Operating Manual

catman® Enterprise 9.1
System Documentation Part II

HBM

Brüel & Kjær

CONTENTS

1 GETTING STARTED

1.1 Introduction

This documentation describes the server part of catman Enterprise, in the following referred to as MD Server system. When installing catman Enterprise to work as a server, programs MD Server and MC Definition List are also copied to the computer.

The client part of catman Enterprise is described in Part III of this documentation: Operating Manual MD Client.

The hardware necessary for working with the MD Server is described in Part I (Operating Manual System Documentation).

1.1.1 What does this documentation contain?

This documentation contains information about:
- The software components supplied with the server part of catman Enterprise
- How to install the software
- Operating the server

1.1.2 Notes on the documentation

This documentation is only part of the catman Enterprise documentation. A list of all documents concerning catman Enterprise can be found in Part I (Operating Manual System Documentation), in Chapter B, *List of Operating Manuals*.

1.1.2.1 Prerequisites

This manual assumes that you:
- are familiar with your Windows® operating system,
- know how to use online help files.

1.1.2.2 Typographical conventions

☞ Unless specified otherwise, all references point to chapters in the same part of a documentation.

For the sake of clear, unambiguous labeling and better legibility, we use the following conventions in this manual:

⚠️ We have flagged important paragraphs with this caution symbol.

☞ This symbol points to a significant detail or specific characteristic.

💡 This symbol marks a tip or an interesting feature.

This marking is used to highlight individual terms in the text. Your **input**, all **push buttons**, **control boxes**, the names of **input fields** etc. are also marked. The **menus**, **commands**, **dialog fields** and **boxes** as well as **register tabs** used in the program are likewise marked. Window or dialog titles and the names of Panel objects start with a capital letter.

<user name>: This type of bracket is used to mark a parameter, in this case the user name, which is inserted or has to be filled in instead.

Text in blue color indicates a link. In a PDF you can click on the text to go to the relevant section, in a printed manual check the page specified.
This symbol means more information is available in the online help.

We hope that these markings will allow you to quickly identify the topics or menus referred to and enable you to find your way round the program.

How to get help on a specific topic in the online help?
Type your search phrase into the **Find in this help** field and press RETURN. As an alternative check the **Index** tab in the help.

1.1.2.3 Basic layout of this documentation

This chapter contains:
- A general description of the system
- The list of operating requirements
- A guide through the installation procedure
- How to install the database server when using MD Trend (optional Software)

Chapter 2, MD Server on page 31, describes the MD Server software:
- System administration and access rights
- Procedures for configuring and starting measuring operations (tests)
- How to define a visualization

- Using a VBScript procedure
- Optional settings

Chapter 3, MC Setup on page 137, describes the MC Setup software:
- How to:
    - set up and calibrate measuring channels
    - set up computation channels
    - define channel groups
    - search for channels
- Information about how to proceed when changing the hardware configuration of a signal conditioning device MGCplus in a cabinet

Chapter 4, MC Definition List on page 225, contains:
- Description of how to proceed when setting up the list of required measuring channels (MC Definition List)
- Explanation of how the necessary sensors may be matched to the existing hardware
- The procedure for creating a wiring scheme

Chapter 5, MD Export on page 243, describes:
- How to proceed when extracting channel data from several tests of a project
- How to combine several tests into one data file

The last chapter contains the Index on page 253

1.2 What does the MD Server do?

1.2.1 Overview

The MD Server is used to control the system and all data acquisition parameters. As a standard the software for setting up the signal conditioning devices (MC Setup) is called up from this software as well. An alternative setup procedure is used by MC Definition List which may be called up from MC Setup or run separately. MD Export allows for extracting channel data from several different tests and exporting these into various formats. You may start MD Export also from within MD Server: **Tools/Script** ribbbon tab, **Channels/Export** group.

The measured data is first collected from the MGCplus or QuantumX/SomatXR signal conditioning devices and—if used—from other external devices by the server. You may then access the data through the server software or from one or more client PCs which are connected via Ethernet to the server PC. The data may be just displayed, stored, or used for evaluation and then visualized.

The software MD Client is used on the client PCs to access and further process the measured data. The relevant Operating Manual contains more information on how to access the data from the MD Server.

With MD Export it is possible to select and extract data from different tests or projects for further processing.

⚠️ Make sure the necessary licenses are available, otherwise the program will not run.

1.2.2 General operating procedure

1. Either connect the transducers being used for the measurement task to the signal conditioning devices or start with program MC Definition List in order to generate a connection scheme for the transducers. Then connect the transducers to the signal conditioning devices according to this list.
See MC Definition List on page 225 and MGCplus or QuantumX/SomatXR Operating Manual for details.
2. Start up the software MD Server.
3. Define general project settings such as the folder for all files relating to the project. All data for the tests made during this project are stored under this folder in separate subfolders, see Project settings on page 33.
4. Configure the required storage space, see Configure storage size on page 35.
5. Check the server options regarding settings for client PCs and options for monitoring, logging, zeroing and periodical export of data, see MD Server options on page 39.
6. Set up the signal conditioning devices according to the transducers connected with program MC Setup on page 137. If MC Definition List was used to create the wiring scheme, the necessary hardware setup is usually done during this process. MC Setup enables you also to include third party devices like a MOOG control system, optical devices like ARAMIS (GOM) or BraggMETERs and Avionics buses like ARINC-429, refer also to "System components" in the System documentation, Part I.

7. If not already done when using MC Setup or creating the MC Definition List, define Calculations (defining computations) on page 190, activate or deactivate channels and group channels and computations as required by the evaluation or visualization process, see Grouping channels and computations on page 210.
8. Define your Measurement settings (DAQ settings) on page 56.
9. You may use Trigger settings on page 58, e.g. for taking snapshots or performing other actions.
10. If necessary, create a visualization for the server PC, see Defining visualization (panels and objects) on page 74.
11. Initialize the system with ⊞ **Initialize** (**Test** ribbon tab).
12. Start the Data acquisition (Test ribbon tab) on page 96 on the server PC.

💡   It is possible to configure a custom tab in the ribbon with VBScript actions, see Custom commands with VBScripts (custom ribbon tab) on page 128.

As soon as the data acquisition is running, the measured data can be accessed by remote client PCs via software MD Client. In this program you may store data, do a real-time or post-process computation, visualize data, etc., see the corresponding Operating Manual or help file for details.

1.3 Operating requirements

☞   How to obtain the required license(s) is described in Part I, Chapter A, Section 3, *Licensing*.

In order to operate MD Server software, you need a PC with the following requirements:
- Intel Pentium 2 GHz or equivalent processor *(minimum)*
- Windows® 10, Windows® 11 or Windows® Server 2016 or 2019 as operating system
    - ☞   Please make sure you use the latest version of your operating system and all recommended updates from Microsoft are installed!
- Microsoft Edge (current version)
- 16 GB or more main memory (RAM)
- Graphics card with a resolution of 1024 x 768 pixel *(minimum)*
- 1 GB free space on your hard disk *(minimum)*
- Microsoft or 100% compatible mouse
- Two 100 Mbit Ethernet (or faster) interfaces. As an alternative for smaller systems one Ethernet interface and one FireWire interface is also possible. For

FireWire only interfaces according to IEEE 1394b are supported. The HBM drivers (including IP over FireWire adapter) have to be installed manually, see Installing the FireWire interface on page 21.

☞ We recommend a RAID mass storage in the server PC for speed (striping) and data safety (mirroring), e.g. a RAID 10.

The following fonts must be available on MD Server and all PCs used:
Arial, Courier, MS Sans Serif, Small Fonts, Tahoma, Times New Roman, Verdana and Wingdings. Usually these fonts are installed with the Windows® operating system.

⚠ All power and screen savers must be deactivated. Please check the system settings for power management concerning hard disk, graphics card, monitor and network cards.

⚠ If an automatic backup is made from your system while running a data acquisition make sure that all files which are accessed during the measurement are also accessible during the backup process. This includes the MD Server (catman) database file.
See also The databases used and files created by the MD Server system on page 134, Configure storage size on page 35.

Notes
🔴 We recommend using the decimal point in numbers, no colon. Please change your Windows system setting accordingly (**Additional date, time, & regional settings -> Change date, time, or number formats -> Additional settings -> Numbers**).
🔴 For safety purposes we recommend using an uninterruptible power supply unit (UPS). For one signal conditioning device MGCplus a minimum output of 620 VA is necessary. With 4 MGCplus and one Ethernet switch 1900 VA is required. With QuantumX/SomatXR a minimum output of 11 VA per module is required.
🔵 Standard user access rights are sufficient for working with catman Enterprise.
🔵 A maximum of 20 000 channels is possible with catman Enterprise.

1.4 Software installation

⚠ Administrator rights are required for installing the software.

💡 We recommend that you close all open programs. You can cancel the installation at any point.

Installing from ZIP file download

Open the **Properties** context menu and **Unblock** the file after downloading from the HBM website. Then unzip it and open the "Start.exe" file in the resulting folder to get the opening screen.

Installing from CD

Insert the CD into your CD-ROM drive. In standard configuration, Windows automatically opens the CD. If you have deactivated this self-starting Windows function, look for the "Start.exe" file in the main directory of the CD and double-click on its icon to get the opening screen.

Further installation procedure

1. Choose in the following screen the program you would like to install, here catman Enterprise.
   You may also install MC Setup and MC Definition List *separately* (without MD Server), so that channel setup definitions may be prepared on another PC than the MD Server PC. However, this option is selected later, so choose here **Install catman Enterprise** in all cases.
2. Click on **Next** in the following screen, read through the license agreements and select in the following screen the folder in which you want the program to be installed: either confirm the suggested target folder or click on **Browse** and choose which one you want. If necessary, Start.exe creates the folder you have specified and then copies all the files to it.
3. Now choose which part of catman Enterprise shall be installed, here MD Server. MC Setup and MD Export are in this case installed additionally.
4. If you would like to install MC Definition List together with MD Server, make the required selection here.
5. Click on **Next** and follow the instructions of the installation program.
6. Quit the installation program and close the opening screen after having installed all required components.

⚠️ If this is the first catman Enterprise installation, you must additionally install the MD License manager. We recommend you install this program on a separate PC. It is not necessary to install the MD License manager on the same PC as MD Server or MD Client. Then start the MD License manager, request a license file

from HBM, register all PCs in the MD License manager and assign the licenses needed for all MD License clients which includes the PC running the MD Server software.

Additional programs required for FireWire and time synchronization

- NTP or PTP time has to be used, if optical devices, external devices, mixed systems with MGCplus and QuantumX/SomatXR or several cabinets with QuantumX backplanes shall be used. You may use a dedicated NTP time server or use the software NTP time server included in catman Enterprise, see Installing the NTP time server/client on page 18. If MGCplus with CP52 or QuantumX/SomatXR devices are used you may use PTP synchronization as an alternative to NTP, see Configure and check synchronization on page 112.
- If you intend to use a FireWire interface on the server PC, install the FireWire drivers after the catman Enterprise installation, see Installing the FireWire interface on page 21.

Notes

- Together with MD Server catmanAP is installed with a PostProcess license (only catman Analysis mode may be used). There are other licenses available for catmanAP, however, only one catmanAP version may be installed on a PC.
- We recommend that you start catman Enterprise immediately after installing it and connect your device(s) in order to set a firewall exception for catman Enterprise (requires administrator rights). However, if you use the Windows firewall, an exception is set by the installation program.
- The MD License client is always installed if a program requiring a license is installed. The installation is made into the default programs folder into subfolder HBM.
- A TEDS editor and the Sensor database editor are installed automatically with MD Server.

1.4.1 Installing the NTP time server/client

When is an NTP time server necessary?

An NTP time server is required in the following cases:

- You use external devices together with HBM signal conditioning devices and these devices cannot use PTP.

- You use systems with MGCplus with CP42 *and* QuantumX/SomatXR (mixed system).
- You use FS22 BraggMETERs DI or Interrogators DI together with HBM signal conditioning devices.

In these cases, the devices are synchronized by using the NTP time stamps associated with the measured values.

However, it is always possible to install the NTP time server at a later point in time, either to have a more accurate time on the server PC or to forward this time to the connected signal conditioning devices. MGCplus and QuantumX/SomatXR can use NTP time stamps even if they are synchronized by hardware which is the standard synchronization method.

☞ We recommend to use PTP when having MGCplus with CP52 or QuantumX B or C type modules or SomatXR modules or a combination of these devices. This also applies when the synchronization cable is not long enough to connect all the back-planes and cabinets. In these cases, the accuracy of the time stamps is much better than with an NTP time server and should therefore be preferred. A PTPv2 Ethernet switch is however required in this case. The time channel data is always in NTP format even when using PTP.

Since the NTP time service available under Windows is not suitable, you must first install a suitable program, e.g. the freeware program NTP from David L. Mills provided with catman Enterprise. The program operates both as a client, for example to connect to a Stratum 1 time server, and as a server to be able to pass the time data to MGCplus, QuantumX/SomatXR and other devices or PCs. NTP (and PTP or IRIG-B) time channels are listed as *Device time stamps MGC* or *Device time stamps QX* in the **Time channels** list.

☞ We recommend you use a hardware based NTP time server, e.g. LANTIME M200 from Meinberg, which gives you excellent stability and better accuracy for the time server itself. If you prefer to use the (software) time server provided by catman Enterprise, set it up on the PC running MD Server as this ensures minimum time delay between the signal conditioning devices connected to the server. Unfortunately, with a software based time server a time difference between devices of more than one millisecond is possible. In case of doubt check with **Run check** (**Set synchronization settings** on the **Tools/Script** ribbon tab in MD Server).

⚠️ You must have Administrator rights for installing the software.

How to proceed (install software working as client and server)

1. Navigate to the installation folder of catman Enterprise (typically "C:\Program Files (x86)\HBM\catmanEnterprise").
2. Open subfolder "NTP".
3. Start the NTP installation program "ntp-xxx-setup.exe". The xxx usually contain the version number and information on the operating system the version is made for.
4. The program from [Meinberg](#) installs the freeware program Network Time Protocol from David L. Mills.
5. If the server PC has an Internet connection, you may want to specify public NTP servers (choose from the drop-down list), this is however not required. You may also use **None** and specify **Add local clock as last resort reference**. We recommend choosing stratum level **8** in this case.
6. If you did not specify a public NTP server, click **OK** in the alert window. (The setting **Add local clock as last resort reference** is required in this case).
7. We recommend using a special account for the NTP service for security reasons. You will be asked whether this shall be done and you can set up the account after clicking **Next**, provided you have the appropriate user rights. Leave all other settings in their default state.
8. After the installation has finished, click on **Finish**.
9. Check whether UDP port 123 (inbound rule on the server) is open for the time clients so they can contact the NTP time server.

The NTP configuration file ntp.conf is created in the NTP installation directory in subdirectory "etc". If you are not sure whether the settings are appropriate, you can also copy the file provided with catman Enterprise in the "NTP" folder to this location.

💡 The installation creates the folder "Meinberg" in the Windows start menu. Start the "Quick NTP Status" program (**Meinberg -> Network Time Protocol**) to check whether the service is running properly (check for delay and offset).

Additional settings required with MOOG systems (RTFE)

As some (older) MOOG systems cannot query the NTP server, a broadcast hast to be used instead. In these cases, insert an additional line before or after the server address line in the ntp.conf file:

```
broadcast <IP address> minpoll 4 maxpoll 4
```

If you are not sure whether this line is needed, contact the HBM Technical support, see Technical support on page 29.

Example:
```
broadcast 172.16.2.1 minpoll 4 maxpoll 4
```

Background information

The NTP time is a figure containing the number of seconds passed since midnight (00:00 h) on 1.1.1970 with a resolution better than one microsecond. In catman Enterprise it is always saved in the 8-byte format (double precision) and a dedicated time channel is used for each sample rate. An NTP time server can be, but does not necessarily need to be synchronized to a time server in the Internet.

You will find further information at https://www.meinbergglobal.com/ and at https://www.pool.ntp.org/.

1.4.2 Installing the FireWire interface

The FireWire drivers supplied by HBM for the QuantumX/SomatXR FireWire interface (IEEE 1394b) have to be installed before this interface may be used with QuantumX/SomatXR. You can however again at any time switch to the original driver of your interface. The installation is not done automatically, because this interface is only sufficient for smaller systems with up to 14 QuantumX or SomatXR modules.

⚠️ The installation requires administrator rights under Windows. If you are using a plug-in card for the FireWire interface, you should insert the card into the PC before the installation. Then the HBM driver is automatically activated for the card when installing catman Enterprise.

👉 For each QuantumX/SomatXR module module-specific information is installed in addition to the driver for the interface, because all FireWire devices must have a unique ID and this information is used in the driver. It is sufficient however to select the option **Install software automatically**, because Windows can find the driver automatically.

After the installation, start MC Setup as usual and in the **Scan** dialog use the HBM Device Manager or specify **24.0.0.0** as IP address for all QuantumX/SomatXR modules. If two FireWire adapters are available, use **25.0.0.0** as second address.

IP over FireWire adapter

The communication with the QuantumX/SomatXR modules takes place similar to Ethernet also with FireWire. In this respect a virtual interface for each physically existing FireWire interface is set up: IP over FireWire. A maximum of two adapters are created and

are visible in the network connections. As IP addresses 24.0.0.1 and 25.0.0.1 are used; consequently, the IP over FireWire interface adapters are named *IP over FireWire 24* and *IP over FireWire 25*. (The latter is only present if more than one physical interface is installed in the PC[1]). Only the Internet protocol TCP/IP (IPv4) is required for the IP over FireWire interface; the client for Microsoft Networks or the QoS Packet Scheduler is not required.

Procedure for the installation of the drivers

1. Install the FireWire interface card into your PC as described in the card's manual.
2. Go to the "DriverSetups" subfolder in the installation folder of catman Enterprise.
3. Double click on "HBM IEEE1394 Driver Setup.exe".
4. Read through the license agreements and follow the instructions of the setup program.
   The setup program copies the required files into the appropriate Windows subfolder and sets up the driver. The driver has been signed for Windows®.
   The installation program creates the folder "HBM IEEE1394 driver" in folder "HBM" (Windows Programs directory). The IP over FireWire adapters are configured automatically if the FireWire interface card is present. You don't need to run program "HBM IEEE1394 IP Adapter Wizard.exe".

Activating the HBM driver

☞ The HBM driver is a modified Thesycon driver.

If your PC has more than one FireWire interface, you must define for which interface the driver is to be used. Up to two interfaces can be used by MD Server.

1. Start the program "t1394bus_installwizard.exe" (Thesycon T1394bus Setup Wizard) in the folder "HBM IEEE1394 driver". The folder is created within the "HBM" folder in the Windows Programs directory (usually "C:\Program Files (x86)\HBM\HBM IEEE1394 driver").
2. Select the required IEEE 1394 host controller by clicking in the list. The "Thesycon T1394bus Setup Wizard" shows all 1394_OHCI_HostControllers located in the PC. If you have several adapters, you can select one of them. Otherwise click the first entry.

---

[1] A FireWire card can have, for example, three connectors, but despite this it is usually registered in the system as only one interface. In this case only one IP over FireWire adapter is set up (*HBM IEEE 1394 IP Adapter*).

3. Click on **Switch to T1394bus driver** to install the HBM driver.
4. Close the program "Thesycon T1394bus Setup Wizard". The driver is now installed and ready for use.

If you want to again activate the original driver of your FireWire interface, restart the "Thesycon T1394bus Setup Wizard", select the relevant IEEE 1394 host controller and click on **Switch to in-box T1394 driver**. This deactivates the HBM driver, but does not uninstall it. You can then switch between both drivers at any time.

Remove T1394bus driver from system removes the HBM driver completely from your PC.

Installing the module-specific driver

1. Connect your QuantumX/SomatXR modules.
   After a short time, a Windows dialog opens with the message that new hardware has been detected and it asks whether a driver is to be sought in the Internet.
2. Activate **No, not this time** and click on **Continue**.
3. Activate **Install software automatically**
   Windows then updates the required files. Wait until the message "New hardware found" appears in the windows taskbar. Then start MD Server or MC Setup.

1.4.3 Upgrading catman Enterprise

☞ We recommend that any older catman Enterprise version is first uninstalled before you install a new *version*.

⚠ Before uninstalling, transfer your licenses from an MD License client back to the MD License manager, because the MD License client is also uninstalled during the uninstallation (**Export license**). However, the licenses are retained even after uninstalling.

If you want to retain the settings of your old version as far as possible, you must however, before the uninstallation, copy various files to a (temporary) folder and move them back to the (new) catman Enterprise folders after the installation. If necessary, these actions must be carried out for each Windows user, because the settings are managed related to the user.

The databases created by MD Server or MD Client will be converted if necessary. A dialog appears where you can choose to update. Otherwise create new databases.

Procedure

1. Remove and export your licenses to the MD License manager, see Licensing in Part I (Operating Manual System Documentation).
2. In the folder for the user logged-in under Windows, find the HBM folder, e.g. "C:\Users\<Username>\HBM" or "C:\Users\<Username>\Documents\HBM". The folder path was changed with Version 5 of catman Enterprise.
3. Copy the content to a temporary folder of your choice.
4. Proceed in the same way with all the other PC users (administrator rights required).
5. Uninstall catman Enterprise.
6. Install the new version and execute a reboot of Windows.
7. Activate your licenses again.
8. Copy the content from the temporary folder into the HBM folder in the Documents folder for the respective user, e.g. "C:\Users\<Username>\Documents\HBM".
9. Start program MD Server.

1.4.4 Database server for MD Trend

☞ See also Snapshot database settings on page 25.

A database and a database server must be set up in order to use MD Trend. This database holds all values for all *conditions* (sometimes also called *load cases, load points* or *end points)* which are to be evaluated.

☞ We recommend setting up the database server on a different PC than the PC running MD Server. This ensures short reaction times when using MD Trend and does not increase the workload on the server PC doing the data acquisition.

You may calculate the hard disk space required for a database according to the following formula:

Number of measured values · Number of channels · 8 + approx. 1 MB = Storage size in bytes

Example

10 000 values in 800 channels result in a storage space of approx. 65 MB.

Install database server

From the MD Trend installation media, the Firebird database server (freeware) can also be installed separately: subfolder TOOLS\FIREBIRD contains the INSTALL.EXE file.

However, it is not mandatory to use this database server. You may use any other database server, e.g. MS Access or Oracle as well, provided standard SQL queries are supported, see Using other database type servers on page 25.

The installation is provided on the MD Trend installation media and automatically called up during installation of MD Trend. Please read through the license agreement, specify where to install the software and choose **Full installation of server and development tools**. Then choose the **Start** menu folder name and accept the default settings in the next screen:

- **Use the guardian to control the server**
- **Run Firebird server as a service**
- **Start Firebird automatically every time you boot up**

Finish the installation with starting the Firebird service.

Set up database

Templates for Firebird and MS Access are provided with software MD Server, see installation folder:

1. Firebird: TEMPLATEDB.FDB (default for MD Trend)
2. Access: TEMPLATEMDSDB.MDB

Copy one of these to the database server and assign a name, e.g. SNAPSHOTS.FDB.

Using other database type servers

Other database servers may be used instead of Firebird, provided standard SQL queries are supported. In this case you must provide the proper connection string in dialog Snapshot database settings on page 25. In addition, the database structure must fulfill certain requirements which are stated in a separate paper. You will find the specification in file TEMPLATEDB.PDF in the DOC folder on the MD Trend installation media.

1.4.5 Snapshot database settings

☞ See also Database server for MD Trend on page 24.

⚠ All settings must be made *before* initializing, see Data acquisition (Test ribbon tab) on page 96.

As mandatory settings you have to:
- specify where the data has to be stored (Database server on page 26),
- use a Condition channel on page 26 on MD Server.

An option is to use a Percentage channel on page 27.

> 💡 If you want to enable triggering the server via MD Trend, use the context menu in the **Trigger settings** window of MD Server, see MD Client/MD Trend trigger on page 68.

1.4.5.1 Database server

☞ See also Database server for MD Trend on page 24.

Open the settings dialog with **Server options** (**Setup** ribbon tab in MD Server). Enable the storage of all snapshots to a database and specify via **Snapshot database settings**:

1. IP address of PC hosting the database.
2. Path and file name of the database to be used.
   A leading backslash is required and the file must already exist (see Set up database on page 25).
3. The connect string valid for your database.

As default, a Firebird database with OLEDB is used. Use in this case the connection string provided. However, depending on your network it might be necessary to state the IP address of the database server after **location**.

Example

provider=IbOleDb;location=192.168.200.124;Data Source=[DBName];User ID=SYSDBA;Password=masterkey

If you want to use a different database server, please check with the documentation of your database which connection string has to be sent for getting access.

How to find the Ethernet address of a PC (Windows® 10)

Open, for example using the 🖥 symbol in the notification area of the taskbar, the **Open Network & Internet settings**. Click on **View your network properties**).

1.4.5.2 Condition channel

A condition channel is mandatory, if you want to use MD Trend. It contains a number representing the load condition applied to the test object. You have two possibilities:

1. Use a channel which is received from an external device or from a CAN bus module in the MGCplus signal conditioning devices and which gets the numbers representing the current condition (load case) from your control system. The channel

itself is treated as a normal channel containing numbers in catman Enterprise. Then indicate this channel in MD Trend, see topic Specify condition channel. As external device either a MOOG control system (MOOG CS) may be used or a device which is connected via a DLL Driver, see MC Setup, Additional external devices on page 149.

2. Use a **Manual input channel**, see Manual input channel on page 191.

💡 You may also use a channel as percentage channel if conditions or load cases are applied partially, see Percentage channel on page 27.

1.4.5.3 Percentage channel

If conditions or load cases are also applied partially, use a percentage channel. Your control system may then forward the according percentage values to this channel. The channel itself is treated in catman Enterprise as a normal channel containing numbers. This allows you to use also e.g. a DC voltage input channel or a transducer to measure a value representing the percentage and specify later in MD Trend which value is equivalent to 100%.

1.4.6 Installing software for Avionics buses

☞ See also Using AIM modules on page 152, Hybrid systems options on page 45, Configure and check synchronization on page 112, Installing the NTP time server-/client on page 18.

catman Enterprise can include data from the avionics buses ARINC-429 and MIL-STD-1553B. You may use e.g. AIM modules ANET429-x or ANET1553-x and combine the signals from these modules with QuantumX/SomatXR or MGCplus data. For synchronization with the HBM devices at least an IRIG B time server is required for the avionics devices. The HBM devices should preferably use NTP or PTP (PTP with QuantumX/SomatXR and MGCplus with CP52 only), but can also work with IRIG B.

Because catman Enterprise does not connect directly to the avionics buses. A separate software, PBA.pro, has to be used. The MD Server connects to this program and reads all data from this program. It is not important whether PBA.pro is running on a separate PC or on the same PC as MD Server.

Prerequisites

- AIM ANET module for ARINC-429 or MIL-STD-1553B avionics bus
- AIM software PBA.pro version 2.76 or higher
- AIM board software package (BSP) version 11.0.1 or higher

Installation of PBA.pro software

1. Extract the downloaded software and start the installation program pbapro_Vxxxx_win32.exe. xxxx stands for the software version number.
2. Click on **Full Version** in the **Install** dialog for installing the full version of the program.
3. Accept the terms in the license agreement and select the install folder (you may use the default folder).
4. Select the relevant component resources in the next dialog: **MIL-STD-1553** or **ARINC 429**, or both, depending on your hardware.
5. Enter your license key.
6. At the end of the PBA.pro installation you are asked to install Python if not yet installed. In this case click on **Yes**.
7. Activate **Install launcher for all users** and click on **Install Now**.
8. You may pin the program to the Start menu. There is no need to launch PBA.pro after the installation. You may deselect the option to launch or to open the release notes and click on **Finish** to exit the installation program.

Configuring PBA.pro for AIM modules

1. Hold the Shift key (⇧) and double-click on the PBA.pro program icon.
2. Activate **Startup Configuration** and click **OK**.
3. Click on the three white points on the right-hand side.
4. Click on the white plus sign at the bottom. You might have to enlarge the dialog to see the sign.
5. Assign a **Title** for your system, e.g. **ARINC429**. Add an identifier if you have several modules, e.g. the last number of the IP address.
6. Under **Options** specify **-svrppa429_94x:<IP address of your AIM module>** for an ARINC 429 module and **-svrppmil:<IP address of your AIM module>** for a MIL-STD-1553 module. Replace the term in pointed brackets with the appropriate IP address, e.g. write **-svrppa429_94x:172.168.169.25**.
7. Click **OK** to close the dialog.
8. Click on ▶ and confirm to save the changes.

9. In PBA.pro click on the **Scripts** tab on the left side.
10. Right-click in the **Scripts** area and select **Add Script(s)** from the context menu.
11. Choose file `param_dist_429.py` or `param_dist_1553.py`, or both, depending on your modules.
12. Click the icon under **Exec** in the Script area to start your script. It might be necessary to allow access to your modules through your firewall (**Domain** and **Private networks** is sufficient).
13. You may also activate **Autostart**, otherwise you have to start the script(s) manually each time you want to work with these modules.

🔆 You may want to create a shortcut for starting PBA.pro with your setup: create a link to PBA.pro.exe, call the **Properties** dialog and add `-startopt` at the end of the **Target** field. Otherwise you have to follow steps 1 and 2 of the configuration for starting with your setup. With this start parameter you only need to click on

▶ in the start window.

How to change the IP address of an AIM module

Use your browser and the ANET WEB Configuration:

1. You may have to change temporarily the IP address of your PC to be in the same subnet as your AIM module.
2. Connect to your module (type in the current IP address).
3. Switch to **Network Settings**.
4. Switch to **Static** (deactivates **DHCP**) and set the required **IP** address and **Subnet mask**.
5. Click on **Apply settings**.

1.5 Technical support

Should your installation not run perfectly or if you have a problem with catman Enterprise, please contact our Technical support in Darmstadt or your local representative.

Email support

✉ support@hbkworld.com

Telephone support

Telephone support is available on workdays from 9:00 to 17:00 hours Central European Time.

+49 6151 803-0

☞   Extended support is available through a service contract.

HBM in the Internet

🌐 https://www.hbm.com

HBM Support and Sales International

🌐 https://www.hbm.com/en/contact/worldwide-contacts/

2 MD SERVER

The MD Server software is the main software of catman Enterprise. From this program the channel setup program (MC Setup) is started and all settings for the measurement process are made. Topic MC Setup on page 137 explains the process of setting up the hardware and how to define sensor scaling, computations, etc.

💡 A different approach is described in topic MC Definition List on page 225. This program enables you to create a list of transducers (sensors) to be measured. This list may then be used to create a wiring scheme and the setup data.

Because these setup procedures are a critical part of the test project, a user access administration is included into the MD Server software. *One of the first things to do* is to provide the proper user settings and restrict access to the software for every user to the part he or she should manage: Administration and access rights on page 32.

☞ When starting MD Server for the first time, a system scan is required to determine the hardware available and create a database containing that information, see General information on MC Setup on page 137 in MC Setup.
It may also be necessary to set a firewall exception for catman Enterprise (requires administrator rights) if you are not using the Windows firewall.
You may then proceed with the topics described here.

Ports required by MGCplus and QuantumX/SomatXR

Depending on the device type connected, release the following ports:

- QuantumX/SomatXR: TCP port 80, 5001, 7411 and all ports above 50 000, UDP for ports 1200, 1201, UPD-Multicast for ports 1300, 1301, 31 416 and 31 417.
- MGCplus with CP52: TCP ports 7 and 80, UDP-Multicast for ports 31 416 and 31 417.
- MGCplus with CP42: TCP port 7 (no UDP used).
- If an NTP time server is used, allow UDP 123 (inbound rule on MD Server).

The general project settings and their importance for the measurement is described in topic Project settings on page 33. The following topics are grouped in the same way you should deal with the corresponding theme starting with the user administration up to diagnosis and troubleshooting.

☞ All settings are stored immediately. Depending on the type of information, settings are stored to different databases, there is no need to save anything.
However, if you want to re-use settings, you may *export* (and save) settings to file.

⚠ Make sure the necessary licenses are available, otherwise the program will not run.

2.1 Administration and access rights

☞ When starting the software for the first time, use the user name **admin** with password **hbm** (only password input is case sensitive). We recommend that you change the password once you start to work with this program.

Use **MD Server -> User administration** to call up the user administration dialog.

2.1.1 What to do first

1. Change the password for the administrator **admin**.
   See also Password restrictions on page 32.
2. Create a **New user**.
3. Specify the user name, that is the name under which the user logs on.
4. You may specify the full name and a description for that user, e.g. what this user is supposed to do, etc.
5. Type in an initial password, e.g. the log-on name. The user should change this password as soon as possible.
6. Define to which parts of the software access may be granted: **Access rights**.
7. Create as many new users as necessary.

2.1.2 Password restrictions

The password input is case sensitive. There is no minimum length for the password and there are no checks made whether the password chosen is appropriate or not. Please observe the standard rules for choosing passwords, e.g. no names of relatives, no birthday dates, etc.

☞ It is recommended that you use a combination of at least eleven letters, digits and symbols which form no complete or meaningful word, e.g. **pa?p12e$7KD**.

2.1.3 Access rights

User can modify project settings

Permission to change the project settings, menu **MD Server -> Project settings**.

User can modify MC Setup

Permission to start the MC Setup program from MD Server and change the hardware configuration.

☞ It is still possible to do this if you start the program direct via the Windows Explorer.

User can change measurement (DAQ) settings

Permission to change anything on the **DAQ settings** ribbon tab.

User can change trigger settings

Permission to change any settings on the **Trigger/Limits** ribbon tab.

User can modify VBScripts

Permission to edit a VBScript via MD Server (**VBScript editor** on the **Tools/Script** ribbon tab).

☞ It is still possible to edit the file via Windows, e.g. with Notepad.

User can modify panels

Permission to configure visualization objects, **Panel** ribbon tab.

User can store panels

Permission to save a panel layout, **Panel files -> Save panel/Save panel as** on the **Panel** ribbon tab.

User has administration rights

Permission to *change settings* in the user administration. The user administration may be *opened* by each user so that he can change his own password.

2.2 Project settings

ⓘ See also .

Click on 🔧 **Project settings** on the **Setup** ribbon tab. The dialog always displays the settings used last.

How to proceed

1. Specify the path for all project files. *It is not possible to type in a path*, you must use the button ⎡ ... ⎤. All data concerning a project will be stored into this folder.

2. If the project folder already exists, all available projects will be displayed in field **Project name**. One of these may be selected and continued. If a new project is to be started, type in a name for the new project.

3. A project may need several measurements or test runs. For each test run, i.e. a start/stop sequence, a new folder is used. Therefore a new name in line **Test base name** *must* be used for each sequence, it is not possible to overwrite data. If a folder shall be used again, you must delete this via standard operating system functions, e.g. from the Windows Explorer. Each test folder contains the data from this test run and the database with the hardware setup information. If necessary, an additional subfolder is created within this folder for trigger measurement data. When using the **Pause** button during a test run, *no new* test run is initiated, this is done only when clicking on **Start** after using the **Stop** button.

    ☞ As default **Auto increment test name when starting a test** is activated in the **Server options** dialog, tab **General options** (**Server options**), see General options on page 39. With this setting deactivated, the dialog is called and you must either specify a new **Test base name** or a new **Project name**, if a new test run is initiated.

4. The channel setting used (MC Setup database file) will be copied to the folder specified above *after* the measurement (after stop). If you make a **New scan** in MD Server or MC Setup this field is filled automatically. Otherwise specify the initial setup file to be used with this project in line **Setup file** with button ⎡ ... ⎤. If a MC Definition List is connected to this setup file, it will be displayed as well.

5. If VBScript is used, a file may be specified with button ⎡ ... ⎤. See Using VBScript on page 124 for details.

    ☞ If a trigger shall run a VBScript, this script has to be in memory (loaded). Files specified here are loaded when opening the project.

6. Panel files containing visualization objects may be chosen with button ⎡ ... ⎤ in line **Panel files**. If you create new panels, these files are appended automatically. See Defining visualization (panels and objects) on page 74 for details.

7. You may add an explanatory comment to the project in the input box **Comment**. This comment is included into every file created later on, also in files exported to Excel.

The name in field **Test performed by** is derived from the logged-on user. Therefore it is *not possible to change* this entry manually. You may however log on under a different user name. As soon as you choose **MD Server -> Log off <user name>** a dialog box prompts you for a new user name.

☞ Logging on under a different user name is only possible if the data acquisition is *not* running.

Each time the MD Server is started, the last settings are recalled. So, if you want to perform a new measurement or a new test, you must change the settings in this dialog. All information about a project together with the number of tests is saved to the project database in the project folder.

2.3 Configure storage size

ⓘ See also (prevent overflow).

For every channel to be measured and saved at least a temporary buffer must exist (before saving to file). It is therefore necessary to reserve space for all channels which are to be stored, for each trigger condition to be monitored and for snapshot data. To create a database file for data storage use **Configure storage** on the **Setup** ribbon tab.

⚠ Every data acquisition is ended if the database is filled with data, i.e. no more temporary storage space is available.

☞ MD Server creates a different database for every user logged on to Windows by default. Therefore the settings are not stored according to the MD Server user names but according to their Windows log-in names. Users having the same Windows log-in name must use the same database settings. When having several users, we recommend to share the same database file by creating the file in a folder which is accessible for all users. However, each user has to do the corresponding setup in MD Server to use this common database file.

In the section to the left of the database configuration dialog you can choose the drive and a folder in which the database is to be created. The drives A: or B: are not allowed.

☞ Do not use a network drive, because then access to the data will be very slow.

The file name does not need to be stated. The database is always created with the file name DB_000.$_$ and not deleted when terminating MD Server.

> 💡 We recommend using a defragmentation tool for the hard disk in question before creating database files bigger than 100 MB. It is also recommended *not* to delete the database file on program end, i.e. use a *permanent* database file for all tests. As this storage space is created as a file on the hard disk (6 GB in the example 2 in topic How to calculate the necessary storage size on page 38), we recommend using a fast hard disk or disk array as drive depending on the hardware available in the server PC.

The database layout is defined in the right-hand section of the dialog: Number of channels, maximum length (depth) and format for each channel. Each channel can be edited individually: Mark the channel(s), enter the desired values and click on **Selected channels** or use **All channels**. In the left section you can see, in the numerical display and in the pie chart, the disk space available on the hard disk as well as the free space and the space needed by the currently set configuration.

2.3.1 Number of channels, channel layout

With **Maximum number of channels** you define the required number of input, output, time, trigger and computation channels. A maximum setting of 20 000 channels is possible. Refer to How to calculate the necessary storage size on page 38 for information on how to calculate the appropriate number.

> 💡 Close the entry with ⏎ in order to re-calculate the necessary storage space immediately.

Auto-size: We recommend not to use this setting for MD Server as it tends to be much slower than with a fixed database layout. Additionally, only up to 512 channels may be used in this case. With this setting a dynamic database is created, which increases in size as long as there is space available on the hard disk.

> ☞ As this type of database is managed by the operating system, data is not secure if there is a system crash.

The **Maximum length** option serves to define the maximum number of measured values for *each* channel. This data depth is only restricted by the space which is available on the hard disk for the database file. If you do not want to mark and set up the channels individually, click on **All channels**.

Refer to How to calculate the necessary storage size on page 38 for information on how to calculate the appropriate number.

☞ We recommend using the same length for all channels. Otherwise make sure that a smaller size does not fill up earlier than the other channels. Every data acquisition is ended, if a database channel has no more space for storing data.

Individual channel formats

The data storage format in the database can be selected. In format **Double** (default) 8 bytes are used for a measured value (double precision), in format **Float** 4 bytes (single precision).

Auto-recover database on program start

When starting, MD Server checks whether a database file which has not been properly closed is present. If yes, MD Server asks whether this database is to be restored except if **Auto-recover database upon program start** is active.

Operate in circular buffer mode

When you operate in this mode, a fixed size must be used (maximum length). Once the database has been filled with values, the new measurements overwrite the oldest ones. With a size of, for example, 100 000 values per channel there are then always only the last 100 000 values (per channel) available for saving to a file or for export.

💡 The setting is, for example, helpful if you are monitoring at a higher sample rate for an event and would like to also save the prehistory over a longer period when the event occurs. With the **Server options** dialog (**Setup** ribbon tab) you may define to export data periodically (before the database is filled up). See Export options on page 42.

Maximum cache size

☞ When operating in circular buffer mode, the cache size is not relevant.

The presetting of 32 kB (per channel) for the size of the cache is sufficient for all normal application cases. You can use higher settings if you are working with high sample rates. With a size of 32 kB 4096 values are buffered in memory (8 bytes per measurement). This is the maximum number of values which can be lost during a system crash. The cache prevents MD Server from writing to disk too often which would reduce the lifetime of a (conventional) hard disk where mechanical head movement is involved when writing data.

⚠️ A corresponding amount of memory (RAM) is required, otherwise the transmission of the measured values will be slowed down.

2.3.2 How to calculate the necessary storage size

The necessary storage size depends on the Measurement mode:

1. Fast trigger mode
   (Here two sample rate groups are available: slow (= standard) and fast.)
   MGCplus: Take the number of active channels and add 11 channels.
   QuantumX/SomatXR: Take the number of active channels and add 4 channels.
   Then take this number and add the number of all computations, trigger conditions and storage channels. For this number of channels set the channel length to the maximum storage time required multiplied with the highest sample rate.

2. Regular DAQ mode
   (Here three sample rate groups are available.)
   MGCplus: Take the number of active channels and add 11 channels.
   QuantumX/SomatXR: Take the number of active channels and add 3 channels per QuantumX/SomatXR module plus 2 channels for all of them.
   Then take this number and add the number of all computations and trigger conditions. For this number of channels set the channel length to the maximum storage time required multiplied with the highest sample rate.

☞ In contrast to a trigger condition the limit checking does not require a database channel for evaluation.

Example 1 (for Fast trigger mode)

A MGCplus system with 1305 active channels is used. 435 computations and 14 trigger conditions are defined, max. 245 storage channels will be used. The total number of values to be stored is 200 000. This leads to:

🔵 2010 channels (1305 + 11 + 435+ 14 + 245) with a channel length of 200 000.

This results in a database file of approx. 3.2 GB.

Example 2 (for Regular DAQ mode)

A QuantumX/SomatXR system with 20 modules having 16 channels each and 2 modules having 8 channels each is used. This leads to 336 active channels. 325 computations and 14 trigger conditions are defined. The total number of values to be stored is 1 000 000. This leads to:

- 743 channels (336 + 22∗3 + 2 + 325 + 14) with a channel length of 1 000 000.

This results in a database file of approx. 6 GB.

2.4 MD Server options

Server options on the **Setup** ribbon tab opens a dialog providing several tabs (some tabs are only visible, if the corresponding hardware has been recognized).

2.4.1 General options

Use **Server options** on the **Setup** ribbon tab to call up this dialog.

Auto increment test name when starting a test

The option provides you with an easy way to change the name for a test before starting the measurement again. As a new name is mandatory in this case, this option saves you having to assign a new name manually.

Store all snapshots for MD Trend

This is required if MD Trend shall be used. Then specify the database settings, refer to Snapshot database settings on page 25.

⚠️ If you plan to use MD Trend (optional software), you must additionally provide a database server, refer to Database server for MD Trend on page 24.

Enable UPS monitoring

This option must be enabled, if UPS related actions or services shall be used. You should deactivate this option only, if no UPS is used in your system. This saves the time necessary for checking the UPS status. See also UPS manager on page 132.

💡 Together with catman Enterprise the UPS Manager program is started if the option is activated. Use the Windows task bar or **Diagnosis -> UPS manager** to

call up the program. There remaining run time, capacity, load, temperature and status are displayed.

Enable communication log during test

The option is helpful to determine communication errors which occur very rarely. During a test the command communication (no data) is written to a ring buffer together with time stamps. This communication log enables the HBM support to track errors, e.g. if devices crash or hang up. In case you experience such an error, open the communication log **Show communication log** on the **Diagnosis** ribbon tab), save it under a different name and send it for evaluation to the HBM Technical support, see Technical support on page 29. The default setting for this option is enabled.
See also Communication logging on page 131.

Enable MGCplus SYNC monitoring

Allows you to see the status of the synchronization between the MGCplus devices in the **DAQ status** window, **Sync monitor** tab. This option is required if a SYNC lost trigger shall be used, refer to MGCplus SYNC lost trigger on page 67.

