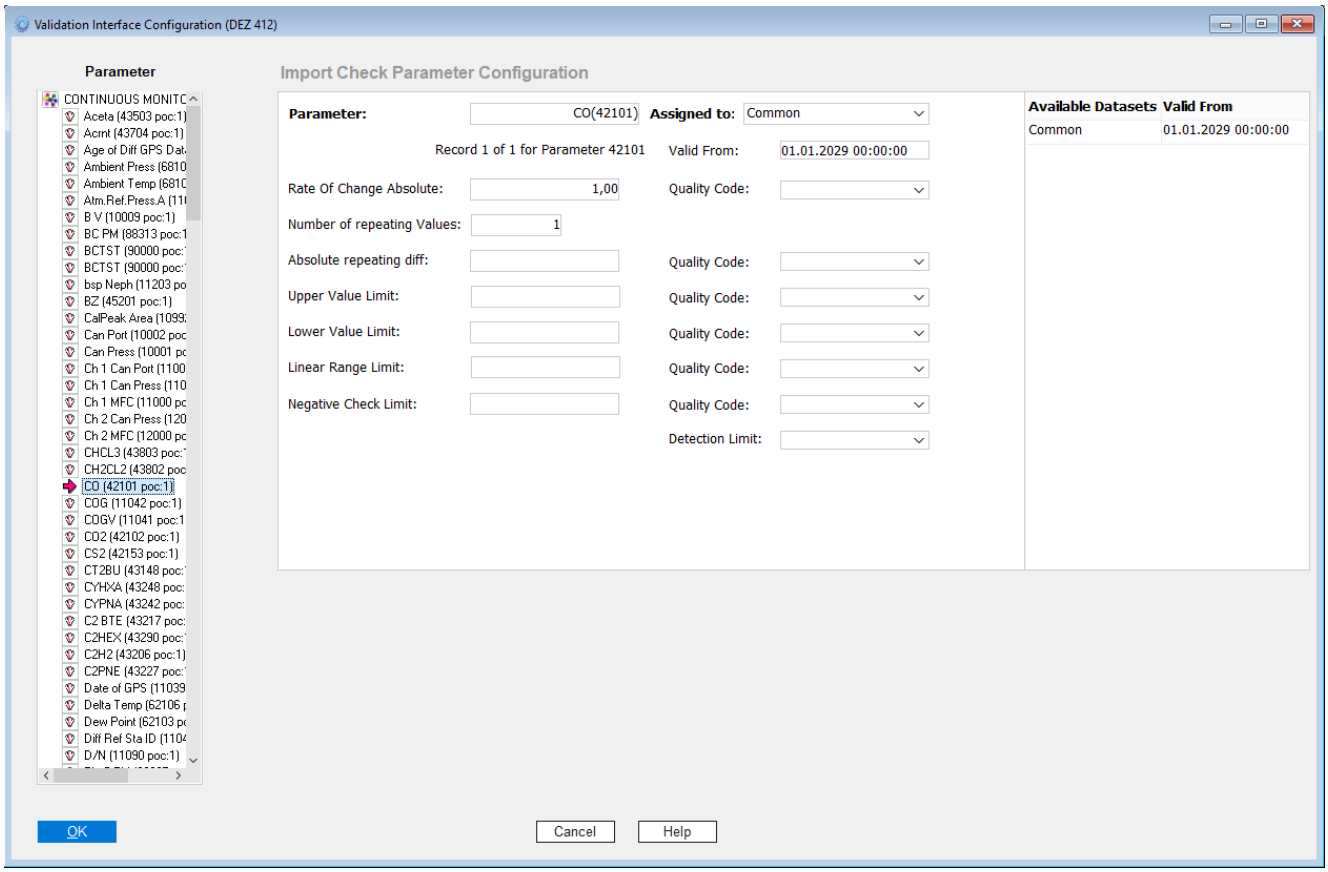
The various checks are dealt with in the following topics:

- Import Validations 111
- Alarm Configuration 113
- Calibration Configuration 114
- Canister Settings 117

12.1 Import Validations

You can configure import validations to be performed on your incoming parameter data.