Enable NTP/PTP sync monitoring

Allows you to check for time synchronization errors.
This option is required if a NTP/PTP service failed trigger shall be used, refer to NTP/PTP sync failed trigger on page 66.

Set the corresponding time values for caution and warning indication in the **DAQ status** window, refer to DAQ status window on page 100. If everything is OK, the LED displayed in the window is green, if the **Caution** value is exceeded it is yellow and above **Warning** it is red.

Enable MGCplus power supply monitoring

Allows you to determine whether the different supply voltages and currents on a fully equipped MGCplus are as required. If the current requested is too high, e.g. due to many strain gages with 120 Ω resistance and high excitation voltages, the supply voltage and hence the excitation voltage drops and all measured values are too low.
This option is required if a MGCplus voltage/current/temperature triggers on page 66 shall be used.

⚠️ The option requires MGCplus devices having an ER003D (rack module), TG003D (desktop housing) or later housing as only these have power supplies supporting this feature.

Signal overflow as

Allows you to choose a specific value which is displayed and saved, if an overflow occurs. You can then recognize, even later, from the measured values that an overflow occurred.

Transfer MGCplus and QX licenses to devices automatically

Enables you to automatically transfer all required licenses for MGCplus or QuantumX (modules of type B or C only, e.g. MX840B, MX1601B or MX471C) and SomatXR modules into the modules resp. devices itself when starting a test. If all devices already have a license, nothing is transferred however. The MD Server PC must have enough licenses which can be distributed otherwise the initialization will fail.
See also .

Read timeout

Specifies the time the server waits for a response from connected HBM devices until an error is issued. The default setting of 6 seconds should be enough for most configurations.

⚠️ If values cannot be read in from a device within the time specified as **Read timeout**, an error message is issued and the device is excluded from the current test (the data acquisition from this device stops).
See also .

Disable MGCplus synchronization

Allows for measuring without A/D converter or measurement start synchronization. In this case *no trigger leader* MGCplus is used, all devices start their measurement one after the other, i.e. *with a short time delay in the milliseconds range (approx. 0.5 ms)*. *The data from different devices is no longer acquired at the same time*. This option may be useful, if no synchronization cable can be used due to long distances between the individual devices and slow sample rates are used.

Please note that the time difference may increase over time due to imperfections of the MGCplus internal time generation.

SY42 synchronization

The standard synchronization with MGCplus works only up to 100 m. For longer distances a special plug-in card, the SY42, is available. Activate this option, if such a card is installed in all MGCplus devices taking part in the measurement and the devices are properly connected. Otherwise NTP synchronization has to be used, see Installing the NTP time server/client on page 18.

Upload setup when init

If MD Server is started, the setup database is only *loaded* by the server, no settings are *transferred* to the devices. This option ensures that the setup data from MC Setup is uploaded to the measuring devices when initializing the MD Server.

Large title font size

This option enlarges the title font size of Digital indicator, Analog indicator and Bar indicator. You have to activate the option before creating these objects. The **Font** context menu of these objects changes only the size of the numbers displayed.

With graphs you can define the size of titles and labels individually with the **Configure** context menu on the **General** tab.

2.4.2 Export options

ⓘ   See also Channels/Export on page 106.

Use **Server options** on the **Setup** ribbon tab to call up this dialog.

Enable disk space check … ensures at the start of a data acquisition that enough disk space is available for exporting the data afterwards. The percentage setting refers to the database size, i.e. 100% means enough space to export the whole database. See also Configure storage size on page 35.

You may use **One file for each storage channel** when exporting or export **All storage channels into one file** (default). When having numerous channels and many values per channel it might be better to use one file per channel as this decreases the file size.

Clear all database channels when stopping MD Server clears the database after exporting the measured data so that the next measurement starts with an empty database. This saves the time for clearing the database at the start of a measurement.

Periodical export

These options let you *save* the storage or snapshot files, or both, *periodically*. This prevents the internal database from overflowing which would stop the measurement.

⚠️ When using a periodical export for all *storage* channels, keep in mind that during the store operation *no new data is read in*. Therefore this time must be short enough, so that the buffers inside the signal conditioning devices do not overflow. We recommend using a small database and export data more often in this case.

Example

1000 active channels with 1000 values each result in a data file of approx. 7.8 MB. 1000 active channels with 10 000 values each result in a data file of approx. 77 MB. To store a file this size may take up to 10 seconds.

The option **Timer interval for flashing database cache** is important, if a data acquisition with a *low sample rate* is performed. Data acquired from a device is not written to the database immediately. Instead it is saved in RAM (the cache) first. The procedure is used to prevent frequent disk access. With this option you may specify an interval at which data is written even if the RAM buffer (cache) is not yet filled. The cache size depends on the number of channels used (usually 4096 values). With a slow data acquisition of e.g. 5 values per second, all data would be cached for about 15 minutes until saving them. If during this time e.g. a mains power failure occurs, all data is lost (without UPS). You may therefore specify a shorter interval (default 5 minutes) for saving this cache if it was not already saved.

File naming

You may use placeholders for the snapshot file names. As default the test base name (see) and the full date when saving is appended to "Snapshot_". If necessary, an additional counter value is included. This ensures unique file names. Both parameters are optional and you may also add other texts.

2.4.3 VBScript editor

Use **Server options** on the **Setup** ribbon tab to call up this dialog.

You can set here font size and family for the editor and the background color of the **Code** window.

2.4.4 Zero balancing

Use **Server options** on the **Setup** ribbon tab to call up this dialog. On this tab, define how the zero balancing should be carried out:

- ● **Standard zero balancing**
  This default method uses either the zero balance or the taring function in the device or—for Calculation channels—a single measurement is carried out.
- ● **High precision zero balancing**
  MD Server determines the zero balance by measuring and averaging several measured values. The measuring is carried out simultaneously on all channels using a sample rate of 10 Hz.
  - ☞ A maximum of 1024 measured values are considered. If the number of measured values resulting from the sample rate used and the time entered is larger, the surplus of measured values will not be considered.
- ● **Store zero history**
  The measurement and especially the storage of many channels can be time consuming and leads to huge files with offset values when performing multiple zeroing operations. The default setting is to store all values from zeroing operations so you have the history of all operations on file. With this option you may deactivate the saving of further zeroing operations.
- ● **Show extended zero balance dialog**
  When pressing the zero button (🟢) a dialog is displayed and the user can select the required method. This method is not available during a running data acquisition.

The second method is better especially for signals with disturbances (noise), as not a single measured value is used. Instead the measured values are averaged over the specified period of time. This means however that more time is required for the zero balance: the zero balance in each device requires approx. 10 ms, for the averaging use measuring times of 1 to 5 seconds.

☞ Do not use precision zero balancing for temperature channels and for digital channels or counter values if these values might change during the balance: lock zero balancing by setting the **Zero enable** column in MC Setup to **off**. Otherwise this can lead to invalid values, e.g. 0.4 as zero value for a digital input that can only be 0 or 1 but has changed its status during the measurement.

2.4.5 Hybrid systems options

Use **Server options** on the **Setup** ribbon tab to call up this dialog.

General settings, options for all external devices (including MOOG control systems)

Size of FIFO (buffer) determines the read buffer size for values from external devices (per channel). With the values contained in this buffer the time matching is done (see below Matching time stamps …). As delays may occur on the network you should set a sufficient number of samples for the buffer, depending on the sample rates used. The default setting is 512 samples which results in 5 seconds buffering time with a sample rate of 100 Hz. Use higher values for higher sample rates to get a buffering time of a few seconds.

The **Output delay** is used to get data from different device types at approximately the same time. catman tries to match the data delivered by e.g. external devices and contained in the FIFO buffer with the values sampled by e.g. an MGCplus according to their time stamps (see below Matching time stamps …). However, the buffer used for this match contains only a limited number of data. If no full match can be found in this buffer, usually the nearest match is used which might result in a time difference. Specify a delay of e.g. 50 ms. Then check on the **Device status** tab whether the read pointers (**SRx data** or **Slow data**) have a high or low number (see also). If the number is close to the maximum buffer size (**Size of FIFO**), decrease the output delay. Increase the delay if the read pointers have a number below 10 with a buffer size of 512 (depending on the systems, values up to 300 ms might be necessary). The number should be around 100 to 150 with a buffer size of 512.

Matching time stamps from hybrid systems

Case 1: Device type 1, e.g. an external device, delivers too late

V_{y1}	V_{y2}	V_{y3}	...	V_{yn-1}	V_{yn}

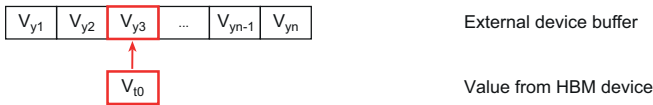External device buffer

V_{t0}

Value from HBM device
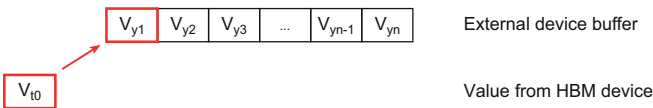
The first value available in the buffer which has been measured at a later point in time than the value measured at t0 by the HBM device and is used as nearest time match. The buffer read pointer (**SRx data** or **Slow data**) has low values, increase the **Output delay**.

Case 2: Output delay of device type 2, e.g. an HBM device, too high
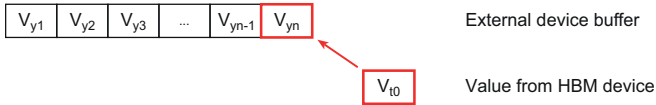
The last value available in the buffer which has been measured at an earlier point in time than the value measured at t0 by the HBM device and is used as nearest time match. The buffer read pointer (**SRx data** or **Slow data**) has high values (**Size of FIFO**), decrease the **Output delay**.

Case 3: Output delay adjusted properly

The value measured at t0 by the HBM device and the value V_{y3} in the buffer have the same time stamp.

PTP/NTP sync control

These settings are only required when:

1. Mixed systems with MGCplus and QuantumX/SomatXR are used.
2. Systems with external devices are used, for which a DLL driver exists.
   The documentation on DLL DAQ Drivers ("Drivers/DLLDriver/DAQ drivers.doc" in the catman Enterprise installation folder) shows how to write such a driver in a high-level programming language.

These settings are not necessary for a MOOG control system, here all settings are made automatically. In all other cases, you must additionally specify before the initialization, which synchronization variant shall be used, see .

> 💡 When using PTP for the HBM devices you should either set the leader HBM device to use the same NTP time server as the external devices or synchronize the PTP time and the NTP time to the same source. Otherwise the PTP time might not be the same as the NTP time.

> 🔵 MGCplus and QuantumX/SomatXR: all QuantumX/SomatXR time stamps should be corrected to the **Nearest match**, option **Manual time stamp correction for MX devices in hybrid systems with MGCplus** (**Hybrid systems** tab in

the **Server options** dialog) is considered. Define additionally an **Output delay** when using high sample rates and PTP synchronization. You *must* select **No synchronization** for measurements where a sample rate above 19.2 kHz (max. on MGCplus) is used for QuantumX/SomatXR.
See also [Output delay](#) above.
- MGCplus and Moog RTFE: all RTFE time stamps should be corrected to the **Nearest match**, option **Manual time stamp correction for external devices** (**Hybrid systems** tab in the **Server options** dialog) is considered.
- MGCplus and other devices: with BraggMETERs and Interrogators the time stamps of these devices should be corrected to the **Nearest match**, option **Manual time stamp correction for external devices** (**Hybrid systems** tab in the **Server options** dialog) is considered.
- QuantumX/SomatXR and Moog RTFE: all RTFE time stamps should be corrected to the **Nearest match**, option **Manual time stamp correction for external devices** (**Hybrid systems** tab in the **Server options** dialog) is considered.
- QuantumX/SomatXR and other devices: with BraggMETERs and Interrogators the time stamps of these devices should be corrected to the **Nearest match**, option **Manual time stamp correction for external devices** (**Hybrid systems** tab in the **Server options** dialog) is considered.

IRIG B sync mode

The synchronization mode determines how the data is forwarded to MD Server:

IRIG B time nearest match: The data in the time channel from the devices is compared against the data from the leader device (MGCplus trigger leader or QuantumX/SomatXR sync-leader). For each leader time value the time value closest to this time from the other device(s) is searched and the data acquired at this time is forwarded together with the data acquired by the leader device at the time value considered.

IRIG B time with interpolation: Instead of searching the nearest match as with the above option, a linear interpolation is made with the values acquired at two points in time (one before and one after the time considered). This interpolated value is then forwarded as data from the device. This method is more accurate but it takes more time to compute.

2.4.6 Client - Server settings

Use **Server options** on the **Setup** ribbon tab to call up this dialog.

TCP/IP Client - Server settings

Port for Client - Server connection: The port used for the client—server communication may be changed here. If a firewall is used, access through this port has to be allowed.

⚠️ Do not forget to specify the port on *each* MD Client if you change the default setting of 2004.

Send timeout for Client—Server connection is the timeout used for the communication between clients and server. The default setting of 30 000 milliseconds is sufficient for most networks. It may be necessary to increase this number if you are using a slow connection e.g. via DSL.

Circular buffer for Client—Server communication

We recommend that you set up circular buffers for all channels, if clients shall be connected. The values of the individual channels are then (temporarily) saved in these ring buffers. Without ring buffers, each value must be transferred *immediately*, otherwise data will be lost as the data is overwritten by the next value.

⚠️ Without circular buffering only **Single point DAQ** may be used on client PCs, i.e. a single value is fetched only with each read, and *no continuous data acquisition* is possible on the client PCs.

The size of the ring buffer depends on the sample rate, the number of clients and the speed of the network. We recommend you set a ring buffer size so that there is a span of at least five to ten seconds until the measurements have to be read from the buffers by the clients. The ring buffers are set up in the RAM of the PC. A larger RAM storage space must be available when using many channels.

Enable MD Server remote control

Using ASCII commands, the MD Server may be controlled from other programs. The operating principle and available commands are identical to the DAQ recorder commands. You will find further information about the available commands and procedures in the DAQ_Recorder document ("Infos/DAQ_Recorder.doc" in the catman Enterprise installation folder). Activate this option if you intend to use these control commands.

UPD settings for data streaming

Using the UDP data output you can specify that all measurements after being read in by MD Server are sent immediately over the network using the UDP protocol. This operating mode is possible in addition to a normal measurement with or without other client PCs.

The application which reads these measurements from the network must be programmed appropriately by the user. You must read the data immediately from the network, because repetition is not possible. Since the time requirements are therefore generally very high, you should use a fast programming language, because otherwise values will be lost.

☞ The UDP data output is not intended for real-time control, because the measurements are only sent approx. every 100 ms. New measurements may therefore only be available after 100 ms.

Specify the **UDP destination**, the IP address (**Local IP of network adapter**) and the **Port** of your network adapter through which the output is to take place and enable **Activate catman UDP data stream**. You may use **4-byte format** to reduce the amount of data transferred. With standard sensor data 4 bytes per value (single precision) are usually sufficient. Then continue with your normal measurement procedure, the UDP output will start as soon as the data acquisition starts.

☞ We recommend using a separate interface for this data streaming, not the interface used for the measuring devices or the MD Client communication.

Notes
● The subnet masks of the PC and of the required network segment for the output must match.
● If NTP time stamps are used the 8-byte format (double precision) is required to transfer the full time-stamp information. The 4-byte format (single precision) may be used, if e.g. only data from strain gages or standard sensors is measured.
● For the port use a port number above 10 000; the default setting is 55 001.

UDP format and output rate

Byte	Parameter	Explanation
0-1	Identifier (always #0)	
2-3	Number of channels in current packet (INT16)	The UDP data stream does not contain any information about channel names or units. The channel layout (i.e. the catman I/O Definition) must be known to every client.
4-7	Sequence counter	May be used to identify lost packets
8-11	Measured value of channel 1 (FLOAT32)	The measured value is scaled to the corresponding physical unit
12-15	Measured value of channel 2 (FLOAT32)	
...	...	

Table 2.1   UDP data (packet) format

UDP packets are sent approx. every 50 to 100 ms. This occurs once all measurements from all channels have been read in, i.e. after the retrieval of a read block. The data itself can be output on an interval of approx. 70 µs.
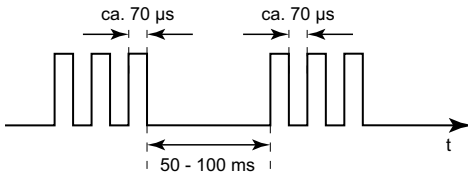
Fig. 2.1   Output rate

2.4.7 Email settings

ⓘ   See also Create email on page 71.

Use **Server options** on the **Setup** ribbon tab to call up this dialog. If you wish to send an email as a trigger action, you must enter the address of the sender and the name of the SMTP email server on this tab.

⚠️ From catman Enterprise 3.1 on the sending of emails takes place via CDO (Microsoft Collaboration Data Objects). Therefore, neither MS Exchange nor MS Outlook needs to be installed, but the SMTP service must be available. Use your Windows installation media to install this component which is part of the Internet Information Service (IIS), if it is not already installed. Additionally, **Mail server name** and **Sender address** must have been set and forwarding to an SMTP email server *without authorization* (anonymous) must be possible. Check with your system administrator, if a Microsoft Exchange Server is used in the intranet. (With default settings this is possible. However, the system administrator can change this so that only specific PCs are allowed to do this.)

2.4.8 ML70 sync (MGCplus only)

☞ Use **Server options** on the **Setup** ribbon tab to call up this dialog. The tab is only available if a ML70 is present in one of the MGCplus signal conditioning devices.

If a ML70 plug-in module is used to deliver values for the data acquisition the sample rate must be set here.

Please note
- 🔴 A special firmware has to be used for this ML70 plug-in module.
- 🔴 The ML70 must reside in the trigger leader MGCplus device and at least one channel with the fastest sample rate has to be acquired by this device, otherwise no measurement is started (error on initializing devices). The channel with the fast sample rate may be a "dummy" channel which does not deliver any measured values, it is just required for the trigger leader MGCplus which has to trigger followers using this sample rate.

2.5 Channel setup (device setup)

ⓘ   See also Autocalibration with MGCplus and QuantumX/SomatXR on page 55.

If you started MD Server the first time, click on 🔲 **MC Setup** on the **Setup** ribbon tab.

Use 🔍 **New scan** if your hardware has changed. This starts MC Setup and asks for a new database name.

Because this procedure plays not only an important part in the measurement but is also time-consuming, there are two possibilities for a hardware channel setup:

1. Define and set up every channel *after* connecting the transducers to the hardware. In this case you need a list showing which transducer is connected to what device and channel.
    Please refer to MC Setup on page 137 for an in-depth cover of this procedure.
2. Make a list of the transducers you *want* to connect (MC Definition List). Then scan the hardware available and let the software try to match the required hardware to your list (as far as possible). This generates a wiring scheme which you may use for connecting the transducers to the hardware and provides the necessary setup database.
    Please refer to MC Definition List on page 225 for an in-depth cover of this procedure.

☞   Please start the measurement only if one of the procedures above has been performed or if the necessary data has been loaded from file.

💡   Use the **Amplifier/Sensors** ribbon tab in MC Setup if you want to see live values, see Checking channels on page 211.

MX840A/B (QuantumX/SomatXR) and CAN bus

CAN channels are included only if Connector 1 is switched to CAN bus and all required CAN signals have been assigned with the MX Assistant.

2.5.1 Time channels

Time channels are created automatically and displayed by catman Enterprise in the time channels group. However, only used time channels are activated. Depending on the DAQ mode you may have one or up to three active time channel types:

●   In **Fast trigger mode** but without having assigned a channel to the fast rate you will have only one time channel type, e.g. no active channels marked SR2 or SR3. Otherwise "Device time stamps … SR2" are available additionally.

- In **Regular DAQ mode** and having assigned channels to all three possible sample rates all possible time channel types are active, i.e. with SR2 and SR3 appended.

Which time value is recorded?
- "DAQ time stamps" contain the time starting with the beginning of the data acquisition.
- "DAQ time stamps absolute" contain the time values from the PC clock.
- "Snapshot time stamps" contain the time starting with the beginning of the data acquisition.
- "Snapshot time stamps absolute" contain the time values from the PC clock.
- "QX trigger time stamps" exists only with QuantumX/SomatXR devices when using **Fast trigger mode**. This channel usually records the date and time values of the standard sample rate in NTP time format (NTP Unix). As soon as a fast trigger occurs, the channel contains the date and time values for the fast channels during the trigger time.
- "Device time stamp" channels are created additionally for QuantumX/SomatXR modules or MGCplus devices and contain the time value present in these devices (date and absolute time). They are listed as "Device time stamps MGC" or "Device time stamps QX" in the **Time channels** list (also for IRIG-B time synchronization). If NTP or PTP time synchronization is used, the NTP time format (NTP Unix) is used for these channels.

Make sure the devices have the correct time if you want to rely on device time channels. Use **Set synchronization settings** (**Special** on the **Tools/Script** ribbon tab) to specify a time source. Also synchronize the PC clock with a time server if you need absolute time channels.

NTP time channels

With an additional *time server*, the HBM devices set the time to this server using *GMT* (Greenwich Mean Time) values. The *local time* or *summer time* is *not taken into account*. If you require local times, you must consider the difference to GMT in the evaluation of the measurement data.

☞ You can also use the NTP time without an NTP time server. The accuracy in this case depends on the accuracy of the internal clock in the HBM devices; only the NTP time *format* is used for the time values.

In order to be able to use the NTP time service, proceed as follows:

1. Provide an NTP time service on the PC running MD Server, see Installing the NTP time server/client on page 18.

    ⚠️ The time server of MS Windows is not suitable.

2. Enter the address of the time server: **Tools/Script** ribbon tab and **Set synchronization settings** in MD Server, see Configure and check synchronization on page 112.

PTPv2 time synchronization

The time is derived from the PTP leader clock. It is recorded in NTP time format, however with much better accuracy than with an NTP time server, especially between all devices using the PTP synchronization. When using non-HBM devices (external devices) without PTP capability you should either set the leader HBM device to use the same NTP time server as these devices or synchronize the PTP time and the NTP time to the same source.

Accuracy of time stamps

- Standard DAQ time stamps depend on the accuracy of the internal clock in the MGCplus (CPxx plug-in module) or QuantumX/SomatXR.
- The accuracy of a standard time channel with *absolute* time stamps depends on the PC clock (which may be synchronized to an NTP or PTP time server as well).
- The accuracy for PTPv2 time stamps is better than 10 µs for MGCplus and 1 µs for QuantumX modules with B or C functionality or SomatXR modules.
- Without an NTP time server, this also applies for NTP time channels. The difference between the time server data and the internal clock can be seen with **Set synchronization settings**: **Offset** (**Tools/Script** ribbon tab), if **Enable NTP/PTP sync monitoring** is activated on the **General options** tab in the **Server options** (click on **Server options** on the **Setup** ribbon tab). The offset applies to the trigger leader MGCplus and all QuantumX/SomatXR modules which serve as Synchronization leader (SYNCLEADER).

    ☞ You may use an NTP/PTP sync failed trigger on page 66 which issues an alert in case the synchronization fails. Additionally the synchronization quality can be shown in the **DAQ status** window, see DAQ status window on page 100, Enable NTP/PTP sync monitoring on page 40.
    The **Synchronization settings** dialog cannot be called up during a measurement run.

2.5.2 Autocalibration with MGCplus and QuantumX/SomatXR

The so-called *autocalibration* (**Auto CAL** column in MC Setup) improves the long-term stability (aging) and—if there are temperature changes where the measurement device is located—also the short-term stability of a measuring amplifier. The autocalibration function differs however depending on the device type.

MGCplus

The autocalibration of the MGCplus interrupts the measurement, because during the autocalibration (approx. 300 ms) no measured values are available from transducers but rather only the internal calibration signals (zero and reference signal) are available. Therefore, the autocalibration is switched off as factory default. If you are carrying out a slow measurement using sample rates below 10 Hz, you can however activate a cyclic autocalibration (approx. every 5 minutes).

⚠️ During autocalibration, the last measured values will be output both via the interface and also via the analog output (if existing) until new measured values are available.

QuantumX/SomatXR

With the QuantumX/SomatXR the autocalibration procedure works different. It is however not with all modules or all sensor types possible to switch the autocalibration on or off.

1. **Off**
   The autocalibration is deactivated. This setting is only recommended for short-term measurements.

2. **On** (Background calibration)
   This is the type of calibration preferred for QuantumX/SomatXR, if available for the module, sensor type and measuring range in question.
   Internally, several QuantumX/SomatXR modules use two measuring amplifiers which are located on one chip and therefore have practically identical behavior. Both carry out the actual measurement, but from time to time one is calibrated with a high precision calibration signal. So every few minutes (configurable with the MX Assistant) this second amplifier is disconnected from the input and connected to the calibration signal for about 300 ms. Then if different values arise on the output of this amplifier, the amplifier acquiring the actual measurement signal is readjusted accordingly. With very small signals and high resolutions

this disconnection and reconnection may however lead to small distortions in the measurement signal.

Background information

To be precise, the term autocalibration is incorrect, because here not only are any changes in the zero point and gain of the measuring amplifier *measured*, they are also *corrected*. So really auto-adjustment takes place. The term autocalibration is however widely used for this process in measurement technology.

2.6 Measurement settings (DAQ settings)

You may set each channel separately or select several channels either by clicking and dragging with the mouse or, to select a range, by clicking on the first channel, then holding down the ⟨⊕⟩ key and clicking on the last channel. The context menu offers all choices as menu items.

☞ For selecting a group of channels, you may click on the left side (**Select** window) on any group. This will display all channels in that group on the right-hand side.

How to proceed

1. Specify whether you need **Fast trigger mode** (allows to switch to a fast sample rate on trigger event) or **Regular DAQ mode** (three sample rates).
   See also
2. Set the sample rates to be used.
   See also and .
3. Define for each channel (hardware *and* computations):
   - whether it should be active or not (participate in the test run or not),
   - whether the data shall be stored,
   - whether data shall be acquired with the fast sample rate additionally (only in **Fast trigger mode**), refer to ,
        or
     which of the three sample rates is to be used (**Regular DAQ mode**).
   ☞ Due to memory restrictions, the **Max. rec.(ording) time for fast trigger** is determined either by the hardware (buffer size of CPxx inside MGCplus devices) or the available PC RAM. The default setting with QuantumX/SomatXR is 58 000 values if this is possible with the number of active

channels. Therefore this setting cannot be changed. You may however define a pre- and post-trigger time within these limits.

💡 If you set **Pre**(-trigger time) to **0** (zero), the total recording time may be used as **Post**(-trigger time) and vice versa.

4. Set a circular buffer size in the **Server options** dialog on the **Client - Server** tab (click on **Server options** on the **Setup** ribbon tab) if you want to connect client PCs, see Circular buffer for Client—Server communication on page 48.

5. If you store channels or plan to make snapshots you can also decide not to store all values but only every nth value: Activate **Data compression** in the ribbon for storage or snapshots.

Use the leftmost column or the **Channel** column to mark several channels and the context menu to perform an action, e.g. to deactivate the marked channels.

💡 You may use the Search facility to limit the display to just the channels found: Use **%** or ***** as wild cards for several characters and **_** or **?** as wild cards for single characters. You may use wild cards more than once in the search string as in **%nnel??04%**. Without any wild card character, an exact match is searched. See also Search functions on page 216.

Fast trigger mode or Regular DAQ mode (two or three sample rates)?

Fast trigger mode (two sample rates) allows you to use a *Fast sample rate trigger*, i.e. all channels marked as **Fast channel** are measured at the beginning with the **Slow rate** (standard time channel). If the trigger action **Fast** (sample rate) occurs, these channels are measured with the **Fast rate** (time channels with extension SR2). It is however not possible to use the fast sample rate for the whole data acquisition without any trigger condition. During the Post-trigger time you cannot stop the data acquisition.

☞ Data acquired with the fast sample rate is stored directly to file and never forwarded to client PCs. Also, data from channels with the slow sample rate acquired during the (fast sample rate) *post-trigger time* is not available for display on client PCs. However, after the post-trigger time this data is stored in the circular buffers (see Client - Server settings on page 47) and can be transferred to client PCs. So if the whole post-trigger time fits into these buffers, the data will be available for storage on client PCs at least after the post-trigger. See also Use fast sample rate on page 70 in topic Setting trigger actions on page 68.

With **Regular DAQ mode** (three sample rates), each channel has to use one of these sample rates for the whole data acquisition and a switchover due to a trigger event is not possible.

☞ Continuous snapshots are not possible in **Regular DAQ mode**.

Notes

🔴 Only active channels participate in a test run. All other settings in this window are valid only if the channel is activated.

🔴 The **Store channel** column has to be set active for a channel which shall be stored. A storage of data occurs however only *if the storage is also activated during the test run*, either manually with **On** and **Off** in group **Storage** (**Test** ribbon tab) or via a trigger.

⚠️ When selecting channels for storage, make sure that enough storage space is available. The procedure how to define storage space is described in Configure storage size on page 35. If the storage space is filled with data, a dialog is displayed and the data acquisition stops.

See also Export options on page 42 for periodical export from the database and Operate in circular buffer mode on page 37.

🔴 In **Fast trigger mode** the fast sample rate (time channel with extension SR2) is used (**Fast channel** column active) if a certain trigger condition occurs. The data acquired is *always* saved directly into a separate file independent of the **Store channel** column setting and this data is not available for client PCs. See also Use fast sample rate on page 70 in topic Setting trigger actions on page 68.

🔵 All settings are stored to the database which contains the project settings (file extension MCS). If you would like to have the same settings available for another project, export these settings via **Save** or **Save as** on the **DAQ settings** ribbon tab.

2.7 Trigger settings

☞ It is not possible to switch from the **DAQ settings** ribbon tab to the **Trigger/Limits** ribbon tab. Click first e.g. on the **Test** ribbon tab, then on the **Trigger/Limits** ribbon tab. Some trigger settings depend on DAQ settings which are activated only when closing the **DAQ settings** ribbon tab.

The trigger settings specify what is to be done when certain conditions arise. Although the actions which can be performed may depend on the Measurement mode (**Fast trigger mode** or **Regular DAQ mode**) or other conditions, e.g. available hardware, all possible settings are shown.

See also Measurement settings (DAQ settings) on page 56, Trigger status window on page 103.

⚠️ A database channel must be available for every trigger condition, see Configure storage size on page 35 for details. An alternative may be the use of limits which do not need a database channel, see Limit checking on page 73.

Possible actions

1. Take a single snapshot or start/stop continuous snapshot mode (**Snapshot** column).
    ☞ Continuous snapshots are only possible in **Fast trigger mode**, see Measurement settings (DAQ settings) on page 56.
    The number of measurements acquired when taking a single snapshot depends on the sample rate: with low sample rates it is a single value, with higher sample rates it is a complete data transfer block i.e. several values. You can see the block size in the **DAQ status** window in field **Data transfer block size**, refer to DAQ status window on page 100.
2. Show a message box on the server PC screen (**Message** column).
3. Set a digital output on a ML78 (MGCplus) or MX879 (QuantumX/SomatXR) module to high or low (**DOut** column).
4. **Fast trigger mode** only: Store data to file on channels marked **Fast channel** in the DAQ settings with the high sample rate specified (**Fast channel** column).
5. Create an email (**Email** column). Please check in this case in the **Server options** (**Setup** ribbon tab) the Email settings on page 51.
6. Create an entry in the project log file SERVERLOG.MDB (**Log** column), see MD Server project log on page 130.
7. Start a (already loaded) VBScript procedure (**VBScript** column), see Using VBScript on page 124.

Double-click in the respective column to set or define an action.

Notes

● There is no restriction on how many trigger conditions may be defined. The only point to consider is the size of the storage space, i.e. the number of

database channels available.

- A trigger condition is recognized even if it exists only shortly inside a read block, see General tab (DAQ status window) on page 100.

  ⚠ If a trigger condition occurs several times in a single read block, the relevant action is also carried out several times.

- The **Unit** column just shows the unit of the selected trigger channel or event. It is not possible to change this setting.

- The **Settings** group in the ribbon provides import and export items. Use **Export trigger as** to save the current trigger conditions to a trigger definition file, which may be imported into a different project.

2.7.1 How to define trigger channels

Define triggers with channels

1. Use the Channel selector window on page 130 to specify the source channel for a trigger monitoring. If this window is not visible, use 📑 **Show channel selector** to open it. Click in the upper part of the window in one of the three groups **System/Function groups/Search results** on a group or device and select a channel from the lower part.

2. Drag this channel with the mouse (left key pressed) out of the **Channel selector** window and release the mouse key: A new line containing this channel as trigger channel is created. As default, the trigger names TRIG_1, TRIG_2, etc. are created. If necessary, change it.

   💡 To select several consecutive channels, use ⬆ (Shift key) and click on the first and last channel to be used. Then, still holding the Shift key, drag the channel you clicked *last* out of the **Channel selector** window (lower part).

   As an alternative you may create a **New** trigger (**Standard triggers** group) from the ribbon and drag a channel in the newly created line.

3. Now specify the trigger condition, see Setting trigger conditions on page 61. You may edit **Tolerance %** and **Threshold** for several selected triggers in one go: choose **Edit selected trigger** from the context menu in these columns.

4. In the last step define the reaction type: show a message, run a VBScript etc., see Setting trigger actions on page 68. Use **Copy** and **Paste** from the context menu, to assign identical actions to several triggers.

💡 You may sort the trigger definition display by each column in ascending or descending order: right-click on the header line.

Define triggers from the ribbon

The ribbon contains several groups for special trigger conditions: Power triggers, Sync triggers and Special triggers.

- **UPS** failure/remaining time trigger
- **MGCplus power supply** (voltage/current/temperature monitoring) trigger
- **NTP/PTP sync failed** trigger
- **MGCplus SYNC lost** trigger
- **Device communication lost** trigger
- **Number of channels in overflow** changed trigger
- **MD Client/MD Trend** trigger

With these special trigger conditions entries in unnecessary columns like **Channel** etc. are ignored.

See Setting trigger conditions on page 61 for the individual trigger settings.

2.7.2 Setting trigger conditions

There are several possibilities available depending on the type of trigger:

1. Check for a certain value in one channel on page 62.
2. Check for certain values in two associated channels (Prediction line) on page 62.
3. Watch for a digital signal state in ML78 or MX879 on page 64.
4. Watch for a digital signal state in the Command channel on page 64.
5. Peak detection on page 65.
6. UPS failure trigger/remaining time alert on page 65.
   - ☞ The UPS must be available (optional device) and of a type approved by HBM, see Operating requirements on page 15. Additionally **Enable UPS monitoring** must be active on tab General options on page 39 (**Server options** on the **Setup** ribbon tab.
7. MGCplus voltage/current/temperature triggers on page 66.
8. NTP/PTP sync failed trigger on page 66.
9. MGCplus SYNC lost trigger on page 67.
10. Device communication lost trigger on page 67.
11. Number of channels in overflow on page 67.
12. MD Client/MD Trend trigger on page 68.

A trigger condition is recognized even if it exists only shortly inside a read block (see also General tab (DAQ status window) on page 100. If a trigger condition occurs several times in a single read block, the relevant action is also carried out several times.

2.7.2.1 Check for a certain value in one channel

This type of trigger is available for all measurement channels and for computation channels.

How to proceed

1. Specify in column **Trigger mode** whether the trigger shall occur if the actual value is above (**>**) or below (**<**) the threshold stated in column **Threshold**.
2. Type in the threshold value (**Threshold** column).

💡 The default threshold values may be defined in MC Setup with the **Upper trigger level** (above **>**) and **Lower trigger level** (below **<**) columns.

2.7.2.2 Check for certain values in two associated channels (Prediction line)

This type of trigger is available for all measurement channels and for computation channels. In contrast to the standard single value trigger, this trigger monitors two channels (a data pair) in reference to a straight line through zero. It can be used, for example, with force-strain measurements if a certain strain value is to be monitored depending on the force level applied. As soon as the strain level at a certain force level is exceeded, an event is triggered. Of course, an event may be triggered also if the strain value falls below a certain value depending on the force level.
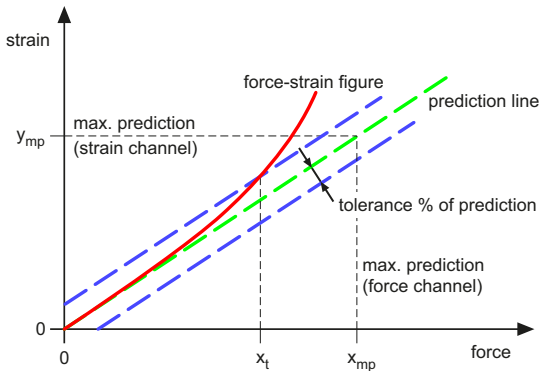
Fig. 2.2 Example of a force-strain measurement with prediction line triggering

The function used for the check is called *Prediction line* and may be pre-defined in MC Setup with columns **Max. prediction**, **Tolerance % of prediction**, see MC Setup column description, scaling functions on page 164. At least the **Max. prediction** value and the **Ref. x channel** has to be defined in MC Setup. When using trigger condition **Value > predict. line** for the figure shown above with **Max. prediction** of the strain channel = y_{mp} and **Max. prediction** of the force channel = x_{mp}, a trigger is fired at force level x_t. The evaluation is continued (and a trigger is fired if necessary) if values above **Max. prediction** occur. However, curve plots are restricted to the **Max. prediction** value, there will be no values plotted higher than this.

How to proceed

1. Specify in column **Trigger mode** whether the trigger shall occur if the actual value is above (**>**) or below (**<**) the prediction line.
2. Type in the values for **Max. prediction** and **Tolerance %** of prediction (deviation from the green line in the figure above specified as percentage of Max. prediction in the strain channel).
3. Specify the x channel by dragging a channel from the **Channel selector** window to this field if not already filled out with the default setting from MC Setup.

⚠️ The x_{mp} value has to be specified in MC Setup as **Max. prediction** of the x channel which is later selected as **Ref. x channel**.

☞ In former versions of catman Enterprise *Max. prediction* was named "100%" or "Physical 100%" and *Tolerance % of prediction* was named "Percent of prediction".

Additionally the prediction line had to be entered using slope and offset values, which is now obsolete.

💡 You may also display the prediction line and the tolerance in graphs, see Real-time graph on page 78 and Post-process graph on page 80.

2.7.2.3 Watch for a digital signal state in ML78 or MX879

⚠️ This trigger type is only available if an MGCplus amplifier plug-in module of type ML78 or—with QuantumX/SomatXR—a MX879 module with digital inputs and outputs is used.

How to proceed

1. Drag from the **Channel selector** window the digital channel from a digital module to the **Channel** column.
2. Select in the **Trigger mode** column whether the high or the low state shall trigger an event.
3. Specify which bit is to be monitored in the **Bit** column.

2.7.2.4 Watch for a digital signal state in the Command channel

⚠️ This trigger type is only available if a MOOG control system (MOOG CS) is connected via Ethernet.

The second channel (the name must be DeviceName_COMMAND) from the MOOG control system transfers 32 bits of status information. The lower 16 bits are used by the signals for MD Server (MOOG command channel on page 118), the higher 16 bits (17 to 32) are free and may be assigned according to your requirements.

How to proceed

1. Drag from the **Channel selector** window the Command channel (**DeviceName_COMMAND**, e.g. **RTFE_HBM_COMMAND**) from the MOOG control system with this channel to the **Channel** column.
2. Select in the **Trigger mode** column whether the high or the low state shall trigger an event.
3. Specify which bit is to be monitored in the **Bit** column.

2.7.2.5 Peak detection

This trigger type is handy, if a periodical signal is to be monitored for local extrema. At the beginning the first measurement is stored as first extremum. During the measurement the last extremum plus or minus a hysteresis value (**Threshold** column) is checked against the actual value. As soon as the actual value is outside this hysteresis band, the next max or min value that occurs is taken as new extremum. In order to have a short time delay, only *one* measured value smaller than the last measurement is required for a maximum or *one* value bigger than the last measurement is required for a minimum.

2.7.2.6 UPS failure trigger/remaining time alert

⚠️ This trigger event is only available if an uninterruptible power supply unit (UPS) approved by HBM (see Operating requirements on page 15) is installed together with the appropriate monitoring software from the manufacturer of the UPS. Additionally **Enable UPS monitor** must be active on the **General options** tab in the **Server options** dialog, see General options on page 39.

Two conditions of the UPS may be used as trigger event:

1.   Power failure and
2.   remaining time in seconds for UPS operation when operating on battery.

The trigger events rely on the event notification system of the UPS. Therefore a notification delay may be configured in the UPS software (wait time) thus suppressing unnecessary notifications due to short power failures. However, there may also be a 60 or 90 seconds time grid used by the UPS when forwarding the remaining time value. We therefore recommend not to use settings which are too close to the shut-down settings (default with most UPS units: 180 seconds). The MD Server software checks approx. every three seconds for new events or data.

💡 Together with catman Enterprise the UPS Manager program is started if the above mentioned conditions are fulfilled. Use the Windows task bar or **UPS manager** (**Diagnosis** ribbon tab) to call up the program. There remaining run time, capacity, load, temperature and status are displayed.

2.7.2.7 MGCplus voltage/current/temperature triggers

⚠ This trigger event is only available with MGCplus devices having an ER003D (rack module), TG003D (desktop housing) or later housing and if **Enable MGCplus power supply monitoring** is active on tab **General options** in the **Server options** dialog, refer to General options on page 39.

In order to be able to set an appropriate value you should watch the values occurring under normal operation for several 10 minutes. When having several devices, check for the device with the maximum current or temperature and the minimum voltage. Then add a safety margin and use these values as trigger levels. The voltage trigger operates always below the **Threshold** value, all others above.

The triggers are not device specific: If the current on the ±5 V or ±16 V power supply lines or the temperature of any device connected exceeds the specified threshold, the corresponding trigger will be fired. The same applies to the voltage trigger: If the +5 V supply voltage of any device connected drops below the specified threshold, the trigger will be fired.

See also **DAQ status** window: Power supply tab (DAQ status window) on page 103.

2.7.2.8 NTP/PTP sync failed trigger

ⓘ See also Configure and check synchronization on page 112, Installing the NTP time server/client on page 18, Configure and check synchronization on page 112, General options on page 39 (Server options).

⚠ This trigger event is only available with QuantumX/SomatXR or with MGCplus devices with CP42 or CP52, if an NTP or PTP time synchronization is set up and if **Enable NTP/PTP sync monitoring** is active on the **General options** tab in the **Server options**.

This trigger type is convenient for monitoring the availability and accuracy of an activated NTP or PTP time service because the **Synchronization settings** window is not available during a running measurement. A check is made whether the NTP or PTP service is running, whether the proper time server is used and—for HBM devices—whether the time delay (offset) between time server and device is smaller than the time specified (**Threshold** and **Unit** columns). The time offset is checked approx. every three seconds. External devices using the NTP or PTP service from the MD Server are checked as well, if the driver setup contains the appropriate information (the drivers.inf file must have the

TimeSyncMonitor entry). In case of doubt, please check with the HBM Technical support, see Technical support on page 29.

2.7.2.9 MGCplus SYNC lost trigger

⚠️ This trigger event is only available with MGCplus devices with CP42 and firmware 4.32 or higher or with CP52 and if **Enable MGCplus SYNC monitoring** is active on tab General options on page 39 in the **Server options** dialog.

This trigger type is convenient for monitoring the status of the MGCplus synchronization. If anything is not working properly with this internal device synchronization, the corresponding fields for the device in question are displayed with a yellow or red background on the **Sync monitor** tab of the **DAQ status** window, see DAQ status window on page 100. As soon as a red background is displayed, this trigger is fired. Please check with the HBM Technical support, see Technical support on page 29 for possible causes and solutions to this problem.

2.7.2.10 Device communication lost trigger

ℹ️ See also General options on page 39.

This trigger type allows you to issue a message, e.g. via email, or start a script to perform further operations if the communication to one of the signal conditioning devices is lost, i.e. a **Read timeout** has occurred (**Server options** dialog, **General options** tab). The data acquisition is usually not stopped in this case, however, the device in question and all channels contained are deactivated, i.e. they do not participate in the further measurement.

⚠️ If the device communication to the trigger leader is lost the data acquisition stops.

2.7.2.11 Number of channels in overflow

This trigger type allows you to issue a message, e.g. via email, or start a script to perform further operations if a certain number of channels changes the overflow state. You may select to trigger an action when the number increases or when the number decreases. The number of channels for the increase or decrease is set under **Threshold**.

This trigger is useful, if damage to cables shall be monitored or if CAN bus sensors might stop transmitting. Both errors would result in an overflow which may be sensed using this trigger.

💡 The number of channels in overflow state is displayed in the **DAQ status** window after initialization, see DAQ status window on page 100.

2.7.2.12 MD Client/MD Trend trigger

This trigger type allows to evaluate a condition on a different PC, e.g. a PC running MD Client or MD Trend. If this condition is fulfilled, this is forwarded to the server PC and—with this trigger condition—the appropriate action is performed on the server. This is also called a server-side trigger.

2.7.3 Setting trigger actions

You may define the following actions:

1. Snapshot and storage on page 68.
2. Show message on page 69.
3. Set digital out (DOut) on page 70.
4. Use fast sample rate on page 70.
5. Create email on page 71.
6. Log entry on page 72.
7. Run VBScript on page 72.

2.7.3.1 Snapshot and storage

ℹ️ See also Measurement settings (DAQ settings) on page 56, Configure storage size on page 35, Export options on page 42.

⚠️ Make sure that enough storage space is available.

Snapshots

Just decide what you want to do if the trigger condition is fulfilled: **Single** snapshot, **Start** or **Stop continuous** snapshot mode. The number of measurements acquired when taking a single snapshot depends on the sample rate: with low sample rates it is a single value, with higher sample rates it is a complete data transfer block i.e. several values. You can see the block size in the **DAQ status** window in field **Data transfer block size**, refer to DAQ

All snapshot data is stored to the internal database. *After* the test run all stored values are exported to binary file SNAPSHOT_<Test base name>_ <count>_<date>.BIN, e.g. Snapshot_NewTest_1_2020_03_31.bin.

⚠️ At least one active channel of all devices must be in sample rate A if **Regular DAQ mode** is used with snapshots.
Continuous snapshots are only possible in **Fast trigger mode**.

💡 It is recommended that you define a stop condition (new trigger) to end the continuous snapshot mode before all internal database memory is consumed. Otherwise you must either use the internal database in circular buffer mode or enable via the **Server options** dialog and the **Export options** tab the **Periodical export of storage channels** option. Use a value which is not too big in this case, as exporting many channels may be time-consuming.

Storage on/off

You may use **Storage on** (or **Storage off**) in the **Snapshot** column, if **Store channel** is active for one or more channels (**DAQ settings** tab). All channels with this setting are then stored to the internal database if the trigger condition is fulfilled. *After* the test run all stored values are exported.

💡 It is recommended that you define a **Storage off** (new trigger) to end the storage before all internal database memory is consumed. Otherwise you must either use the internal database in circular buffer mode or enable via the **Server options** dialog and the **Export options** tab the **Periodical export of storage channels** option. Use a value which is not too big in this case, as exporting many channels may be time-consuming.

2.7.3.2 Show message

This displays a message on the server screen. Just double-click on the button in the column to be able to type in the message.

💡 Use **Insert system texts** to insert special text strings. Strings between @ characters are converted before saving the log entry, please do not change them.

2.7.3.3 Set digital out (DOut)

⚠️ This trigger action is only possible if
either a QuantumX/SomatXR with MX879
or a MGCplus with a ML78 plug-in module with digital I/O and AP75
or a MOOG control system (MOOG CS) connected via Ethernet is available.

MOOG control system (CS)

Click on **DOut** and specify **DeviceName_COMMAND**, e.g. **RTFE_HBM_COMMAND**, from the MOOG control system as channel. Then activate the bit to be changed and whether the bit shall become high or low if the trigger occurs.

MGCplus with ML78 or QuantumX/SomatXR with MX879

Click on **DOut** and specify the MX879 module or the channel containing the ML78 (the names of the device and the channel are displayed). Then activate the bit to be changed and whether the bit shall become high or low if the trigger occurs.

2.7.3.4 Use fast sample rate

⚠️ This trigger action is only possible if **Fast trigger mode** (two sample rates) is used. With **Regular DAQ mode** (three sample rates) you have to assign a higher sample rate for the channel in question which is then used for the complete test. See also .

This setting uses some background operation: it is even possible to specify a pre-trigger time, i.e. a certain time before the trigger condition occurs, in which this fast measurement is used. Of course, it is not possible to go back in time and redo the measurement! The trick is that all channels marked as **Fast channel** in the DAQ settings are always measured with this fast rate. Under normal conditions this high sample rate is then reduced to the slow sample rate specified and provided as normal measurement data. Depending on the device used, all values are either stored to a special buffer inside the MGCplus or in the RAM of the server PC. As soon as the trigger condition is true, data visualization and all data forwarding (to client PCs) stops (data acquisition continues). Only the visualization on the server PC continues (with the fast sample rate). The buffers with all pre-trigger values are read out and stored to file FastTriggerData_<Test base name>_<count>.BIN in subfolder TRIGGER, e.g. FastTriggerData_MyTest_001.bin. Data acquired during post-trigger time is appended to this trigger data file and the time channels (extension SR2) are stored additionally. After the post-trigger time everything is

again like normal data acquisition, i.e. all real-time visualization panels on the client PCs are updated. Also as much data as possible is stored in the buffers set up for the Client-Server communication (see Client - Server settings on page 47) and data is again forwarded to client PCs. This means that if the buffer size is smaller than the post-trigger time, values are lost.

Define the **Pre-** and **Post-**trigger time in the Measurement settings (DAQ settings) on page 56. During the Post-trigger time you cannot stop the data acquisition.

☞   The (relative) time channel valid for the fast trigger channels starts at the pre-trigger time with negative values so that the trigger point in time equals 0.

⚠️   Data acquired with a fast sample rate in **Fast trigger mode** is not stored to the internal database: it is always *directly stored to file*. During saving of the pre-trigger values and during the post-trigger time the display of all channels on client PCs is stopped and *no data* from these channels is forwarded to Client PCs. The data is however stored into the server PC storage channels—if activated—in the background and then forwarded to the buffers set in the Client-Server options.

2.7.3.5 Create email

ℹ️   See also Email settings on page 51 (**Server options** dialog).

This action allows an email message to be created for a user or user group with special text like time or measured data.

⚠️   From catman Enterprise 3.1 on the sending of emails takes place via CDO (Microsoft Collaboration Data Objects). Therefore, neither MS Exchange nor MS Outlook needs to be installed, but the SMTP service must be available. Use your Windows installation media to install this component which is part of the Internet Information Service (IIS), if it is not already installed. Additionally **Mail server name** and **Sender address** must have been set in the **Server options** dialog, **Email settings** tab) and forwarding to an SMTP email server *without authorization* (anonymous) must be possible. Check with your system administrator, if a Microsoft Exchange Server is used in the intranet. (With default settings this is possible. However, the system administrator can change this so that only specific PCs are allowed to do this.)

Type in the address of the recipient or click on **System -> Address book** to open an address book. Use **Insert system texts** to insert special text strings. Strings between @ characters are converted before sending the message, please do not change them.

Example

Address: **Administrator@Company.com**

Subject: **Measurement in critical phase**

Text: **@MV/Channel_01_01_001@ has exceeded value xyz at time @TI/Channel_01_01_001@**

2.7.3.6 Log entry

This creates an entry in the project log file SERVERLOG.MDB. Use **Insert system texts** to insert special text strings. Strings between @ characters are converted before saving the log entry, please do not change them.

2.7.3.7 Run VBScript

Runs a VBScript procedure. The script must already be in memory (loaded), otherwise no selection is possible.

☞ Include all script procedures necessary into one VBScript project file (∗.ESP, **E**nterprise **S**cript **P**roject). Then specify this file in the project settings (see). This saves you having to load the VBScript manually.

Example

The following VBScript example contains a procedure which displays a message on the server screen, if the measurement value in Channel_03_03_002 exceeds the specified threshold (**Trigger mode**: **Value > threshold**).

```
Sub TriggerSample()

Dim idxChan
Dim valChan

idxChan = catman.IO.GetChannelFromName("Channel_03_03_002")
valChan = catman.IO.MVal(idxChan,0)
MsgBox "Limit of Channel_03_03_002 exceeded, " _
    & "current value is " valChan

End Sub
```

2.8 Limit checking

☞ It is not possible to switch from the **DAQ settings** ribbon tab to the **Trigger/Limits** ribbon tab. Click first e.g. on the **Test** ribbon tab, then on the **Trigger/Limits** ribbon tab.

For all channels upper and lower limits may be monitored. In contrast to Trigger settings on page 58 the limit checking does not require a database channel for evaluation. Choose on the left side of the dialog, which channel or function group to display.

The values for the limit levels are pre-defined in MC Setup, you may however change these values here (changed values are shown with bold numbers). Activate the box in the **Check** column to enable the monitoring. The background color in the fields for upper and lower level indicate whether the limit condition is still fulfilled or whether it has been fulfilled since the start of the acquisition (the color may not be changed, an explanation for the colors is given at the bottom of the dialog). Activate **Auto popup** to call up the dialog whenever a limit level is exceeded or undercut.

Settings

☞ Limit checking may only be activated if at least either **Upper limit** or **Lower limit** column has an entry unequal to zero and the upper limit level is higher than the lower level.

Check prediction line: Activate this option if a **Reference x channel** is specified in MC Setup and a prediction line shall be checked. Enable **Show prediction line** in the plot configuration to show the line, see Real-time graph on page 78 or Post-process graph on page 80.

☞ Prediction line checking may only be activated if a **Reference x channel** is defined for the channel in question in MC Setup.
See also Check for certain values in two associated channels (Prediction line) on page 62 for an explanation of the operation principle.

Tolerance %: This entry enables you to display a tolerance band around the prediction line. The tolerance band is drawn as two parallel lines with the width as positive and negative shift, if you activate **Show tolerance band** in the plot configuration, see Real-time graph on page 78 or Post-process graph on page 80. Specify here (or in MC Setup) the width of the tolerance band as a percentage of the **Max. prediction**. The value specified here takes precedence over the value in MC Setup and is therefore displayed in bold font.

Max. prediction: This value is used as full scale value for the channel. It is derived from the setting in MC Setup. The value stated here is also used as reference for the **Tolerance %**. The value specified here takes precedence over the value in MC Setup and is therefore displayed in bold font.

See also

VB Proc: Be careful when assigning a VBScript procedure to a limit check: as no hysteresis is defined, the procedure is run every time the condition is met. If the signal is close to the limit specified, a small noise level triggers the procedure several times. Therefore, the procedure should check if it has been called recently or is still running. See also

> 💡 You may save and load limit check definitions: 📂 (open)/💾 (save) in the **Limit checking** dialog. They can also be transferred from or to MD Client PCs. However, when importing on a MD Client, only definitions from channels available in the pre-selection are imported.

Options

Auto popup: If a limit is exceeded the dialog pops up (if not already shown).

Show enabled channels: Restricts the display of channels to the activated ones.

Show channels exceeding limits: Displays only channels which are currently exceeding a limit. If you activate additionally **Include once exceeded**, channels having exceeded a limit in the past are displayed too.

Reset status: Resets all indications (background colors) and the status whether a channel has been exceeded in the past.

2.9 Defining visualization (panels and objects)

This visualization is for the MD Server window only. It may be used to monitor important channels during a measurement from the server PC. As a rule, the visualization of measurement data is done on the client PCs.

> ⚠️ Each visualization object needs processing power and may decrease the acquisition speed.

We recommend therefore using the visualization objects with caution and not to use too many objects besides the status information windows.

💡 You may save or load a particular set of visualization objects with **Save panel** and **Open panel** (**Panel** ribbon tab). As soon as you save a panel it is included into the project. Use the **Project settings** dialog (**Setup** ribbon tab) to add other or remove certain panel files from your project. A panel (file) contains not only the visualization objects but also the configuration of these objects.

Only *one* panel is shown at a time. Click on 📄 + to add another panel, then use **Save panel as** to change the name of a (newly created) panel. The panel file name is also used as panel name and the panel is automatically included into the **Project settings**. See also Project settings on page 33.

Use the **Indication** group to create the various visualization objects. Use the Channel selector window on page 130 to assign channels to objects. The objects most often used are directly accessible, all others are grouped: click on the triangle to the right of the group to open the menu.

The **Panel navigator** window (**Show panel navigator**, **Control** group) shows all objects of the active panel and their position. Change the numbers and click on **Change** to move an object.

Background picture (for a panel)

ℹ️ See also Background picture in a graph on page 78.

You can assign a background image for the whole panel with **Open** in the **Background pictures** group. Formats BMP (bitmap) and GIF (Graphics Interchange Format) are possible. Use the other entries in the group to delete or stretch (scales to the panel size) the image.

☞ No new background images can be assigned *during* a measurement.

2.9.1 Available visualization objects

There are different object types available:
- Real-time displays including a Spreadsheet on page 80 object for real-time values,
- a Post-process graph on page 80,
- Snapshot displays: a Snapshot spreadsheet on page 86 and a Statistical spreadsheet on page 87,

- Other display objects:
    - a Label (text or image) on page 87 which can be used for displaying texts or pictures, e.g. company logos. You may also display the status of a MOOG control system (just activate the context menu item), see MOOG CS options on page 116.
    - a Command button on page 87 for starting VBScript actions,
    - the Flex table on page 88,
    - a Load level visualization on page 96,
- a Background picture object (for the whole panel) is available on the ribbon in group **Control**.

How to configure these objects is explained in Configuring visualization objects on page 77.

Real-time and post-process objects

These objects display either data in real time or show all snapshots stored in the database (post-process). Most real-time objects display the current value only; the real-time graph displays a certain amount of values only, see Real-time graph on page 78.

Nearly all objects allow various export formats and destinations: to clipboard, file or printer and in Windows metafile format (∗.WMF), as bitmap (∗.BMP) or JPEG (∗.JPG). In the **Export graph** dialog (context menu) it is even possible to export **Data only** which results in a table with all values currently displayed.

VBScript command button

ⓘ See also Field type, formatting (Cell tab) on page 89.

This object shows a button which triggers the connected VBScript (**Configure** menu). See Using VBScript on page 124 for details concerning VBScript.

☞ There is also the possibility to have command buttons on an additional tab in the ribbon. This tab and the defined actions are then available for all panels as soon as the scripts are loaded. See Custom commands with VBScripts (custom ribbon tab) on page 128.

Load level visualization

ⓘ See also Enter manual input, Manual input channel on page 191.

This object allows for monitoring certain load levels for different channels. The different levels are indicated by coloring the **Load level** column of the respective channel. This

makes it easy to determine immediately, whether a number of channels has reached a certain load step. The color levels are set in percent, the value for 100% is defined individually per channel.

💡 You can also specify the 100% value in MC Setup in the **Max. prediction** column, see MC Setup column description, scaling functions on page 164.

2.9.2 Configuring visualization objects

With the visualization objects you may:

- Move and re-size an object.
  - ☞ Do not click on the top toolbar to move a *graph.* Use any point inside the object instead. With all other objects use the object caption (top line with object title).
- Decide which colors shall be used for graphs or the display of values.
- Change the scaling.
- Configure the layout.
- Display or hide statistics or change the data display format.

Use the context menu or 📐 **Configure** (**Graphic setup** group) to configure each object. Depending on the possible settings the context menu contains the appropriate items. After selecting an entry, the necessary dialog window is opened. Just specify the settings you require. The configuration may be called up before, after or during a running data acquisition.

💡 Use **Show panel navigator** to access objects outside the current screen area (**Center object** on screen) or to position objects accurately (type in a new position and click on **Change**). The numbers displayed are *TWIPS* (**TW**entieth of an **I**nch **P**oint, depending on the screen resolution approx. 17 TWIPS are one pixel) and refer to the left upper corner of an object. All objects are shown sorted by object types.

For all objects one setting is important: the data source. If the **Channel selector** window is not yet visible, just use 📇 **Channel selector** (**Graphic setup** group) to call up this window and drag a channel into the object. All other settings are optional.
See also Channel selector window on page 130.

☞ We recommend caution when using autoscaling for the y-axis with the Real-time graphs since this might be time-consuming when redrawing a graph. Use a fixed

scaling instead.

Refresh rate

The refresh rate for real-time displays showing one value only is set from **Refresh rates** (**Control** group). Be aware that real-time displays which show only one value at a time, e.g. a digital indicator, display already only one value per read block (the last = newest one). The setting influences the number of read blocks skipped before forwarding a read block and refreshing the object again.

See also General options on page 39, read block size in topic General tab (DAQ status window) on page 100.

Background picture in a graph

All graph objects support a background image in bitmap (∗.BMP) format. Click on in the **Configure graph** dialog to select an image. The image is automatically scaled to the graph size. Use either or delete the file name (blank field) to remove the image.

See also Background picture (for a panel) on page 75.

☞ No new background images can be assigned *during* a measurement.

Zooming in a graph

Use or to zoom into a graph (horizontal, vertical or rectangular zoom). Click on a graph to display the zoom option, choose a zoom function and then click with the left-hand mouse key inside the graph and drag to indicate the zoom section. Click on the symbol again to return to the original state. will take you back to the last zoom states step by step. The **Zoom** group is only visible in graphs where zooming is possible.

2.9.2.1 Real-time graph

☞ Create a Real-time graph from the **Panel** ribbon tab (**Indication** group).

If an y(x) graph is required instead of an y(t) graph drag the required channel directly to the x-axis. Drag a time channel to the x-axis or use context menu **Use dt for x axis** to get a y(t) graph again.

☞ If a **Reference x channel** (see MC Setup column description, scaling functions on page 164) is defined, this is used as default for the x-axis in y(x) graphs.

Clicking on a plot legend[1] (left-hand mouse key) leads to a popup menu which lets you configure or delete this plot (![icon]) or delete all plots (![icon]).

Curve fit

You may draw an equalizing plot with ![icon] (**Points and Best Fit Line**, linear interpolation) or ![icon] to ![icon] (**Best Fit Curve** or **Spline**, polynomial interpolation of 2nd to 4th order) in the graph over all values displayed (**Configure plot** dialog).

Control plots

The **Control plots** tab (**Configure** dialog of the graph) allows you to display reference curves or limits. Just specify x and y coordinates (64 max.); a straight line is drawn between these points. Up to 4 control plots are possible.

☞ You cannot edit or create new control plots during a measurement.

Prediction line display

You may display the prediction line with the **Configure plot** context menu (left-hand mouse key) of the plot legend[1] (**Show prediction line**) if a **Reference x channel** is specified in MC Setup.

Also the display of a *tolerance band* around your prediction line may be activated here if either a value is specified in the **Limit checking** dialog or in MC Setup (**Tolerance % (of prediction)**). A trigger definition is not required.

☞ See also Check for certain values in two associated channels (Prediction line) on page 62 for an explanation of the operation principle.

Notes

● Use **History buffer size** in the configuration dialog, **General** tab, to specify how many values are visible (x-axis min/max). The maximum number of data shown is 32 000 per channel. A min/max compression occurs for y(t) graphs (a whole-number compression factor is used) if you specify more than 10 000 values; y(x) graphs are never compressed.

---

[1] The plot legend (curve legend) displays the color, style and name of a curve. It is located at the top edge of a graph: ▬▬ Device_1 CH 0.

- We recommend caution when using autoscaling for the y-axis with the Real-time graph since this might be time-consuming when redrawing the graph. Use a fixed scaling instead.
- If you want to display absolute time values, drag the **DAQ time stamps absolute** or **Device time stamps …** (**Channel selector** window) to the x-axis. Then open the **Configure** dialog via the context menu and activate **Date/time** *and* **Auto** for the **x axis** on the **Axis** tab.
- With ‖ **Freeze graph** in the context menu you may freeze the graph temporarily, e.g. the display is paused.

2.9.2.2 Spreadsheet

🛈 See also Flex table on page 88.

☞ Create a Spreadsheet from the **Panel** ribbon tab (**Indication** group).

The Spreadsheet displays the real-time data of a channel (one channel per row). Just drag the required channel(s) to a row. You may select several channels as usual with ⟨⇧⟩ or ⟨Ctrl⟩ keys.

☞ Values for the min./max. display are retrieved from the values this object receives. Depending on the setting at **Refresh rates** a maximum of one value per read block (read cycle) is provided for this object. Therefore these values are not identical with the minimum or maximum of the acquired values. Hence this indication is only recommended for low sample rates up to 10 Hz and with a setting of **Update indicators after 1 read cycle**.

2.9.2.3 Post-process graph

☞ Create a Post-process graph from the **Panel** ribbon tab (**Indication** group).

The Post-process graph is usually displaying snapshot values stored in the database during a measurement. If an y(x) graph is required drag the required channel directly to the x-axis. Drag a time channel to the x-axis to get a the original display again.
Update every x snapshot refreshes the graph automatically. However, if **Continuous snapshots** is active, you have to use the **Update graph** context menu.

☞ If a **Reference x channel** (see MC Setup column description, scaling functions on page 164) is defined for a channel, this is used as default for the x-axis in this graph making it a y(x) graph.

Clicking on a plot legend[1] (left-hand mouse key) leads to a popup menu which lets you configure or delete this plot (🗙) or delete all plots (🗙).

Curve fit

You may draw an equalizing plot with 〰 (**Points and Best Fit Line**, linear interpolation) or 〰 to 〰 (**Best Fit Curve** or **Spline**, polynomial interpolation of 2nd to 4th order) in the graph over all values displayed (**Configure plot** dialog). The calculated coefficients may also be displayed in the graph: activate **Show best fit equation**; a figure such as **1.5e-2** signifies $1.5 * 10^{-2}$. However, you will only get meaningful values if you assign a channel to the x-axis.

Control plots

The **Control plots** tab (**Configure** dialog of the graph) allows you to display reference curves or limits. Just specify x and y coordinates (64 max.); a straight line is drawn between these points. Up to 4 control plots are possible.

☞ You cannot edit or create new control plots during a measurement.

Prediction line

You may display the prediction line with the **Configure plot** context menu (left-hand mouse key) of the plot legend[1] (**Show prediction line**) if a **Reference x channel** is specified in MC Setup.

Extrapolate three/all points to Max prediction: This uses either the last three or all points to calculate a straight line from these points to the Max prediction value. **Show extrapolated value** displays this value as number in the graph (plot legend context menu).

Also the display of a *tolerance band* around your prediction line may be activated here if either a value is specified in the **Limit checking** dialog or in MC Setup (**Tolerance % (of prediction)**). A trigger definition is not required.

🛈 See also Check for certain values in two associated channels (Prediction line) on page 62 for an explanation of the operation principle.

---

[1] The plot legend (curve legend) displays the color, style and name of a curve. It is located at the top edge of a graph: ▬▬▬ Device_1 CH 0.

2.9.2.4 Digital indicator

☞ Create a Digital indicator from the **Panel** ribbon tab and **Real-time displays** (**Indication** group).

As a rule the Digital indicator displays minimum values (lower left, in blue) and maximum values (lower right, in red). Use **Hide statistic** from the context menu to suppress this display.

Context menu item **Alarm** allows you to specify certain value ranges which are then used to change the background color (alarm checking must be enabled). It might be necessary to change the background and foreground color of the display to have a good contrast with all alarm colors.

Notes

🔵 If you want to display absolute time values, drag **DAQ time stamps absolute** or **Device time stamps …** (**Channel selector** window) into the object. Then apply the **Date + Time** format for the **DAQ time stamps absolute** or the **NTP time/IRIG-B time** format for the **Device time stamps …** via the **Format output** context menu.

🔴 Values for the min./max. display are retrieved from the values this object receives. Depending on the setting at **Refresh rates** a maximum of one value per read block (read cycle) is provided for this object. Therefore these values are not identical with the minimum or maximum of the acquired values. Hence this indication is only recommended for low sample rates up to 10 Hz and with a setting of **Update indicators after 1 read cycle**.

2.9.2.5 Bar indicator

☞ Create a Bar indicator from the **Panel** ribbon tab and **Real-time displays** (**Indication** group).

Besides several different appearances with horizontal or vertical bar, it is also possible to use an inverted scale. Axis labels may be displayed left, right or on both sides.

Notes

🔴 With logarithmic scale a proper **Min** value has to be used, e.g. **0.0001**.

🔴 Values for the min./max. display are retrieved from the values this object receives. Depending on the setting at **Refresh rates** a maximum of one value

per read block (read cycle) is provided for this object. Therefore these values are not identical with the minimum or maximum of the acquired values. Hence this indication is only recommended for low sample rates up to 10 Hz and with a setting of **Update indicators after 1 read cycle**.

2.9.2.6 Analog indicator

☞ Create an Analog indicator from the **Panel** ribbon tab and **Real-time displays** (**Indication** group).

With the Analog indicator several different appearances are available, e.g. dial, knob, horizontal or vertical gauge display with a 90° wide pointer scale. It is also possible to use an inverted scale.

Notes
● With logarithmic scale a proper **Min** value has to be used, e.g. **0.0001**.
● Values for the min./max. display are retrieved from the values this object receives. Depending on the setting at **Refresh rates** a maximum of one value per read block (read cycle) is provided for this object. Therefore these values are not identical with the minimum or maximum of the acquired values. Hence this indication is only recommended for low sample rates up to 10 Hz and with a setting of **Update indicators after 1 read cycle**.

2.9.2.7 LED

☞ Create a LED from the **Panel** ribbon tab and **Real-time displays** (**Indication** group).

The channel name of the data source is displayed as default, but you may change this any time. Just type in a new caption.
Use **Alarm** from the context menu to set the colors. The **Default color** is used as OFF color.

☞ Right-click on the LED, not the text field, to get the context menu.

💡 If **Bit test** is enabled in the **Configure LED** dialog (**Alarm** context menu) you may assign **Default color** and **ON color** to display status information: if the bit specified is set (**1**), the **ON color** is displayed, otherwise (**0**) the **Default color**.

2.9.2.8 Real-time bar graph

☞ Create a Real-time bar graph from the **Panel** ribbon tab and **Real-time displays** (**Indication** group).

Each column displays another channel, just drag a channel into the graph. New channels are always appended as a new bar on the right-hand side. The scale is automatically set to the maximum of all Max. prediction values (see MC Setup column description, scaling functions on page 164), if these values are available.

Notes
- Left-click on a bar to be able to delete this or all plots or to change the text displayed (**Configure plot**). As default the channel name is used.
- Right-click in the graph if you want to configure, duplicate or export the graph.
- With ▐▐ **Freeze graph** in the context menu you may freeze the graph temporarily, e.g. the display is paused.

2.9.2.9 Real-time FFT graph

☞ Create a Real-time FFT graph from the **Panel** ribbon tab and **Real-time displays** (**Indication** group).

Displays the frequency distribution in the present signal. The object should only be used when having sample rates above 100 Hz.

💡 You can also define a calculation which may then save the corresponding frequency and amplitude values, see FFT peak channels on page 208.

Configure real time FFT

Specify the **FFT window function** and the number of measurements (**FFT buffer size**) over which the spectrum is to be calculated together with the **FFT output unit** on the **FFT settings** tab in the **Configure graph** dialog.

Peak detection

Activate **Enable peak detection** to get an additional window in the graph showing the peak frequencies and amplitudes. Set with **Peak threshold** the minimum amplitude which is required for a peak. **Peak width** determines how many samples have to be available for a peak to be considered valid. This eliminates temporary spikes. There must be at least 3 points (calculated) for a peak to be valid.

Notes

- ● We recommend caution when using autoscaling for the y-axis with this graph since this might be time-consuming when redrawing the graph. Use a fixed scaling instead.
- ● Clicking on a plot legend[1] (left-hand mouse key) leads to a popup menu which lets you configure or delete this plot ( ) or delete all plots ( ).
- ● With ❚❚ **Freeze graph** in the context menu you may freeze the graph temporarily, e.g. the display is paused.

2.9.2.10 Channel row graph

☞ Create a Channel row graph from the **Panel** ribbon tab and **Real-time displays** (**Indication** group).

This graph is similar to a Real-time bar graph but enables you to display channel values on the y-axis at *certain* x-axis positions. For each channel to be displayed in a plot, an x position must be specified. All y values of a plot are then connected by a line.

How to proceed

1. Open the channel list with **Add plot** in the context menu.
   The first plot (PLOT_1) is displayed, however no channel is assigned so far.
2. Open the **Channel selector** window ( )
3. Drag the required channel(s) to the Plot list.
   ⚠ Drag&drop on the graph itself like with the other graphs does not work in this case.

   Use the context menu **Delete selected channels** in the graph or channel list to delete selected channels.
4. If required, modify the text to display (x-axis label) in the **x data label** column.
   The default x-axis label is the channel name. The labels are displayed in separate lines, if the x-axis values of channels are identical.
5. Assign in the **x Value** column where the current value of each channel shall be displayed in the graph.

---

[1] The plot legend (curve legend) displays the color, style and name of a curve. It is located at the top edge of a graph: ━━━ Device_1 CH 0.

6. Use the **Add** button to create more plots.
   ☞ Up to 6 Plots are possible.
7. Close the channel list to display the data (click **Add plot** in the context menu again).

Use and to display the previous or next plot in the list, if available. You may assign each plot a different color and use the legend context menu as usual.

💡 Use **Rotate x channel labels** from the context menu to turn the x-axis labels clockwise by 90°, if the texts overlap.

Control plots

The **Control plots** tab (**Configure** dialog of the graph) allows you to display reference curves or limits. Just specify x and y coordinates (64 max.); a straight line is drawn between these points. Up to 4 control plots are possible.

☞ You cannot edit or create new control plots during a measurement.

Notes

● Clicking on a plot legend[1] (left-hand mouse key) leads to a popup menu which lets you configure or delete this plot ( ) or delete all plots ( ).
● As long as the channel list for a plot is displayed, the graph size is restricted in width. The original width is used again, if this list is hidden (click **Add plot** in the context menu again).
● With **Freeze graph** in the context menu you may freeze the graph temporarily, e.g. the display is paused.

2.9.2.11 Snapshot spreadsheet

☞ Create a Snapshot spreadsheet from the **Panel** ribbon tab and **Snapshot displays** (**Indication** group).

The Snapshot spreadsheet displays data similar to an Excel spreadsheet, however only snapshot data is shown. Just drag the required channel(s) to a column.

---

[1] The plot legend (curve legend) displays the color, style and name of a curve. It is located at the top edge of a graph: Device_1 CH 0.

2.9.2.12 Statistical spreadsheet

☞ Create a Statistical spreadsheet from the **Panel** ribbon tab and **Snapshot displays** (**Indication** group).

The upper part of the table is the same as a Snapshot spreadsheet. This spreadsheet uses the snapshot data to calculate with linear regression slope and the point of intersection with the x-axis (offset) against a selected y-channel. Linear predictions for the channels are then calculated using the determined slope and offset. After that, the residuals of the prediction are calculated compared to the actual channel to obtain the standard deviation.

The first channel assigned to the object is used as y-channel. The statistical part of the table is therefore not available for this channel. All other channels assigned are calculated against this channel.

Slope: the calculated slope of this channel against the selected y-channel.

Intercept: the calculated point of the x-axis where the y-channel intersects it.

2 Sigma: the standard deviation of the linear regression residuals multiplied by 2 (confidence level of 95%).

FSE in %: the Full Scale Error in percent is expressed here as 2 sigma divided by the full scale of the y-channel.

2.9.2.13 Label (text or image)

☞ Create a Label from the **Panel** ribbon tab and **Other displays** (**Indication** group).

If you specify a picture file, make sure the file is available when opening a panel with this object.

The following file formats are supported: bitmap (BMP), Windows Metafile Format (WMF), Icon file format (ICO), Graphics Interchange Format (GIF) and JPEG (JPG).

2.9.2.14 Command button

☞ Create a Command button from the **Panel** ribbon tab and **Other displays** (**Indication** group).

Runs a VBScript procedure. The script must already be in memory (loaded), otherwise no selection is possible.

💡 Include all script procedures necessary into one VBScript project file (∗.ESP, **E**nter-prise **S**cript **P**roject). Then specify this file in the project settings (see Project settings on page 33). This saves you having to load the VBScript manually.

ℹ See also Using VBScript on page 124, Custom commands with VBScripts (custom ribbon tab) on page 128.

2.9.2.15 Flex table

ℹ See also Spreadsheet on page 80.

☞ Create a Flex table from the **Panel** ribbon tab and **Other displays** (**Indication** group).

The Flex (flexible) table offers you more possibilities than the Spreadsheet: each cell can be configured individually. As many channels and values may be displayed as contained in the database, see Configure storage size on page 35.

Change column width/row height

You may change the *width* of columns or *height* of rows: Drag the border between two cells in the header row or column.

In order to be able to change *all* rows or columns, click in the field in the upper left corner of the table (   ) to mark the complete table and then move the separator between the two cells in the header row or column.

Marking cells
- You mark individual cells by clicking with the mouse.
- You mark several cells with the mouse by clicking and moving the mouse with the left mouse key pressed.
- You mark several cells with the mouse by clicking in the upper left cell and in the lower right cell of the required area while holding down the ⬆ key.
- You mark several cells with the arrow keys by pressing the appropriate arrow keys while holding down the ⬆ key.

Assigning the value to be displayed

Drag the relevant channel from the **Channel selector** window onto a cell and select the required information from the popup menu.

Field type, formatting (Cell tab)

Using the **Cell** tab, you can define whether a cell is to contain text, a number, an image or a button. For numbers the formatting can also be defined. With a button specify the desired action using the **Action** tab.

Formula (Cell tab)

With the **Numeric (editable)** cell type you can also enter a formula in the **Formula** field similar as in Excel. Specify the cell references using the column letter and the row number: A1 (first row and column) or C4 (third column and fourth row). You enter cell ranges separated by a colon: A1:B27 for the range of columns A and B from the 1st to the 27th row.

⚠️   Numbers in formulas must be entered with a decimal point; the decimal comma is not permissible.

Action

This tab allows you to specify a script procedure if your project contains a VBScript. The tab is only accessible if the cell is formatted as button. The procedure is executed when clicking on the button. Please read for details.

Formulas available for the Flex table

Parameters written in angled brackets (<>) have to be replaced. Also delete the angled brackets when copying a formula from this help. You may write the formulae with small or capital letters.

☞   In several formulae you can specify a range instead of a single value, e.g. AVERAGE(A1:B25) or MIN(A3:B9).

ABS(<value>)
Calculates the absolute value.

ACOS(<value>)
Calculates the angle for the specified cosine.

ACOSH(<value>)
Calculates the inverse hyperbolic cosine.

ADD(<value1>, <value2>)
Adds two values.

AND(<bool1>, <bool2>, …)

Returns 1 if all arguments are true, otherwise 0. Up to 30 arguments are possible.

ASIN(<value>)

Calculates the angle for the specified sine.

ASINH(<value>)

Calculates the inverse hyperbolic sine.

ATAN(<value>)

Calculates the angle for the specified tangent.

ATAN2(<x-value>, <y-value>)

Calculates the angle to the specified tangent taking the arithmetic sign into account.

ATANH(<value>)

Calculates the inverse hyperbolic tangent.

AVERAGE(<value1>, <value2>, …)

Calculates the mean of the arguments. Up to 30 arguments are possible.

CEILING(<value1>, <value2>)

Rounds a number to the nearest largest multiple of <value2>. CEILING(4.65, 2) produces 6, CEILING(-2.78, -1) produces -3.

CHAR(<value>)

Returns the character which corresponds to the numerical value passed in the ANSI code.

CLEAN(<text>)

Removes all non-printable characters from the passed text.

CODE(<text>)

Returns the ANSI code of the first character of the passed text.

COMBIN(<value1>, <value2>)

Calculates the number of possible combinations (<value1> over <value2>)

CONCATENATE(<text1>, <text2>, …)

Links the passed texts. Up to 30 arguments are possible.

COS(<value>)

Calculates the cosine of the specified angle.

COSH(<value>)

Calculates the hyperbolic cosine of the specified angle.

DATE(<year>, <month>, <day>)

Calculates the date in the serial date/time format (Windows format).

DAY(<Date>

Calculates the day of the month for the specified date.

DEGREES(<value>)

Calculates the angle for the specified radian value.

EVEN(<value>)

Rounds to the nearest even integer (INTEGER). EVEN(5) returns 6, EVEN (-2.5) returns -4.

EXACT(<text1>, <text2>)

Compares two texts and returns 1 if they are identical, otherwise 0. Uppercase/lowercase is not differentiated.

EXP(<value>)

Calculates e^{value}.

FIND(<text1>, <text2>, <value>)

Seeks <text1> in <text2> starting at the position <value>. Returns the start position or 0.

FLOOR(<value1>, <value2>)

Rounds a number to the nearest smallest multiple of <<value2>>. FLOOR(4.65, 2) produces 4, FLOOR(-2.78, -1) produces -2.

HOUR(<<time>>)

Returns the hour for the specified time.

IF(<Condition>, <value1>, <value2>)

Evaluates the condition and returns <value1> if the condition is true, otherwise <value2>.

INT(<value>)

Rounds a number to the nearest integer.

INVERSE(<value>)

Calculates the inverse of the specified number (1/<value>).

ISBLANK(<cell>)

Returns 1 if the cell is blank, otherwise 0. ISBLANK(B1) produces 1, when B1 is blank.

ISEVEN(<value>)

Returns 1 when the specified value is an even number, otherwise 0.

ISNONTEXT(<value>)

Returns 1 when the specified value is not a text, otherwise 0.

ISNUMBER(<value>)

Returns 1 when the specified value is a number, otherwise 0.

ISODD(<value>)

Returns 1 when the specified value is an odd number, otherwise 0.

ISREF(<value>)

Returns 1 when the specified value is a cell reference, otherwise 0.

ISTEXT(<value>)

Returns 1 when the specified value is a text, otherwise 0.

LEFT(<text>, <value>)

Returns the first <value> characters of <text>.

LEN(<text>)

Returns the number of characters in <text>.

LN(<value>)

Calculates the natural logarithm of <value>.

LOG(<value1>, <value2>)

Calculates the logarithm of <value1> to the base of <value2>.

LOG10(<value>)

Calculates the decadal logarithm of <value> (logarithm to the base 10).

LOWER(<text>)

Converts text to lowercase letters.

MAX(<value1>, <value2>, …)

Calculates the maximum of the arguments. Up to 30 arguments are possible.

MID(<text>, <value1>, <value2>)

Returns <value2> characters from the position <value1> in the <text>.

MIN(<value1>, <value2>, …

Calculates the minimum of the arguments. Up to 30 arguments are possible.

MINUTE(<time>)

Returns the minute for the specified time.

MOD(<value1>, <value2>)

Carries out a modulo division. MOD(255, 16) produces 15.

MONTH(<date>)

Returns the month for the specified date.

NEG(<value>)

Inverts the arithmetic sign of the argument

NOT(<value>)

Inverts the logical value of the argument.

NOW()

Returns the date and time in the serial date/time format (Windows format).

ODD(<value>)

Rounds to the nearest odd integer. ODD(4) returns 5, ODD(-2.5) returns -3.

OR(<value1>, <value2>, …)

Returns 1 if one of the arguments is true, otherwise 0. Up to 30 arguments are possible.

PI()

Returns the number pi.

POWER(<value1>, <value2>)

Calculates $<value1>^{<value2>}$.

PRODUCT(<value1>, <value2>, …)

Calculates the product of all arguments. Up to 30 arguments are possible.

PROPER(<text>)

Converts all first letters of the words in the text to uppercase.

RADIANS(<value>)

Calculates the radian value of the specified angle.

RAND()

Produces a random number between 0 and 1.

REPLACE(<text1>, <value1>, <value2>, <text2>)

Replaces <value2> characters with <text2> in <text1> from position <value1>.

REPT(<text1>, <value>)

Produces a text with <value> repetitions of <text>.

RIGHT(<text>, <value>)

Returns the last <value> characters of <text>.

ROUNDDOWN(<value1>, <value2>)

Rounds <value1> down to <value2> places.

ROUNDUP(<value1>, <value2>)

Rounds <value1> up to <value2> places.

ROUND(<value1>, <value2>)

Rounds <value1> to <value2> places.

SECOND(<time>)

Returns the second for the specified time.

SIGN(<value>)

Returns the arithmetic sign of <value>.

SIN(<value>)

Calculates the sine of the specified angle.

SINH(<value>)

Calculates the hyperbolic sine of the specified angle.

SQRT(<value>)

Calculates the (positive) square root.

SQUARE(<value>)

Calculates the square.

STDEV(<value1>, <value2>, …)

Calculates the standard deviation of the arguments. Up to 30 arguments are possible. n (n-1) is used.

STDEVP(<value1>, <value2>, …)

Calculates the standard deviation of the arguments. Up to 30 arguments are possible. n is used.

SUBSTITUTE(<text1>, <text2>, <text3>[, <value>])

Replaces <text2> with <text3> in <text1>. If the text occurs more than once, you can specify with <value> which occurrence is to be replaced. If this is not specified, all occurrences are replaced.

SUM(<value1>, <value2>, …)

Calculates the sum of the arguments. Up to 30 arguments are possible.

SUMSQ(<value1>, <value2>, …)

Calculates the sum of square roots of the arguments. Up to 30 arguments are possible.

TAN(<value>)

Calculates the tangent of the specified angle.

TANH(<value>)

Calculates the hyperbolic tangent of the specified angle.

TIME(<hour>, <minute>, <second>)

Returns the time in the serial date/time format (Windows format).

TODAY()

Returns the date and time in the serial date/time format (Windows format).

TRIM(<text>)

Removes all multiple and leading space characters as well as any space characters at the end of the text.

UPPER(<text>)

Converts text to uppercase letters.

VAR(<value1>, <value2>, …)

Calculates the variance of the arguments. Up to 30 arguments are possible. n(n-1) is used.

VARP(<value1>, <value2>, …)

Calculates the variance of the arguments. Up to 30 arguments are possible. n is used.

WEEKDAY(\<date\>, \<value\>)

Returns the weekday for the specified date. With \<value\> you specify how the week is to be defined: 1 = from Sunday (1) to Saturday (7), 2 = from Monday (1) to Sunday (7), 3 = from Monday (0) to Sunday (6).

XROOT(\<value1\>, \<value2\>)

Calculates $\langle value1 \rangle^{1/\langle value2 \rangle}$

YEAR(\<date\>)

Returns the year for the specified date.

2.9.2.16 Load level visualization

☞ Create a Load level visualization from the **Panel** ribbon tab and **Other displays** (**Indication** group).

Call the **Configuration** dialog with **Configure** from the context menu. The dialog consists of two tables. Drag from the **Channel selector** window the channels you want to monitor to the left one and specify the 100% value in units of this channel, e.g. **800** for a value of 800 µm/m. On the right table you specify the percentages you would like to change the color together with the color which shall be displayed, if this percentage is exceeded. The table must be filled in consecutive order with lowest (negative) values starting at the top, e.g. **-50, -25, 0, +25, +50**.

💡 You can also specify the 100% value in MC Setup in the **Max. prediction** column, see MC Setup column description, scaling functions on page 164.

2.10 Data acquisition (Test ribbon tab)

🛈 See also Measurement settings (DAQ settings) on page 56, Configure storage size on page 35.

☞ If you plan to save data, please check your storage size settings before starting a measurement.

Before starting the data acquisition, you have to initialize the system: click on ⊞ **Initialize**.

⚠ Do not connect or re-connect a sensor with TEDS to a QuantumX/SomatXR module during a measurement. The data acquisition in this channel is stopped until

the TEDS settings are read out and the channel is adjusted.

2.10.1 Principle measurement procedure

After initialization you may **Start** (▶), **Pause** (❚❚) or **Stop** (■) the measurement. If you stop the measurement, all data in the internal database is stored. A *new test name* will be used for the next test run (default setting in the **Server options** dialog, **General options** tab, see General options on page 39.

Use pause, if you want to stop the measurement temporarily. In this case the icon

changes to ❚▶ **Resume** and data is appended if you continue the data acquisition. However, in pause state it is not possible to change any setup data or the channel configuration.

☞   If errors occur during data acquisition, e.g. a defective channel, you will find a descriptive entry in the MD Server project log on page 130. Channels which report hardware errors are deactivated automatically.

Visualization

During the measurement all data measured is displayed in the visualization objects defined and is available for the client PCs. In **Fast trigger mode** (two sample rates) data from channels marked as **Fast channel** is not forwarded to client PCs during the post-trigger time (see trigger behavior below). The data is however displayed in the graphs on the server PC.