- Start **VALIDATIONS** from the **IMPORT** > **INTERFACE** main menu.
- Select the parameter for which you want to specify the validation settings in the list on the left.



- Add new settings with the context menu item **ADD**.
- Specify the settings.
- For each setting, specify a quality code for values that violate the threshold value. Thus, if the configured threshold is exceeded, the values get the quality code defined for the condition.
  Note: The quality codes are configured here 27.

Element/Condition	Description
Parameter	Here the selected parameter for which settings are configured is displayed.
Assigned to	Select a station if you want to configure validations for single stations or single station groups. If left empty, the validation settings are valid for all stations where the parameter is measured. This is indicated by **COMMON**.
Valid From	Specify the start of validity for the current configuration.
Available Datasets/Valid From	You can configure import validations for different time ranges and with different values. New configuration sets are automatically added to this list with all existing configurations. You can identify **COMMON** settings or settings for single stations or station groups. Click en entry to load the respective settings into the dialog.
Rate Of Change Absolute	Defines the absolute rate of change. This specifies how much two consecutive values can differ from each other.
Number of Repeating Values	Defines the maximum number of repeating values. This specifies the number of values that are checked for difference to trigger the quality code for the next (repeating difference) setting. Use to detect e.g. frozen measuring devices.
Absolute Repeating Diff	Defines the absolute value of repeating differences. This specifies when two repeating values are identified as same or different. Use to detect e.g. frozen measuring devices.

Element/Condition	Description
	Example: Number of repeating values = 5 and absolute repeating difference = 100 means: If 2 of 5 repeating values differ more than 100, they are identified as not repeating but different values.
Upper Value Limit	Defines the upper limit of acceptable values. Values above that limit get the quality code configured.
Lower Value Limit	Defines the lower limit of acceptable values. Values below that limit get the quality code configured.
Linear Range Limit	This specifies
Negative Check Limit	This specifies
Detection Limit	Defines the quality code for values below the parameter's detection limit.

- Save new sets with [**OK**].
  Note that you can display the parameter long names with

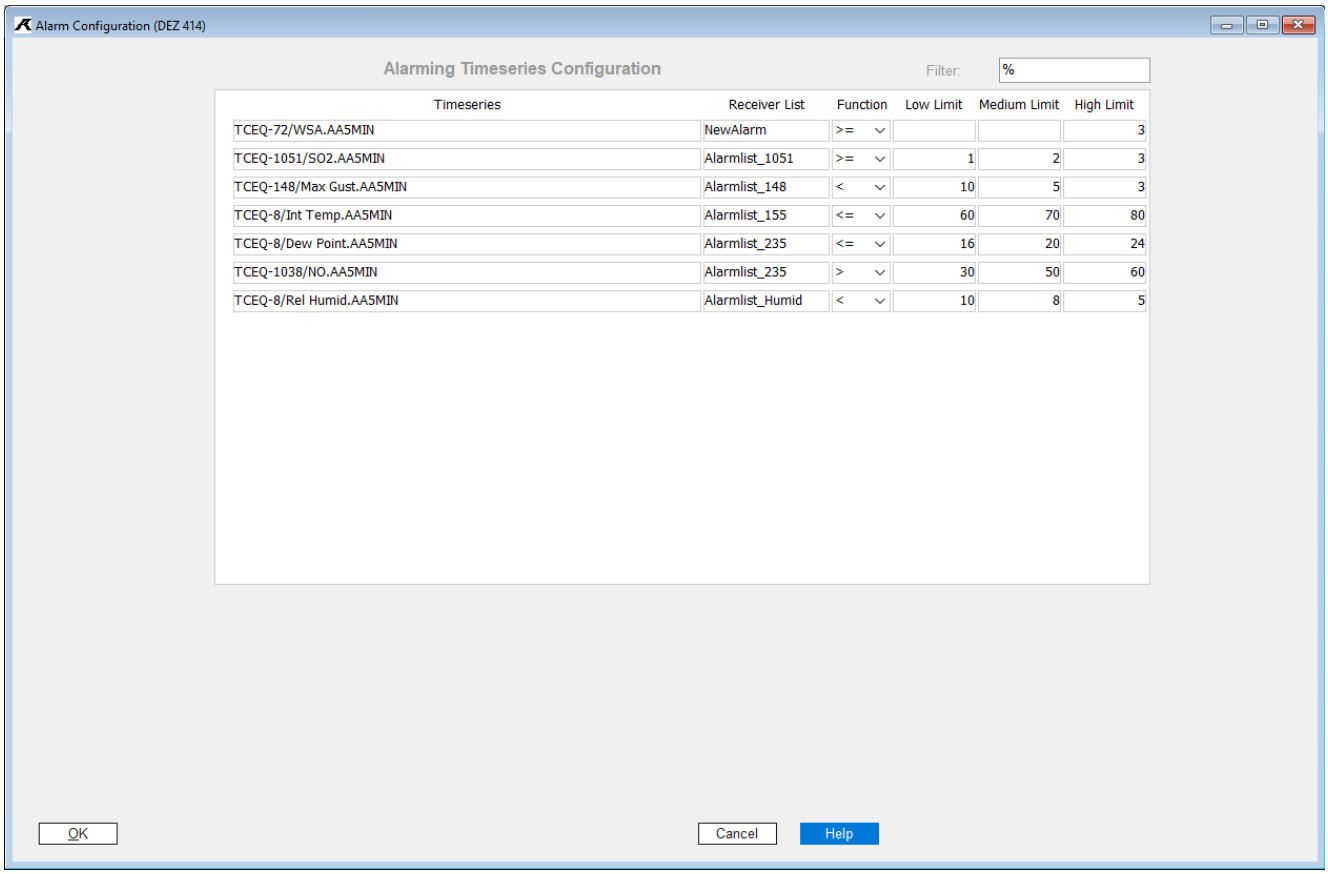
```
[TREE]
DETAILED_ITEMS=1
```

in the `dez.ini` in your installation folder.

12.2 Alarm Configuration

You can configure alarm configurations to be used when working with the KISTERS AlarmManager (KiALM). For this, you specify a threshold value for each base production time series (not for aggregated time series!) to be monitored; if the time series values exceed or fall below this threshold, an alarm is triggered. During data import, the system checks whether alarm thresholds need to be monitored.

- Start **ALARM LIMITS** from the **IMPORT** > **INTERFACE** main menu.



- Add new alarm configurations for your time series with the context menu item **ADD**.
- For each alarm setting, specify:

Element	Description
Filter	In case of long lists of time series, filter the time series by their names.
Time Series	Select the time series for which you want to trigger alarms.
Receiver List	Select a receiver list. These lists are used in the KISTERS AlarmManager (KiALM to send alarms to certain receivers by e.g. email. This list is transferred to the KISTERS AlarmManager (KiALM) together with the alarm data in XML format.
Function	Here you select the operator to be used for the alarm condition. Select e.g. equal to/smaller than or larger than.
Low Limit	Specify the time series value that will trigger a low-level alarm according to the selected operator. If your time series values are e.g. larger than the value specified here, a low-level alarm is triggered.
Medium Limit	Specify the time series value that will trigger a medium-level alarm according to the selected operator.
High Limit	Specify the time series value that will trigger a high-level alarm according to the selected operator.

Note: You do not need to specify all limit levels.

- Save new alarm configurations with [**OK**].