Triggers

If a trigger condition occurs, the specified action is taken.

☞   During the execution of a trigger action, no other trigger events are monitored.

If fast measurement is specified as trigger condition (**Fast trigger mode**), then all trigger data is stored to separate files for each trigger event (and displayed in real-time objects on the server PC only). During this time no data is forwarded to client PCs. Only after the post-trigger time as much data as fits into the client-server communication buffers is forwarded to client PCs, however not displayed in graphs. This data is only stored. With MGCplus the **DAQ status** window will show an increasing number of values in the data buffers of the participating devices (the pre-trigger values). If the trigger condition is met again during the post-trigger time, no action is taken (no re-trigger).

The **Trigger status** window allows you to watch the trigger conditions. If necessary, you can even change, disable or enable a trigger in this window, see .

Storage

Values are stored to the internal database if either taking snapshots or if channels have been marked for storage *and* storage of data has been activated. Except for snapshots a storage of data appears only, if you additionally either click **On** in the **Storage** group or activate the storage with a trigger condition (**Storage on/off** in the **Snapshot** column). See also .

Status information

As a rule, the **DAQ status** window is displayed during the measurement. However, you may close this window. Additionally the **Trigger status** window and the **Limit checking** window may be displayed, allowing you to watch triggers and limits.
See also .

2.10.2 Available Test actions (after initialization)

🛈   See also .

⚠️   At least one active channel of all devices must be in **Rate 1** if **Regular DAQ mode** is used and snapshots shall be taken.

☞   Apart from zeroing most actions are only available after starting a measurement.

🟢**On**/▶️**Off** (**Storage** group): activate this option and data from all channels marked in the **Store channel** column of the DAQ settings is saved to the database.

Zero measurement: performs a zeroing on all channels *not excluded* from this operation in MC Setup (column **Zero enable** set to **off**). All zero measurement values are stored to a separate file ZERO.BIN in the folder of the test. The last 10 zero values per channel are also saved in the MC Setup database file. After the measurement a check is made, whether the MC Setup database is derived from a MC Definition List, see also . If so, the (last) zero values are written back to the

corresponding channels of that MC Definition List. This ensures that the MC Definition List always has the newest zero values.

Snapshot: stores the actual value to the internal database. Requires enough space in this internal database, see Configure storage size on page 35.

Continuous snapshots: stores the actual and *all further values* acquired with the slow sample rate to the internal database.

☞ Continuous snapshots are only possible in **Fast trigger mode**, see Measurement settings (DAQ settings) on page 56.

Trigger fast sample rate: a fast data acquisition is triggered manually. This option is not available in **Regular DAQ mode**.

The values given in the trigger dialog for pre- and post-trigger apply here as well. All channels marked in the **Fast channel** column of the DAQ settings are stored to a file in subfolder TRIGGER.

Enter manual input is necessary for specifying values, e.g. a certain load level, *manually* before doing a snapshot.

⚠ This menu item is only available if a manual input channel has been defined and the measurement has been started.
See also Manual input channel on page 191.

The dialog shows all manual input channels defined. Type in the appropriate value(s) and click on **Snapshot**.

☞ In former versions of catman Enterprise Manual input was named *Load level input*.

Use ![icon] ![icon] or ![icon] to zoom into a graph (horizontal, vertical or rectangular zoom). Click on a graph to display the zoom option, choose a zoom function and then click with the left-hand mouse key inside the graph and drag to indicate the zoom sec-

tion. Click on the symbol again to return to the original state. ![icon] will take you back to the last zoom states step by step. The **Zoom** group is only visible in graphs where zooming is possible.

2.10.3 DAQ status window

The **DAQ status** window is activated after the initialization but may be closed at any time, if you do not need it. If it is closed, open it from the **Test** ribbon tab with **DAQ status**.

2.10.3.1 General tab (DAQ status window)

☞   See also .

This tab shows some project settings, the sample rates used and the number of active and storage channels together with the channels in overflow state. Important for data acquisition is the field **Data transfer block size** which indicates how big the blocks transferred (read blocks) are and the **Database filled up to** value. If this value reaches 100%, the measurement is stopped. The free space on the disk used for storage is also indicated (**Data disk filled up to**).
See also .

With the exception of fast trigger measurements, the read block size shows whether the reading-in speed of the server PC used is higher than the speed of data acquisition in the measuring devices. If not, the internal buffers in the devices fill up and the measurement stops. This would be indicated with a growing block size in field **Data transfer block size**.

The NTP or PTP synchronization quality is displayed as LED if NTP or PTP time is used and **Enable NTP sync monitoring** is activated in the **Server options** dialog (use **Server options** on the **Setup** ribbon tab before starting a test). Depending on the settings, the LED may be green, yellow or red, see .

Read block size/transfer block size
Data is retrieved from the signal conditioning devices in *read blocks*. The size of these read blocks is determined by the software. It depends on the sample rate used and is shown in the **DAQ status** window in field **Data transfer block size**. With low sample rates it is 1 and increases for high sample rates. The read-block data is used for updating visualization panels and may be accessed by client PCs. If continuous data storage is activated, these data are stored to the internal database.

⚠ If values cannot be read in from a device within the time specified as **Read timeout**, an error message is issued and the device is excluded from the current test (the data acquisition from this device stops).

If the read-block size increases during the test run, this points to a hardware problem: the transmission rate is too low. If the value exceeds the internal buffer size of the devices participating in the test, the device is excluded from the current test (the data acquisition from this device stops).

2.10.3.2 Device status tab (DAQ status window)

ⓘ See also Measurement settings (DAQ settings) on page 56, General options on page 39 in Server options, Hybrid systems options on page 45.

You may watch details for the data transfer on this tab, which displays the status and contents of the device buffers. As long as a green background is displayed in the **Slow/Fast DAQ** fields during measurement, the recording is error free. After initialization a yellow background and **Ready** is displayed.

Information displayed in Fast trigger mode

The **Slow/Fast DAQ** fields may be in stopped mode (red background) if either no fast data acquisition is specified for one of the channels contained in this device (**Fast DAQ**) or if no channel contained in this device takes part in the test run (**Slow DAQ**).

QuantumX/SomatXR

Under normal measurement conditions the **Slow/Fast data** fields show the number of values in the device buffers. If the buffer increases, the data acquisition speed in the QuantumX modules is higher than the speed of the data processing of the server PC. In this case the internal device buffers in the QuantumX modules will fill up and the measurement eventually stops.

If a **Fast channel** has been defined, the number displayed in **Slow data** starts to increase as this buffer is temporarily not read out. The **Slow data** buffer is read out only *after* the post-trigger time and shows then again the number of data in the device buffer.

MGCplus

Under normal measurement conditions and if a **Fast channel** has been defined, the number displayed in **Fast data** is counted up after measurement start until the buffer is filled with the pre-trigger data. The number to be reached depends therefore on the pre-trigger time. As soon as a trigger condition for using the fast sample rate occurs, this value

decreases as the data is read out and transferred to the server PC. After having transferred the pre-trigger data, the number of data in the device buffer is displayed again. At the same time the **Slow data** number starts to increase as this buffer is temporarily not read out. The **Slow data** buffer is read out again after the post-trigger time and shows then again the number of data in the device buffer.

If the number in the Slow data buffer increases, the data acquisition speed in the measuring devices is higher than the speed of the data processing of the server PC. In this case the internal buffer in the device will fill up and the measurement eventually stops.

Information displayed in Regular DAQ mode

The buffer sizes for all three sample rates of all devices are displayed. If one of these buffers increases, the data acquisition speed in the measuring devices is higher than the speed of the data processing of the server PC. In this case the internal device buffers in the devices will fill up and the measurement eventually stops. The **DAQ status** field may be in stopped mode if no channel contained in this device takes part in the test run.

Information displayed in both cases for external devices

For external devices the input buffer *read pointer* is displayed. This shows where data with the same time stamp as the MGCplus or QuantumX/SomatXR data is read from. If these values are close to 1, the external devices deliver their values quite late compared to the HBM devices. You may increase the **HBM devices output delay** in the **Server options** dialog, **Hybrid systems** tab (**Setup** ribbon tab). On the other hand, if the value is close to the **FIFO buffer size** (**Hybrid systems** tab), the delay is already too long, decrease the **HBM devices output delay** to get a better match.

See also Hybrid systems options on page 45.

2.10.3.3 Sync monitor tab (DAQ status window)

☞ The tab is only accessible if **Enable MGCplus SYNC monitoring** is active on the **General options** tab in the **Server options** dialog (**Setup** ribbon tab) and MGCplus devices with CP42 and firmware 4.32 or higher or with CP52 are used. The tab contains information for MGCplus devices only (no QuantumX/SomatXR or other devices).

This tab shows status information about the process used to synchronize several MGCplus devices via SYNC OUT and SYNC IN on the CP42/CP52 plug-in modules. The background of the individual fields changes to yellow for minor errors and eventually to red, if severe errors occur. Each color change is also entered into the log file of MD

Server, refer to MD Server project log on page 130. This enables you to find time sections with faulty synchronization. However, the measurement data is no longer acquired synchronously during this time.

💡 Use a MGCplus SYNC lost trigger to be able to react immediately to synchronization errors, see MGCplus SYNC lost trigger on page 67.

Check the synchronization cables and connections of the relevant devices and the termination plugs, if a severe error occurs. It may be necessary to power the complete system off and on again (reboot) to initiate a new synchronization procedure. Please note down, which fields show what color and contact the HBM Technical support, see Technical support on page 29, if severe errors still appear.

2.10.3.4 Power supply tab (DAQ status window)

🛈 See also General options on page 39 in Server options.

This tab shows you the voltage, current and temperature values for all MGCplus devices if **Enable MGCplus power supply monitoring** is active on the **General options** tab in the **Server options** dialog (**Setup** ribbon tab).

Use this display to monitor the "normal" conditions over several 10 minutes if you want to use a MGCplus voltage or current or temperature trigger. Check for the maximum value of all devices here which value is best suited as trigger level.

Refer also to MGCplus voltage/current/temperature triggers on page 66.

2.10.4 Trigger status window

The **Trigger status** window shows the trigger channels, whether they are enabled and how often a trigger occurred.

Currently active triggers are displayed with a yellow background color. It is possible to **Show active triggers only**.

Use **Threshold** and 📝 to change the threshold value for all selected triggers. ❌ disables and ✅ enables all selected triggers.

2.10.5 Recover procedure

If during a test a mains power failure occurs, this is detected by MD Server on restart as the final entry for the acquisition is missing in the SERVERLOG.MDB (project folder, refer

to). In this case the program tries to recover everything as far as possible and data not yet stored is exported.

2.11 MD Server tools

🛈 See also .

☞ Some items on the **Tools/Script** ribbon tab are only accessible if the system has been initialized.

Licenses group

As default, the licenses required for operating the MGCplus or QuantumX/SomatXR signal conditioning devices are stored on the PC running MD Server. **Transfer MGCplus and MX licenses** allows you to store the licenses in the devices itself. This facilitates operation with changing device configurations and more than one MD Server license. **Retrieve MGCplus and MX licenses** transfers the licenses back to the PC running MD Server. See .

Channels/Export group

The entries in this section provide you with different options for exporting data.
See .

Special group

Set synchronization settings allows you to define the IP address of an NTP time server or PTP or IRIG B settings for a device.
See .

Set TCP/IP settings (MGCplus with CP42/CP52 only) allows to change the CP42/CP52 Ethernet interface parameters to the recommended settings (e.g. no auto negotiation). If "TCP/IP settings on all devices are correct" is shown in the dialog, the setting has been performed already.

MOOG CS options allow for up to 64 catman Enterprise channels to be output to the MOOG system or a MOOG command channel (digital inputs and outputs) to be used.
See .

Open catmanAP project starts catmanAP in Analysis mode. This allows you to further evaluate your test data. Refer to .

The **Script control** options refer to VBScript which allows you to extend the functionality of catman Enterprise, see .

2.11.1 Transfer/retrieve MGCplus and MX licenses

☞ With QuantumX it is possible to transfer licenses to modules of type B or C only, e.g. MX840B, MX1601B or MX471C, or to SomatXR modules.

Usually the required licenses for HBM devices are handled by the MD Server software and therefore transferred to the PC running the MD Server software. However, the licenses may also be transferred permanently to the MGCplus devices or QuantumX/SomatXR modules. This facilitates operation with changing device configurations and more than one MD Server license. The transfer is done with the MD Server software, i.e. the required licenses must first be transferred from the MD License manager to an MD Server instance.

Procedure for transferring licenses to MGCplus and QuantumX/SomatXR

1. Transfer the required licenses for all devices to a PC which has program MD Server installed.
2. Connect all relevant devices to this PC.
   ☞ It is not possible to select which devices receive a license. *All* connected devices will get a license if enough licenses are available.
3. Start program MD Server and scan the new system (**New scan** on the **Setup** ribbon tab).
4. Download the licenses to the connected devices with **Transfer MGCplus and MX licenses** on the **Tools/Script** tab.

If more licenses are available than devices connected, these are retained on the PC running MD Server, each device or module gets only one license.

If more devices or modules are connected than licenses are available, the devices or modules with the highest IP addresses will get no license. Please note that no data acquisition is started if the required licenses are not found.

☞ You may specify that every time the measurement is initialized, any available licenses are distributed to the connected devices, if required.

Procedure for transferring licenses back to MD Server (retrieving licenses from devices)

1. Connect all devices where the license shall be retrieved to the PC from which the licenses were transferred.

   ☞ It is not possible to select from which devices a license is retrieved. The licenses from all connected devices will be removed.

2. Start MD Server and scan the new system (**New scan** on the **Setup** ribbon tab).
3. Retrieve the licenses from the connected devices with **Retrieve MGCplus and MX licenses** on the **Tools/Script** tab.
4. You may then use the MD License client to transfer these licenses back to the MD License manager.

If a device or module connected has no license, nothing is retrieved. Only the number of licenses which has been transferred from this MD Server installation may be retrieved. If more licenses are found than transferred originally, these surplus licenses will remain in the devices or modules.

2.11.2 Channels/Export

The ⊞ **Database channel manager** shows for all channels, how many values per channel have been acquired and how much space is still available. The window is intended for service and error tracking purposes. You may export selected channels also from here into formats catman and Excel, if Excel is installed. However, 🖥 **MD Export** in this group offers more options for exporting data. Click on 📝 to open the **Database editor** window. This window displays all values in the database and allows you to change individual values. Additionally min. and max. values for each channel are displayed.

🖥 **MD Export** (also available as a separate program) is used to extract data from several tests of a project and export it into various other formats. Open the project from which you want to export with 📁 or **Project -> Open project**.
See also .

🔖 **Data export** calls up the export dialog of MD Export. However, the channels used for the last export are then used again. If you need to change that setting, start with 🖥 **MD Export** first and call the dialog from within MD Export.

2.11.2.1 Building a channel list

How to proceed

- Mark in the upper part of the window either the Devices entry of a test or a single device. This displays the relevant channels in the lower part of the window.
- Open the **Channel list** window with 📄 or **Channel -> Channel list**.
- Mark the required channel(s) from your project (lower part of the window) and drag these to the **Channel list** window.

  📑 allows you to search for specific channel names using part of the name: Use **%** or ***** as wild cards for several characters and **_** or **?** as wild cards for single characters. You may use wild cards more than once in the search string as in **%nnel??04%**. Without any wild card character, an exact match is searched. Check the Search result section for channels found.

- You may also import channels contained in a MC Definition List (🔴), a MC Setup database file (🔴) or an Excel file (🔴).
- Use 🔺 to include the information about zero values acquired either for storage channels (Zero_Tag slow DAQ) or for snapshot channels (TagZero). However, the channels contain only references. The zero values themselves are stored in file ZERO.BIN in the MD Server project. The number shown here is the index under which the zero value and time point can be found in file ZERO.BIN.
- Use 🔺 to include time channels.
- If more channels are added by these functions than required, mark and delete unwanted channels with 🖊.
- You may rearrange channels: mark the required channels in the left column (number column), then use the context menu to cut and paste these channels. A red dotted line shows the insertion position.

- ☞ Close the **Channel list** window with **Accept** if all required channels are selected.

2.11.2.2 Available export options

Export "Stored data" into one file: If the data has been saved to multiple files by MD Server (one file per channel) you may use this option to again combine all files into one. Otherwise one file per channel is saved.

Concatenate split test files: This option saves all data into one file if several tests have been made producing several data files.

Export empty channels: Even channels in overflow are exported if this option is activated. The default setting is not to export these channels. Overflow values can be defined for non-HBM devices and HBM devices (e.g. MGCplus, QuantumX) separately, see Null and Overflow values on page 247.

Log missing channel names: All channels are listed which could not be exported because they were not there (available in this project). This might e.g. happen if you import a channel list from an older project.

2.11.2.3 File formats and format options

☞   We recommend to use **catman** or **ASCII + channel information** as format, because here all traceability data is stored as well. The same applies to **Microsoft Excel**, if **Include channel information** is activated on the **Excel** tab.

For most data formats you can choose which type of data is to be included and how **Null values** and **Overflow values** shall be treated: Use the **General** tab to define your settings. A **File comment** will be used in the catman and ASCII formats only.

If other than the default file names (MD Server file names) for the respective export data (Stored data, Snapshot data etc.) are required, specify these on the **General** tab.

Null and Overflow values

A **Null value** is a user defined overflow value for a non-HBM device. For example, if a MOOG channel sends -55555 as overflow, use this field to define that value.

An **Overflow value** is for HBM devices like MGCplus or QuantumX. If an HBM device signals an overflow, the value specified here is stored in the database. This makes it easier to find those values because they do not depend on scaling and resulting maximum amplifier output.

In default channels with these values are not exported. Activate **Export empty channels** in the **Data export** dialog to include those channels.

The (binary) catman standard format (∗.BIN)

Files in the **catman standard format** can only be read in again by catman itself. However, the speed of the export is substantially higher than in other formats. In addition the channel information (acquisition parameters of a channel) are also saved.

ASCII and ASCII + channel information (∗.ASC)

⚠️ When exporting values in one of these formats, the currently set number formatting for each channel in the *Database editor view* is used. The default format is with three decimal places. For very small values, this could lead to a loss of data. Increase the number of post decimal places in this case. A time channel that contains absolute time values (serial date-time format) must have a date-time formatting. A change can only be carried out in the **Database editor**.

Example

With a sample rate of 9600 measurements per second, then in the setting with three decimal places, the first five values for the time channel are saved with 0.000 because the value is less than 0.0005. From the sixth value there is then 0.001, etc. Either use more decimal places or one of the scientific formats with an exponent.

Data exported in this form is present as ASCII text file and can be read directly by numerous other programs. The data of a channel is located in a column with the columns being separated from one another by a freely selected **ASCII separator,** e.g. ⌨ (tab key) or semicolon. Each row is terminated by CRLF (Carriage Return/Linefeed, standard for DOS and Windows). The first rows of the file contain, if required (**+ channel information**), acquisition parameters (traceability data).

⚠️ The traceability data (channel information) can only be restored from a file saved in format **ASCII + channel information**, if the *parameter IDs* have been saved additionally (appended).

☞ Please note that export in the ASCII format is slower than the binary export due to the formatting processes.

MS Excel (∗.XLS, ∗.XLSB)

⚠️ Export in this format is only possible if MS Excel is installed.

Files in this format can be directly read by MS Excel. When exporting, you may use several options, just click on the **Excel** tab. The channel information contains the traceability data. If these are transferred, it is useful to set the start row for values to 35 or higher,

because not every channel might have the same amount of traceability information, e.g. the time channel has no scaling etc.

☞ The entry for **Data starts at...** is always evaluated, even if no channel information is transferred.
If data is stored with **Channels into rows**, the corresponding number of columns is used. If more than 256 columns are necessary, the export routine creates the required number of worksheets and uses those for consecutive columns.

⚠️ With MS Excel 97 to 2003 the export is restricted to 65,000 values, because these worksheets cannot have more than 65,536 rows. If more data is present in the channel, they are ignored or the import is terminated with an error message. For MS Excel 2007 or higher (default setting) even 1,000,000 values are possible. Make sure the format **Excel workbook (∗.xlsx)** or **Excel binary workbook (∗.xlsb)** is the default setting for saving in Excel in this case. In both cases a binary workbook is saved (extension xlsb).

Further information can be found at https://www.microsoft.com/.

NI DIAdem (∗.DAT)

Files exported in this format can be directly read in by the data analysis software DIAdem® from National Instruments Engineering GmbH & Co KG. A header file and a data file are produced when exporting data in this format. The header file can be directly interpreted by DIAdem® and the data file is in the format REAL64.

☞ You may import files in this format into catman Professional again.

Further information can be found at https://www.ni.com/.

FlexPro (∗.FPD)

Files in this format can be directly processed by the data analysis software FlexPro®.
Further information can be found at https://www.weisang.com/.

CAEMAX RemusLAB (∗.RMS)

Files in this format can be directly processed by the data analysis software CAEMAX Remus.
Further information on CAEMAX can be found at https://www.caemax.de/en/.

MTS RPC III (∗.RSP)

Files in this format can be processed directly from the data analysis software RPC$^®$ III from the MTS$^®$ Systems Corporation. As the format does not allow individual channel lengths, so-called groups each with 2048 values are always saved. If the channels to be exported do not contain a multiple of this value, the missing values will be filled with zeroes. You should therefore try to acquire or export a multiple of 2048 values. As integer values with 16 Bit must always be saved in this format, catman Enterprise uses the minimum and maximum of the respective channels and scales all values accordingly.

Further information on MTS Systems Corporation can be found at https://www.mts.com/.

Vector MDF 3 (∗.MDF)

MDF (**M**easurement **D**ata **F**ormat) is a binary file format for measurement data which was developed in 1991 by Vector together with Robert Bosch GmbH. In the meantime, the MDF format is widely used for measurement data in the automotive industry. Refer also to MDF 4.

Further information on Vector Informatik GmbH can be found at https://www.vector.com/.

ASAM MDF 4 (MDF 4.1, ∗.MDF)

MDF 4 (**M**easurement **D**ata **F**ormat) is a further development of the MDF 3 format and in this version is an ASAM standard (Association for Standardization of Automation and Measuring Systems). With this development there is no 2 GB restriction as with MBF 3 (theoretically 2^{64} bytes, i.e. 10^{10} TB per file can now be written). In addition the format has far-reaching possibilities of saving metadata. Also, special functions of MDF 4.1, such as data compression or Quick Preview, are not supported by catman Enterprise.

Further information on ASAM can be found at https://www.asam.net/.

nSoft (∗.DAC, ∗.MDF)

catman Enterprise supports the export of Database channels for the evaluation and analysis package nSoft$^®$ from nCode. catman Enterprise can produce two different nSoft$^®$ formats:

Time series format (∗.DAC)

An equidistant measurement series is written to a file, i.e. one Database channel produces one file. The file name is created with the name provided by you, the channel name (appended with an underscore) and the file extension DAC.

X/Y pairs format (∗.MDF)

Two channels are written to one file, one as the x-axis, e.g. displacement, and the other as the y-axis, e.g. force. The file extension is MDF.

Further information on nSoft and nCode can be found at https://www.ncode.com.

EDASWin

Files in this format can be directly processed by the data analysis software EDAS Win®.

Further information can be found under https://www.mh-gmbh.de/.

nCode s3t

Files in this format can be directly processed by the software GlyphWorks and nCode DesignLife.

GlyphWorks is a software program for the analysis of test data; it processes a large amount of data and offers a graphical, process-orientated user interface. You can create a workflow for the analysis using drag & drop of analysis modules.

nCode DesignLife is a program for lifetime analysis. nCode DesignLife uses FEM results and operational strength tests for the lifetime assessment.

Further information on HBM's nCode software can be found at https://www.hb-mprenscia.com/.

2.11.2.4 Save/Load export layout

Save your export settings if you want to use the same or similar settings again: **Export -> Save export layout**. The Channel list is saved together with all export settings. You may still modify settings after loading an export layout (📥 or **Export -> Load export layout**).

The last used settings are always retained for the next export as *LastAutoExportlayout* and are also used for menu item 📥 **Data export** in MD Server.

2.11.3 Configure and check synchronization

🛈 See also

☞ **Set synchronization settings** (**Tools/Script** ribbon tab) is only accessible if an initialization has been made. Synchronization via NTP, PTP or IRIG B is only

necessary if no hardware synchronization is possible, e.g. with distributed devices, or when having external devices.

Specify in this dialog the type of time synchronization and either the IP address of the NTP time server (MD Server PC) or the PTP or IRIG B time settings which may then be distributed to the connected devices. An IRIG B time source may be selected only with some device types. If a time source has already been set for a device, the setting (e.g. address) is displayed in the **External time source** column.

💡 Check deviations with **Run check**. The **Offset** column then shows the current deviation. With MGCplus only the trigger leader is relevant, with QuantumX/SomatXR only Sync-leader modules are relevant, e.g. CX27. The offset entry for all other devices is **0** and **Follower** is displayed in the **Internal synchronization** column.

Time format

Depending on the hardware, either **NTP**, **PTP** or **IRIG B** may be used. When using IRIG B or PTP synchronization, the time is recorded in NTP format and the settings for NTP are applied as far as possible (buffer size and time matching). NTP (and PTP or IRIG-B) time channels are listed as *Device time stamps MGC* or *Device time stamps QX*, e.g. in the **Time channels** list.

⚠ Only one type of time format should be used, mixing NTP and IRIG B formats is not recommended. If you want to use it anyway, make sure the IRIG B channel is not in the trigger leader MGCplus (first device).

You may also provide external devices with the IP address of the PC running MD Server, if an NTP time service is running on this PC. However, in order to be able to use the service, the external devices driver must have the appropriate information as well (the drivers.inf file must have the TimeSyncMonitor entry). In case of doubt, please check with the HBM Technical support, see Technical support on page 29.

Procedure for NTP

1. Select all devices/modules which are marked as leader devices (Leader or Sync-leader, depending on the device type). Click on the leftmost column and use 🔼 or Ctrl keys.
2. Type in or select the **Server address**.

   ⚠ If your PC has several network cards, make sure you specify the address of the network card to which the MGCplus signal conditioning devices or Quan-

3. Click on **Apply settings**.
    After the distribution the time service is (re)started on all connected Quan-tumX/SomatXR or MGCplus devices.

☞ After the **first** configuration or changing the server IP address settling may take up to 2 hours so therefore allow the devices sufficient time. When switching on the devices the next time, allowing 5 minutes settling time is sufficient.

You may switch back to the standard synchronization procedure of the signal con-ditioning devices with **System clock**.

☞ We recommend you use a hardware based NTP time server, e.g. LANTIME M200 from Meinberg, which gives you excellent stability and better accuracy for the time server itself. If you prefer to use the (software) time server provided by cat-man Enterprise, set it up on the PC running MD Server as this ensures minimum time delay between the signal conditioning devices connected to the server. Unfor-tunately, with a software based time server a time difference between devices of more than one millisecond is possible. In case of doubt check with **Run check** (**Set synchronization settings** on the **Tools/Script** ribbon tab in MD Server).

Procedure for PTPv2

☞ You can only use the PTP time in conjunction with special Ethernet switches.

1. Select all devices/modules which are marked as leader devices (Leader or Sync-leader, depending on the device type). Click on the leftmost column and use ⟨⇧⟩ or ⟨Ctrl⟩ keys.
2. Activate **PTP** and specify the settings for the protocol if you are using a PTP Grand Master Clock. Otherwise, use the settings **Auto** and **UDP IPv4**. The setting for **IPv6 Multicast-Scope** is only interesting if you are using several PTP time servers. You can then select the time server to be used. Otherwise, leave the setting on **Link-local** (first or no special time server).
3. Click on **Apply settings**.
    The setting is transferred to the device(s) and the synchronization process is started. The modules synchronize themselves automatically; with a system without an external PTP time server the leader is also negotiated automatically. The deviations in module times with this method are less than 1 µs and no wait

time is necessary after switching on. The maximum distance between devices is 100 m for a connection via Ethernet cable.

MGCplus

With MGCplus the IRIG B time source is used only for the device it is connected to. The time values are not forwarded to any other devices.

⚠️ The IRIG B channel must not be used in the trigger leader MGCplus (first device).

1. Select the device with the special version of the ML01B plug-in card.
2. Activate **IRIG B**.
3. State the current year, if this information is not transferred with your IRIG B format.
4. State the difference of your time zone to the **UTC time zone** (derived from **C**oordinated **U**niversal **T**ime, i.e. universal time, corresponding to Greenwich Mean Time or London time), if you want to use local time. Also take into account any difference due to summer time.
5. If you know the time delay between the IRIG B time generator and your device and it is constant, you can correct it using the entry for **Time offset**.
6. Click on **Apply settings**.
   The setting is transferred to the device(s).
7. Assign the IRIG B *sensor* to the slot to which the IRIG time source is connected. The deviations in module times with this method are less than 1 µs and no wait time is necessary after switching on.

QuantumX/SomatXR

With QuantumX/SomatXR (only MX840 or MX440 A and B types) proceed as follows:

1. Select all modules which are marked as leader devices (Leader or Sync-leader). Click on the leftmost column and use ⬆ or [Ctrl] keys.
2. Activate **IRIG B**.
3. State the current year, if this information is not transferred with your IRIG B format.
4. State the difference of your time zone to the **UTC time zone** (derived from **C**oordinated **U**niversal **T**ime, i.e. universal time, corresponding to Greenwich Mean Time or London time), if you want to use local time. Also take into account any difference due to summer time.
5. If you know the time delay between the IRIG B time generator and your device and it is constant, you can correct it using the entry for **Time offset**.

6. Click on **Apply settings**.
    The setting is transferred to the device(s).
7. Assign the IRIG B *sensor* to the channel to which the IRIG time source is con-
    nected.
    The deviations in module times with this method are less than 1 µs and no wait
    time is necessary after switching on.

Notes

● Activate **Enable NTP sync monitoring** on the **General options** tab in the **Server
  options** dialog (click on **Server options** on the **Setup** ribbon tab) if an NTP/PTP
  sync failed trigger on page 66 shall be used.
● Please check with external devices whether additional settings are necessary,
  either in a (NTP) configuration file or in the configuration file of the respective
  device.

2.11.4 MOOG CS options

☞ The menu item is only accessible if a MOOG control system (MOOG CS) is avail-
able in the system.

This option also allows for up to 64 catman Enterprise channels to be output to the
MOOG system. Use **Configure channel output** to specify the channels which shall for-
ward their data to the selected MOOG device. The data is output after each read block,
i.e. with approximately 10 Hz.

Select in the dialog which type of connection is to be used:

1. Connection via Ethernet: **TCP/IP remote control system** (preferred connection)
    Specify a **FIFO buffer size** additionally in the **Server options** dialog, **Hybrid sys-
    tems** tab. This buffer is created on the MD Server PC and used to store the val-
    ues received from the MOOG control system. Together with the NTP time data
    from MOOG control system and from the signal conditioning devices all data is
    then synchronized.
    See Hybrid systems options on page 45, IRIG B sync mode on page 47.
2. Connection via a CAN bus module in a MGCplus signal conditioning device: **CAN
    remote control system**
    No buffer setting is necessary in this case as all buffering is done in the
    MGCplus. However, the CAN bus modules need a Vector database specifying all

settings for operation, see Configure CAN bus modules for remote control on page 117.

Required channels, naming conventions

The first two channels must be the NTP time data (DeviceName_NTP) and a 32-bit digital channel (DeviceName_COMMAND). All other channels appear with the names set up in the MOOG control system and cannot be changed in MD Server or MC Setup.

Please check the channel names in your MOOG CS system before connecting: channel names containing certain characters are not only not imported (and present), the import of further channels is also terminated. The following characters are not allowed: colon (**:**), period (**.**) and angled brackets (**< >**).

Enable MOOG command channel

With this option activated, 16 digital inputs (Watch for a digital signal state in the Command channel on page 64) and 16 digital outputs (Set digital out (DOut) on page 70) are available for a command channel. This channel may be used to exchange process information and to control the MD Server, see MOOG command channel on page 118.

Using a CAN bus module (MGCplus only) for the command channel

In this case the *first* channel of the *first* ML71B CAN bus module in the system (in whichever device) must be used as command channel and it is mandatory to disable zeroing for this channel (see MC Setup on page 137, Setting-up channels on page 157). The ML71 receives the messages with ID 000 (highest priority), the sending device must read message ID 001. The message itself consists of 2 bytes in Motorola format.

Using a TCP/IP remote control system for the command channel

Choose the command channel device, i.e. the device which will deliver the command channel. Up to four devices are displayed sorted by their Ethernet addresses.

2.11.4.1 Configure CAN bus modules for remote control

☞ This function may only be used, if a ML71B is available in one of the MGCplus and a MOOG control system is used. If additionally a Moog command channel shall be used, this must be the *first* channel of the *first* ML71 CAN bus module in the system, refer also to MOOG command channel on page 118. The devices are sorted according to their Ethernet addresses, so if in doubt use a device with a lower Ethernet address.

A separate CAN database may be loaded for each module in the **MOOG CS options** dialog, up to four modules are supported. The dialog displays one input box per ML71, with

⊡ a file dialog may be called up to specify file and path. The same CAN database can be loaded for more than one ML71. The files are project dependent.

After closing this dialog, the original CAN database files are copied to the test subfolder of the project. During initialization (see Data acquisition (Test ribbon tab) on page 96) a check is made whether the original files specified and the copies are still identical. If not, the original files are copied again and then transferred to the ML71. There is no transfer as long as the files are unchanged.

⚠ The download of a new CAN database may change the channel configuration (more or less channels available). In this case it is mandatory to perform a re-scan, which is asked for automatically when MD Server detects the changed hardware. If done by MD Server, all settings are retained as far as possible and mapped to the new hardware structure automatically.

☞ A CAN database may, for example, be created with the CAN database editor from Vector Informatik GmbH. The MGCplus Operating Manual contains detailed information about CAN and ML71.
See also https://www.vector.com/ for further information on Vector Informatik GmbH.

2.11.4.2 MOOG command channel

A MOOG command channel enables you to initiate actions on MD Server from your control system and to transfer certain information from MD Server back to the control system. A MOOG control system (MOOG CS) is required, otherwise the feature cannot be used. You may however use *one* of the following devices for the MOOG command channel, if a MOOG control system is connected via Ethernet:

1. The MOOG control system (preferred method).
2. A remote control system, which is sending/receiving the data via the *first* ML71 CAN bus module found in the MGCplus signal conditioning devices.
3. A remote control system for which a DLL Driver exists and which is connected via Ethernet. The Command channel must be the second channel (32 bits) sent by the device, the first channel must be a time channel in either NTP or IRIG B time format. IRIG B is only possible if either MGCplus or QuantumX/SomatXR devices with IRIG B time are used in the system.

☞ The documentation on DLL DAQ Drivers ("Drivers/DLLDriver/DAQ drivers.doc" in the catman Enterprise installation folder) shows how to write such a driver in a high-level programming language.

Actions you may start via a MOOG command channel

- a snapshot, see Available Test actions (after initialization) on page 98 (topic Data acquisition (Test ribbon tab) on page 96).
- the continuous snapshot mode, see Available Test actions (after initialization) on page 98 (topic Data acquisition (Test ribbon tab) on page 96).
- trigger a fast data acquisition, see Available Test actions (after initialization) on page 98 (topic Data acquisition (Test ribbon tab) on page 96).
- a zeroing operation, see Available Test actions (after initialization) on page 98 (topic Data acquisition (Test ribbon tab) on page 96).
- a check from an external system whether the MD Server still runs in acquisition mode (WATCHDOG message)

A 2-byte message (16 bits) is used to determine the required action and another 2-byte message is used for sending the actions performed. The MD Server performs a handshake with the sending device and updates the log file. The procedure is explained in the following with a snapshot as an example. The terms in this example are assigned to certain bits of the 2-byte messages, see Bit usage in input message on page 120 and Bit usage in output message on page 121.

1. The sending device sets ACTION_SNAPSHOT in the input message.
2. The MD Server sends SNAPSHOT_STARTED once and makes a snapshot.
3. The MD Server sends once SNAPSHOT_COMPLETE with SNAPSHOT_STARTED still set.
4. The sending device resets ACTION_SNAPSHOT in the input message.
5. The MD Server sends once a message with SNAPSHOT_COMPLETE and SNAPSHOT_STARTED cleared.

If continuous snapshots shall be triggered, the ACTION_CONTINUOUS bit must be set during the whole process, not just at the beginning. Because every measurement (scan) is acknowledged in steps 3 and 4, the continuous mode is terminated as soon as these bits are reset by the sending device.

💡 For further information on using a command channel please refer to document Integration of non-HBM Systems ("Infos/Integration of nonHBM Systems.doc" in the catman Enterprise installation folder).

Bit usage in input message

Bit	Action name	Comment
Bit 1	—	no longer used
Bit 2	ACTION_ SNAPSHOT	perform a snapshot
Bit 3	ACTION_ CONTINUOUS	switch to continuous snapshot mode
Bit 4	ACTION_ ZEROING	perform a zeroing operation
Bit 5	TEST_OFF	no pressure request
Bit 6	TEST_READY	pressure request
Bit 7	TEST_ACTIVE	pressure request and spectrum active
Bit 8	TEST_HOLD	pressure request and spectrum on hold
Bit 9	TEST_SCRIPT	system in script state
Bit 10	—	no longer used
Bit 11	ACTION_WATCH_ DOG	perform handshaking
Bit 12	ACTION_ FASTTRIGGER	trigger a fast data acquisition
Bit 13 … 16	—	not yet specified

The actions with TEST_… have to be set up in the MOOG CS system. You may display these states with a Label (text or image) on page 87 and context menu item **MOOG control system status**.

Bit usage in output message

Bit	Action name
Bit 1	—
Bit 2	—
Bit 3	SNAPSHOT_STARTED
Bit 4	SNAPSHOT_COMPLETE
Bit 5	CONTINUOUS_STARTED
Bit 6	CONTINUOUS_COMPLETE
Bit 7	ZEROING_STARTED
Bit 8	ZEROING_COMPLETE
Bit 9	—
Bit 10	—
Bit 11	WATCH_DOG_STARTED
Bit 12	WATCH_DOG_COMPLETE
Bit 13	FAST_TRIGGER_STARTED
Bit 14	FAST_TRIGGER_RUNNING
Bit 15	—
Bit 16	reserved

2.11.5 Open catmanAP project

This menu item starts catmanAP in Analysis mode and the **Test Explorer** tab is displayed. The Test Explorer provides an overview of the tests and measurement data stored on your PC. Similarly to the Windows Explorer, it displays the folders on your PC on the left, the tests/files in the selected folder in the center, and the already loaded tests on the right. You use it to specify which tests you want to load for further processing.

How to load a test file

1. Select a folder on the left side in the **Search directories** frame where the structure of all files on the server PC is shown.
    The middle section displays all files contained in this folder.

    🔅 Set up folders as *Favorites* for quick access: From the folder's context menu choose **Add to favorites**. You can then access folders in the network or deep nested folders with one click.

2. Select a file and drag it to the right side into frame **Analysis project**.
    If necessary the file is converted, e.g. Fast trigger data, to a binary format and loaded.

3. Switch to tab **Visualization** for display or tab **Computations** for further calculations.

☞ catmanAP is a separate program and contains an in-depth online help system. Please refer to this help for further information on catmanAP.

2.11.5.1 Find files (search functions in catmanAP)

In the middle frame of the Test Explorer you can set various filters so that, for example, only files containing the specified character string, are displayed. As test parameters are not used in catman Enterprise you cannot use this option.

Search sub folders also displays files in sub-folders of the folder selected on the left. In this case the path is displayed below the files.

Display file comments displays the comments in the ∗.BIN files below the files. Select a file and click on the **Channels** tab at the bottom to display the channels with various information like number of samples, sample rate etc. contained in the file.

Filter by file type

In the selection field in the middle Panel specify which files you want to see in the file selection list. You can use one of the entries here or enter your own file extension, e.g.

.TXT to only display text files. Specify the file extension **.BAK** to be able to load files which have arisen due to a power failure occurring during periodical saving and where only the backup file T_DATABACKUP.BAK is available.

Use search filters

You can use search filters to search for specific files or test parameters, or to search for specific channels within a file. The search box may contain **∗** or **?** as wildcards.

1. Search for files
    Enter the character string you are looking for in the search field and press ⏎
    (Return), or click on 🔍. Only files matching the search box criteria are displayed.

    If so selected, this also applies to all files in sub-folders. ⊗ clears the input
    field.
2. Search for channels
    Select a file on the **File selection** tab. Then open the **Channels** tab to display the
    channels in the file. Enter the character string you are looking for in the search
    field and press ⏎ (Return), or click on 🔍. Only channels containing that char-

    acter string are then displayed. ⊗ clears the input field.

2.11.5.2 Load additional tests in catmanAP

The **Use channels automatically** option assigns the *first* test loaded a special role: It then acts as a template. When you load an additional test with the option enabled, catmanAP performs the following actions:

- All the computations you created from the first test by way of the **Algebra**, **Frequency analysis**, **Filter**, **Strain gauge stress analysis**, **Curve operations**, **Interpolation**, **Peak values** or **Class counting** tabs are also generated for the new test if the channel names match.
    - ☞ Computations using data from multiple tests cannot be generated automatically.
- All the channels (and computations) of the first test already displayed in graph objects and existing in the new loaded test under the same channel name are likewise displayed. You may choose to create new panels for the new data or display the data in the same objects as the first test.

💡 You also get the same functionality when saving a project without references to the test data. When you open this Analysis project and load measurement data into it, they are also computed and displayed if the channel names match.

2.11.5.3 Remove or delete tests in catmanAP?

When *removing* a test (**Remove test** in the **Analysis project** group) the test plots displayed in graphs may be deleted as well as computations which use these test data. If you decide not to delete computations and plots, the defined plots will be re-displayed and the computations carried out as soon as you load a new test with matching channel names.

When *deleting* a test, the test with *all measurement data* is deleted from the folder; the deletion must therefore be first confirmed. Deletion is *only* possible via the **Delete test** context menu for the test marked in the test selection (central frame with the **File selection** tab).

2.12 Using VBScript

🛈 See also MD Server tools on page 104, MD Server VBScript help ("MDServer-ScriptRef.htm" in folder "MDSScriptHelp" in the catman Enterprise installation folder).

⚠️ The VBScript engine has been changed in version 5.0 of catman Enterprise. However, on loading an older VBScript project the code is converted. Please save the script under a different name after this conversion. We recommend you check the new code whether adjustments have to be made.

☞ Access to the script functions requires proper user access rights, see Administration and access rights on page 32.

The **Script control** group in the **Tools/Script** ribbon tab allows you to create or load VBScript code with procedures and functions which are executed at certain events. Only procedures or functions of a loaded script may be executed.

☞ A good understanding of VBA is essential when using this feature. It is sufficient if you have already written macros in Word or Excel (recorded macros are not comparable). Otherwise we recommend that basic knowledge is first acquired through

appropriate literature, e.g. from Microsoft Press. The VBScript help contains in-depth information about how to write a script and which commands are available.

As soon as a script is loaded, you may e.g. define command buttons which will execute procedures from this script. These are defined with the **Configure** context menu, see Command button on page 87.

It is however not necessary to define buttons, it is also possible to define other starting conditions for procedures from within the VBScript editor or via a trigger action, see Run VBScript on page 72.

💡 If you specify a script in the project settings, this will be loaded automatically on start up. This saves you having to load the script manually.

☞ There is also the possibility to have command buttons on a custom designed tab in the ribbon. This tab is then available with all panels as soon as the script is loaded, see Custom commands with VBScripts (custom ribbon tab) on page 128.

2.12.1 The built-in VBScript editor

The VBScript editor provides you with an interface for editing. On the right-hand side the object library, a code building aid, is displayed. It contains in several categories the most important objects and methods for access to the program:

- catman objects: IO interface, DB interface and General
- MD Server object
- GUI manager object

In each category you will find several descriptions for certain actions. Double-click on one description to insert the corresponding script command into the code on the center window. If the command uses parameters you must specify those. Parameters are indicated by pointed brackets: **<Parameter>**. Replace the pointed brackets as well.

☞ Please observe that not all commands are accessible via one of these categories. The MD Server VBScript help ("MDServerScriptRef.htm" in folder "MDSScriptHelp" in the catman Enterprise installation folder). explains *all* MD Server commands and their parameters. Additionally the ActiveX reference (catInterface_Help.chm in the installation directory) lists and explains *all* catman commands and parameters.