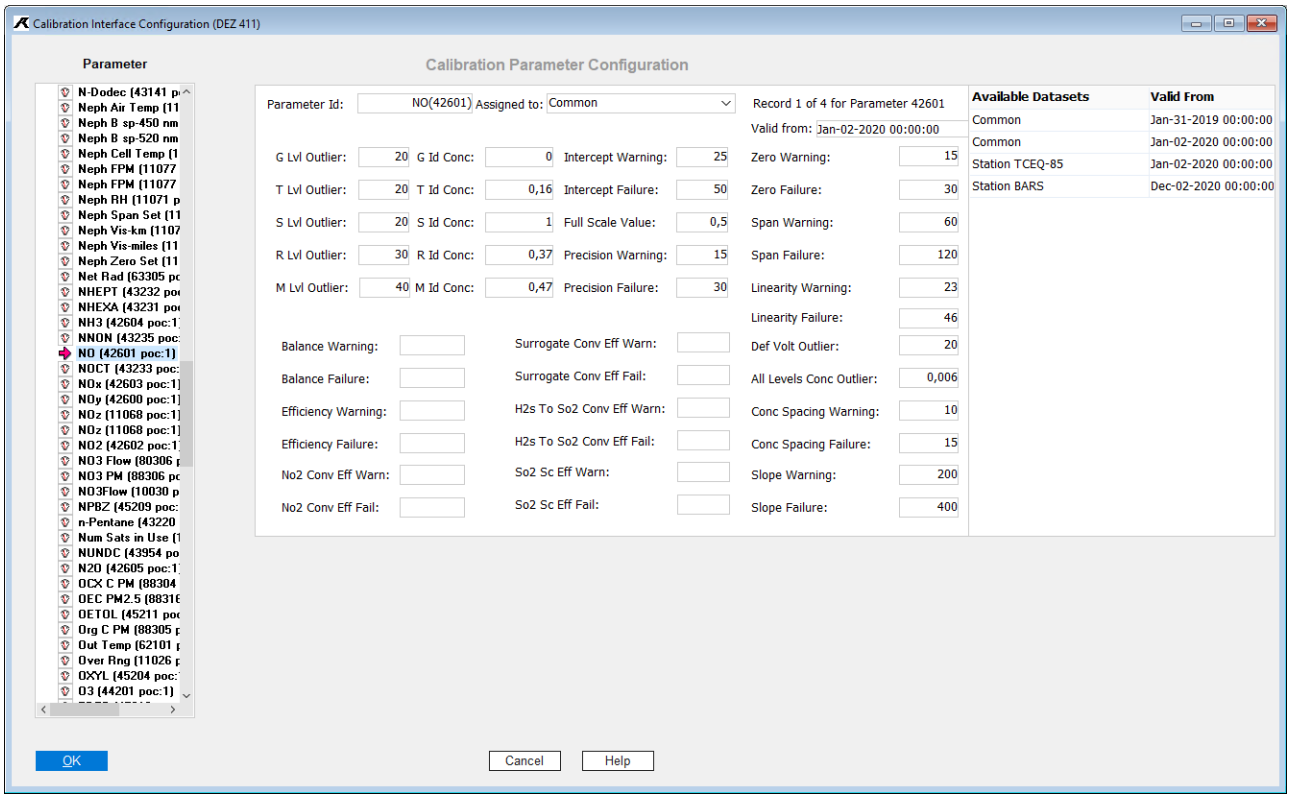
12.3 Calibration Configuration

You can create calibrations with their thresholds 114 and define your own calibration sequences 116 for your measuring devices.

Calibrations

To configure calibrations and their settings, proceed like this:

- Start **CALIBRATIONS** from the **IMPORT** > **INTERFACE** main menu.
- Select the parameter for which you want to specify calibration settings in the list on the left.



- Add a new set of calibration settings with the context menu item **ADD**.
- Specify the following settings:

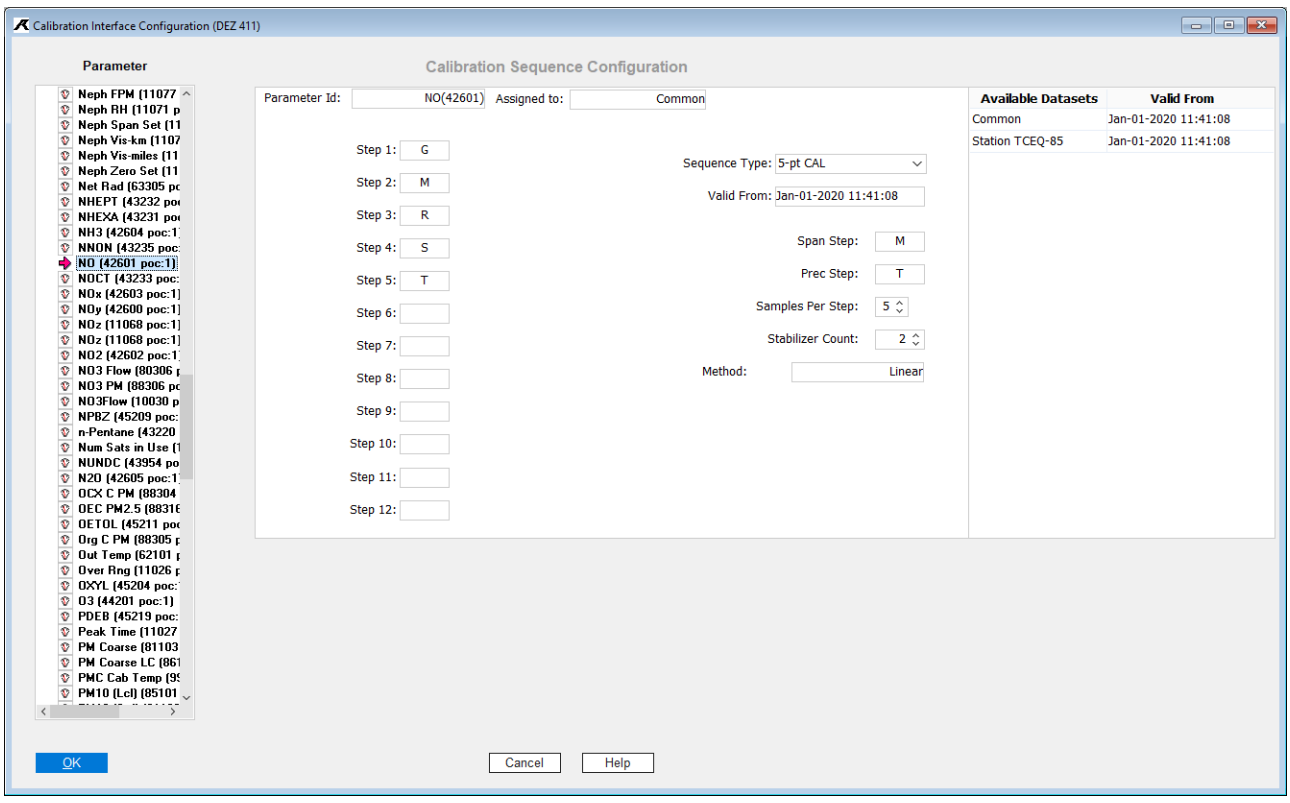
Element	Description
Parameter	Here the selected parameter for which settings are configured is displayed.
Assigned to	Select a station if you want to configure calibrations for single stations or single station groups. If left empty, the calibration settings are valid for all stations where the parameter is measured. This is indicated by **COMMON**.
Valid From	Specify the start of validity for the current configuration.
Available Datasets/Valid From	You can configure calibrations for different time ranges and with different values. New configuration sets are automatically added to this list with all existing configurations. You can identify **COMMON** settings or settings for single stations or station groups. Click en entry to load the respective settings into the dialog.
G/T/S/R/M Lvl Outlier	Calibration values for level outliers. Separated by levels G, T,S, R, M.
G/T/S/R/M ID Conc	Calibration values for concentration.
Intercept Warning	Calibration value for the intercept deviation to trigger a warning.
Intercept Failure	Calibration value for the intercept deviation to trigger a message of failure.
Full Scale Value	Calibration value for
Precision Warning	Calibration value for
Precision Failure	Calibration value for
Balance Warning	Calibration value for
Balance Failure	Calibration value for
Efficiency Warning	Calibration value for