Basic procedure

1. Create a new function or subroutine via **New** (**Procedures** group).
    A subroutine or function is also called a procedure. You may use more than one code module in a VBScript project, each module consisting of several functions or subroutines.
2. Insert the code necessary to perform a specific action.
3. If required, assign via **Automatic procedure execution** (**Procedures** group) the execution time point, e.g. **After DAQ start**.
    Mark the subroutine (procedure) you wish to assign to a certain execution time point, next mark the execution time point, e.g. **After DAQ start**, then click on **Assign**.

General rules

- All variables are just defined with `Dim` (as variant), there is no type declaration necessary.
- Global variables may be defined before the code section (before `Sub xxx`) with `Public`.

You may test the script with ▶ (**Execution** group).

☞ Do not forget to save the script project (which includes all script modules)! If this script is necessary for the project, include the VBScript project file into the project settings (**Project settings** in the **Setup** ribbon tab).

2.12.2 Example for VBScript

The following VBScript example contains two procedures. One performs certain actions and computations after each DAQ scan, i.e. after every read block. The second procedure performs only one action: it displays a message, if a command button is pressed.

Procedure "CheckIt"

```
Public Buf(1000)

Sub CheckIt()

Dim Chan
Dim Count
```

```
With catman.IO
'Get I/O channel index from name; code blocks from catman
    base services -> I/O channel buffer are used from the
    object library
    Chan = .GetChannelFromName("Channel_02_07_001")
'Read most current value from I/O readblock buffer and
    check whether this is bigger than 400
    If .MVal(Chan,1) > 400 Then
Set flag for signal snapshot in MD Server
        MDServer.SignalSnapshot
    End If
'Repeat for other channels
    Chan = .GetChannelFromName("Channel_03_07_002")
    If .MVal(Chan,1) > 500 Then
        MDServer.SignalSnapshot
    End If

    Chan = .GetChannelFromName("Channel_02_07_003")
    If .MVal(Chan,1) > 500 Then
        MDServer.SignalSnapshot
    End If

    Chan = .GetChannelFromName("Channel_02_07_004")
    If .MVal(Chan,1) > 600 Then
        MDServer.SignalSnapshot
    End If

    Chan = .GetChannelFromName("Channel_02_07_005")
    If .MVal(Chan,1) > 600 Then
        MDServer.SignalSnapshot
    End If
'Read single data line (1st value) from input buffers of
    channels 1 to 1000 and write values to buffer Buf
    .GetMVLine 1, 1000, 1, Buf
    For i = 1 to 1000
        If Buf(i) << 100 And Buf(i) > 0 Then
            Count = Count + 1
```

```
        End If
    Next
'Set text (content) for panel object "Label_1" on server PC
    to Count
    GUIManager.SetLabelText "Label_1", Count

End With

End Sub
```

Assign **After DAQ scan** as execution time point via **Automatic procedure execution** (**Procedures** group).

Procedure "SendMsg"

```
Sub SendMsg()
'Use ShowMessage (MD Server) to display a note when press-
    ing a command button connected with this (SendMsg())
    procedure
    MDServer.ShowMessage "You clicked a command button"

End Sub
```

This procedure has then to be linked to a command button with the **Configure** context menu of the required Command button: choose **SendMsg** as **VBScript procedure**.

2.12.3 Custom commands with VBScripts (custom ribbon tab)

Procedures which shall be available as soon as a project is loaded may also be accessed from a separate tab in the ribbon: **Custom commands**.

Preparations

First the file "VBCUSTOM.INI" has to be created in folder "C:\ProgramData\HBM\MDServer". Access the hidden folder ProgramData by typing %ProgramData% into the path entry field of a File Explorer window. This file specifies which buttons to use with what script. Make sure the file is in format UTF-8 without BOM (not UTF-8Y).

Creating VBCUSTOM.INI

Use a plain text editor, e.g. MS Notepad, and create the file as text file. The first line must be `[VBSCRIPT]`, the next line specifies how many buttons are defined. After that each button is defined with the procedure or function connected, the caption, image and a tool-tip text. The images used *must* reside in a subfolder of the installation folder of catman Enterprise called "BITMAPS". Images must be in icon format (∗.ICO).

All functions and procedures must reside in one script which has to be loaded before-hand, e.g. when opening a project. In line `FUNCTION_X` you specify the corresponding function or procedure name.

Example

```
[VBSCRIPT]

FILECOUNT=2

FUNCTION_1 = MyFunctionNo1
CAPTION_1 = This is No 1
IMAGE_1 = FunctionNo1Picture.ico
TOOLTIP_1 = Start MyFunction No 1

FUNCTION_2 = MyProcedureNo2
CAPTION_2 = This is No 2
IMAGE_2 = ProcedureNo2Picture.ico
TOOLTIP_2 = Start Procedure No 2
```

VBScript for a custom ribbon tab

The functions or procedures are written as usual. Then include the script into the . As an example, a script which starts program Notepad from the ribbon is given.

Example

```
Sub StartNotepad()

  Dim objWSH

  Set objWSH = CreateObject ("WScript.Shell")
  objWSH.RUN "Notepad.exe"
  set objWSH = Nothing

End Sub
```

Save the script, include it into the MD Server project and state e.g. `FUNCTION_1 = StartNotepad` in the VBCUSTOM.INI.

2.13 Channel selector window

The **Channel selector** window (click on 🔢) is divided into two parts: on the upper part all connected devices with their channels, your Calculations and the Function groups with Time channels and Manual inputs are shown. Click on one of the groups or entry in one group and the corresponding channels are shown in the lower part.

If you search for specific channel names, the result is also shown in the lower part of the dialog. Click on the Search results group in the upper part to again display the result of your last search.

Use the Shift key (⬆) to select consecutive channels in the lower part and drag the channels (still holding the Shift key) to graphs or tables. Use the Ctrl key (Ctrl) to select non-consecutive channels. Objects capable of showing only a single channel will not change if several channels are dropped.

How to search

Use **%** or ***** as wild cards for several characters and **_** or **?** as wild cards for single characters. You may use wild cards more than once in the search string as in **%nnel??04%**. Without any wild card character, an exact match is searched.

2.14 Diagnosis and troubleshooting

2.14.1 Diagnosis (ribbon tab)

These menu items enable you to track problems and to switch all cabinets on or off if an UPS is used.

MD Server project log

☞ The log file is always copied to the project folder. So when starting a new project a new log file is created as soon as the system is initialized (⬛).

Show project log

The MD Server project log (file SERVERLOG.MDB) shows several events (levels): inform-
ation ◆, warning ⚠ and all errors ⛔, which occur during operation. It is possible to
edit the log messages, add comments and specify an auxiliary file with additional inform-
ation or data. However, the original log level and log message will always be visible
together with date and time of the event.

The information given in the **Log message** column is useful if problems arise during a
test: the cause and history of warnings or errors should point to the problem in question.
Start to search for the last user log-in and then check where the problem originated.
Then control the settings of this program part, e.g. with MC Setup or the DAQ settings
etc.

💡 Use **Asc** or **Desc** in the drop-down list to sort the entries in ascending or des-
cending order in the **Date/Time** column.

There is a separate log file for MC Setup (MC Setup log file) and for the communication
logging.
See also and Terminal and communication logging below.

Delete log entries

This dialog lets you remove and archive either all entries or just all entries prior to a cer-
tain date from the project log file. Archived log entries are saved to subfolder *Archive* in
the project folder.

Communication logging

The communication log items are for users with special knowledge only. Usually these
menu items are used by HBM personnel when solving interface problems. This might be
necessary if you contact the HBM Technical support, see Technical support on page 29,
otherwise we recommend not to change these settings.

By default (**Server options**, tab) all communication during a
test is logged in a circular buffer (starting after the initialization). This enables the tech-
nical support to trace the communication if a problem arises during a test. In this case
the log file may be exported and send to the support. If during setup or other actions a
problem occurs, the communication log can be activated manually and may prove helpful
for tracking down these problems. However, the information contained is only relevant
for the HBM Technical support. If the default setting of 50 000 entries (**Number of lines**)
is not enough, increase the size with **Configure log size**.

UPS manager

⚠️ For all UPS related actions and services like UPS trigger or UPS monitor an UPS of a type approved by HBM must be available (see Operating requirements on page 15) and **Enable UPS monitor** must be active in the **Server options**, see General options on page 39.

Together with catman Enterprise the UPS Manager program is started if the above mentioned conditions are fulfilled. Use the Windows task bar or **UPS manager** (**Diagnosis** ribbon tab) to call up the program.

⚠️ Do not close the window as this terminates the program. Just minimize the window if you don't need it.

There remaining run time, capacity, load, temperature and status are displayed. You may also switch the conditioning devices *on* or *off* in this window. All devices supplied by a UPS are switched off or on together. Use **Settings** in the window to specify network settings and SNMP properties. **Scan** searches for UPS devices on the network.

As soon as one battery capacity falls below a certain operating time limit, the server is shut down or the corresponding cabinet is switched off. Various settings may be specified with the UPS administration software. Please read the Operating Manual of the UPS for further details and for information on how to exchange the battery.

⚠️ Please observe the two year service interval for the UPS units.

Reset QuantumX/SomatXR communication

QuantumX/SomatXR modules are able to communicate with more than one PC or program at the same time. However, the number of connections possible is limited. After having a server breakdown or timeout, e.g. due to network problems, all QuantumX/SomatXR modules still have a communication/command queue reserved for this connection. If this happens several times, e.g. during system setup or when experiencing network problems, the maximum number of connections for the QuantumX/SomatXR modules may be reached and no further connection is possible. Use this command to reset all reserved communication connections. MD Server initiates a new connection automatically after this command.

Terminal

The **Terminal** item (MGCplus only) is for users with special knowledge only. Usually it is used by HBM personnel when solving interface problems. This might be necessary if you contact the HBM Technical support, see Technical support on page 29.

2.14.2 Troubleshooting tips

In most cases four different error sources lead to problems and should be checked:

1. Licensing problems
    If during start up, when initializing or during the measurement the required licenses are not found, an error message indicating this is issued. Please check topic Licensing in the System Documentation, Part I, whether all settings are made as required.

2. Hardware problems
    Examples are device or channel not present, switched off accidentally etc. Hardware problems may also occur if the channel setup is not done according to the transducer wiring, e.g. if a half bridge is connected and a full bridge specified. This would lead to false readings or errors. Therefore we recommend to perform a channel check after the setup in MC Setup, refer to Checking channels on page 211. Use then the search option to find channels with errors.

3. Communication problems
    An example would be, if a channel could not be accessed even though the device or channel is physically there. Check the Ethernet wiring and scan again (Checking channels on page 211). Check also the synchronization connections (cables, plug-ins and MGCplus termination plug-ins). Then switch all units off and on again (see UPS manager on page 132) or—with QuantumX/SomatXR—Reset QuantumX/SomatXR communication on page 132 (**Diagnosis** ribbon tab). If the problem persists, you might have a hardware problem: defective channel or device.
    See also **Set TCP/IP settings** (Special group on page 104 in topic MD Server tools).
    If you are experiencing communication problems between the server and client PCs, check the port and timeout settings and make sure a circular buffer is activated.
    See also Client - Server settings on page 47.

4. Measuring problems
    An example would be, if not all data necessary for a trigger condition has been defined or no channel is active. If just no data is visible in a certain panel, check whether a valid data source is defined for the panel (re-define if necessary). If a high value is shown, i.e. -1 000 000, a signal overflow has occurred. Specify the value to be shown with **Server options**, **Signal overflow as**, see General options on page 39 (**Server options**, **Setup** ribbon tab).

Try to find out into which category your problem falls and use the appropriate program part to solve it. If you still make no headway, contact the HBM Technical support, see Technical support on page 29.

2.15 The databases used and files created by the MD Server system

- DB_000.$_$: the MD Server database file for temporary data storage, see Configure storage size on page 35.
- MCLOG.MDB: log file for MC Setup in the "My Documents" folder of the logged-on Windows user, subfolder \HBM\MCSComponent.
- SERVERSETTINGS.MDB: instead of using the Windows registry, preferences and basic settings of MD Server are stored to this database in the "My Documents" folder of the logged-on Windows user, subfolder HBM\MDSERVER. Please do not change any entries manually except if specified in this help otherwise as this might result in faulty operation of MD Server.
- MDPROJECT.MDB: project settings in the project path.

In the test folder (subfolder of the project folder) the following files are stored for this test run:

- SERVERLOG.MDB: the log file for MD Server. This file contains all log entries for each project and is stored into the project path, see also MD Server project log on page 130.
- MCSETUP.MCS: all settings concerning the transducers and the signal conditioning devices MGCplus.
- StoreChannels_<Test base name>_<date>[_<count>].BIN, e.g. StoreChannels_ NewTest1_2020_03_31.bin: a file containing the measured data for all channels. It depends on the **Export** options (**Server options** dialog), whether data is stored into one file (default) or not. The <count> entry is only used when several files are used. See also Export options on page 42.
- Channel_name_xxx.BIN: a file for each channel containing the measured data if this store option is activated (**One file for each storage channel**). The file name corresponds to the channel name. See also Export options on page 42.
- DAQ TIME STAMPS.BIN: one file containing the time stamps (marks) in relative time since measurement start.
- DAQ TIME STAMPS_ABSOLUTE.BIN: one file containing the time stamps (marks) in absolute time (day, month and year etc.).

- ZERO_TAG SLOW DAQ.BIN: one file containing references to the channel data and zero values if one or several zero measurements were made during the test run.
- SNAPSHOT_<Test base name>_<date>[_<count>].BIN (default setting): one file containing all snapshot data. The <count> entry is only used when periodic export is specified. See also Export options on page 42.
- ZERO.BIN: one file containing all zero measurement values (history) from the complete test run.
- Subfolder TRIGGER with
    - FastTriggerData_<Test base name>_<count>.BIN: a file with data from all channels marked in the **Fast rate** column in the DAQ settings (Fast trigger mode and MGCplus only). Every **Fast** trigger generates a new file with <count> as the respective number, see Measurement settings (DAQ settings) on page 56, Use fast sample rate on page 70 and Data acquisition (Test ribbon tab) on page 96.

3 MC SETUP

This program is either started via the MD Server software or it may be run on a different PC (without MD Server installation) as a separate software. In both cases the databases (setup database) created by the program are later used by the MD Server software.

> 💡 It is also possible to make a plan first: which transducers have to be connected? The database produced by this plan is then merged with the database of the available hardware created by MC Setup in the system scan (Changing the hardware configuration on page 217) and results in a wiring diagram. This procedure is described in topic MC Definition List on page 225.

Restart with changed hardware

The following assumes that the number of devices and channels is unchanged. If new hardware inside existing devices is found or if Ethernet addresses have been changed, MC Setup asks for a re-scan. If you agree, all settings are retained as far as possible: they are automatically imported into the new database created by this re-scan. You have to start a new scan manually if you add new devices: click on **New** in the **MC database** group (**Database** ribbon tab).
See also Setting-up channels on page 157.

3.1 General information on MC Setup

After the catman Enterprise installation a default database with one MGCplus device is loaded. To connect to your system, you must start with a *system scan* (Step 1: Scan hardware on page 141). This procedure creates a setup database file which contains the available hardware configuration. When later starting MC Setup from within MD Server, the active setup database file is loaded automatically. If you start MC Setup as a standalone application either state an existing database file or create a new one which has then to be transferred to the PC running MD Server.

After starting-up MC Setup, you are asked whether a connection to the system (hardware) shall be made. You have two choices:

1. Connect to the devices available for the test
    You are then working "online" and all settings may be transferred immediately to the devices.

2. Work with the setup database file
You may then perform the setup even on a separate PC without any possibility to connect to the devices. All relevant information is contained in the database file of MC Setup.

MC Setup main window

The MC Setup main window shows the available data acquisition channels as lists on the left-hand side in three main categories:

1. System with Devices (hardware channels) and Calculations
2. Function groups with Time channels, Manual inputs and user defined groups
3. Search results

The list of channels available under the selected category is displayed on the right-hand side. Operation is done from ribbon or when the right mouse key is pressed over an entry on the left-hand side or in a column of a channel displayed on the right-hand side.

Rename device

Double-click on a device to rename it. Up to 50 characters are possible for device names.

Handling sensors with TEDS or T-ID

With MGCplus, TEDS modules are not considered automatically. You must use **Activate TEDS (system wide)** on the **Device** ribbon tab to activate the TEDS contents. Without this action you may even override all TEDS settings with MC Setup by manually defining scaling, unit etc.

With QuantumX/SomatXR, the TEDS contents are usually read out when connecting the sensor to the module. However, the TEDS usage might have been set manually to "Ignore TEDS" with the MX Assistant. Use **Activate TEDS (system wide)** to enable and read out all TEDS contents again. This will take approx. one minute.

☞ Do not connect or re-connect a sensor with TEDS to a QuantumX/SomatXR module during a measurement. The data acquisition in this channel is stopped until the TEDS settings are read out and the channel is adjusted.

💡 For sensors with T-ID we recommend using MC Import: Import from a custom defined database (Excel, Access) on page 173.

Possible settings for hardware channels

The possible settings for the hardware devices are retrieved from a database supplied by HBM. If new hardware with new features is installed, it might therefore be necessary to

update the hardware capability database as well. Please contact HBM in this case to obtain new database files.

Copy/paste channel settings

Several columns allow for copy/paste operations with single or multiple cells. Mark the cells you would like to copy and use context menu **Copy** (right-hand mouse key). Then mark all cells the setting shall be transferred to and choose **Paste** from the context menu.

Drag & drop to a (user defined) Function group

It is possible to drag channels displayed on the right-hand side to a (user defined) Function group: press the *right-hand* mouse button and drag. It is *not possible* to use the *left-hand mouse button* in this case.

Print/export channel list

Print table on the **Database** ribbon tab prints the list displayed on the right-hand side directly to the default printer.

⚠️ MS Excel 2007 or higher is required for saving in Excel format.

Table (MS Excel) in the **Export** group saves the settings for *all* device channels (except time channels) to an Excel file (Excel 2007 binary format (∗.xlsb)). A file dialog asks you for storage path and file name.

With **Selected columns (MS Excel)** in the **Export** group you get a dialog which allows you to select the columns you want to export to the T_Channel spreadsheet. The settings for *all* device channels (except time channels) may be exported.

Table (ASCII) in the **Export** group saves the settings for *all* device channels (except time channels) to a text file. A file dialog asks you for storage path and base file name.

With both MS Excel exports four spreadsheets (in one worksheet) are created: T_Channel, T_Calc, T_UserScale and T_TriggerChannel. If you export into ASCII, four files are created: BaseName_T_Channel.txt with the channel information, BaseName_T_Calc.txt with information on the calculation channels, BaseName_T_UserScale.txt with information on the user scaling of channels and BaseName_T_TriggerChannel with the trigger settings. The files are always created, even if no information is contained. Trigger settings are not defined in MC Setup, they are however exported together with the other settings.

💡 You may also import any of these Excel spreadsheets with MC Import, e.g. just the Trigger definitions, see MC Import: Import from a custom defined database (Excel,

Update MC Definition List with current MC Setup settings

Menu **MC Setup -> MC Definition List -> Update MC Definition List** provides a convenient way to update the MC Definition List used for creating the active MC Setup database: measured zero values from the **Amplifier/Sensors** ribbon tab, **Reading ZERO** column, and the current scaling information (columns **Electrical zero** to **Physical nominal**) are written back to the original MC Definition List database.

Notes

● For all MGCplus 8-channel amplifier plug-in modules the settings **Sensor type**, **Uexc** (excitation voltage) and **Filter** are restricted: only identical settings for all 8 channels may be chosen. If you assign different settings to these channels, the last setting is used for all channels.

● All settings and setups are stored in databases immediately, there is no need to save anything. However, if you want to re-use settings, you may export (and save) settings to file.

● You may always save the database under a new name (**Save as** on the **Database** ribbon tab) for trying out new settings. However, do not forget to load the original database again before ending MC Setup. The database active when ending MC Setup will be used as database for MD Server and even entered as setup file in the project settings.

● Except those items where **... (system wide)** is appended, most menu or ribbon actions in MC Setup affect only those channels displayed on the right-hand side. The channels shown on the right-hand side are called *selection* in contrast to *marked* channels.

● You may change the column width: click and drag the separator between two columns. You can also hide a column. To display the column again you can either drag the right separator of the column left to the hidden one to the right-hand side or use the context menu **Show all columns** in the header line.

● Use the **Layout** tab in the MC Setup options dialog (**MC Setup -> Options**) to switch certain columns to bold face in the **Amplifier/Sensors** ribbon tab or change the font size on all ribbon tabs.
See also Layout options tab on page 223.

3.2 Getting started with MC Setup

If you do the setup for the first time you have to use this manual approach as described in the following or work with the MC Definition List on page 225. However, even in this case it might be useful to generate the channel names in Excel. We recommend you first read the topic Getting started with MC Import on page 174. The following steps describe the principle procedure. In-depth information is provided in the other sections in MC Setup. Links in the following steps point to the relevant sections.

3.2.1 Step 1: Scan hardware

If you have just installed MD Server, the first thing to do is to perform a system scan in order to find all hardware connected to the server PC. With systems provided by HBM each MGCplus has its Ethernet address labeled on the left side of the front and each QuantumX/SomatXR module has its Ethernet address labeled on the right upper side of the front. Ex-works the first MGCplus has Ethernet address 172.16.1.101 and is configured as leader. The first QuantumX/SomatXR module has Ethernet address 172.16.2.1. In both cases it is the lowest address.

☞ Only analog input and CAN bus channels are supported with QuantumX/SomatXR modules, no hardware computation channels (math channels).

Device names

You may name devices for easier differentiation: double click on the device in the list of devices on the left side. The maximum length is 50 characters.

MX840A/B and CAN bus

CAN channels are included only if Connector 1 is already switched to CAN bus and all required CAN signals have been assigned with the MX Assistant.

☞ You must install the required MX Assistant software manually, it is not included in the catman Enterprise setup. Either download it from the HBM website (HBM downloads) or use the CD included with your QuantumX/SomatXR module delivery.

First scan

1. Use the HBM Device Manager for QuantumX/SomatXR modules and for MGCplus devices with CP52, see HBM Device Manager on page 143.
    With the signal conditioning devices MGCplus with CP42 you have to type in the

corresponding IP address range; the HBM Device Manager cannot be used here. The scan may be performed in ascending or descending order, just type in the address range accordingly. Leave the input field empty if no device of this type is connected.

QuantumX/SomatXR modules connected to a CX27 need not to be specified, it is sufficient to use the HBM Device Manager or to state the CX27 IP addresses and to activate **with CX27 scan** (the digital I/Os of the CX27 are supported too). When using MX modules connected via FireWire, you may state **24.0.0.0** (and **25.0.0.0** if two interfaces are available) as IP address which will find all modules connected on all FireWire interfaces.

☞ MX modules and MGCplus with CP52 use DHCP (factory setting). We recommend you assign static addresses: Click on **HBM device manager** in the **Scan System** dialog. This allows for setting a static address regardless of the original setting. As an alternative you may use DHCP if your network administrator uses the MAC address (see label on device) to ensure a static address.

2. Please check the appropriate topics if other or special devices are used: (e.g. for Avionics buses or a MOOG control system), , , or .

3. Click **Start scan**.

After the scan all devices found are displayed with their addresses and assembly. Close the system view to return to the main window of MC Setup. Here the devices are displayed on the left-hand side under System. With MGCplus, the SYNC leader device is marked with **(Leader)**.

If no device is found, you may create virtual channels, e.g. for testing purposes. See .

Leave the dialog, e.g. with ⬅ **Return to channel setup**.

☞ MGCplus devices and QuantumX/SomatXR modules connected via Ethernet are displayed in the channel list sorted according to their IP addresses. QuantumX/SomatXR modules connected via FireWire or via a CX27 module are sorted according to their module names. The module names may be changed with the HBM Device Manager (see above step 1). You may specify the relevant IP

addresses in the required sequence in the scan dialog, if you need a special device sequence when using Ethernet.

New scan

The (last) addresses or ranges used are already displayed in the dialog box if a scan has been made before. If you wish to create a new configuration, create a new database: **New scan** on the **Setup** ribbon tab from the MD Server ribbon or **New** on the **Database** ribbon tab from MC Setup. You will be prompted for a file name for the setup database to be created and the dialog is opened, with which a scan of the relevant range may be started, see also *First scan* above.

☞ Information about a system re-scan (and possible error messages) is provided in Changing the hardware configuration on page 217.

3.2.1.1 HBM Device Manager

MX modules with firmware 4.0 or higher and MGCplus with CP52 (CPU module) automatically announce themselves cyclically in the network. These messages are collected and evaluated. Therefore the IP address set on the device is unimportant as all devices are found which are in the (Ethernet) network (or, with MX modules, can be reached through FireWire).

☞ MGCplus devices with CP42 are not found.

Procedure

● Click on **New module scan** if the device or module is not shown on the **Modules found** list.
All MGCplus devices with CP52 or MX modules are listed, even modules or devices to which no connection is possible. When having QuantumX/SomatXR modules connected to a CX27, it is sufficient to activate **CX27 sub-device scan**.

● For an identification with ❌ change the IP address.

🔵 is shown before the module address if the device or module is accessible and measurements can be supplied. Generally, the PC and device must be located in the same network segment.
Refer also to Changing the IP address (MX module or CP52) on page 145.

● Click on **Add devices** to get the previous dialog again.

- Proceed as usual with **Start scan** and .

MGCplus devices and QuantumX/SomatXR modules connected via Ethernet are displayed in the channel list sorted according to their IP addresses. QuantumX/SomatXR modules connected via FireWire or via a CX27 module are sorted according to their module names. Mark the relevant device, click on **Rename** and change the name in the **Name** column. In order to transfer MGCplus devices into the channel list in alphabetical order it is sufficient if you sort the list in the HBM Device Manager: Click on the **Name** column before adding the devices.

No device or no module found?

If no module appears in the **Modules found** section, there are several possible causes. Check the following points and then repeat the search:

- Is the QuantumX/SomatXR module or MGCplus device switched on.
- Does the MX module LED light up green?
- With a connection via a CX27 module does this module also have the appropriate firmware?
- Have you used the correct Ethernet cable?
- Is your Ethernet switch operating correctly?
  If you have no other devices operating on the switch with which you could check the function, try to establish a direct connection between the PC and the module or device in question.
- Have you waited long enough for the PC to define its address?
  If the PC does not find any server in the network, then the server is first sought (the symbol for the interface in the Windows tray indicates the search, but the symbol may only appear, when the display is configured). After about 20 to 30 seconds either an automatic address or the specified alternative address is set. Similarly, during this period no module or device is found with a scan.
- Is it possible that your firewall is blocking the scan?
  Depending on the device type connected, release the following ports:
  - QuantumX/SomatXR: TCP port 80, 5001, 7411 and all ports above 50 000, UDP for ports 1200, 1201, UPD-Multicast for ports 1300, 1301, 31

416 and 31 417.
- 🔴 MGCplus with CP52: TCP ports 7 and 80, UDP-Multicast for ports 31 416 and 31 417.
- 🔴 MGCplus with CP42: TCP port 7 (no UDP used).
- 🔴 Is the interface adapter used activated in Windows and ready for operation?
- 🔴 Is the correct driver installed for the adapter?
FireWire interfaces require the driver for the FireWire hardware and the virtual adapter IP over FireWire is also needed; refer to Installing the FireWire interface on page 21.
- 🔴 Is it possible that your virus protection program is responsible for blocking the scan?
Try deactivating your virus protection to find the cause. If the MX modules or MGCplus devices are then found, refer to the documentation of your virus protection program to find out how to enable access to the network (local subnet) for individual programs. You must enable the same ports as described under Firewall.
- 🔴 If a WLAN is also active with your PC, test whether the module is found when you temporarily switch off the WLAN (only for the time of the scan). With some WLAN configurations problems may occur with a scan over all interfaces.
- 🔴 If your PC has several Ethernet interfaces, try deactivating all other Ethernet interfaces.
- 🔴 If you are using the devices in a larger network, contact your network administrator. In managed networks there is a range of methods for restricting or completely preventing the data transfer between individual devices. Administrative releases may therefore be needed here.

Changing the IP address (MX module or CP52)

With systems provided by HBM each module or device has a fixed address. However, the factory setting is DHCP. We recommend to use fixed addresses. The module must be present in the list in the **Modules found** section if you want to change the address.

Procedure

1. Mark the module or device in the list of modules in the **Modules found** section.
2. Enter the desired **IP address** and **Subnet mask** in the **Change address setting of module** section.

The default subnet mask is 255.255.0.0 and normally does not need to be changed. In the case of doubt, consult your network administrator.

3. Click on **Apply settings**.

After changing the IP address the MX module must be restarted to activate the address. You are therefore asked whether a restart is to be carried out. A new module scan is then carried out automatically.

Renaming a device (MX module or CP52)

Mark the module in the list in the **Modules found** section and click on **Rename** to be able to assign a name to the module. Then edit the name in the **Name** column.

Identifying an MX module (LED)

If you are not certain which MX module belongs to which address, you can flash the module LED.

1. Mark the line with the IP address and the module name in the **Modules found** section of the HBM Device Manager
2. Click on the **Flash LED** button.
3. Click again to stop the flashing.

3.2.1.2 Using CANHEAD (MGCplus only)

ℹ️   See also Perform CANHEAD wiring compensation (cable length compensation) on page 214.

You have the option to assign all CANHEAD modules automatically if unassigned modules are found. This function corresponds to the auto-assign feature used in the MGCplus Assistant. Otherwise you must first start the ML74 Channel Assignment with **Advanced setup** (**Device** ribbon tab) and manually assign all modules.

You may prefer the first approach the first time you scan a new system. The second approach might be more convenient, if you expand your system: the auto-assign feature assigns all modules sorted according to their serial number.

⚠️   Because the auto-assign feature assigns all modules sorted according to their serial number this might change the channel order compared to an earlier scan. You may deactivate this feature on the **Diagnosis** ribbon tab in **MC Setup ->**

CANHEAD baud rates

CANHEAD modules may use a (higher) baud rate of 500 kBaud, starting with the following combination of plug-ins and modules: CA1030 with hardware revision 2.0 and firmware version 2.14, ML74B with firmware version 5.14, CP42 with firmware version 4.46 or CP52. The higher baud rate results in a higher data transfer speed and is automatically activated if all modules connected to the same CAN bus support this feature. A maximum of 5 CANHEAD modules may be used per CAN bus interface in this case. If at least one older CA1030 is detected, the default baud rate of 250 kBaud is used.

Older CANHEAD modules (the amplifier hardware part) may be upgraded, however, they have to be sent to Darmstadt. Please contact the HBM Technical support, see Technical support on page 29 for details on the procedure.

Phase delay between CANHEAD and ML801

Depending on the number of CANHEAD modules connected to a ML74 plug-in module a phase delay occurs between CANHEAD channels (which are synchronized also between different ML74) and other plug-in modules of the MGCplus. The resulting phase delay is approx. 5 ms for one CANHEAD up to 80 ms for seven or more CANHEAD modules and 250 kBaud (see specifications in the CANHEAD Data Sheet). It is possible to compensate this phase delay on ML801B plug-in modules, if a ML801B with firmware version 6.56 or higher and either a CP42 with firmware version 4.46 or higher or a CP52 are used. The compensation is used if you

1. Select the appropriate **Bessel CANHEAD** filter in the **Filter** column of MC Setup for your ML801. This ensures that these plug-in modules use the same filters (filter characteristic and phase delay) as your CANHEAD modules.
    (The option **CANHEAD filter** (**Advanced setup** on the **Device** ribbon tab) is automatically activated, if a CANHEAD module is detected.)
2. Use dialog **Output delay/baud rate** (**Device** ribbon tab) to fine-tune the delay value, because it depends on various settings.

Adjust output delay and baud rate

You may have to find the best compromise on the delay settings, if with more than one MGCplus device different combinations of ML74 and the number of connected CANHEAD modules are used (ML74 variants). In the dialog the maximum allowed number of

CANHEAD modules (marked with M) and the actual (assigned) number of CANHEAD modules (marked with A) together with the baud rate used is displayed for each ML74 module.

1. Set the delay on the ML74 with **Advanced setup** and **ML74 -> Set minimum number of CANHEADs** (mark the ML74 module in question). This setting ensures that the ML74 has the same delay regardless of the number of actually assigned (connected) modules. If this setting is higher than 5, only 250 kBaud may be used. We recommend you use 5 or less CANHEAD modules per ML74.

2. Call the dialog to adjust the ML801 output delay with **Output delay/baud rate**.

3. Check for the numbers displayed in the left upper table of the dialog. Click on one of the entries to get the corresponding module marked in the table below. The highest of these numbers (the delay values) is forwarded to the input field on the right upper side. However, you have to decide whether this value shall be used for all ML801 or whether you prefer to change this number.
   If, for example, three modules have a value of 6 and one module a value of 13, it might be better to use 6 as this gives best results for three of the four modules. When using 13 in this case, the delay for three modules is too long and only one module has the proper delay value.
   The number is multiplied with 410 μs and used as delay time in the ML801B module.

4. Click on **Set** to transfer the setting to all ML801 amplifier plug-ins.

3.2.1.3 Configuring the HBM devices for optical sensors

🛈 See also Using an ARAMIS camera system from GOM on page 150, Configure and check synchronization on page 112, Optical sensor computations on page 209, and topic "Optical strain measurement" in the System Documentation, Part I.

With the appropriate license you may use the HBM devices for optical strain measurement (e.g. FS22 BraggMETERs) with MD Server. A maximum of four devices is supported. Each device can use up to 8 connectors (depending on device type) with (in theory) up to 15 channels (measuring positions or optical sensors) for each connector.

The optical strain measurement uses fiber Bragg gratings in optical fibers. A wavelength is output as measured value. The change of this wavelength resulting from the fiber Bragg grating under mechanical stress is a measure for the strain of the grating. As only the wavelength is output from the devices, you must still enter a computation in MC Setup that calculates the strain from the wavelength change.

Setting the Ethernet address on a FS22 BraggMETER

The IP address must be currently set using a terminal program and the commands given in the operating manual.

FS22 BraggMETERs require TCP ports 3500 and 3365.

☞ Enter the IP address used in the **Scan options** or ensure that the IP address used lies within the scan range entered when performing the system scan.

⚠ After switching power on the devices require some time for their initialization. Therefore please wait until the devices are ready before you start the scan.

Configuring channels

1. Start MC Setup.
2. Perform a system scan: Enter the appropriate IP address(es) for the BraggMETERs or Interrogators as external devices addresses.
3. Close the system view window after the scan.
4. Mark the first device found in the System group on the left upper side.
5. Open the configuration window by selecting **Advanced setup** on the **Device** ribbon tab.
6. Adjust the device to the sensors used and check that they are functioning correctly.
   ☞ Activate the connector used on the **Analyze spectra** window, so that values are acquired and displayed.

💡 Further information on the setup can be found in the Operating Manual for your device.

3.2.1.4 Additional external devices

With the appropriate license you may use other devices together with HBM equipment. During a system scan in MC Setup specify in frame **External devices** those devices for your measurement project, if DLL drivers exist, e.g. a MOOG control system or an AIM ARINC-429 device.

☞ Only devices are shown in the list for which a license exists.

With devices connected via Ethernet specify the appropriate address. Depending on the device used, it might be necessary to specify a configuration file which is to be loaded on initialization. For example with ARINC-429 boards, a *.PDI file has to be loaded (use the

board configuration utility to create the file) and an *.INI file has to be present in the same directory. In case of doubt, please contact the HBM Technical support, see Technical support on page 29 for information on the necessary drivers and files.

Refer also to the documentation in the System Documentation, Part I, concerning Avionics buses and to topics Using AIM modules on page 152 and MOOG control system (MOOG CS) on page 153.

☞ The documentation on DLL DAQ Drivers ("Drivers/DLLDriver/DAQ drivers.doc" in the catman Enterprise installation folder) shows how to write such a driver in a high-level programming language. Further information may also be found in Integration of non-HBM Systems ("Infos/Integration of nonHBM Systems.doc" in the catman Enterprise installation folder).

Settings which have to be made after the setup in MD Server

1. Specify in MD Server, which *time format* is used (**NTP**, **PTP** or **IRIG B**), see Configure and check synchronization on page 112.
2. Specify in the **Server options** on tab **Hybrid systems** (**Setup** ribbon tab) which *synchronization procedure* shall be used, see PTP/NTP sync control on page 46 in **Hybrid systems** options.

Zeroing channels from external devices

Usually zeroing is done by the MGCplus or QuantumX/SomatXR signal conditioning devices. If you would like to use this function for external devices too, define a *Software scaling* for all channels from these devices requiring a zeroing, refer to Software scaling / strain gage temperature compensation on page 170. Use **0/0** (first row), **1/1** (second row) as scaling and repeat the unit in the **Unit** row in the **Linearization table**. Then a standard zero operation also sets these channels to zero.

Using an ARAMIS camera system from GOM

With the appropriate license you may use an ARAMIS 3D camera system.

Background information

The ARAMIS camera system is a stereo camera system which supplies 3D coordinates based on triangulation and using stochastic patterns or reference point marks. The GOM (**G**esellschaft für **O**ptische **M**esstechnik) Testing Controller controls the image data and analog value recording and the light management. The system measures samples and components made of any materials by non-contact means based on the principle of

digital image correlation. It permits wide-area and spot analysis of samples from just a few millimeters in size up to structural components on a scale of several meters. The measurements can be performed independently of the geometry and temperature of the samples.

See also https://www.gom.com/de-de/produkte/3d-testing.

☞ catman Enterprise communicates exclusively with the PC running the ARAMIS software (GOM Testing Controller).

General procedure at the Testing Controller

1. Create a project on the Testing Controller or load an existing project.
2. Enable the **SCPI Interface** option (**Tracking settings**) in the deformation tracking dialog.
3. Choose the desired sample rate for deformation tracking. The sample rate must be one of the sample rates also available in catman Enterprise.
4. Start deformation tracking.

⚠ Do not stop deformation tracking while MD Server is measuring. Otherwise you have to re-initialize MD Server.

How to proceed in catman Enterprise

1. Use the HBM Device Manager, for example, to find your QuantumX/SomatXR modules (see HBM Device Manager on page 143). Otherwise specify the IP addresses in the appropriate fields as usual.
   Then integrate the GOM Testing Controller via the **External devices** area. Select **GOM_camera** as **Device type**.
2. Enter the Ethernet (IP) address of your Testing Controller (you do not have to specify a port) in column **Address**.
3. Click on **Start scan**.
4. The selected HBM devices and the Testing Controller (and other external devices) are displayed in the channel list. For the Testing Controller, all the channels defined there, the NTP time channel and the Diag_Sec_Number channel are displayed. Diag_Sec_Number is a sequential numbering of the received data packets. Missing numbers would indicate lost data.
5. Activate synchronization via NTP—see also Installing the NTP time server/client on page 18, Configure and check synchronization on page 112, NTP/PTP sync failed trigger on page 66.

6. Set the same sample rate for the channels from the Testing Controller in catman Enterprise as you selected above in the Testing Controller.
7. Configure your measurement and visualization as usual, and start your measurement.

Notes

- 🔴 catman Enterprise cannot check whether the configuration of the Testing Controller still matches the one in the catman Enterprise project.
- 🔴 You must create a new catman Enterprise project if the Testing Controller configuration changes.
- 🔴 catman Enterprise cannot carry out any configuration on the Testing Controller, and in particular cannot change the sample rate. So make sure that the sample rates set in catman Enterprise and in the Testing Controller for deformation tracking are identical.
- 🔴 Note that the Testing Controller's sample rate is not guaranteed under all circumstances. Interference on a camera image or a high CPU load might reduce the sample rate for example. We therefore recommend dragging the NTP_TIME channel onto the x axis of the graph with the Testing Controller data in order to obtain a correct display. The channel contains the NTP time for each data packet sent by the Testing Controller. The NTP time is in UTC and seconds since 1.1.1970 (**C**oordinated **U**niversal **T**ime, i.e. world time, corresponding to Greenwich Mean Time [London time]).
- 🔴 Exit catman Enterprise before stopping deformation tracking.

Using AIM modules

☞ See also Installing software for Avionics buses on page 27, Hybrid systems options on page 45, Configure and check synchronization on page 112.

⚠️ Start PBA.pro *before* trying to connect to an AIM module.

AIM modules can only be used together with HBM devices.

Procedure

1. Start PBA.pro

2. In PBA.pro start your configuration with a click on ▶️.
   See also Installing software for Avionics buses on page 27.
3. Make sure the script for you type of module is started in PBA.pro.

4. Start MD Server and /or MC Setup.
5. Start a **New scan** in MD Server or create a **New** MC Setup database.
6. Select **HBM_AIM_Driver** in frame **External devices** of the **Scan system** dialog and specify the IP address of the PC where PBA.pro is running.
7. Click **Start scan**.

☞ The IRIG B time from AIM modules is converted into NTP time stamps.

MOOG control system (MOOG CS)

☞ Using a MOOG control system together with MGCplus or QuantumX/SomatXR, or both, requires an NTP time server for synchronization, see Installing the NTP time server/client on page 18.

You may connect a MOOG CS either via Ethernet or—with MGCplus—via CAN bus. It is even possible to forward data from MD Server to the MOOG control system. The data is output after each read block, i.e. with approximately 10 Hz, see also General tab (DAQ status window) on page 100 for read blocks.

⚠ Please check the channel names in your MOOG CS system before connecting: channel names containing certain characters are not only not imported (and present), the import of further channels is also terminated. The following characters are not allowed: colon (**,**), period (**.**) and angled brackets (**< >**).

Refer to the following topics for detailed descriptions:
- NTP time: Additional settings required with MOOG systems (RTFE) on page 20.
- Hybrid systems options on page 45.
- MOOG CS options on page 116 and subtopics Configure CAN bus modules for remote control on page 117 and MOOG command channel on page 118.

3.2.2 Step 2: Decide on how to specify your sensor connections

Two options are available:

1. If the transducers are already connected or shall be connected according to a plan already made, you just enter information about which transducer type is connected to what channel and device. For details concerning how to connect a specific transducer to a channel, see the Operating Manual of the respective device. If you want to use this option, please proceed with Step 3.

2. If an MC Definition List exists, you may merge this list with the hardware available. Use menu **MC Setup -> MC Definition List -> Merge with MC Definition List**. Otherwise create a new list by specifying which transducer you would like to use in the test. This configuration is then matched to the available hardware. As a result, the connection scheme is provided and the database for the MC Setup program is updated with this information. Please read MC Definition List on page 225 for a detailed description of this process. Depending on the MC Definition List created, you may then proceed with Step 4: Define computations on page 155 or Step 7: Perform system (channel) check on page 156.

☞   MOOG control system only: The first two channels must be the NTP time data (DeviceName_NTP) and a 32-bit digital channel (DeviceName_COMMAND). All other channels appear with the names set up in the MOOG control system and cannot be changed in MD Server or MC Setup. Please check the channel names in your MOOG CS system before connecting: channel names containing certain characters are not only not imported (and present), the import of further channels is also terminated. The following characters are not allowed: colon (**,**), period (**.**) and angled brackets (**< >**).

💡   You may prepare the channel settings on several PCs: create a MC Setup database file as described in Step 1: Scan hardware on page 141. Then copy this file to several PCs having MC Setup/MC Definition List installed. Here several people may set up the individual devices used. Then import from those changed MC Setup database files just those devices which were set up in the corresponding file: **From existing MC Setup** (**Import** group on the **Database** ribbon tab). Use import **Mode**: **By hardware address** and specify a **Device index range**.

3.2.3 Step 3: Set up your channels

In this step the channels are set up according to their transducers. The bridge type or strain gage resistance used as well as the supply voltage or gage factor and other relevant settings have to be specified.

You have these options:

1. Using sensors with TEDS: use **Activate TEDS (system wide)** on the **Device** ribbon tab and you are done.
2. Using a Sensor database on page 159.

3. A Manual procedure for setting-up channels on page 162, using a calibration protocol or technical data sheet.
4. Import the settings, see Merge with MC Definition List on page 172, Import from an existing MC Setup database on page 172, MC Import: Import from a custom defined database (Excel, Access) on page 173.
5. You may also manually set up the devices with the MGC Assistant or MX Assistant and use **Get setup from devices**. However, this type of setup is time consuming and therefore not recommended for systems with more than one device.

Refer to Setting-up channels on page 157 for a comparison and description of these procedures.

3.2.4 Step 4: Define computations

Now computations may be defined. If no computation is necessary, this step can be omitted.
See Calculations (defining computations) on page 190.

As an alternative these settings may be imported from an older MC Setup or from a custom defined database, even from an Excel spreadsheet.
See Import from an existing MC Setup database on page 172, MC Import: Import from a custom defined database (Excel, Access) on page 173.

3.2.5 Step 5: Manual inputs required?

If necessary, you may also define manual inputs: **Manual input channel** on the **Database** ribbon tab or context menu **Create channel -> Manual input channel** on the Manual inputs group. This channel type allows you to type in data via keyboard, e.g. readings from analog instruments, load levels etc. (**Enter manual input** on the **Test** ribbon tab in MD Server). You may define up to 8 channels for manual input.
See also Manual input channel on page 191, Enter manual input.

☞ In former versions of catman Enterprise *Manual input* was named Load level input.

3.2.6 Step 6: Arrange channels into groups

It is very useful to arrange the channels and computations used into *groups* in section Function groups. These groups may later be accessed by the client PCs which makes it

easier to decide which channels should be used for evaluation on what client PC. A channel may occur in more than one group.

See also Grouping channels and computations on page 210.

💡 It is also possible to search for a specific type of channel and use the search results to add those channels to a certain Function group.

☞ Dragging is only possible with the right-hand mouse button.

3.2.7 Step 7: Perform system (channel) check

We recommend you check the system here, e.g. check whether the wiring is done properly, the transducers are all working and calibrated etc.: **Amplifier/Sensors** ribbon tab. See Checking channels on page 211.

3.2.8 Step 8: Save settings

Finally the hardware part of the settings has to be transferred to the devices: click on

Transfer setup to devices (**Device** ribbon tab). This saves all settings to the RAM of MGCplus or QuantumX/SomatXR. However, when closing MC Setup you are also asked whether you want to transfer all or only changed channel settings to your devices.

Additionally, all data is stored in the MC Setup database. The name of this database is shown in the caption of the MC Setup window and this database should then be used in the project settings (default setting, if MD Server is installed on the same PC).

MGCplus

To preserve these settings permanently on MGCplus devices, all settings are saved in the internal Flash-EPROM of each channel when quitting MC Setup. You may use **Save to EEPROM** on the **Device** ribbon tab to save settings manually to all channels.

☞ Use menu **MC Setup -> Options** and define on the **EEPROM** tab whether the settings should never be saved automatically (preferred option) or whether you would like to get a dialog. The setting is only important for MGCplus and has no effect with QuantumX/SomatXR, see EEPROM options tab on page 221.

QuantumX/SomatXR

With QuantumX/SomatXR, if anything is saved, all settings are automatically saved in the internal Flash-EPROM for all channels of a module. An additional saving procedure for

permanent storage is not necessary. Use **Exit without saving** on the **EEPROM** tab in the **Options** dialog (EEPROM options tab on page 221).

3.3 Setting-up channels

ℹ️   See also MC Definition List on page 225, Using CANHEAD (MGCplus only) on page 146.

This topic corresponds to Step 3: Set up your channels on page 154 in topic Getting started with MC Setup. Most settings referring to hardware are made with the **Device** and **Amplifier/Sensors** ribbon tabs.

💡   Click on one of the devices to limit the number of channels displayed on the right-hand side.

The proper settings must be specified for every channel connected to a transducer (sensor). There are several possibilities for a channel setup:

1.   Activate TEDS when using sensors with TEDS module.
      Use **Activate TEDS (system wide)** on the **Device** ribbon tab which scans all devices for sensors with TEDS and activates the settings defined in the TEDS. If required settings cannot be made an error message will be issued. Check the log file in this case (**Show** in the **MC Setup log** group on the **Diagnosis** ribbon tab). See also Diagnosis and Error handling on page 216.

      ⚠️   With MGCplus, TEDS modules are not considered automatically. You must use **Activate TEDS (system wide)** on the **Device** ribbon tab to activate the TEDS contents. Without this action you may even override all TEDS settings with MC Setup by manually defining scaling, unit etc.
            With QuantumX/SomatXR, the TEDS contents are usually read out when connecting the sensor to the module. However, the TEDS usage might have been set manually to "Ignore TEDS" with the MX Assistant. Use **Activate TEDS (system wide)** to enable and read out all TEDS contents again. This will take approx. one minute.

2.   Use a Sensor database.
      This enables you to specify all sensor data once only in the database and then assign sensors with drag & drop.
      See Using a Sensor database on page 159.

3.   Specify the settings manually.
      See Manual procedure for setting-up channels on page 162.

4. Import settings from other databases.
    See Import from an existing MC Setup database on page 172, MC Import: Import from a custom defined database (Excel, Access) on page 173.
5. You may also use MGC Assistant or MX Assistant to manually set up the devices. This type of setup is time consuming because you have to set up each channel individually, there are no copy and paste or import options available as with the above described procedures. Therefore we do not recommended this procedure for systems with more than one device. After having set up all channels, perform a system scan and use **Get setup from devices** which will read out all channels of all devices and overwrite any existing setting in the MC Setup database.

💡 We recommend checking *all* channels after a setup. This ensures proper operation and valid data. See Checking channels on page 211, on how to check hardware channels.

⚠️ For all MGCplus 8-channel amplifier plug-in modules the settings **Sensor type**, **Uexc** (excitation voltage) and **Filter** are restricted: only identical settings for all 8 channels may be chosen. If you assign different settings to these channels, the last setting is used for all channels.

Channel setup

MGCplus: displays a small window, in which all settings for the marked channel may be made.

QuantumX/SomatXR: starts the MX Assistant, if the software is installed and the path is set, see Diagnosis options tab on page 222.

☞ You must install the required MX Assistant software manually, it is not included in the catman Enterprise setup. Either download it from the HBM website (HBM downloads) or use the CD included with your QuantumX/SomatXR module delivery.

Advanced setup

With MGCplus this menu allows for access to specific functions. Depending on what you mark on the left side you get different dialogs:

- Mark the device to change the CP options.
- Mark a slot with an ML74 to assign CANHEAD modules, rename CANHEADs or use **ML74 -> Set minimum number of CANHEADS** to specify the CANHEAD delay, see also Using CANHEAD (MGCplus only) on page 146.

● Mark a multi-channel plug-in card to select whether this card uses 2, 4 or 8 channels.

With QuantumX/SomatXR this menu starts the MX Assistant, if the software is installed and the path is set, see Diagnosis options tab on page 222.

☞ You must install the required MX Assistant software manually, it is not included in the catman Enterprise setup. Either download it from the HBM website (HBM downloads) or use the CD included with your QuantumX/SomatXR module delivery.

💡 You may also pre-define here Trigger settings, pre-set the Prediction line or zero channels, refer also to Trigger settings on page 58, Check for certain values in two associated channels (Prediction line) on page 62.

3.3.1 Using a Sensor database

What do you need the Sensor database for?

The sensor database is used for the easy setup of the measuring chain for the sensor used. You only need to enter the sensitivity/characteristic and technical data of a sensor *once* and then you can assign this sensor to the channel to which it is connected with a mouse click. Then the device is appropriately set up provided the sensor type is supported by the device.

Using a sensor database is convenient if you have many different transducers without TEDS and if you cannot use one of the importing functions or MC Definition List. With strain gages it might be easier to use one of the other setup methods.

Which Sensor database is used?

Use **HBM -> SensorDatabase_B** in the programs list of Windows to call up the Sensor database program, then open the Sensor database used by MD Server and MC Setup: C:\Users\<Username>\Documents\HBM\MCSComponent\HBM_Sensordatabase.sdb.

💡 Use one file as main database, e.g. with MX Assistant, and copy it after a sensor recalibration to the server PC.

All information about how to define specific sensors can be found in the "Sensor database help" of program SensorDatabase_B.

The Sensor database is compatible with a User sensor database loaded into the MX Assistant and the Sensor database used by the MGCplus Assistant. If you use one of these programs there is the same help on the Sensor database available. Make sure you use the correct Sensor database file when modifying.

☞ Even though units in the Sensor database may contain more than 4 characters this is not supported for hardware channels by MC Setup. All entries longer are shortened.

How to search for a sensor

Use **Show Sensor database** in the **Device** ribbon tab to display the **Sensor database** window. Type part of the sensor name into the search field. There is no need for wildcards, all sensor names containing the entered string will be displayed (see **Name/Description** field in the Sensor database).

3.3.1.1 Add your sensors to the Sensor database

Start the Sensor database program (separate program) with **HBM -> SensorDatabase_B** in the programs list of Windows.

Before you enter a transducer into the Sensor database for the first time, you should consider the future structure of your database. After the installation, several transducer types are available to you as sensor templates under **Sensor groups**. These templates cannot be modified and are therefore identified with 🔒. You can however enter more groups under **Sensor groups** or use—preferably—the **My sensors** group.

Define the structure of the Sensor database

With many transducers (over 50) you may you either create subgroups in the **My sensors** group or create several main groups. Depending on the size and organization of your company, you can, for example, group the transducers according to departments and therefore create a main group for each department. However, as with the templates, you can also use the individual types of transducers as main groups or define your company name as a main group and the transducer types as subgroups of it. The structure is not needed to find transducers, but it is only recommended with many transducers to obtain a quicker overview. In the case of doubt, you can always search the database for transducers.

How to enter a sensor?

1. Mark the group to which you would like to add a new entry.
2. Using the context menu or the **Edit** group, create a new sensor or a new CAN signal (or a new group).
3. Enter the transducer data.

As an alternative to creating a new sensor you may copy a similar sensor, paste it into your group and then modify the data, e.g. the sensitivity. All information about how to define specific sensors can be found in the online help for the Sensor database (access from the Sensor database).

Notes

- A maximum of 1024 sub-entries (lower half) per group can be shown. Therefore use an appropriate structure which ensures that no group exceeds this number of entries.
- If you enter a **Serial number**, it is displayed in round brackets after the name in the **Sensor database** window in MC Setup.
- Further information on how to define specific sensors can be found in the online help for the SensorDatabase_B program and in programs MX Assistant and MGC Assistant.

3.3.1.2 Assigning sensors from the Sensor database

Procedure

1. Click on **Show Sensor database** to display the **Sensor groups/Sensors** windows.
2. Mark the channel(s) to which the sensor(s) should be assigned.
3. Either click on a sensor group, e.g. **HBM transducers -> Force transducers -> C9B**, or enter **C9B** in the search field (🔍) in the lower half of the **Sensor database** window if you have connected this type of transducer.
4. In the lower section of the **Sensor database** window, double-click on the correct sensor.

   💡 You can also assign a sensor with drag & drop to *one* unmarked channel or to *all marked* channels.

In the Sensor database it can be defined that a sensor may only be used on *one* channel. In this case you obtain an error message for multiple or renewed assignment.

☞ Assign CAN signals with the corresponding Assistant program for your device or use one of the other methods for assigning sensors. The assignment of CAN signals from the Sensor database is currently not supported.

5. Repeat these steps until all connected transducers have been assigned.

☞ Any transducers already defined in the device, e.g. using the Assistant program, are *not* recognized by MC Setup. Only the relevant settings may be transferred to MC Setup with **Get setup from devices** or **Load from EEPROM** (MGCplus only).

Immediately after assigning a transducer, the setting will be checked. Should an error message occur, check whether this transducer type may be connected to this amplifier type (sensor type, measuring range, connection plate etc.). When transferring the setup to the devices, the settings are checked again and if not all of the settings specified in the Sensor database can be accepted, e.g. the desired excitation voltage is not available, the next possible setting is used. Because the setting made is transferred back to MC Setup, the data shown after transferring a setup to the devices or modules always reflects the current state.

3.3.2 Manual procedure for setting-up channels

The proper settings must be specified for every channel connected to a transducer (sensor). We recommend marking all channels having identical settings and select the setting from the context menus of the columns. This saves you having to set each channel separately. If the channels are not in consecutive order, it might be easier to form a group with identical channels and do the setup then. As an alternative you can also copy many settings to other rows.
See also Grouping channels and computations on page 210.

☞ The settings you can choose depend on the hardware used in the devices. Every MGCplus slot corresponds to a certain amplifier plug-in module and a connector plate on the rear side. *Both* determine the possible settings. With QuantumX/SomatXR the *module type* may restrict the type of sensors which can be attached to a connector.
Also some settings are inaccessible as soon as certain units are selected: for example, all scaling information entry is blocked and only gage factor setting is

possible as soon as **µm/m** is chosen as unit. Another example is the setting of half or full bridge for strain gage transducers: this setting is only accepted when choosing additionally a (valid) supply voltage, the setting 0 V is not allowed.
A polynomial sensor scaling is not supported. If such a scaling is stored in a Quantum X/SomatXR channel, do not use *any* scaling in MC Setup. Otherwise the configuration in the QuantumX/SomatXR channel is overwritten on initialization.
The entry length of units for hardware channels is restricted to 4 characters. Software channels may have up to 8 characters.

If you want to use a tabular scaling, right-click in the **Software scaling** column and choose **Linearization table**. For temperature compensation with strain gages, use context menu **SG temperature compensation** in the **Software scaling** column.

See also .

Rescan sensor IDs / Activate TEDS

You may initiate a sensor ID re-scan from the **Device** ribbon tab. If transducers have been moved to a different channel, this is indicated in the log, if a sensor ID match is found. Sensor IDs are contained in T-ID and TEDS modules (sensors with TEDS).

for a scenario involving the use of T-IDs.

If you have TEDS sensors connected we recommend you use **Activate TEDS (system wide)** (**Device** ribbon tab) which re-scans all devices for sensors with TEDS and activates the settings defined in the TEDS.

⚠️ With MGCplus, TEDS modules are not considered automatically. You must use **Activate TEDS (system wide)** on the **Device** ribbon tab to activate the TEDS contents. Without this action you may even override all TEDS settings with MC Setup by manually defining scaling, unit etc.
With QuantumX/SomatXR, the TEDS contents are usually read out when connecting the sensor to the module. However, the TEDS usage might have been set manually to "Ignore TEDS" with the MX Assistant. Use **Activate TEDS (system wide)** to enable and read out all TEDS contents again. This will take approx. one minute.

Load from/save to EEPROM

Save to EEPROM: this function is useful for MGCplus devices: all settings are transferred to the internal EEPROM and therefor saved permanently. Otherwise settings from MC Setup are saved only to RAM and lost if the MGCplus is switched off long enough (approx. 8 hours).

With QuantumX/SomatXR all settings transferred to these devices are permanently saved within a few seconds. So this action is not required for these modules.

Load from EEPROM: loads the current setting from the device and overwrites any setting in the MC Setup database.

Restore factory settings

MGCplus: loads the factory setting.

QuantumX/SomatXR: resets the channel names, *no other setting* is changed. If you use sensors with TEDS and a channel name is set there, this name is used as channel name afterwards.

☞ A new system scan has to be made if these settings shall be used in MC Setup.

3.3.3 MC Setup column description, scaling functions

The following table lists all columns and provides information about the settings. See also the MGCplus Operating Manual for a description of all hardware related items like excitation voltage or filter.

Column title	Settings and comment
Name	Name of the channel. This is a very important setting, because *all program parts* rely on unique channel names and identify channels by this name, even if different hardware (other MGCplus devices) is used. In most cases this will be a default or generated name. (1), (2), etc. is appended if identical names occur e.g. on merging or import.
Sensor type	Depending on the hardware and sensor used, the sensor type must be specified. For most sensors, this is the bridge type or wiring scheme connected. Use the right mouse key to get the context menu with the possible settings.
Exc	Excitation voltage for the transducer if applicable. This column has no meaning for DC input. Use the right mouse key to get the context menu with the possible settings.

Column title	Settings and comment
Filter	Depending on the hardware a filter type and frequency may be selected. Check with the MGCplus Operating Manual for details concerning filter behavior. Use right mouse key to get the context menu with the possible settings. ☞ When having ML801B plug-in modules with firmware 6.56 or higher you may also select the **Bessel CANHEAD** filters for ML801. This ensures that these modules use the same filters (filter characteristic and phase delay) as your CANHEAD modules, see also Using CANHEAD (MGCplus only) on page 146.
Gage factor	For strain gages only: "sensitivity" of a strain gage.
Bridge factor	For strain gages only. The setting depends on the hardware or the wiring scheme used: it is not possible to input a bridge factor for a quarter bridge. A bridge factor may only be typed in for half or full bridges.
Wiring comp. factor	For CANHEAD modules with strain gages in three-wire circuit only. The signal loss occurring in this case can be measured by the hardware and is compensated by a higher gain factor. The number indicated is the loss factor. See also Perform CANHEAD wiring compensation (cable length compensation) on page 214.
Electrical zero	Electrical value for zero point, see Hardware scaling on page 168.
Physical zero	Physical value (quantity) for the zero point, see Hardware scaling on page 168.
Electrical nominal	Electrical value for nominal load, see Hardware scaling on page 168.
Physical nominal	Physical value (quantity) for nominal load, see Hardware scaling on page 168.

Column title	Settings and comment
Engineering unit	Unit of the physical quantity to be measured. Use the right mouse key to get the context menu with the possible settings. You may however also type in a unit. For software channels up to 8 characters are possible. Because the MGCplus can display only 4 characters, all entries for hardware channels are restricted to 4 characters.
Max. prediction (formerly Physical 100%)	Allows you to pre-set a certain value as 100% value, e.g. for a load level visualization, as maximum value for the prediction line trigger or as default for the 100% load condition (also called load point or end point), if you use MD Trend. See also Prediction line in Real-time graph on page 78 and Post-process graph on page 80, Load level visualization on page 96, Check for certain values in two associated channels (Prediction line) on page 62 and topic "Full scale value of percentage channel" in MD Trend help.
Upper trigger level	Allows you to pre-set a certain value for the above (>) trigger condition.
Lower trigger level	Allows you to pre-set a certain value for the below (<) trigger condition.
Tolerance % of prediction (formerly Percent of prediction	Allows you to pre-set a certain value as tolerance value for the prediction line display and trigger.
Software scaling	Enables you to specify a scaling with *more than two points.* This scaling is done in software, *not inside the MGCplus hardware,* see Software scaling / strain gage temperature compensation on page 170.

Column title	Settings and comment
Zero enable	Allows zeroing of the channel. If in the main window of the MD Server the zero button is clicked (0), only channels with this field activated (**On**) participate in the zeroing process. ☞ This setting should be deactivated for temperature channels. This applies also to digital channels or counter values if the values change during the balance. Otherwise this can lead to invalid values, e.g. 0.5 as zero value for a digital input that can only be 0 or 1 but has changed its status during the measurement.
Zero value	Displays the current (last measured) zero value. The last 10 zero values per channel are always stored in the MC Setup file.
Auto CAL (Autocalibration)	Shows the state of the autocalibration feature. With MGCplus autocalibration is always switched off during a measurement, an ON setting enables autocalibration only during periods without data acquisition. With QuantumX/SomatXR the default calibration method is activated in ON state: With some modules every few minutes the second amplifier of each channel is separated from the sensor input and calibrated. The zero and range calibration of the first amplifier chip is then corrected with the information derived from this calibration and the comparison of the output between the two amplifiers (they reside on a single chip). It depends however on the measuring range and module and sensor type whether autocalibration is available. See also Autocalibration with MGCplus and QuantumX/SomatXR on page 55.
Ref. x channel	Double-click into the column to get a channel selection dialog. Any x channel (e.g. a load channel) to be used as reference may be specified here. With a prediction line trigger this channel is used as x channel. In y(x) graphs this channel is used as default for the x-axis if the channel of this row is assigned to the y-axis.

Column title	Settings and comment
Sensor ID	If a sensor with T-ID or TEDS is connected, the Sensor ID, a world-wide unique number, is provided in this column. See also Setting-up channels on page 157, Getting started with MC Import on page 174.
Sensor description	In these fields additional information may be specified. As an option some fields may already contain specific data from a customer database. There is no information retrieved from the signal conditioning devices MGCplus.
Descriptor file	For computations the formula used will be displayed in the comment line. The maximum length for a comment is 128 characters. If a descriptor file is specified, this file may contain further information and can be accessed from different parts of the software as
Comment	well as from a client PC. We recommend to check the access rights in Windows because client PCs may access the file only when the proper access rights are assigned in the Windows operating system.
Slot	
Amplifier (Amplifier plug-in module)	These fields contain information derived from the signal conditioning devices (MGCplus and QuantumX/SomatXR modules) and cannot be changed.
Connector (Connector plate on rear side)	
Serial number(s)	

3.3.3.1 Hardware scaling

ℹ See also Software scaling / strain gage temperature compensation on page 170.

As a rule, hardware scaling is used in MD Server. This means that all computations necessary for scaling an electrical value which is measured by the signal conditioning devices to a read-out representing the physical quantity is done inside the MGCplus or QuantumX/SomatXR. Therefore no workload is added to the CPU of the server PC for this

task. The scaling principle for a standard force transducer with a sensitivity of 2 mV/V for nominal load is shown in the following picture.
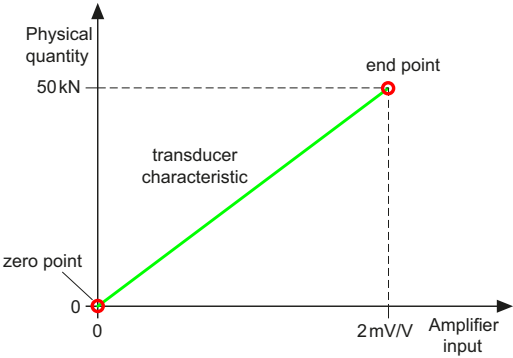
Fig. 3.1   Scaling principle

For two points, usually zero and nominal load of a transducer, a pair of values is given in the calibration certificate: the electrical value which corresponds to a measured value and the value of the physical quantity at this point. The line connecting these two points is called sensor characteristic. Most calibration certificates use 0/0 for the zero point (electrical/physical zero) and nominal output (electrical nominal, e.g. in mV/V) at nominal load (physical nominal, e.g. in kN). However, there may also be specifications like 0.0123 mV/V at 0 kN and 2.021 mV/V at 500 kN. In this case enter **0.0123** for the **Electrical zero, 0** for **Physical zero, 2.021** for **Electrical nominal** and **500** as **Physical nominal**.

☞   In order to ensure proper operation, at least one of the two zero values must be zero when using hardware scaling. It is not allowed to specify, for example, **5 kN** and **0.2 mV/V** as zero value pair. If such values are typed in, the program re-calculates a corresponding zero value for either electrical or physical quantity.

After specifying the characteristic, a zero measurement (, **Amplifier/Sensors** ribbon tab) is mandatory. This stores the actual electrical zero value which considers pre-loads due to the current transducer mounting etc. in the zero buffer inside the signal conditioning devices, in the MC Setup database (**Zero value** column) and—most important for the measurement data—during a test run in file ZERO.BIN. The MC Setup database file contains the last 10 zero measurements, the ZERO.BIN file all zero measurements made during a test run.

☞   The hardware scaling in MC Setup/MD Server uses a two-point characteristic, not a zero-span characteristic as is the case in the MGCplus signal conditioning devices. Hence the values displayed with the MGCplus Assistant may be different from the values entered here.

3.3.3.2 Software scaling / strain gage temperature compensation

🛈   See also Hardware scaling on page 168.

Software scaling works similar to the hardware scaling normally used. In contrast to hardware scaling, the CPU power of the server PC is used. This enlarges the workload and has to be used with caution therefore. On the other hand, this procedure has some advantages:

- With strain gages special temperature compensation methods are available.
- With standard transducers (sensors) more than two points may be specified for scaling. This enables you to linearize sensor characteristics with up to 16 points.
- The restriction for the length of unit entries for hardware channels (4 characters) does not apply for software scaling. Here 8 characters are possible.
- For channels from external devices, a zeroing may be done when using a software scaling. Use a "fake" scaling with 0/0 (first row) and 1/1 (second row) as scaling points in this case.

⚠   In all cases, the scaling computation is made only during a *later data acquisition*. It is not made when using functions of MC Setup like wiring check etc. The measured values displayed in these windows are always "raw data" directly from the measuring devices without any computation.

Use the context menu in the **Software scaling/SG temperature compensation** column to define either a **Linearization table** or provide the data for a **SG temperature compensation**.

Linearization of sensors (Linearization table)

The input of up to 16 pairs specifying the linearization points is possible on tab **Linearization table**. A linearization is carried out with linear interpolation between these points. Instead of typing in values from e.g. a calibration protocol it is also possible to measure the electrical values: double-click in the left column in the required line and press [F5] or click on **Measure**.

SG (strain gage) temperature compensation

☞ Use button ⌊ ... ⌋ to open the Channel selector.

Two possibilities are available here apart from the "classical" solution which uses a half bridge with an active strain gage and another strain gage of the same type (same lot number of the foil) which only acquires the thermal expansion of the component:

1. **Temperature compensation with compensation strain gage**
   By using a separate DAQ channel with the same type of strain gage ("dummy" strain gage) which only acquires the thermal expansion of the component (separate piece from identical object material). This measurement can be subtracted from many active channels, each containing only one active strain gage. This means that many active channels can be compensated with one compensation strain gage if the temperatures of these strain gages are similar enough. In this case select **Temperature compensation with compensation S/G**. Then specify the channel measuring this dummy strain gage.

2. **Compensation with temperature channel and polynomial function**
   Here too, the compensation for several active strain gages can also take place using one channel measuring temperature in this case. This is possible if the temperature range is not too big and if the material used is the one the strain gages are matched for. Select **Compensation with temperature channel and polynomial function**. Then specify the channel which acquires the temperature at the measuring point(s). In addition to the polynomial coefficients **P(i)** you must also specify the thermal expansion coefficients and the reference temperature: **alpha SG** is the thermal expansion coefficient of the strain gages in ppm (10^{-6}/K) as stated on the strain gage package, **alpha material** is that of the material on which the strain gages are installed and T_{ref} is the temperature in °C for which the polynomial has been determined with a deviation of zero. This is the point at which the polynomial curve passes through the line at zero strain (usually 20°C). To additionally compensate for the temperature dependency of the gage factor (**alpha gage factor**, approximately 1% per 100°K with Constantan strain gage), also specify the value (in ppm/K). Otherwise leave the field blank or enter **0**.

Formula for temperature compensation with strain gages

$$\varepsilon = \varepsilon_{\text{measured}} \cdot \frac{\text{Gage factor}}{C} -$$

$$- \left(a_0 + a_1 T + a_2 T^2 + a_3 T^3 + \ldots \right) \cdot \frac{2}{C} -$$

$$- \left(\alpha_{\text{Material}} - \alpha_{\text{SG}} \right) \left(T - T_{\text{Ref}} \right)$$

with $C = \text{Gage factor} \cdot \left(1 + \alpha_{\text{Gage factor}} \left(T - T_{\text{Ref}} \right) \right)$

3.3.4 Merge with MC Definition List

Menu item **MC Setup -> MC Definition List -> Merge with MC Definition List** allows you to merge an existing MC Definition List (database) to a MC Setup database created by a system scan. This procedure is described in detail in MC Definition List on page 225, topic Merging MC Definition List and hardware on page 237.

3.3.5 Import from an existing MC Setup database

There are several scenarios where an import from an existing MC Setup database is convenient:

1. You use a system which has only minor changes compared to an older setup or which needs the same type of data already used in a previous setup.
2. You have a system with hundreds of channels and several people who can set up the system simultaneously. In this case scan the whole system as usual (Step 1: Scan hardware on page 141), distribute this file to all of them and assign everyone a certain range of devices. You may then "combine" these setups with the import feature using the **By hardware address** mode and specify a **Device index range**.
3. With this type of import you can also import some other settings besides channel names and scaling like Function groups or trigger definitions.

Before starting any action, a backup of the currently used database file is made.

How to proceed

1. Choose **From existing MC Setup** in the **Database** ribbon tab.
2. Select with ... the MC Setup database file from which the import is to be made.
3. Specify whether the channel name (**By name**) or the channel hardware position (**By hardware address**) shall be used for identification. In the latter case you may

restrict the range of channels to be imported: **Device index range**.
If using **By name** and the name does not exist, the corresponding settings are not imported. This applies also to computations, if the channels (channel names) used do not exist. Missing channels in Function groups are just not added to the group.

4. Go through the **Hardware independent options** and deselect the options not required. As default all options are selected here.

5. Frame **Hardware dependent options** shows further options for import. Whether these settings of the database to be imported are valid or even possible for the actual MC Setup database depends on the hardware used for a channel. For example, with MGCplus 8-channel amplifier plug-in modules these channels may have certain settings only as identical settings for all channels. But also settings like **Bridge factor** are only valid for channels connected to strain gages, not e.g. for channels connected to force transducers.
    If one of the settings is not allowed or possible for a specific channel, no setting is imported and this channel remains unchanged. After the import a message is displayed and the MC Setup log file contains entries specifying the channels not imported, see MC Setup log on page 216.

6. Start the import with **OK**.

7. Update/transfer all data to the devices and check the log file for import problems (**Show** in the **MC Setup log** group on the **Diagnosis** ribbon tab).
    See also Diagnosis and Error handling on page 216.

💡 The dialog allows not only importing trigger definitions but also a complete set of Function groups. Of course, the channels (channel names) defined in the Function groups must be available, otherwise these channels are skipped.

☞ During the import a message may appear, asking you whether a specific (already existing) table shall be overwritten or not. If you do not know the table and its meaning just click on **No** and the import will continue.

3.3.6 MC Import: Import from a custom defined database (Excel, Access)

🛈 See also Getting started with MC Import on page 174 for different scenarios explaining the advantages of the different import possibilities

Databases of two formats may be imported: MS Excel and MS Access. Before starting any action, a backup of the currently used database file is made.

⚠️ The import just imports the data specified. There is no check made, whether this data is appropriate and might be used as setting for a specific channel. This will be detected when later transferring the data to the channels. In that case these channels which could not be set according to the information provided will be marked in MC Setup. Use the Find channels section (bottom left) with filter **which are not OK** and the group Search results to display the channels in question.

The message "Fields with invalid values detected" indicates that for some fields the formatting could either not be detected or may be different in the same column. Entries considered invalid are marked with a *red question mark*. Usually this is no problem as the format will be set as soon as this column is dedicated to a destination column. If however the column is superfluous, empty content is irrelevant.

💡 If you plan to import data we recommend you set up a test system first and export the settings e.g. via **Table (MS Excel)** (**Export** group) to an Excel file in order to see the possible entries and tables (spreadsheets) in the Excel workbook. Depending on the export option you use, spreadsheets T_FGroup and T_SubAmp may be exported too. You may delete T_SubAmp, it is for internal use only. Spreadsheet T_FGroup can be used to specify function groups and subgroups. Use square brackets around each name and separate subgroups with a semicolon. Then specify the appropriate group(s) on the T_Channel sheet (FunctionGroups column): `[Group1;subgroup1];[Group2]` etc.
If you plan to use Access, import worksheet T_Channel from this Excel file into Access to get the appropriate tables and fields.

3.3.6.1 Getting started with MC Import

MC Import is a very versatile program with many features and offers several alternatives how to use it. In order to provide you with some ideas we describe in the following several scenarios. However, these scenarios are not the only ones possible, they just describe commonly used practices.

If you plan to use Access, create a small test system with the relevant sensor types first and export the settings to an Excel file, refer to . Then import worksheet T_Channel from this Excel file into Access to get the appropriate tables and fields.

Operating scenarios

Principle procedure

- Select your database file.
- Specify the source table or worksheet you want to use.
- Select which column of this table shall be used to extract what information: name, supply voltage, unit etc.
- Drag & drop the required channels or start one of the auto import/update processes.

The import process can be done automatically for *several* channels, if the information necessary for selecting the appropriate channels is provided by the database file: either device, slot and channel number (**Start auto import**) or an identification column, e.g. the channel name or Sensor ID (Update). Otherwise a "manual import" must be made: mark the rows to be copied and drag them to the appropriate destination rows (lower table) using the *right-hand* mouse key. A dotted blue line indicates where the data will be inserted.

Creating special channels (non-hardware channels)

You may import computations from an Excel file: state them in Excel with the same syntax as you would in MC Setup, see Calculations (defining computations) on page 190. For creating a calculation channel use the following columns in worksheet T_Calc.

Calculation channel

Worksheet	Column	Value
T_Calc	Name	Channel name
	Formula	Calculation syntax
	Unit	Channel unit (optional)
	Type	0 (code for standard calculation channels)

For creating a Manual input channel on page 191 or Virtual channels on page 207 use the following columns in worksheet T_Calc for providing the necessary information.

Manual input channel

Worksheet	Column	Value
T_Calc	Name	Channel name
	Formula	Name of a variable from GP17 to GP24, e.g. GP17
	Unit	Channel unit (optional)
	Type	10 (code for Manual input channel)
	TransSensA	Value for manual input

Virtual channel

Worksheet	Column	Value
T_Calc	Name	Channel name
	Unit	Channel unit (optional)

	Type	Code for function:
		    21: continuous ramp
		    95: sawtooth
		    22: constant

	TransSensA	Function parameter 1:
		    ramp: interval
		    sawtooth: interval
		    constant: value

	TransSensB	Function parameter 2 (sawtooth only): max. value (calculated, see below)

The sample rate for the continuous ramp is determined by **Rate 1** in **Regular DAQ mode** and **Slow rate** in **Fast trigger mode**.

The sawtooth function always uses 10 steps before having the falling edge. Additionally the max. value for the sawtooth has to be calculated: subtract 1/10 of the interval value from your desired maximum as parameter 2. Example: desired maximum is 2500, interval is 20. In this case, enter 2498 as max. value.

Using Excel spreadsheets and using drag & drop for assigning settings

This is the easiest way of copying settings when no special identification for the channels is necessary, e.g. because they are all of the same type. You may also use manual drag & drop with the right-hand mouse key for selected channels only, even if you already used one of the other methods, see .

How to proceed

1. Create an Excel file. Use three spreadsheets, if applicable: one for channel data like name and (later) scaling (T_Channel), one for computation channels (T_Calc) and one for sensors requiring a special user scaling (T_UserScale). You may omit spreadsheets which are not required.

2. Define for all data you want to set up the required columns (use the first row as header name) and enter the appropriate data.

    ☞ Set up a test system first and export the settings to an Excel file in order to see the required entries, see Manual procedure for setting-up channels on page 162.

3. If required, you may also set up the other spreadsheets for computations or a special sensor scaling.

4. Save your Excel workbook.

5. Start MC Import (**From custom database** in the **Import** group on the **Database** ribbon tab) and open this Excel file (**Open source file**).

6. Select for the upper table which displays the contents of your first Excel spreadsheet (sorted by name) the proper worksheet: **Source table**.

7. Specify for all columns which type of data is contained: use the context menu in the header line to set e.g. **Channel data -> Name**, **Sensor type**, **Gage factor** etc. You may switch to another source table and, for example, specify a formula channel.

    You may save this channel assignment with so that you can use it again for your next import.

8. Now mark the rows (channels) containing the data you want to assign and drag them from the upper table to the lower table using the *right-hand* mouse key. A dotted blue line indicates where the data will be inserted.

9. Repeat this procedure as often as necessary to copy all required rows from your Excel spreadsheet(s) to the appropriate rows of your MC Setup database.

10. Close MC Import and update/transfer all data to the devices. Check the log file for import problems (**Show** in the **MC Setup log** group on the **Diagnosis** ribbon tab) or use the Find channels option with filter **which are not OK**.
    See also Diagnosis and Error handling on page 216.

Using Excel spreadsheets and identifying channels by name

A common approach is to set up the channel names first and then import all other settings according to these channel names. The drawback to this approach is that the channels must be capable of using the assigned sensor type. There is no check made in the process itself, just after trying to upload the settings later you might get warnings or error reports if a sensor connection is not available for a particular channel.

How to proceed

1. Create an Excel file and define the channel names in an Excel workbook. Use three spreadsheets, if applicable: one for channel data like name and (later) scaling (T_Channel), one for computation channels (T_Calc) and one for sensors requiring a special user scaling (T_UserScale). You may omit spreadsheets which are not required.
2. Start MC Import (**From custom database** in the **Import** group on the **Database** ribbon tab) and open this Excel file (📥 **Open source file**).
3. Select in the upper table which displays the contents of your first Excel spreadsheet (sorted by name) the proper worksheet: **Source table**.
4. Click on the header of the column containing the channel names and select from the context menu **Channel data -> Name**.
5. Either drag the whole column or mark the required number of rows and drag them from the upper table to the lower table using the *right-hand* mouse key. A dotted blue line indicates where the names will be inserted.
6. Close MC Import and update/transfer all data to the devices. Now the channel names are defined.
   See .
7. Export this setup to a new Excel workbook (**File -> Export table (MS Excel)**).

This will now act as a template for your future setups. If you want to add more channels or change a setting, change it here. Then:

1. Call up MC Import (**From custom database** in the **Import** group on the **Database** ribbon tab) and open this Excel file (📥 **Open source file**).
2. Click on the channel name (**ChanName**) column and use context menu item **Set as identification column**.
3. Specify for all columns which type of data is contained: use the context menu in the header line to set e.g. **Channel data -> Name**, **Sensor type**, **Gage factor** etc. You may switch to another source table and, for example, specify a formula channel. You may save this channel assignment with 🔧 so that you can use it again for your next import.
4. Mark the whole table (left upper field **Source**) or the required rows (channels) and click on 🔧 (**Update**). This will transfer all settings to the appropriate channels, identifying the channels by their name.

Using Excel spreadsheets and identifying channels by T-ID or TEDS

In this scenario all (or at least most of the) sensors are equipped with either a T-ID or a TEDS module. The unique number contained in this module is used to identify the channel.

When having TEDS sensors, just read out the information and set the channels accordingly: **Activate TEDS (system wide)** on the **Device** ribbon tab. When having T-ID modules, make sure all T-IDs are read out: **Rescan Sensor IDs (system wide)** on the **Device** ribbon tab. Then use in both cases the following procedure.

⚠️ With MGCplus, TEDS modules are not considered automatically. You must use **Activate TEDS (system wide)** on the **Device** ribbon tab to activate the TEDS contents. Without this action you may even override all TEDS settings with MC Setup by manually defining scaling, unit etc.
With QuantumX/SomatXR, the TEDS contents are usually read out when connecting the sensor to the module. However, the TEDS usage might have been set manually to "Ignore TEDS" with the MX Assistant. Use **Activate TEDS (system wide)** to enable and read out all TEDS contents again. This will take approx. one minute.

💡 HBM provides special hardware and software for connecting T-ID modules via USB to a laptop and reading out the Sensor ID into any Excel spreadsheet cell (HBM 1-Wire). This simplifies the process considerably as the Sensor ID may be read out directly after the installation without any measuring device connected. Please contact HBM Technical support, see Technical support on page 29 for further information.

How to proceed

1. Export the current setup with your hardware configuration to an Excel workbook (**Table (MS Excel)** in the **Export** group on the **Database** ribbon tab).
2. Define the channel names in the Excel workbook on spreadsheet T_Channel. You may delete the other spreadsheets, if they are not required. One is used for computation channels (T_Calc) and one for sensors requiring a special user scaling (T_UserScale).
3. Delete columns ChanAddress, SlotAddress and DevIndex in the Excel spreadsheet T_Channel.
4. Define for all data you want to set up the required columns (use the first row as header name) and enter the appropriate data.

☞ Set up a test system first and export the settings to an Excel file in order to see the required entries, see Manual procedure for setting-up channels on page 162.

5. Save your Excel workbook.
6. Start MC Import (**From custom database** in the **Import** group on the **Database** ribbon tab) and open this Excel file (📄 **Open source file**).
7. Select in the upper table which displays the contents of your first Excel spreadsheet (sorted by name) the proper worksheet: **Source table: T_Channel**.
8. Click on the header of the column containing the Sensor IDs (**Identifier**) and select from the context menu **Set as identification column**. Additionally specify that this column contains the Sensor ID: context menu **Channel data -> Sensor ID**.
9. Specify for all columns you want to import which type of data is contained: use the context menu in the header line to set e.g. **Channel data -> Name**, **Sensor type**, **Gage factor** etc. You may switch to another source table and, for example, specify a formula channel.

   You may save this channel assignment with 🖼 so that you can use it again for your next import.
10. Mark the whole table (left upper field **Source**) or the required rows (channels) and click on 🖼 (Update). This will transfer all settings to the appropriate channels, identifying the channels by their Sensor IDs.
11. Close MC Import and update/transfer all data to the devices. Check the log file for import problems (**Show** in the **MC Setup log** group on the **Diagnosis** ribbon tab) or use the Find channels option with filter **which are not OK**.

    See also Diagnosis and Error handling on page 216, Search functions on page 216.

Using Excel spreadsheets and identifying channels by their hardware position (slot/device/channel address)

This scenario is useful in all cases where a plan has been made to which device and channel all sensors have to be connected. You may then use this information to set up the system.

How to proceed

1. Define the channel names in an Excel workbook. Use three spreadsheets, if applicable: one for channel data like name and (later) scaling (T_Channel), one for computation channels (T_Calc) and one for sensors requiring a special user scaling (T_

UserScale). You may omit spreadsheets which are not required.

2. Define on the spreadsheet for the channel data three columns for the device/slot/channel numbers and enter the appropriate data.

3. Define for all data you want to set up the required columns and enter the appropriate data.

   ☞ Set up a test system first and export the settings to an Excel file in order to see the required entries, see Manual procedure for setting-up channels on page 162.

4. If required, you may also set up the other spreadsheets for computations or a special sensor scaling.

5. Save the Excel workbook.

6. Start MC Import (**From custom database** in the **Import** group on the **Database** ribbon tab) and open this Excel file (📄 **Open source file**).

7. Select in the upper table which displays the contents of your first Excel spreadsheet (sorted by name) the proper worksheet containing the channel data: **Source table**.

8. Click on 📄 (**Options**) and specify the names for you device/slot/channel columns.

9. Specify for all columns you want to import which type of data is contained: use the context menu in the header line to set e.g. **Channel data -> Name**, **Sensor type**, **Gage factor** etc. You may switch to another source table and, for example, specify a formula channel.

   You may save this channel assignment with 📊 so that you can use it again for your next import.

10. Mark the whole table (left upper field **Source)** or the required rows (channels) and click on 📊 (Start auto import). This will transfer all settings to the appropriate channels, identifying the channels by their hardware position.

11. Close MC Import and update/transfer all data to the devices. Check the log file for import problems (**Show** in the **MC Setup log** group on the **Diagnosis** ribbon tab) or use the Find channels option with filter **which are not OK**.

    See also Diagnosis and Error handling on page 216, Search functions on page 216.

Using Excel spreadsheets and identifying channels by their CANHEAD serial numbers

This scenario is useful in all cases where CANHEAD modules are used (MGCplus only) and the sensor data shall be changed/updated.

How to proceed

1. Export the current setup with your hardware configuration (and serial numbers) to an Excel workbook (**File -> Export table (MS Excel)**).
2. Use up to three spreadsheets of this Excel workbook, if applicable: one for channel data like name and (later) scaling (T_Channel), one for computation channels (T_Calc, optional) and one for sensors requiring a special user scaling (T_UserScale, optional). You may delete all spreadsheets which are not required (e.g. T_SubAmp). You may also delete unnecessary channels.
3. Delete columns ChanAddress, SlotAddress and DevIndex in the Excel spreadsheet T_Channel (if existing).