Element	Description
Efficiency Failure	Calibration value for
No2/Surrogate/H2s to SO2/So2 Sc/ Conv Eff Warn	Calibration value for
No2/Surrogate/H2s to SO2/So2 Sc/ Conv Eff Fail	Calibration value for
Zero Warning	Calibration value for
Zero Failure	Calibration value for
Span Warning	Calibration value for
Span Failure	Calibration value for
Linearity Warning	Calibration value for
Linearity Failure	Calibration value for
Def Volt Outlier	Calibration value for
All Levels Conc Outlier	Calibration value for
Conc Spacing Warning	Calibration value for
Conc Spacing Failure	Calibration value for
Slope Warning	Calibration value for
Slope Failure	Calibration value for

Calibration Sequences

To configure user-defined calibration sequences (up to 12 at the moment), proceed like this:

- Start **CALIBRATIONS SEQUENCES SETTINGS** from the **IMPORT** > **INTERFACE** main menu.
- Select the parameter for which you want to specify a calibration sequence in the list on the left.



- Add a new user-defined calibration sequence with the context menu item **ADD**.

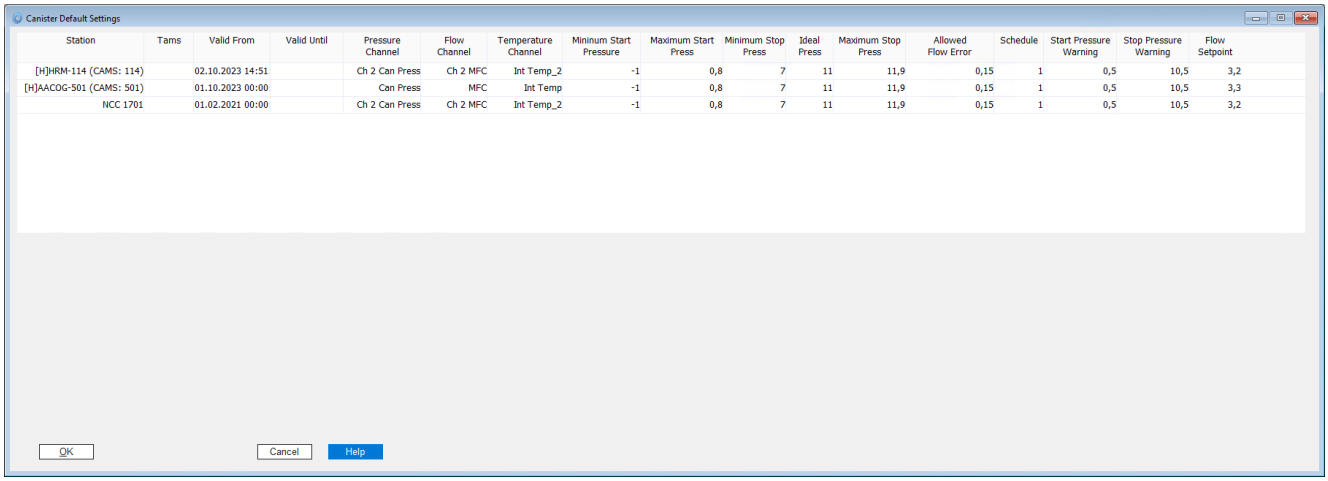
- Specify the following settings:

Element	Description
Step1 through 12	Specify the sequence of the calibration. Separated by levels G, T,S, R, M.
Sequence Type	Select the type of calibration sequence (e.g. a 5-point calibration). Here, also an enhanced calibration type can be selected.
Valid From	Specify the start of validity for the current configuration.
Available Datasets/Valid From	You can configure calibration sequences for different time ranges and with different values. New configuration sets are automatically added to this list with all existing configurations. You can identify **COMMON** settings or settings for single stations or station groups. Click en entry to load the respective settings into the dialog.
Span Step	Specify the step by levels G, T,S, R, M to be used to calibrate the span of calibration.
Prec Step	Specify the step for calculation of the precision.
Samples Per Step	Specify the maximum number of samples to be used for each calibration sequence step. Example: If set to 5, each step of the calibration will examine 5 samples to get the calibration values.
Stabilizer Count	Specify the number of samples that are not taken into account when calibrating since the serve during startup for stabilization of the calibration conditions.
Method	If **SEQUENCE TYPE** is set to enhanced, you can select a method (single or dual linear or square/cubic polynoms) to define the calibration equation differently from linear with slope, etc.

12.4 Canister Settings

You can configure canister settings for your test gas samplers in the stations. After the canister run (normally 24 hours) you can evaluate the different parameters of the test run by the settings defined here. Use these settings to judge how the filling process worked.

- Start **CANISTER SETTINGS** from the **IMPORT** > **INTERFACE** main menu.



- Add a new set of settings for your canisters with the context menu item **ADD**.
- Specify the following settings:

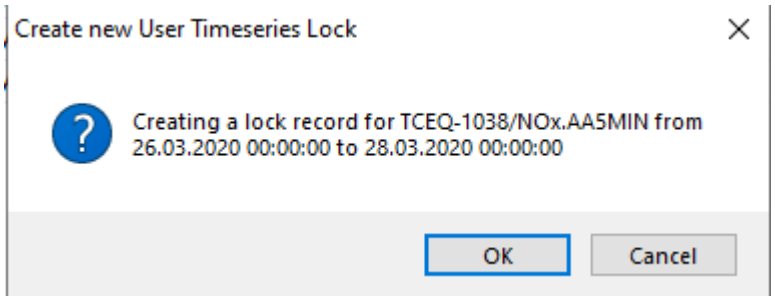
Element	Description
Station	Select the station where the canister is installed for which you define settings here.

Element	Description
Tams	Any arbitrary string for the current settings.
Valid From	Specify the start of validity for the canister settings.
Valid Until	Specify the end of validity for the canister settings. If no end date is specified, the canister run settings are valid indefinitely.
Pressure Channel	Specify the channel that provides the pressure data for the canister run. The values from this channel can be used to monitor the correctness of the gas filling during the canister run.
Flow Channel	Specify the channel that provides the flow values to be used to monitor the correctness of the gas filling during the canister run.
Temperature Channel	Specify the channel that provides canister temperature data to evaluate the correctness of the canister run.
Minimum Start Pressure	Specify the minimum pressure at the start of the canister run. The pressure on start of the canister run must lie between the minimum and maximum start pressure. Valid for **all pressure values defined here**: If exceeded or underrun, the canister run is judged as invalid.
Maximum Start Pressure	Specify the maximum allowed pressure at the start of the canister run.
Minimum Stop Pressure	Specify the minimum allowed pressure at the end of the canister run.
Ideal Pressure	Specify the pressure value inside the canister that is regarded as optimum.
Maximum Stop Pressure	Specify the maximum allowed pressure at the end of the canister run.
Allowed Flow Error	Specify the maximum allowed error for flow measurement during canister run.
Schedule	Select a canister sampling schedule. Normally, the canister runs every 3 or 6 days. **Note**: The schedules are managed in the SYS_REP module.
Start Pressure Warning	Specify the pressure value where a warning is provided at the start of the canister run.
Stop Pressure Warning	Specify the pressure value where a warning is provided at the end of the canister run.
Flow Setpoint	Specify the set point (target value) for the flow.

See canister functions in general DMO documentation.

13 Time Series Data Locks

Use this to create an editing lock on the time series.



Index

P

S

T

V

Z