This Excel workbook will now act as a template for your future setups. If you want to add more channels or change a setting, change it here. Then:

1. Start MC Import (**From custom database** in the **Import** group on the **Database** ribbon tab) and open this Excel file (📩 **Open source file**).
2. Click on the **CBSerNum** column and use context menu item **Channel data -> CB Serial number**.
3. Click on the **SubChanAddress** column and use context menu item **Channel data -> Sub channel address**.
4. Click on the **CBSerNum** column and use context menu item **Set as identification column**.
5. Specify for all columns which type of data is contained: use the context menu in the header line to set e.g. **Channel data -> Name**, **Sensor type**, **Gage factor** etc. You may switch to another source table and, for example, specify a formula channel. You may save this channel assignment with 🖼 so that you can use it again for your next import.
6. Mark the whole table (left upper field **Source**) or the required rows and click on 🖼 (Update). This will transfer all settings to the appropriate channels, identifying the channels by their CANHEAD serial number and their sub channel number in the CANHEAD base module.
7. Close MC Import and update/transfer all data to the devices. Check the log file for import problems (**Show** in the **MC Setup log** group on the **Diagnosis** ribbon tab) or use the Find channels option with filter **which are not OK**.
    See also

3.3.6.2 How to import settings

1. Specify your database file with 📥 (**Open source file**). Choose the proper file type (**Access files** or **Excel files**) in the dialog.
2. Specify the table name (MS Excel: worksheet) from which the data shall be read under **Source table**.
3. Assign for each column of the MC Setup database (**Name**, **Sensor type** etc.) the corresponding field of *your* database in the **Dest.** row (upper table). Depending on the table selected in the lower part (**Channel data** or **Computations**), only fields for this table are displayed in the context menu of the upper table. Fields already selected are grayed out in the menu and cannot be assigned twice.

   You may save this channel assignment with 🖼️ so that you can use it again for your next import.
4. Decide on the method to use for the import: manually with drag & drop, using an identification column (e.g. channel name or T-ID) or using hardware references (device, slot and channel number).
   In the second case set the corresponding column with the context menu **Set as identification column**.

   In the third case use 📇 (**Options**) to specify the required column header texts. The setting is retained so you have to do this once only.
5. You may also use manual drag & drop with the right-hand mouse key if an identification column or hardware references are defined: in this case all settings from the selected group only are transferred. Otherwise mark the whole upper table and select either 🔧 (**Start auto import**) or 🔧 (**Update**).

Any errors during import are listed in the MC Import log. Use 📄 to open the log.

You may then continue with a new source file (step 1) or source table (step 2).

3.3.6.3 Data format for on/off information

Columns **Zero enable** and **Auto CAL** use **ON** or **OFF** as *string*. Therefore a string has to be used as format for this column when importing from MS Excel or MS Access, numerical values are not allowed here.

3.3.6.4 MC Import column (context menu items) description

🛈   See also MC Setup column description, scaling functions on page 164.

The following table lists all context menu entries for specifying columns and provides information about their significance.

Channel data columns

Menu item	Explanation
Name	Name of the channel. This is a very important setting, because *all program parts* rely on unique channel names and identify channels by this name, even if different hardware (other MGCplus devices) is used. (1), (2), etc. is appended if identical names occur e.g. on merging or import.
Sensor type	Depending on the hardware and sensor used, the sensor type must be specified. For most sensors, this is the bridge type or wiring scheme connected.
Exc	Excitation voltage for the transducer if applicable. This column has no meaning for DC input.
Filter	Depending on the hardware a filter type and frequency may be selected. Check with the MGCplus Operating Manual for details concerning filter behavior. Use right mouse key to get the context menu with the possible settings. ☞ When having ML801B plug-in modules with firmware 6.56 or higher you may also select the **Bessel CANHEAD** filters for ML801. This ensures that these modules use the same filters (filter characteristic and phase delay) as your CANHEAD modules, see also Using CANHEAD (MGCplus only) on page 146.
Gage factor	For strain gages only: "sensitivity" of a strain gage.
Bridge factor	For strain gages only. The setting depends on the hardware or the wiring scheme used: it is not possible to input a bridge factor for a quarter bridge. A bridge factor may only be used for half or full bridges.

Menu item	Explanation
Wiring comp. factor	For CANHEAD modules with strain gages in three-wire circuit only. The signal loss occurring in this case can be measured by the hardware and is compensated by a higher gain factor. The number to be used here is the loss factor.
Electrical zero	Electrical value for zero point.
Physical zero	Physical value (quantity) for the zero point.
Electrical nominal	Electrical value for nominal load.
Physical nominal	Physical value (quantity) for nominal load.
Engineering unit	Unit of the physical quantity to be measured. Use the right mouse key to get the context menu with the possible settings. You may however also type in a unit (max. eight characters, however only four can be displayed by the AB22A of the MGCplus).
Max. prediction (formerly Physical 100%)	Allows you to pre-set a certain value as 100% value, e.g. for a load level visualization, as maximum value for the prediction line trigger or as default for the 100% load condition (also called load point or end point), if you use MD Trend.
Upper trigger level	Allows you to pre-set a certain value for the above (>) trigger condition.
Lower trigger level	Allows you to pre-set a certain value for the below (<) trigger condition.
Tolerance % of prediction (formerly Percent of prediction	Allows you to pre-set a certain value as tolerance value for the prediction line trigger.

Menu item	Explanation
Zero enable	Allows zeroing of the channel. If in the main window of the MD Server the zero button is clicked ( ), only channels with this field activated (**On**) participate in the zeroing process. ☞ This setting should be deactivated for temperature channels. This also applies to digital channels or counter values if the values change during the balance. Otherwise this can lead to invalid values, e.g. 0.5 as zero value for a digital input that can only be 0 or 1 but has changed its status during the measurement.
Zero value	Zero value (offset) after a zeroing. You may use this to preset a certain value.
Auto CAL (Autocalibration)	Shows the state of the autocalibration feature. With MGCplus autocalibration is always switched off during a measurement, an ON setting enables autocalibration only during periods without data acquisition. With QuantumX/SomatXR the default calibration method is activated in ON state: Every 5 minutes the second amplifier chip of each channel is separated from the sensor input and calibrated (takes approx. 300 ms). The zero and range calibration of the first amplifier chip is then corrected with the information derived from this calibration and the comparison of the output between the two chips. It depends however on the measuring range and module and sensor type whether autocalibration is available. See also Autocalibration with MGCplus and QuantumX/SomatXR on page 55.

Menu item	Explanation
Ref. x channel	Any x channel (e.g. a load channel) to be used as reference may be specified here. With a prediction line trigger this channel is used as x channel. In y(x) graphs this channel is used as default for the x-axis if the channel of this row is assigned to the y-axis.
Sensor ID	If a sensor with T-ID or TEDS is connected, the Sensor ID, a worldwide unique number, is provided in this column.
Sensor description	In this field additional information may be specified.
Comment	In this field additional information may be specified. The maximum length for a comment is 128 characters.
ExpectedShuntDeflection	Signal change in case an external (custom specific) shunt is used. This external shunt is only available for half and full bridges and QuantumX/SomatXR MX1615, MX1616 or MGCplus with AP14, AP810, AP814, AP815 or CANHEAD.
Function group name	In this field the name of a function group may be specified. The corresponding channel will be added to this function group. You may specify more than one function group and even a subgroup for a channel. If the function group does not exist, it will be created. You may define function groups on the T_Fgroup Excel spreadsheet and use square brackets around each name. Separate subgroups with a semicolon. Then specify the appropriate group(s) on the T_Channel spreadsheet (FunctionGroups column): `[Group1;subgroup1];[Group2]` etc.
CB Serial number	CANHEAD only: Use this field for specifying which CANHEAD base module shall be used.
Sub channel address	CANHEAD only: If CB Serial number is used, this field specifies the channel in the CANHEAD base module. The column is used automatically, of CB Serial number is set as *identification column*.

User scale data columns

Menu item	Explanation
USName	Name of the channel using this user scaling.
USThermChan	Channel name for the channel necessary for the strain gage temperature compensation, if USScaleMethod is COMPCHAN or COMPTEMP.
USUnit	Engineering unit for the user scaling (unit of the physical quantity which is measured).
ScaleType	LINTAB (table), COMPCHAN for strain gage temperature compensation with a second strain gage or COMPTEMP for strain gage temperature compensation with a channel measuring the temperature.
X point/y point	x and y values for the linearization table. There is no need to specify the number of points used as with MC Setup: the value is calculated from the number of value pairs available.
USThermCofx	Coefficients for the temperature compensation of strain gages, if temperature compensation with the coefficients from the package is used (COMPTEMP).
AlphaSG / AlphaMaterial	Expansion coefficients of the strain gage and testing material used, if temperature compensation with the coefficients from the package is used (COMPTEMP).
AlphaTref	Reference temperature at which the expansion coefficients have been measured, if temperature compensation with the coefficients from the package is used (COMPTEMP).

Descriptor files

If a descriptor file is specified, this file may contain further information and can be accessed from different parts of the software as well as from a client PC. We recommend to check the access rights in Windows because client PCs may access the file only when the proper access rights are assigned in the Windows operating system.

Rosette columns

Menu item	Explanation
Result type	Letter/number extension used by the computation, e.g. AG for angle. See Available formulae and abbreviations used for strain gage rosette computations on page 199.
Rosette grid angle (formerly Strain angle)	Angle between the strain gage grids which indicates the rosette type: 45, 60 or 90 (as number).
Strain channel	Name of the channel measuring the respective grid.
Transverse sensitivity	Transverse sensitivity of each grid.
Young's modulus	Modulus of elasticity.
Poisson's ratio	Poisson's ratio.

3.4 Calculations (defining computations)

ⓘ See also Import from an existing MC Setup database on page 172, MC Import: Import from a custom defined database (Excel, Access) on page 173.

After clicking on the **Calculations** group (left-hand side) the first time, no channel (no computation) is defined on the right-hand side. Either use one of the entries in the **Calculations** group (**Database** ribbon tab) or right-click on the **Calculations** entry to get the context menu. From there choose which type of computation you would like to create (**Create channel**):

1. Manual input channel on page 191.
2. Online computations on page 191.
3. Strain gage (SG) rosette computations on page 198.
4. Statistical functions on page 204 (and filters).
5. Additional computations on page 205.
6. Optical sensor computations on page 209.

Computation channels may be zeroed as well. However in the default setting this is disabled: the **Zero enable** column is set to **OFF** after creating a new computation.

Modifying calculations

Use context menu **Edit** in a result channel name column to call up the corresponding computation dialog. Click on **Change** and **Close** after modifying your calculation.

Deleting calculations

Unlike hardware channels, computation channels may be deleted: click in the column on the far left (consecutive numbers are displayed there) to mark the row in question and select **Delete channel** from the context menu. You may also select several channels in this column by dragging the mouse while holding the left mouse key or by using the CTRL-key and marking individual rows.

3.4.1 Manual input channel

This topic corresponds to in Getting started with MC Setup.

Define manual inputs with **Manual input channel** (**Database** ribbon tab). This channel type allows you to type in data via the keyboard, e.g. readings from analog instruments, load levels etc. (**Enter manual input** on the **Test** ribbon tab). You may define up to 8 channels for manual input.

See also and Data acquisition, Enter manual input.

☞ In former versions of catman Enterprise *Manual input* was named *Load level input*.

3.4.2 Online computations

Define Online computations with **Online computation** (**Database** ribbon tab).

This computation type shows you a dialog with a pocket calculator. Use the integrated Channel selector to specify a source channel. Double-click to enter it into the computation formula.

The formula uses an abbreviation for the channel name so that the equation length is shortened. For each abbreviation, e.g. c7, the full name is displayed in the **Argument channels used** list.

On the left-hand side a numeric keyboard together with the usual functions are grouped as buttons, just click to select. Some **Special functions** are available from the drop-down

list. It is possible to use a computation already defined in another computation, just drag the result channel to the **Edit computation formula** field.

☞   Please use brackets to specify the processing sequence when using computations with mixed plus/minus and multiply/divide operators.

💡   Do not forget to click on **Add** to transfer the computation defined to the list of Calculations.

3.4.2.1 Mathematical functions

The online computation provides several standard functions on buttons and several other functions from the **Special functions** drop-down. All functions are listed here in alphabetical order.

Derivation, deriv

deriv(Arg1, Arg2) computes the derivation of channel Arg1 to channel Arg2. Only channels are allowed as parameters. Use 0 for dt (Arg2) if you need the derivative over time.

Example

```
deriv(Chan_Force, Chan_Displacement)
REM Computes the derivation of the force over the dis-
    placement.
```

ABS (absolute value)

abs(Arg) computes the absolute value of Arg.

Examples

```
abs(-25.3)    REM results in 25.3
abs(32.79)    REM results in 32.79
```

acos, asin, atan (inverse trigonometrical functions)

acos(Arg) computes the angle for the specified cosine, asin(Arg) computes the angle for the specified sine and atan(Arg) computes the angle for the specified tangent. The result is given in radians (rad).

Examples

```
acos(0.7604)    REM results in 0.7069 = 40.5°
asin(0.7604)    REM results in 0.8639 = 49.5°
atan(0.7604)    REM results in 0.6501 = 47.25°
```

AND, Boolean operator

Returns 1 if the arguments on both sides are true, otherwise 0.

See also OR.

Examples

```
1 AND 1    REM results in 1
1 AND 0    REM results in 0
```

Bit test

bit(Arg, BitNo.) tests whether a specific bit is set in the argument. For this the argument number is converted into a 32 bit integer. The lowest bit is bit 0. Both numbers and channels are permissible as argument. Produces 1, if the bit is set.

Examples

```
bit(25, 3)
```

REM The result is 1, because the third bit (2^3) is set (1 + 8 + 16 = 25).

```
bit(25, 2)
```

REM The result is 0, because the second bit (2^2) is not set.

ceil

See Next smaller integer.

cos

See sin, cos, tan (trigonometrical functions).

RMS value

rms(Chan, dt, T) calculates the RMS value of a signal. Specify the time **T** in milliseconds which is to be used for the computation. For the parameter **dt** you can either specify the interval between two measured values (1/sample rate) in milliseconds or 0; then the value is automatically determined from the channel sample rate. Specifying a parameter value not equal to zero is only necessary if an auxiliary channel (AutoSequence, EasyScript) or a computed channel is involved for which no DAQ channel is included in the computation and so no sample rate can be determined.

Example

```
rms(Chan_Force, 0, 500)
```

```
REM Produces new function values every 500 ms. The values
    are produced with the sample rate of the Chan_Force
    channel.
```

EXP (exponential function)

exp(Arg) computes the value of the exponential function (e function) for the specified operand.

Example
```
exp(5)    REM results in 148.413
```

floor

See [Next smaller integer](#).

If (condition)

if(condition, TRUE, FALSE) outputs, depending on the condition, either the expression for TRUE or the one for FALSE. Both numbers and channels are permissible as arguments.

Examples
```
if(Channel1 > 0, Channel1, -1 * Channel1)
REM This computation rectifies the signal
if((Channel1 > 350) OR (Channel2 > 740), 1, 0)
REM With this computation you can monitor multiple con-
    ditions. Trigger on this computation channel, once the
    value is e.g. greater than 0.5.
```

Integral

integral(Arg1, Arg2) computes the integral of channel Arg1 over channel Arg2. Only channels are allowed as parameters.

Example
```
integral(Chan_Force, Chan_Displacement)
REM Computes the integral of the force over the dis-
    placement.
```

INT (integer)

With a computation in DAQ mode int(x) cuts off the decimal places from x. This method is used here to speed up the computation. In Analysis mode, int(x) rounds an argument

to the nearest whole number.
See [Floor/Ceiling](#).

Examples

```
int(2.3)    REM results in 2
int(2.8)    REM results in 2
int(-2.8)   REM results in -2
int(-2.1)   REM results in -2
```

LN, LOG (logarithm)

ln(Arg) computes the natural logarithm, log(Arg) computes the logarithm to the base 10. If the argument Arg is smaller than or equal to 0, this will result in a run-time error.

Examples

```
log(100)    REM results in 2
ln(100)     REM results in 4.605…
```

Max/Min

max(Arg1, Arg2) determines the greater, min(Arg1, Arg2) determines the smaller of two numbers. Both numbers and channels are permissible as arguments. However, the channels must be acquired with the same sample rate, because in each case the measurements are examined at the same point in time.

Examples

```
max(25.3, 27)    REM The result is 27
min(25.3, 27)    REM The result is 25.3
```

Modulo division, mod

The modulo division Arg1 mod Arg2 finds the integer remainder from the division of two integer numbers. If you specify floating point numbers, the decimal places are not considered. Both numbers and channels are permissible as arguments.

Example

```
14 mod 3
REM Since only 12 or 15 is divisible by 3, 2 is left as the
        result (indivisible remainder)
```

Next smaller/larger integer, floor, ceil (round)

floor(Arg) rounds Arg to the next smaller integer. The arithmetic sign of Arg is considered in this case.
See also INT.

Examples

```
floor(2.3)   REM results in 2
floor(2.8)   REM results in 2
floor(-2.8)  REM results in -3
floor(-2.1)  REM results in -3
```

ceil(Arg) rounds Arg to the next higher integer. The arithmetic sign of Arg is considered in this case.

Examples

```
ceil(2.3)   REM results in 3
ceil(2.8)   REM results in 3
ceil(-2.8)  REM results in -2
ceil(-2.1)  REM results in -2
```

OR (Boolean operator)

Returns 1 if one of the arguments on both sides is true, otherwise 0.
See also AND.

Examples

```
1 OR 0   REM results in 1
0 OR 0   REM results in 0
```

POW (power function)

Raises the operand preceding the operator ^ to the power specified in the subsequent operand.

Example

```
2^3   REM results in 8
```

Random number, rnd

rnd(0) returns a random number between 0 and 1.

Example

```
rnd(0)
```

sin, cos, tan (trigonometrical functions)

sin(Arg) computes the sine, cos(Arg) the cosine and tan(Arg) the tangent. The argument Arg must be given in radians (rad).

Examples

```
sin(1.5708)    REM results in 1
cos(1.5708)    REM results in 0
tan(0.7854)    REM results in 1
```

SQR/SQRT (square root)

sqrt(Arg) computes the square root of Arg. If the argument Arg is smaller than or equal to 0, this will result in a run-time error.

Example

```
sqrt(9)    REM results in 3
```

System time

Outputs the elapsed time in milliseconds since the PC was started.

3.4.2.2 Mathematical operators

You can use the following operators within mathematical expressions.

Brackets

Use brackets around numerical expressions or function arguments or to enforce a specific analysis sequence.

Examples

```
sin(%P1%)
(%P1% + 5) * 2    REM without brackets %P1% + 10 would be
calculated
```

Boolean operator: AND

Logical AND. The expression is 1 if both expressions on the left and on the right-hand side of the operator are true (equal 1), otherwise 0.

Example

```
%MaxTemp% > 100 AND %MaxPressure% > 50
```

Boolean operator: OR

Logical OR. The expression is 1 if either the expression on the left or on the right-hand side of the operator is true (equal 1), otherwise 0.

Example

```
%MaxTemp% > 100 OR %MaxPressure% > 50
```

Less than, greater than, equal to, not equal to, etc.

Less than (<), greater than (>), does not equal (<>), equal to or greater than (>=), less than or equal to (<=). The expression is 1 if the condition is met, otherwise 0.

The equal operator (=) compares the expressions to its right and left. Results in 1 if both expressions are mathematically equal, otherwise in 0.

+, -, *, /

Add (+), subtract (-), multiply (*), divide (/). If when dividing the divisor = 0, this will result in a run-time error.

3.4.3 Strain gage (SG) rosette computations

Define Strain gage (SG) rosette computations with **SG rosette** (**Database** ribbon tab).

This computation may be used when having strain gage rosettes. It enables you to calculate the principle strains and stresses together with the orientation angle of the main principle strain or stress. Also the equivalent stress according to von Mises may be calculated or the transverse sensitivity may be stated.

See also SG (strain gage) temperature compensation on page 171.

☞   It is not possible to drag a channel from the integrated Channel selector to the strain channel **A** field of the first rosette. Please double-click on the channel in question (list of channels on the right-hand side) or mark the channel and click on **Select**.

How to proceed

1. Specify the channels where the rosette grids are measured. Grids b and c are filled in automatically after the channel has been specified for grid a. If these are not the correct channels, please change them (mark the input field and double-click on the correct channel).
2. If required, specify the transverse sensitivity.

3. If more than one rosette is used and all grids are in *consecutive hardware channels,* type in the number of active rosettes and the corresponding number of computations will be created. You can later use a Function group (see Grouping channels and computations on page 210) to collect all hardware and computed channels.
4. Mark the computations the program shall make. Keep in mind that some computations only make sense, if a certain rosette type is used (see next step).
5. Next the type of rosette must be specified: 0/45/90° grid angle or 0/60/120° grid angle. If the orientation angle of the principle stress is already known, a 0/90° rosette with two grids only may be used as well. In this case no orientation angle may be computed.
6. Now type in the Young's modulus (modulus of elasticity) and the Poisson's ratio for the material used in the test.
7. As last point either use the default setting for the base name which will be used for these computations or specify one and click on **OK**.
    Now all computations are created in the background using the base name specified. As default the channel name for grid a of each rosette is used as base name together with a combination of two or three letters or two letters plus a digit to indicate the type of computation (see table below). If you specify a name to be used, to each name first a number (rosette number) and then the above mentioned identification is appended.

Available formulae and abbreviations used for strain gage rosette computations

ⓘ See also SG (strain gage) temperature compensation on page 171.

Abbreviation	Computation
AG	Orientation angle
ES	Equivalent stress according to von Mises
SN1	Principle strain 1: strain in the direction of principle nominal stress 1
SN2	Principle strain 2: strain in the direction of principle nominal stress 2
SNA	Shear strain: strain under 45° to grid a. The angle is evaluated in mathematical positive direction (counter-clockwise)
SNX	Strain X: strain in the direction of the grid a

Abbreviation	Computation
SNY	Strain Y: strain under 90° to grid a. The angle is evaluated in mathematical positive direction (counter-clockwise)
SS1	Principle nominal stress 1
SS2	Principle nominal stress 2
SSH	Shear stress
SSX	Stress X: stress in the direction of the grid a
SSY	Stress Y: stress under 90° to grid a. The angle is evaluated in mathematical positive direction (counter-clockwise)

⚠ It is very important that the order of the grids a-b-c is counter-clockwise (mathematically positive) at the stated grid angles. The formulae used are based on this condition and would otherwise result in false values. With HBM strain gage rosettes the grids are labeled with a, b and c and can be used directly. If rosettes from other manufacturers are used, check the measuring grid arrangement and labeling.

Formula: SG rosette, angle

The following applies to 0°/45°/90° rosettes:

$$\alpha = \frac{1}{2} \operatorname{atan} \left| \frac{2\varepsilon_B - \varepsilon_A - \varepsilon_C}{\varepsilon_A - \varepsilon_C} \right|$$

The following applies for 0°/60°/120° rosettes:

$$\alpha = \frac{1}{2} \operatorname{atan} \left| \frac{\sqrt{3}\,(\varepsilon_B - \varepsilon_c)}{2\varepsilon_A - \varepsilon_B - \varepsilon_C} \right|$$

☞ The Arcustangens is calculated so that the angle is output in the correct quadrants I to IV.

Formula: SG rosette, reference stress according to von Mises

The following applies to 0°/45°/90° and 0°/60°/120° rosettes:

$$\sigma_v = \sqrt{\sigma_1^2 + \sigma_2^2 - \sigma_1\sigma_2}$$

Formula: SG rosette, principle strain 1

The following applies to 0°/45°/90° rosettes:

$$\varepsilon_1 = \frac{\varepsilon_A + \varepsilon_C}{2} + \frac{1}{\sqrt{2}}\sqrt{(\varepsilon_A - \varepsilon_B)^2 + (\varepsilon_C - \varepsilon_B)^2}$$

The following applies for 0°/60°/120° rosettes:

$$\varepsilon_1 = \frac{\varepsilon_A + \varepsilon_B + \varepsilon_C}{3} + \frac{1}{\sqrt{3}}\sqrt{\frac{(2\varepsilon_A - \varepsilon_B - \varepsilon_C)^2}{3} + (\varepsilon_B - \varepsilon_C)^2}$$

Formula: SG rosette, principle strain 2

The following applies to 0°/45°/90° rosettes:

$$\varepsilon_2 = \frac{\varepsilon_A + \varepsilon_C}{2} - \frac{1}{\sqrt{2}}\sqrt{(\varepsilon_A - \varepsilon_B)^2 + (\varepsilon_C - \varepsilon_B)^2}$$

The following applies for 0°/60°/120° rosettes:

$$\varepsilon_2 = \frac{\varepsilon_A + \varepsilon_B + \varepsilon_C}{3} - \frac{1}{\sqrt{3}}\sqrt{\frac{(2\varepsilon_A - \varepsilon_B - \varepsilon_C)^2}{3} + (\varepsilon_B - \varepsilon_C)^2}$$

Formula: SG rosette, shear strain

The following applies to 0°/45°/90° rosettes:

$$\gamma = 2\varepsilon_B - \varepsilon_A - \varepsilon_C$$

The following applies for 0°/60°/120° rosettes:

$$\gamma = \frac{2}{\sqrt{3}}(\varepsilon_B - \varepsilon_C)$$

Formula: SG rosette, strain X

The following applies to 0°/45°/90° and 0°/60°/120° rosettes:

$$\varepsilon_x = \varepsilon_A$$

Formula: SG rosette, strain Y

The following applies to 0°/45°/90° rosettes:

$$\varepsilon_y = \varepsilon_C$$

The following applies for 0°/60°/120° rosettes:

$$\varepsilon_y = \frac{2}{3}\left(\varepsilon_B + \varepsilon_C - \frac{\varepsilon_A}{2}\right)$$

Formula: SG rosette, principal nominal stress 1

The following applies to 0°/45°/90° rosettes:

$$\sigma_1 = \frac{E}{1-\nu} \cdot \frac{\varepsilon_A + \varepsilon_C}{2} + \frac{E}{\sqrt{2}(1+\nu)} \cdot \sqrt{(\varepsilon_A - \varepsilon_B)^2 + (\varepsilon_C - \varepsilon_B)^2}$$

The following applies for 0°/60°/120° rosettes:

$$\sigma_1 = \frac{E}{1-\nu} \cdot \frac{\varepsilon_A + \varepsilon_B + \varepsilon_C}{3} + \frac{E}{1+\nu} \cdot \sqrt{\left(\frac{2\varepsilon_A - \varepsilon_B - \varepsilon_C}{3}\right)^2 + \frac{1}{3}(\varepsilon_B - \varepsilon_C)^2}$$

Formula: SG rosette, principal nominal stress 2

The following applies to 0°/45°/90° rosettes:

$$\sigma_2 = \frac{E}{1-v} \cdot \frac{\varepsilon_A + \varepsilon_C}{2} - \frac{E}{\sqrt{2}(1+v)} \cdot \sqrt{(\varepsilon_A - \varepsilon_B)^2 + (\varepsilon_C - \varepsilon_B)^2}$$

The following applies for 0°/60°/120° rosettes:

$$\sigma_2 = \frac{E}{1-v} \cdot \frac{\varepsilon_A + \varepsilon_B + \varepsilon_C}{3} - \frac{E}{1+v} \cdot \sqrt{\left(\frac{2\varepsilon_A - \varepsilon_B - \varepsilon_C}{3}\right)^2 + \frac{1}{3}(\varepsilon_B - \varepsilon_C)^2}$$

Formula: SG rosette, shear stress

The following applies to 0°/45°/90° and 0°/60°/120° rosettes:

$$\tau_{xy} = \frac{\sigma_1 - \sigma_2}{2} \sin 2\alpha$$

Formula: SG rosette, stress X

The following applies to 0°/45°/90° rosettes:

$$\sigma_x = \frac{E}{1-v^2}(\varepsilon_A + v\varepsilon_C)$$

The following applies for 0°/60°/120° rosettes:

$$\sigma_x = \frac{E}{1-v^2}\left(\varepsilon_A + v\frac{2}{3}\left(\varepsilon_B + \varepsilon_C - \frac{\varepsilon_A}{2}\right)\right)$$

Formula: SG rosette, stress Y

The following applies to 0°/45°/90° rosettes:

$$\sigma_y = \frac{E}{1-v^2}(\varepsilon_C + v\varepsilon_A)$$

The following applies for 0°/60°/120° rosettes:

$$\sigma_y = \frac{E}{1-v^2}\left(\frac{2}{3}\left(\varepsilon_B + \varepsilon_C - \frac{\varepsilon_A}{2}\right) + v\,\varepsilon_A\right)$$

3.4.4 Statistical functions

Define Statistical functions with **Statistical functions** (**Database** ribbon tab).

This computation type allows for calculating minimum or maximum over a channel range, e.g. the minimum or maximum of several non-consecutive channels, not just a single channel. The other statistical function available is running average (sliding mean). As a new function Bessel and Butterworth filters are available.

Procedure

1. Select the required computation: **Select function**.
2. Use the integrated Channel selector to specify one or more source channels. Select a device ot connector in the System display to show the channel(s) in the right-hand field. Then double-click the channel(s) in question to enter it/they into the **Argument channels used** field. Usually the abbreviated channel specifier (c and a number) is also inserted into the formula. If more than one channel is required, the cx are separated with a colon (,), e.g. **Chan_min(c1,c2,c3,)**.
3. Depending on the function you might have to specify additional parameters, see below.

☞ The computations use an abbreviation for the channel name so that the equation length is shortened. For each abbreviation, e.g. **c7**, the full name is displayed in the **Argument channels used** list.

💡 Do not forget to click on **Add** to transfer the computation defined to the list of Calculations.

Available functions

1. **Chan_min, Chan_max**
   These functions calculate the minimum or maximum over a channel range (several channels).
2. **Running average**
   The mean value is calculated over the specified number of points. Because it is a

running average the number of results is equivalent to the number of input points, e.g. the computation shifts by one value for the next average computation.

3. **Running average with tolerance**
   Similar to the running average computation, however the tolerance value is added to the calculated average value.

4. **Digital filter**
   Calculates a Bessel or Butterworth filter of 8th order. catman Enterprise tries to match the cut-off frequency (-3 dB) as close as possible to the specified **Filter frequency**. Please be aware that the sample rate must be more than twice the cut-off frequency. Otherwise an error occurs during initialization and the measurement cannot be started.

5. **Compress**
   Compresses the specified number of measurements in one result value by calculating minimum, maximum, peak-to-peak or mean value over this range.

3.4.5 Additional computations

Use **Additional computation** and one of the sub-entries there (**Database** ribbon tab) to define this type of computation.

Some of the additional computations are computations made within a DLL (Dynamic Link Library), just choose **Additional custom calculation** for these. The following special computations are installed with MD Server:

1. Curve fit on page 205.
2. Peak/valley analysis on page 206.
3. Matrix computation on page 207.
4. Virtual channels on page 207.
5. FFT peak channels on page 208.

💡 If you require any special real-time computation you may either write a DLL yourself or ask HBM for a solution. The documentation on Math Drivers ("Drivers/OnlineMathDriver/Math Drivers.doc" in the catman Enterprise installation folder) shows how to write such a driver in a high-level programming language and how to install it in MD Server/MC Setup.

3.4.5.1 Curve fit

ⓘ   See also Curve fit in a graph.

Define a curve fit with **Additional computation -> Additional custom calculation** and click on any entry under **Curve fit** in the following dialog (**Database** ribbon tab).

This computation uses several channels distributed over an object which measure e.g. a force or strain. With this (geometrical) distribution a curve fit for the data acquired by the channels is made. This enables you to specify a geometrical point for which the corresponding force or strain value shall be calculated.

The example below assumes a distribution of strain gages over an object at the specified x values (positions) x_n to x_{n+4}. The blue crosses mark the measured strain values. At position x_b the strain value shall be calculated because no strain measurement is available.
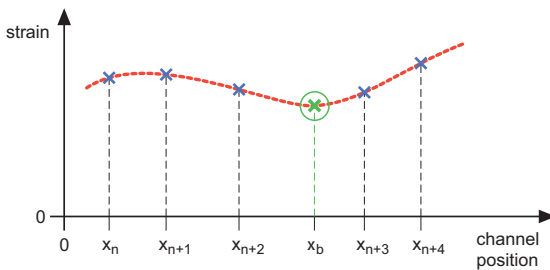
Fig. 3.2   Curve fit principle

Enter the channels which contain the measurements for the positions x_n to x_{n+4} in the **y channel** column (mark the channel and click on **Select channel**). Then either specify the x values (position values) in the **x value** column, **Use fix x values** (same distance between all channels) or use a channel specifying the x values. In the latter case, click first on the corresponding row in the **x channel** column, then mark the channel to assign and click on **Select channel**. As a result, the y value (green circle, strain value in this example) is calculated which would be measured at position x_b.

3.4.5.2 Peak/valley analysis

Define a peak analysis with **Additional computation -> Additional custom calculation** and click on any entry under **Peak analysis** in the following dialog (**Database** ribbon tab).

Carries out a peak-value analysis of a channel. You may calculate the x values corresponding to the maxima (Peak) or minima (Valley) found in the y channel and count how often a y maxima or minima occurred (**Y peak/valley counter**). Specify a hysteresis in the unit of the measurement signal so that noise levels in the measurement signal are not detected as extreme values.

Computational algorithm

1. Search for a local extreme value.
2. If found, set a marker and put this value into the extreme value memory.
3. Provided the following value does not undercut, resp. exceed this extreme value by the value specified in the **Hysteresis** column, wait for a further local extreme value and update the extreme value memory as necessary.
4. If a following value undercuts or exceeds the extreme value memory by the value specified in the **Hysteresis** column, the momentary content of the extreme value memory and the corresponding x value are written into the results channels and the marker is reset, so that the search for another extreme value can start.

☞ The hysteresis determines the resolution of the computation: A low value supplies more, but possibly unwanted, extreme values than a large hysteresis value.

3.4.5.3 Matrix computation

Define a matrix computation with **Additional computation -> Additional custom calculation** and click on **HBMMATH32** in the following dialog (**Database** ribbon tab).

This computation allows for multiplying several channels with a matrix. The matrix elements may be calculations themselves. This enables you, for example, to linearize a three component force transducer or to transform geometric coordinates into a different coordinate system.

In principle it is also possible to perform the necessary calculations with standard online computations, however it is easier and more convenient to use the matrix calculation.

You may either specify the matrix coefficients manually or open an ASCII file or Excel spreadsheet and copy a cell or a range of cells: mark the cell or cell range, then use the *right-hand* mouse key and drag it to the matrix area. A dotted blue line marks where the range is to be inserted.

3.4.5.4 Virtual channels

Virtual channels are channels for e.g. testing panels or Client—Server communication etc. without hardware. You can create several channels with different test signals and use these in a measurement. Values are created like having values from measurement channels.

You can create virtual channels either when scanning instead of device channels or additionally to measurement channels from devices. In both cases these channels are created from within MC Setup.

Creating virtual channels without connected devices

- Create a new MC Setup database.
- Define in the **Scan System** dialog a non-existing IP address.
- Start the scan.
- Because the scan cannot find anything a dialog appears asking you whether you want to create virtual channels.
- Create the required channels.

Creating virtual channels additionally

- Go to the **Database** ribbon tab in MC Setup.
- Use **Additional computation** (**Calculations** group) to create additional virtual channels.

Virtual channel settings

Three different types of virtual channels are available:

1. **Continuous ramp**
   Define **Interval** and **Sample rate**. With e.g. an **Interval** of **0.01** and a **Sample rate** of **100** the values start at zero and each second the increment amounts to 1 (100 x 0.01).
2. **Saw tooth**
   Define **Interval** and **Max. value**. With e.g. an **Interval** of **0.01** and a **Max. value** of **3** the values start at zero and each second the increment amounts to 1 (100 x 0.01). However, after 3 seconds the max. value is reached and values start at zero again.
3. **Constant value**
   As the name indicates, the **Value** you specify is output constantly.

Unit is an optional entry.

3.4.5.5 FFT peak channels

Define FFT peak channels with **Additional computation -> Additional FFT peak channels** (**Database** ribbon tab).

The calculation is equivalent to the peak calculation described for a . The values from the calculation may however be saved.

Procedure

1. Select in the dialog the device with the channel from which the calculation shall be made.
2. Double-click on the relevant channel in the right-hand frame.
   This channel is used as **FFT argument channel**.
3. Type in a **Results channel name**.
4. Define how many peaks you want to be evaluated at most (**Number of peaks**), what the minimum amplitude must be (**Peak threshold**) and how many values must be there to be evaluated as peak (**Peak width**, at least 3).
5. Specify the conditions for the FFT computation: number of values (**FFT buffer size**), **FFT window function** and **FFT output unit**.
6. Click on **Add**.
   Depending on how many peaks you wanted to be calculated several channels are created: an amplitude and a frequency channel per peak.

You may use these channels for display in Digital indicators or graphs as usual.

Modifying FFT peak computations

It is not possible to change the number of peaks as this would change the number of result channels. Apart from that all other settings may be changed, even the base name for the result channels. You may however delete channels by clicking in the column on the far left (consecutive numbers are displayed there) and use **Delete** from the context menu or ribbon.

3.4.6 Optical sensor computations

ⓘ   See also , and topic "Optical strain measurement" in the System Documentation, Part I.

Define Optical sensor computations with **Optical sensor computation** (**Database** ribbon tab).

This dialog provides the necessary computations for converting wavelength data into either strain values or into temperature values. In both cases a temperature correction is possible. The **Base wavelength** depends on which fiber Bragg grating is measured in the

respective channel. The **Gage factor or sensitivity of FBG strain sensor** is determined by the fiber, please check the data sheet.

Optionally you can also carry out a temperature compensation here. Further information on this topic can be found in the Operating Manual for your device.

☞ The measuring channels with the wavelengths do not allow zero balancing, this is only possible with the computation channels.

3.5 Grouping channels and computations

The Function groups entry is very convenient for grouping channels or computations with identical settings, transducers etc. It may also be used to limit the number of channels visible on the right-hand side or to group channels for client PCs. A hardware channel or computation may belong to several different groups, there is no limit to how often a channel may be used for different groups. Of course, the data is *measured or calculated only once*.

☞ Except those items where **... (system wide)** is appended, all ribbon actions in MC Setup affect only those channels displayed on the right-hand side.

💡 Amplifier and sensor check show only those channels displayed on the right-hand side before switching to the diagnosis window. By selecting a certain group you may therefore limit the number of channels displayed.

How to create a group of channels

1. Right-click on Function groups (left side of the window) to get the context menu and choose **New function group**.
2. Type in a name for that group.
   ☞ Group names must be unique.
3. Now mark the desired channels on the list of channels on right-hand side. You might need to first click on another group, e.g. Calculations or Devices, to see available channels. Of course, you may also use another group to select some channels from this group. It is possible to select several consecutive channels by holding down the ⇧ key or individual channels by holding down the Ctrl key.
4. Copy the marked channels (use the context menu in the **Name** column).
5. Go to the Function group and select context menu **Paste channels in function group**.

> 💡 As an alternative you can drag & drop the channels in question, however, you must use the *right mouse key* for dragging, the left mouse key is not working here.

You may also copy, paste or delete channels from a group. Just use the context menu to apply the respective action to the selected channels.

☞ It is not possible to delete channels from a device since these channels are built-in hardware.

3.6 Checking channels

☞ You can only check hardware channels, it is not possible to check e.g. a computation. The devices must be connected, otherwise no check is possible. Use **Reconnect system** (**System** group, **Device** ribbon tab) if necessary.

The **Amplifier/Sensors** ribbon tab shows all channels currently displayed in the **Device** ribbon tab also here. You should therefore first select the channels you want to check and then choose this tab. You may select channels from all available devices, e.g. MGCplus, QuantumX/SomatXR, from MOOG, ARAMIS (GOM) or BraggMETER devices. All actions made are done with all channels displayed here, it is not possible to select individual channels here.

After selecting the **Amplifier/Sensors** ribbon tab a continuous measurement is started and the result displayed. This shows you whether the value displayed is in the expected range. Column **Reading absolute** displays the value without zero offset (zero balance).

> 💡 Use the Search facility to limit the display to just these channels which are to be checked, see Search functions on page 216.

This program part is useful for
- checking that the channels in question are working properly by showing currently measured values,
- performing a zero adjustment,
- deleting an old zero value,
- checking that all settings concerning the hardware have been made,
- checking that the wiring is done properly,
- checking for changes of sensor IDs,
- performing a shunt check (measurement),
- compensating sensors attached to CANHEAD for wire resistances.

💡 With MGCplus we recommend you initiate an autocalibration before starting a test: **Calibrate (system wide)** (**Amplifier/Sensors** ribbon tab). This ensures proper operation, because all long-term drifts etc. are excluded by this internal re-calibration. This is not necessary with QuantumX/SomatXR as here either a permanent autocalibration is used or no autocalibration necessary (device dependent).
See also Autocalibration with MGCplus and QuantumX/SomatXR on page 55 and QuantumX/SomatXR and MGCplus Operating Manuals.

The whole table with all columns visible plus information about Device, Slot, Channel and Sub-channel can be exported to file in Excel or ASCII format: **Table (MS Excel)**, **Table (ASCII)** or **Table (MS Access)** in the **Export** group on the **Amplifier/Sensors** ribbon tab.

⚠ Export in Excel format is only possible if MS Excel is installed.

If a check fails, this is also indicated in MC Setup after closing this tab: the channel name is displayed with a red background color. A tooltip indicates which test has failed. However, with QuantumX/SomatXR the information displayed might not be sufficient to find the cause. Then use the MX Assistant instead.

3.6.1 Checking internal zero and calibration values (ZERO-CAL)

☞ This check is only available for MGCplus signal conditioning devices.

With **Check ZERO-CAL** (**Amplifier/Sensors** ribbon tab) the *internal* zero and calibration values (reference values for autocalibration) of your devices are tested. If these values are not OK an error state is displayed. Usually this means that the corresponding channel has to be serviced. The numbers displayed are for service purposes only.

3.6.2 Check sensor wiring

ⓘ See also Measure/Edit shunt deflection on page 215 for testing half and full bridges and Perform CANHEAD wiring compensation (cable length compensation) on page 214.

The wiring check enables you to control strain gage installations and whether there are possible problems due to faulty connections or damaged strain gages. The wiring check may be performed with certain devices only: MGCplus with AP14, AP810, AP814, AP815 or CANHEAD or QuantumX/SomatXR MX1615, MX1616 as well as MX238, MX410,

MX430, MX440 and MX840A/MX840B (not with MX840) with the connection of strain gages via the adapter plug SCM-SG120/350/700/1000 in quarter bridge configuration.

Quarter bridges

When using quarter bridges, a hardware check may be performed with the built-in shunt resistor of some devices. If this function is not supported by the hardware, **NA** (not available) is displayed when a check is initiated. During the check a shunt resistor is applied in parallel to the internal completion resistor. Thus a certain signal step is produced. The result depends on

- shunt and completion resistor values,
- the resistance of the strain gage used,
- the wiring,
- the set gage factor.

Because the values of shunt and completion resistor are well known and the resistance of the strain gage used has only little influence on the result, it can be checked whether the wiring is done properly: no short circuit to ground or any other strain gage wiring fault.

With CANHEAD modules a custom shunt may be used. However it is not possible to set the value of this resistor in MD Server. Use instead the MGCplus Assistant and the procedure described in topic "Correction factor and shunt calibration" (MGCplus Assistant help). The setting is then saved in the CANHEAD and therefore necessary once only.

Half and full bridges

When using half or full bridges a wiring check is also possible when using one of the following devices: QuantumX/SomatXR MX1615, MX1616 or MGCplus with AP14, AP810, AP815 or CANHEAD. However, the signal is not defined in this case as with quarter bridges. Therefore you have to specify the expected signal. Use either **Edit shunt deflection** (mark channels first) or context menu **Edit shunt deflection** to specify a value, see .

Allowed maximum deviation (wiring check tolerance)

Specify the maximum deviation between shunt reading and shunt nominal value allowed in percent with **Set tolerance (system wide)** (**Wiring check** group on the **Amplifier/Sensors** ribbon tab). The current value is indicated in the **Wiring OK/NOK** column header. The default setting is 0.5%, which is sufficient for most applications.

Test results

OK is displayed in the **Wiring OK/NOK** column if the wiring check succeeds. Otherwise a more detailed explanation in the **Error status** column (MGCplus only) describes the possible error source, e.g. **Transducer error** if the wiring is faulty due to missing sense lines. The values shown in the **Reading shunt** column (after a check has been made) may vary between different channels due to distinct gage factors or strain gage resistances.
Any errors during this wiring check are also written to the project (MC Setup) log file MCLOG.MDB (see also MC Setup options on page 221).

⚠️ Channels where the wiring check failed are deactivated for any subsequent tests (default). You may however change this behavior via the MC Setup **Options** dialog on the **Diagnosis** tab, refer to MC Setup options on page 221.

☞ More information about this test procedure and the principles behind can be found in the HBM literature which is also available on the web https://www.hb-m.com as PDF file.

3.6.3 Perform CANHEAD wiring compensation (cable length compensation)

Some versions of the CANHEAD module use an unregulated three-wire circuit for the connection of the strain gage. This means that although a drift of the zero point due to cable temperature changes is (approximately) eliminated, the loss of sensitivity due to the resistance of the wires is however not taken into account. This menu item may be used to measure the signal loss due to wiring and calculate a correction factor.

⚠️ If you have built your own shunt resistance into the CANHEAD, you must specify the resistance value with the MGCplus Assistant. Use the procedure described in topic "Correction factor and shunt calibration" (MGCplus Assistant help). The setting is then saved in the CANHEAD and therefore necessary once only.

Perform CANHEAD wiring compensation (system wide) activates the built-in shunt, calculates the loss due to the cable resistance and indicates the determined loss factor. The factor is saved and from now on all measurements are multiplied by the reciprocal. The measurement is carried out for all channels and—if present—for the compensation strain gage.

☞ The correction only applies to the temperature at which the measurement has been carried out. Since the resistance of (copper) wires changes with

temperature, measurement at a different temperature also produces a different correction factor. For low to medium cable resistances the deviations can often be tolerated when viewed in relation to the strain gage measurement uncertainty.

3.6.4 Check sensor ID

Checks whether the Sensor IDs from TEDS and T-ID modules connected to the signal conditioning devices are identical to those stored in the setup database. An error message is displayed and an entry in the MC Setup log file is made if differences occur. See also MC Setup options on page 221.

☞   This check is done automatically every time you connect to the system (devices).

3.6.5 Measure/Edit shunt deflection

🛈   See also Check sensor wiring on page 212.

When using half or full bridges with QuantumX/SomatXR MX1615, MX1616 or with MGCplus and AP14, AP810, AP815 or CANHEAD a standard wiring check is not possible: you have to specify the expected signal because it cannot be calculated. However, if you don't know the expected signal either you have the option to measure it with this menu item. Then use these values as reference (nominal) values.

How to proceed

1. Use **Measure shunt deflection** (**Amplifier/Sensors** ribbon tab) to perform a (precise) measurement of the actual value with the shunt activated. The resulting value is indicated in the **Shunt nominal** column.
2. Enter this value as default for further tests with **Edit shunt deflection** (**Amplifier/Sensors** ribbon tab).
3. Set the allowed deviation to this value for further tests with **Set tolerance (system wide)** (**Amplifier/Sensors** ribbon tab). Default is 0.5%, which is sufficient for most applications. The current value is indicated in the **Wiring OK/NOK** column header.

The next time you would like to check that your system has not changed, use **Check sensor wiring** (**Amplifier/Sensors** ribbon tab). As long as the deviation to the above measured value is less than the specified tolerance, the **Wiring OK/NOK** column will indicate this with **OK**.

3.7 Search functions

The **Find channels** drop-down list (bottom left) provides you with several search criteria. This makes it easy to locate specific channels from a hundred or thousand active channels. This search function is available in different parts of MD Server.

☞ Search operations are done with *all channels* in groups Devices, Calculations and Function groups.

You may search for channels:

1. Which are OK.
2. Which are not OK.
3. With change flag set, i.e. which have been changed and where the changes have not yet been written back into the channel (activated).
4. By name, comment, sensor type, sensor description, unit, serial number or gage factor.
   > 💡 Use **%** or ***** as wild cards for several characters and **_** or **?** as wild cards for single characters. You may use wild cards more than once in the search string as in **%nnel??04%**. Without any wild card character, an exact match is searched.
5. Which are active.
6. Which are Store channels (**by stored**) or Fast channels (**by fast**).
   See also Measurement settings (DAQ settings) on page 56.

☞ After the search all channels matching the search criteria are listed in group Search results in window **Select**.

3.8 Diagnosis and Error handling

MC Setup log
This log contains a protocol for every change to your channels made in MC Setup. Assigning a channel name or sensor or if differences are found when connecting to your devices—all information of this type may be forwarded to the MC Setup log. With **MC Setup -> Options** you specify on the **Logging** tab, which information shall be logged in file MCLOG.MDB.
See MC Setup options on page 221.

The location of the MC Setup log file is displayed as window title if you click on **Show**. With **Clear log entries** you can either delete all entries or just all entries before a certain date.

MC Setup Communication log

The MC Setup Communication log is helpful to determine errors during setup: the *command communication* between MC Setup and the signal conditioning devices (no data) is written to a ring buffer together with time stamps. This communication log enables the HBM support to track errors, e.g. if devices crash or hang up. In case you experience such an error, activate the communication log (**Log communication**, **Diagnosis** ribbon tab) and save it. Then send it for evaluation to the HBM Technical support, see Technical support on page 29. As default the communication log is disabled when starting MC Setup.

Terminal

The **Terminal** item (MGCplus only) is for users with special knowledge only. Usually it is used by HBM personnel when solving interface problems. This might be necessary if you contact the HBM Technical support, see Technical support on page 29.

Error handling

Channels having errors are displayed with a red background color and are deactivated for any subsequent tests (default, you may however change this behavior via the **Diagnosis** tab in the MC Setup **Options** dialog (**MC Setup -> Options**), refer to MC Setup options on page 221). In this case please check your hardware, e.g. with Check sensor wiring on page 212. The background color is reset if all errors have been resolved, e.g. after a wiring check which resulted in no errors (OK) for the channels in question.

3.9 Changing the hardware configuration

There are several possibilities for changing the hardware configuration:

1. Changing the number or type of channels in one or more signal conditioning devices MGCplus.
2. Moving channels of a certain type between different signal conditioning devices MGCplus.
3. Changing the number of QuantumX/SomatXR modules in a backplane.
4. Moving certain QuantumX/SomatXR modules to another backplane.

5. Changing the number of MGCplus signal conditioning devices or Quan-tumX/SomatXR modules.
6. Changing the number of cabinets.

The first four points are explained in the following, the last two points are dealt with in topic Expanding or re-organizing the MD Server system, see System Documentation, Part I.

Important when using QuantumX/SomatXR

If you add new QuantumX/SomatXR modules to an older system it might be necessary to update the firmware of all older modules. All modules should use the same firmware version.

☞ Decimal sample-rate domains are not supported by MD Server.

3.9.1 Changing a MGCplus hardware configuration

It does not matter whether you want to move a certain type of channel from one MGCplus to another or whether you want to add or remove channels—it is the same procedure.

How to proceed

1. Switch off the cabinets (see UPS manager on page 132).
2. Insert the new channels in a signal conditioning device MGCplus of your choice. Make sure that the amplifier plug-in module and the connector plate for the rear fit together, because not every connector plate can be used with every amplifier module. Check that both parts are sitting in the same slot and fix them with the screws provided. If you want to remove a channel, just unscrew the amplifier module and the connector plate and remove them.
3. Switch on the cabinets.
4. If you want to retain your settings from the last project, just start up MC Setup. The program will detect the change and ask for a re-scan. If you agree, all recent settings will be imported into a new MC Setup database after the scan. This saves you the effort of re-defining all those settings for the unchanged channels. However, you may also start the MC Setup program (do not transfer the setup to the devices) and create a new database: **New** (**MC Database** group on the **Database** ribbon tab). In this case you have to perform a new scan.
    See also Getting started with MC Setup, Step 1: Scan hardware on page 141.

⚠️ An IRIG B plug-in module may not be inserted into the MGCplus device which is used as trigger leader, if NTP *and* IRIG B time sources shall be used together in a system: The MGCplus cannot use both NTP and IRIG B time channels in one housing.
Use another MGCplus device in this case (preferably the last one in the synchronization chain), the hardware synchronization takes care of synchronizing the IRIG B time channels with the NTP time channels. The NTP time is deactivated for the MGCplus having the IRIG B plug-in module.

3.9.2 Changing a QuantumX/SomatXR hardware configuration

If you add new QuantumX/SomatXR modules to an older system it might be necessary to update the firmware of all older modules. All modules should use the same firmware version.

☞ Decimal sample-rate domains are not supported by MD Server.

How to proceed when adding a module to a backplane

1. Switch off the cabinets (see).
2. Remove any case protection, case clips or lateral covers from the modules. The modules can only be fastened in backplanes in housings with IP20 protection (metal housing only, no cover, no SomatXR modules). Refer to the QuantumX Operating Manual for details on how to remove case protection, cover or housing clips.
3. Remove also the cover from the rear connection plug.
4. Unscrew the upper and lower screwed clamping glands of the backplane up to the stop (the screws are secured against falling out).
5. Position the module vertically on a backplane of your choice and push it in carefully on the lower guide rail back up to the stop. Up to 14 QuantumX modules may be connected to one CX27 (two backplanes).
6. Tighten the lower then the upper screwed clamping gland.
7. Switch on the cabinets.
8. If you want to retain your settings from the last project, just start up MC Setup. The program will detect the change and ask for a re-scan. If you agree, all recent settings will be imported into a new MC Setup database after the scan. This saves you the effort of re-defining all those settings for the unchanged channels. However, you may also start the MC Setup program (do not transfer the setup to the devices) and create a new database: **New** on the **Database** ribbon tab. In this case you have

to perform a new scan.
See also Getting started with MC Setup, Step 1: Scan hardware on page 141.

How to proceed when removing a module from a backplane

1. Switch off the cabinets (see UPS manager on page 132).
2. Unscrew the upper and lower screwed clamping glands of the backplane up to the stop (the screws are secured against falling out).
3. Pull the module vertically off the backplane.
4. Tighten the upper and lower screwed clamping gland.
5. Switch on the cabinets.
6. If you want to retain your settings from the last project, just start up MC Setup. The program will detect the change and ask for a re-scan. If you agree, all recent settings will be imported into a new MC Setup database after the scan. This saves you the effort of re-defining all those settings for the unchanged channels. However, you may also start the MC Setup program (do not transfer the setup to the devices) and create a new database: **New** on the **Database** ribbon tab. In this case you have to perform a new scan.
   See also Getting started with MC Setup, Step 1: Scan hardware on page 141.

How to proceed when moving a module to another backplane

1. Remove the module in question as described in How to proceed when removing a module from a backplane, Steps 1 to 4.
2. Insert the module in the new backplane as described in How to proceed when adding a module to a backplane, Steps 4 to 8.

How to proceed when changing systems without backplanes

The procedure depends on the system used. With only a few QuantumX/SomatXR modules you may have one of the following systems:

1. You use FireWire and connect the modules directly to the server PC.
   Up to 14 modules may be connected. Make sure, the connection between the modules is properly made (from X102 to X101 on the next module, there from X102 to X101 on the next module and so on).
2. You use an Ethernet switch and connect several modules via the switch to the server PC.
   Also here the FireWire connection should be made in order to synchronize the modules. Otherwise an NTP time server has to be used, see Installing the NTP

time server/client on page 18, Time channels on page 52, Configure and check synchronization on page 112.

3. You use a CX27 for up to 14 modules and connect the CX27 via Ethernet to the server PC.
   Make sure, the connection between the modules is properly made (from X102 to X101 on the next module, there from X102 to X101 on the next module and so on). We recommend to use the CX27 as last module in this chain.

4. You use several CX27 for summing up several QuantumX/SomatXR modules and transferring the data over Ethernet switches to the server PC.
   For synchronization this requires either a connection between these CX27 modules as described for cabinets with backplanes, see topic "Adding QuantumX backplanes" in System Documentation, Part I, or an NTP time server.

As soon as a CX27 is used, you connect the QuantumX/SomatXR modules via FireWire with the CX27, either by using a backplane or by using the FireWire connectors at the back side of your QuantumX/SomatXR modules as described in the Operating Manual.

☞ Without a backplane, make sure a power supply cable is attached on every third module: Only three modules may be supplied from one connector through the FireWire cabling.

3.10 MC Setup options

Open the MC Setup **Options** dialog via **MC Setup -> Options**.

EEPROM options tab

Determines the behavior of MC Setup on exit when using MGCplus devices: save settings always (default and recommended setting), ask or save never. The save option will only be used if a connection to the MGCplus devices has been established.

The option has no influence when using QuantumX or SomatXR modules.

Logging options tab

MC Setup log

Allows you to define which events are logged to the MC Setup log file MCLOG.MDB. You open the log file via **Show** in the **MC Setup log** group on the **Diagnosis** ribbon tab.

Communication log

The communication log logs the last 10 000 communication events (default setting) and works as a circular buffer (ring buffer). This communication log enables the HBM support to track errors, e.g. if devices crash or hang up. In case you experience such an error, activate and then save the communication log (**Save communication log** on the **Diagnosis** ribbon tab) under a different name and send it for evaluation to the HBM Technical support, see Technical support on page 29. The default setting for this option is disabled at startup.

Diagnosis options tab

Automatically deactivate channels on failed wiring check …: The default setting after a failed wiring check is to deactivate those channels and all connected computations. You may however decide not to deactivate these channels as this changes the list of active channels. With an active option all channels previously deactivated are activated again if the (next) wiring check succeeds.

Deactivate the reporting of channel error status: Suppresses the dialogs usually popping up if an error occurs.

Hide dialog asking for CANHEAD auto assign: Because with auto-assign the sequence of the CANHEAD modules may change, you can suppress the dialog. In this case you have to assign all modules manually. Use the **Advanced setup** (**Device** ribbon tab) and click on the corresponding ML74, see also Advanced setup on page 158.

Suspend calculation channels from Measurement: This option allows you to restrict the measurement to hardware channels only. The the primary task of MC Setup is: You set up hardware channels and check that everything is working properly. If these channels are checked and OK, all calculations should work as well (if properly defined).

Select path for MX Assistant: With the path specified you may use **Advanced setup** (**Device** ribbon tab) to call up the MX Assistant. This program enables you to perform special setups or actions, e.g. switching from analog to CAN bus input on a MX840 or reading out the calibration protocol of a device.

☞ You must install the required MX Assistant software manually, it is not included in the catman Enterprise setup. Either download it from the HBM website (HBM downloads) or use the CD included with your QuantumX/SomatXR module delivery.

Layout options tab

Decide here, which font family and font size you would like to use for the channels display. You may also select which columns shall be marked with bold face in the **Amplifier/Sensors** ribbon tab, see also Checking channels on page 211.

4 MC DEFINITION LIST

ⓘ   See also MC Setup on page 137, Getting started with MC Import on page 174.

The MC Definition List software enables you to define all channels and sensors necessary for a specific test project and later generate a wiring scheme for these channels according to the hardware available. The program is either started *from the MC Setup* software with **MC Setup -> MC Definition List -> Edit MC Definition List** or—when installed on a PC without MC Setup—as a separate program.

The following diagram shows the principle operating scheme when using the MC Definition List: A list of the channels required for the transducers is composed and results in the MC Definition List database. On the hardware side a new scan is made or an existing MC Setup database is used. Both databases are then merged together. The result is a new MC Setup database and a connection diagram which can be used to connect the transducers to the appropriate channels.

Be careful when *merging* the MC Definition List with an existing database. After the matching process the database contains all information from the MC Definition List, the original information about computations and Function groups is lost. Although a backup copy is made, we recommend that you copy the active database first: use **Save as** on the **Database** ribbon tab in MC Setup to store an existing database under a new name. Then open this database for merging. As an alternative, you may also create a new database and perform a new system scan. It is always possible at a later point in time to import settings like trigger definitions or computations from another (old) database file.
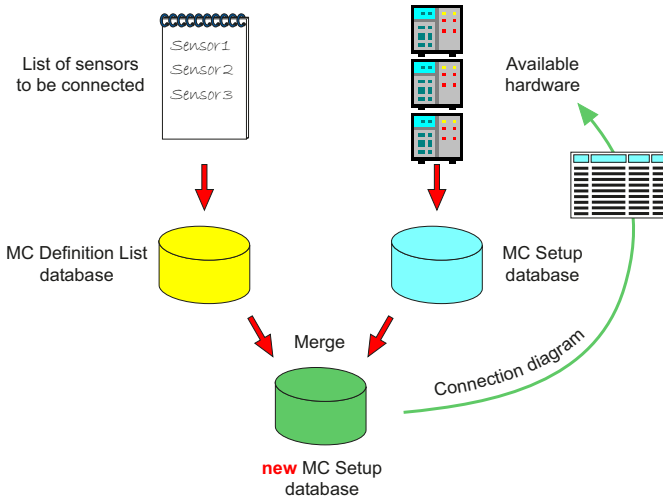
Fig. 4.1   MC Definition List operating principle

☞ It is not mandatory to use the MC Definition List program, it is also possible to connect all transducers and then specify the connections made directly in MC Setup or use MC Import.

💡 If during a later test run the zero point of a channel is changed (**Zero channels**, see Available Test actions (after initialization) on page 98), this change is forwarded to the original MC Definition List. Therefore the MC Definition List always contains the *latest* information about the transducers of a test and may be used again for a new merge process with different hardware. Zero values acquired during a test run are saved to ZERO.BIN. **MC Setup -> MC Definition List -> Update MC Definition List** provides a manual way to update the MC Definition List with the measured zero values and *all scaling information* (columns **Electrical zero** to **Physical nominal** in MC Setup).

4.1 General procedure (MC Definition List)

🛈 See also MC Setup on page 137, MC Setup column description, scaling functions on page 164.

💡 The merging process reads the MC Definition List from top to bottom according to the channel numbers displayed in the far left-hand column. Therefore hardware for the channels defined first is searched for first. Consequently channels important for the measurement should be positioned at the beginning of the list.

The general procedure is as follows

1. Compose a list with the required number of channels.
   ☞ You may also import an old list (menu **File -> Import)** or compose several lists and combine these later with menu **File -> Append**, see Importing a channel list on page 234, Appending MC Definition Lists on page 236.
2. Specify the setup for these channels according to the transducers/sensors used.
3. If required, define all computations necessary for the measurement task and group channels and computations.
4. Define fast and slow measurement rates and, if required, set pre- and post-trigger times or filter parameters.
5. Merge the database file containing the hardware configuration of the MD Server system (from MC Setup) with the MC Definition List.
6. Check that all channels could be connected.
   If not, you must decide which channel may be omitted or whether it is necessary to install new hardware in one of the MGCplus signal conditioning devices. Then make a new system scan in MC Setup and use this database for merging. If adding more hardware is not possible, remove less important channels from the MC Definition List.
7. Print out the wiring scheme.
8. Do the wiring and **Open** in MC Setup the database specified during the merge process (**Database** ribbon tab), which will contain all information from the MC Definition List.

💡 If you need to zero balance or compute channels from non-HBM devices (refer to Additional external devices on page 149), you may use channels with sensor type **External**. We recommend to import the channel names into the MC Definition List, refer to Importing a channel list on page 234. The channel names must match exactly the names you receive when performing the Scan (refer to Step 1: Scan hardware on page 141). Then assign the sensor type **External**. After creating these channels, you may use them in further computations as usual.

General information on using MC Definition List

⚠️ For all MGCplus 8-channel amplifier plug-in modules the settings **Sensor type**, **Uexc** (excitation voltage) and **Filter** are restricted: only identical settings for all 8 channels may be chosen. In the merging process the first channel of a specific sensor type which is connected to such an amplifier determines therefore the settings for excitation voltage and filter for the following 7 channels. If the following sensors (with identical sensor type) in the MC Definition List have different settings for excitation voltage or filter, these are ignored and a warning entry is made in the log file for the merging process (see also . In order to use different settings for identical sensor types, dummy channels may be inserted, see .

Mark several cells, copy/paste/delete actions

🔵 If you want to have the same data in several rows or a range, just type the data into one cell or row and copy this cell or range, e.g. with ⌐Ctrl⌐-C. Then mark the range where you want the same data to appear and press ⌐Ctrl⌐-V. This copies data from one table cell to other cells *of the same type.* As an alternative you may position the cursor over the lower right corner of a marked cell. As soon as the cursor changes, click and drag to select consecutive cells. After releasing the mouse key, the contents of the first cell are copied to the cells just marked. If columns on the left or right side of the originally marked cell are also marked, these cells are copied as well. If a range is copied, you may also mark a single cell as starting point and insert the copied block. Another possibility is to copy a range with multiple blocks: if for example 2 columns and 3 rows are copied, you may insert this block into a marked range of 2 columns and 9 rows. (If you mark only 8 rows, 6 will be filled with data (2 blocks) and 2 will be left empty.)

⚠️ Please observe:

🔵 Copying a cell or range to *multiple* cells with item **Check block size of selection on paste** activated in the **Preferences** dialog (menu **File -> Preferences**) is only possible, if the destination block is an integer multiple in rows of the block to copy.

🔵 Rosette computations and the corresponding hardware channels are always treated as a channel block: changing settings or copying works with the whole block only and it is not possible to insert

channels in between. You may however group the channels as required later, e.g. for having all computed channels in a separate block, see .

- ⟨⇧⟩ together with the space bar marks complete rows, ⟨Ctrl⟩ together with the space bar marks complete columns. If several rows or columns were already selected before using one of these shortcuts, the selection is *extended* accordingly.
- If all channels are shown (**Show all** selected as display filter in the drop-down list on the top), you may drag & drop channels with the right mouse button: just click on one of the selected channel numbers in the left column and drag. When the ⟨Ctrl⟩ key is pressed additionally, the selection is copied.
- ▬ removes one or more selected channels, ✕ just clears the contents of selected cells. You may also use the ⟨Del⟩ key (clear contents) or the ⟨Del⟩ key together with ⟨⇧⟩ (remove channels).

Sorting or hiding columns, rename channels

- It is also possible to hide unnecessary columns: right-click in the top (heading line) and choose **Hide**. As soon as one column is hidden, a new menu item **Show** is displayed in this context menu, even in the **Channel name** column. Select which columns should be displayed again from the **Show** sub-menu.
- Sorting is easy: right-click in the top (heading line) of a column to get the context menu and choose **Sort**. This will sort the table according to this column in ascending order, the column is marked with ▽ afterwards. Choosing **Sort** a second time changes the sort order to descending.
- If the naming scheme starts to become distorted after you have defined a lot of channels, you may renumber all channels or just those having a specific base name. Use menu **Channels -> Rebuild index of channel names**. If *all* channels are to be renamed, check **Rebuilt indices for all base names**. Don't forget to specify the new base name!

Find channels, display certain channels only

- Use the drop-down list next to 🔍 to restrict the channels shown to certain channel groups, e.g. to rosette channels and rosette computations, to other computations or to those channels *used* in a computation. The latter allows for

dependency checking, especially if anything in the merge process fails. If channel groups are defined, their names are listed as well.

- ![binoculars icon] allows searching in the columns **Channel name** or **Channel group**. The dialog is displayed beneath the first channel matching the search condition. Press ⏎ to find the next occurrence. A search for an empty string finds all dummy channels.

Dummy (empty) channels

If hardware channels shall be spared for later use, "dummy" channels may be added. These channels use the same transducer type as the preceding channel but have *no channel name*. If a group name is used both channels (preceding and dummy channel) must have the same group name. These channels are connected to hardware channels following immediately those channels above in the MC Definition List.

This feature may also be used if e.g. from an eight-channel amplifier plug-in module only 6 channels shall be used for two strain gage rosettes and the other two channels shall be left free. Just insert after the second group of strain gage rosette and computation channels two dummy channels.

ⓘ See also Creating dummy channels (channels, which provide hardware without a connection) on page 240.

4.2 Creating a channel list

ⓘ See also General information on using MC Definition List on page 228.

It is either possible to load an older list of channels, save it under a new name and change it or to create a new (empty) list. You may also create a new list by appending a list to an already existing or an empty list, refer to Appending MC Definition Lists on page 236. The procedure is described in the following sections.

Important notes

- If you want to mark a cell for copying do not double-click: this enters cell edit mode. Just click once to get a frame around the cell and use then e.g. $\boxed{\text{Ctrl}}$-C or the **Edit** menu.
- If a context menu exists for the column in question it is enough to mark the channels and choose from the context menu.

4.2.1 Step 1: Create new channels

Create some new channels with [+] or **Channels -> Insert**. Use **Start with index** if already the first channel should get a running number appended.

We recommend that you create as many channels of the same type or group as required in one go and assign a characteristic channel base name. The appended number consists of five digits (with leading zeros) which makes it easier to sort according to name. The channels are always created (and inserted if necessary) starting at the row of the cursor position.

4.2.2 Step 2: Define sensors/computations

Set the sensor type or computation you want to use with these channels: mark the channels in column **Sensor type** and choose from the context menu.

The channels to be used in a computation formula must already exist. Select **Online calculation** from the context menu. The dialog is equivalent to the dialog used in MC Definition List. For strain gage rosettes you have to mark the channel containing the first grid and call the dialog with context menu **Rosette…**.

See also Online computations on page 191 in MC Setup.

> 💡 If you need to zero balance or compute channels from non-HBM devices (refer to Additional external devices on page 149), you may use channels with sensor type **External**, refer to General procedure (MC Definition List) on page 226. After creating these channels, you may use them in computations as usual and select the computed channels e.g. for zero balancing.

> 👉 It is important to choose the proper type of sensor and wiring scheme, e.g. three-wire quarter bridge for 350 Ohm strain gages: **Quarter bridge 3-wire 350 Ohm**. Otherwise a hardware connection is selected later, which may not be available. You can check the type of sensor and wiring scheme available for a specific hardware channel in MC Setup in column **Sensor type** via the context menu.

> 💡 The context menu displays only some sensor types, see Preferences and options (MC Definition List) on page 241 on how to change this menu.

Strain gage rosette computations

Before specifying a rosette computation, make sure all channels required for the strain measurement (hardware channels) are existing. Only the result channels will be

generated. Mark the first channel containing grid a and call the **Rosette computation** dialog with the context menu from the **Sensor type** column.

The sensor type of the hardware channels may be set/changed when defining the rosette computation, just activate in the **Rosette computation** dialog **Override the current sensor-type settings** and specify the appropriate one.

When setting up several rosettes, the strain channels must be in consecutive order: grids a-b-c (counter-clockwise, i.e. mathematically positive). The grid is indicated in the (generated) channel name.

With **Use the first present channel name ...** the names of the grid channels use the channel name of grid a and append the grid letter (A/B/C). With **Use the present channel names ...** the grid channel names remain unchanged.

The results channels may also use the name of the channel with grid a and append a combination of two or three letters or two letters plus a digit to indicate the type of computation: Activate **Use first strain channel (A) of EACH rosette as its base name** (default). As an alternative you may specify a base name to which a running number with five digits (leading zeros) is appended.

The computations used are the same as described in the table in .

☞   It is not possible to change or remove a single rosette computation or a grid channel at a later point in time as is the case with other computations. You may only delete all rosette results channels including the grid channels with **Channels -> Remove** and re-create them.

4.2.3 Step 3: Define scaling/engineering unit

Depending on the selection made, cells in other columns might be locked or already filled with default data. For example, when typing in a gage factor (except **0**), columns **Engineering unit**, **Electrical nominal** and **Physical nominal** are locked. With strain gages just the unit and the gage factor must be stated, input of, for example, transverse sensitivity is optional.

Otherwise select the correct unit and specify other transducer data like electrical and physical nominal values. With a standard force transducer for 5 kN this would be **kN**, **2** (mV/V) and **5**.

💡   You may change the default values for the *active* MC Definition List (further settings) with dialog **Preferences** and **Edit default values** (**File -> Preferences** menu).

4.2.4 Step 4: Set excitation voltage for sensors

Set the required sensor excitation voltage in **Uexc** column.

☞ Although you may type in any value as excitation voltage, only certain voltages can be used depending on the amplifier modules. When merging the list, this is checked and the next lower or—if this is not possible—the next available voltage is used. In both cases a warning is issued and an entry in the log file is made.

4.2.5 Step 5: Define sample rate, zero and storage options

Specify measurement related data like **Fast rate**, **Active**, **Store**, **Zero enable**, **Zero value** or **Shunt nominal**. These settings correspond to either the settings in MC Setup, e.g. **Zero enable**, **Zero value**, or MD Server in **DAQ settings**, e.g. **Active** or **Fast rate** (only available in measurement mode **Fast trigger mode**.

💡 If you need to zero balance or compute channels from non-HBM devices (refer to Additional external devices on page 149), you may use channels with sensor type **External**, refer to General procedure (MC Definition List) on page 226. After creating these channels, you may use them in computations as usual and select the computed channels for zero balancing.

4.2.6 Step 6: Arrange channels into groups

You may group channels by assigning a group name to several channels. This group is transferred to MC Setup section Function groups when merging the list. As soon as a group name is assigned, this name is added to the context menu of column **Channel group** and to the **Show...** drop-down list.

4.2.7 Step 7: Define filters or triggers, or both

ℹ See also General information on using MC Definition List on page 228.

The last step almost completed the process. Additionally it is possible to (pre-)set the fast and slow sample rates for measurement mode **Fast trigger mode** as well as pre- and post-trigger times and filter type and frequency to be used with ⏲ (**System parameters**).

☞ Click on the **Set...** buttons to apply settings.

After specifying the sample rates, you may also set the pre- and post-trigger times. These times depend on the hardware and are changed as soon as a different sample rate is used. As default pre- and post-trigger times are equal, but you may use the available time for pre- *or* post-trigger exclusively. Trigger definitions/channels may not be defined in MC Definition List, they have to be imported after the merge process in MC Setup, see Import from an existing MC Setup database on page 172.

☞ It is not possible to set three sample rates for measurement mode **Regular DAQ mode** and assign these to the individual channels. Use the DAQ settings dialog in MD Server.

If all channels necessary for the measurement task are defined, you may proceed with the merging process described in Merging MC Definition List and hardware on page 237.

4.3 Importing a channel list

Instead of creating a new list you may also import data from

1. an Excel spreadsheet or
2. an Access database.

In both cases the same principle procedure is used to extract the information. When using an Excel file, the first row must contain column headers. The procedure is similar to the import function in MC Setup, see MC Import: Import from a custom defined database (Excel, Access) on page 173.

How to proceed

1. Open the **Import** dialog with **File -> Import…**.
2. Specify the **Source file** with 📂.
    The file is loaded and checked. An error message is displayed if fields have wrong or invalid data types, e.g. string format instead of number format. These fields are displayed in the table on the left with a *red question mark.* Usually this is no problem as the format will be set as soon as this column is dedicated to a destination column. If however the column is superfluous, empty content is irrelevant.
3. Indicate the worksheet (Excel) or table (Access) to be used (**Table**). Otherwise the first one contained in the file is used.
4. Except for the sensor coding column specify which columns to use for what data in the MC Definition List: right-click on the left side in the **Dest.** row in the column to

be assigned and choose the destination column from the context menu.

This process is called *column assignment*. Save this assignment with ![icon] because you need it again later. Also when reading in a file with the same column names, just load this assignment with ![icon] next to **Column assignment**.

☞ Hide unnecessary columns with **Hide unused columns** (context menu in **Dest.** row).

5. The next steps depend on the information available.

You have two options for specifying the sensor type

1. Load or create a look-up table, if the sensor type is coded into the channel name or an extra column.
2. Create first the channels required by dragging the channels from the **Channel name** column to the table on right-hand side. The table should be empty at the beginning as all channels are appended. Then close the import dialog and select the sensor type in MC Definition List. Now open the import dialog again, see below.

Procedure when using a look-up table

1. Click on ![icon] to open the look-up table editor.

   ☞ You may also edit an existing look-up table from this dialog. Open the table with ![icon].

2. Enter in column **Sensor coding** the abbreviations or codes used in the import chan-nel list column you want to specify as sensor coding column. Use **?** as wild card for a single character, e.g. **???A1** for sensor type A1 which is coded into the fourth and fifth character (if you want to use this column). Do not use **∗** as wild card for sev-eral characters. Further trailing characters are ignored, e.g. with 721A1_SG only the A1 part will be evaluated. As soon as something is entered into a row, a new empty row is created.

   ⚠ The input is case sensitive, the same letter as small and capital letter is con-sidered a different entry!

3. Choose in column **Sensor type** the circuit type from the context menu.
   Here the same settings are possible as in the MC Definition List.
4. The **Unit** column cannot be changed. You may however change the default unit used in dialog **Edit default values** (**File -> Preferences**).

5. Save the look-up table with ![save icon].
   The file is saved as an Access database with extension SLT.
6. Close the **Edit sensor look-up table** dialog.
7. Set the column containing the abbreviations or codes used **as sensor coding column**.
8. Load your look-up table with ![load icon] into field **Look-up table**.
9. Mark the channels to transfer in any column, as always *all* columns are selected.
   ☞ Click in the left upper corner of the table to mark the whole table.
10. Right-click on the marked area and drag it (right mouse key still pressed) to the table on the right-hand side.

A dialog informs you whether all channels could be imported. If errors occurred, open the

log file with ![log icon] and check for possible problems.

Procedure after having assigned the sensor type in MC Definition List

1. Open the **Import** dialog again with **File -> Import…**.
2. Specify the **Source file** with ![file icon].
3. Indicate the worksheet (Excel) or table (Access) to be used (**Table**).
4. Activate **Update mode** for the following. Data is assigned automatically to the channels having the *same* name, even if they are not in the same order in both tables.
   An error message is issued and an entry in the log is made, if channels are not available in the destination table.
5. Mark the channels from which data shall be transferred in the left table.
   Click on **Dest.** or **Source** in the upper left corner to mark the whole table.
6. Right-click on the marked area and drag (right mouse key still pressed) this area to the table on the right-hand side.
   All data from the marked area will be transferred to the proper channels and columns on the right-hand side (columns without assignment are ignored).

4.4 Appending MC Definition Lists

Instead of creating a new list you may also append several already existing lists. This is also useful, if several people compile their own MC Definition Lists, which are then combined into one for system setup.

How to proceed

1. Open the dialog with menu **File -> Append**.
2. If a file with a definition which files to append already exists, open it with 📂. Otherwise open the Windows Explorer with 🔍.
3. Select the MC Definition List(s) to append and drag these to the table **Source files**.
4. If the new list shall be saved under a specific file name, state it under **Destination file**. Otherwise the current MC Definition List file is used.
5. Click on **Append** to create the new MC Definition List file.
    In the default setting this list is opened afterwards.

☞   A warning is issued and a log file entry is made, if duplicate channel names are detected. Check then which channels were omitted by opening the log file with 📄.

4.5 Merging MC Definition List and hardware

Actually, two lists or databases are merged in this process: the MC Definition List database and the MC Setup database. The latter contains all information about the available hardware. This database is *changed* during the process and contains afterwards the new setup data which must be transferred to the hardware in order to become active. A backup copy is made as a rule so that the original information in the setup database is not lost (*.BAK). As an additional safety measure, you may copy the active database first: use **Save as** in MC Setup (**Database** ribbon tab) to store an existing database under a new name. Then open this database for merging. As an alternative, you may also create a new database and perform a new system scan.

4.5.1 Starting the process

1. Choose menu **Merge -> Merge with MC Setup** from the MC Definition List program. You may also use menu **MC Setup -> MC Definition List -> Merge with MC Definition List** from the MC Setup program.
2. Specify either the MC Setup database file or the MC Definition List database file, depending on which program you used to call up the dialog. If no MC Setup file is available, start the MC Setup program from MD Server, create a new database file and perform a system scan.

The symbol ⚠ next to the number of channels just indicates that you are trying to connect more transducers than channels are physically available (not enough hardware). It does not mean that the merging process will fail completely.

3. Start the matching process with **Merge**.
4. The status of the process is displayed (☞ shows which part is executed) and the progress indicated in the bar at the bottom of the dialog. Finally the results are shown as numbers and symbols. ⚠ means that something is not OK, ✔ means everything worked properly. If ❌ is displayed, a serious error occurred and the merging process failed. Please check whether the database files are accessible (not accessed by another program). Otherwise this type of error points to a corrupt or outdated version of the database files used for the merge process.

If errors occurred, check with to find out how to solve them. Otherwise proceed with .

☞ Details may also be found in the log file, click on **Show log file** which will open the log file. All entries are ASCII strings, so the log file may be opened by a standard text editor as well. Every action made during the merge process and the corresponding result are entered into this log.

4.5.2 Solving merging problems

There are several reasons for merging failures:

1. Not enough hardware channels available.
2. The proper type of hardware is not available.
3. Computations could not be defined, because channels on which the computations are based on could not be connected.

☞ Details may also be found in the log file, click on **Show log file** which will open the log file. All entries are ASCII strings, so the log file may be opened by a standard text editor as well. Every action made during the merge process and the corresponding result are entered into this log.

💡 The merging process reads the MC Definition List from top to bottom according to the channel numbers displayed in the far left-hand column. Therefore hardware for the channels defined first is searched for first. Consequently channels important for the measurement should be positioned at the beginning of the list.

⚠️ Every change you make is saved at once in the MC Definition List database file. If you want to keep the original file, make a copy first: use **File -> Save as**. You are then working on this copy.

How to proceed

1. Click on **Show results**.
2. In the following dialog choose **Show not connected** from the drop-down list at the bottom.
3. Now check why those channels could not be connected.

If hardware is missing, you must either

● decide to add more channels to the system. Refer to topic Changing the hardware configuration on page 217, and the System Documentation, Part I, for details.
    or
● delete some unnecessary channels from the MC Definition List or re-arrange the channel order (and skip the last channels for *this* test).

If the hardware is not of the proper type, try to find out whether you can substitute a certain sensor wiring type with another one. You can, for example, try to connect a strain gage in three-wire or four-wire configuration or you can use a dummy resistor and connect both as a half bridge. This might enable you to use some channels which are not yet connected.

After having decided which measures to take, change the MC Definition List and perform a new merging.

☞ You may print out the displayed part (**Show…**) of the list with **Print**.

4.5.2.1 Re-arranging channels

If you decide to move channel upwards to have these channels connected beforehand, proceed as follows:

1. Select **Show all** as display filter in the drop-down list on the top.
2. Mark the row(s) to move in the first (numbering) column (use ⊕).
3. Click with the *right* mouse button on one of the numbers of the selected channels and drag to the destination row.

☞ Channels containing a strain gage rosette and the relevant rosette computations always form a *channel block.* It is not possible to insert channels in between. When trying to move or copy one of those channels the complete block is marked.

4.5.2.2 Creating dummy channels (channels, which provide hardware without a connection)

Sometimes it is required to have "free" channels between connected channels:

- It could be beneficial to use only 6 channels of the last 8 channel amplifier in a MGCplus device for two rosettes.
- It could be necessary to have some spare channels available in a certain device.
- Another reason might be that with all MGCplus 8-channel amplifier plug-in modules the settings **Sensor type**, **Uexc** (excitation voltage) and **Filter** are restricted: only identical settings for *all* 8 channels may be chosen.
  In the merging process the first channel of a specific sensor type which is connected to such an 8-channel amplifier determines therefore the settings for excitation voltage and filter for the following 7 channels. If now 7 dummy channels are inserted after the first sensor, the next sensor with identical sensor type is connected to the next available amplifier and may have a different setting for excitation voltage or filter.

☞ Dummy channels are not connected to a transducer in the merge process.

How to proceed

1. Click on the row after that row, where the last active transducer is connected to the amplifier plug-in module in question.
2. Insert the required number of channels with the **Use base name** feature in the insert dialog deactivated. It is mandatory that these channels have *no channel or group name*.
3. Leave column **Sensor type** of the newly created dummy channels free. The type will be chosen according to the last sensor connected to this amplifier module.

☞ Channels containing strain gage rosettes and rosette computations always form a *channel block.* It is not possible to insert channels in between.

4.5.3 Printing out the wiring scheme

If the merging process succeeded, choose **Show hardware list** from the drop-down list in the results window. You may sort this list in ascending or descending order by right-clicking in a column. Then print out this list. The last three columns show you to which hardware the transducers have to be connected.

- **Device number**: this corresponds to the last digits of the Ethernet address. For example, **6** would mean Ethernet address xxx.xxx.xxx.106.
- **Slot number**: this corresponds to the slot inside the device. Slot 1 is the far-right connection-plate slot, as seen from the rear side of the signal conditioning unit MGCplus.
- **Channel number**: this corresponds to a sub-channel number. Please check with the MGCplus Operating Manual which pins have to be used for connecting what channel.

Now connect all transducers according to this wiring scheme.

⚠️ You must transfer the new MC Definition List to the signal conditioning devices before starting a measurement with the newly connected transducers (use the MC Setup program). We also recommend that you check the wiring as described in .

4.6 Preferences and options (MC Definition List)

In the **Preferences** dialog (menu **File -> Preferences**) you can choose:
- Which default settings are used when specifying a certain sensor and wiring type (**Edit default values**).
- Which colors are used to display the different parts of the table.
- Whether paste operations work only on the same number of rows and columns as was copied (**Check block size of selection on paste**). Normally a range is also copied to multiple cells. For example, if 2 columns and 3 rows were copied, you may insert this block into a range of 2 columns and 9 rows. If only 8 rows were marked in this example, 6 will be filled with data (2 blocks) and 2 will be left empty. If the block size is checked, the range marked *must equal* the range copied in size.

☞ The dialog displays only the most frequently used sensor types. If a specific type is needed, just contact the HBM Technical support, see Technical support on page 29 for a modified sensor type database.

5 MD EXPORT

5.1 Introduction

MD Export is used as an addition to MD Server to extract data from several tests of a project and export it into other formats. You may extract single channels only or use sets of data. You can call up MD Export directly from MD Server (**Tools/Script** ribbon tab) or as a standalone program after terminating MD Server.

5.1.1 Program window

The upper part, the *project explorer*, contains the project with all tests and related data as tree view. The lower part, the *data window*, shows *either* further *information* about the part marked above, i.e. for the project, a test or a certain data category *or* the available *data channels* are displayed. If you click on an entry, all available related information is displayed in the data window (lower part).

These two windows are an integral part of the MD Export window. All other windows are separate floating windows.

💡 You may change the sizes of those two parts by positioning the mouse pointer over a border. As soon as the cursor changes, you may drag that border and change the size as desired.

☞ As all data which shall be processed is temporarily imported into the (internal) database, the size of this database determines, how many channels and values may be exported. The current number of channels available is displayed in the status bar on the right-hand side (**Max DB channels**).
See MD Server database for MD Export on page 244 for further details.

5.1.2 General operating procedure

ℹ️ See also MD Server database for MD Export on page 244.

1. Open a project by choosing the MD Server project folder you want to examine with 📁 or **Project -> Open project**, see Selecting data, information displayed on page 244.
2. Either compile a Channel list to export certain channels or choose in the **Export** dialog (**Export -> Data export**) which project or test to export.

3.	Then set the export properties like file format, file name etc. and start the export, see .

5.2 Selecting data, information displayed

Selecting a project is done by specifying the MD Server project folder (**Project -> Open project**). If the project folder contains a valid database file MDPROJECT.MDB, all tests and their data is available for further examination.

The project explorer (upper part) shows all tests and data concerning the project chosen. If you click on an entry, all available related information is displayed in the data window (lower part). Additionally the info item (🛈) shows general statistics like time and date, user etc. The sub-entry *Measurement details* lists the number of data acquired for each set of data.

☞	Red color indicates errors, e.g. a test which has not been performed contains no MCSETUP file. This is also indicated in the information part. If data of a certain category was not acquired, this category is grayed out.

💡	You may change the sizes of those two parts by positioning the mouse pointer over a border. As soon as the cursor changes, you may drag that border and change the size as desired.

📇 allows you to search for specific channel names using part of the name: Use **%** or ***** as wild cards for several characters and **_** or **?** as wild cards for single characters. You may use wild cards more than once in the search string as in **%nnel??04%**. Without any wild card character, an exact match is searched.

5.3 MD Server database for MD Export

All data to be processed is temporarily imported into the MD Server database. Then the data is exported from there. Therefore the size of this database determines the maximum number of channels and the maximum number of values per channel which may be processed. Change the setup with 📇 or from menu **Database -> Database configuration**: the database file for data storage is created. However, the database defined in MD Server is also used by MD Export.

⚠️ If you change the definition from within MD Export, the MD Server database is changed too.

The database configuration is described in topic Configure storage size in Chapter B, MD Server.

The **Database channel manager** shows for all channels, how many values per channel have been acquired and how much space is still available. The window is intended for service and error tracking purposes. You may export selected channels also from here into formats catman and Excel, if Excel is installed. Click on 📝 to open the **Database editor** window from the **Database channel manager**. This window displays all values in the database and allows you to change individual values. Additionally min. and max. values for each channel are displayed.

5.4 Exporting data from MD Export

Click on 📥 or use menu **Export -> Export dialog** to open the dialog from MD Export.

At the top of the **Data export** dialog the path and format currently selected is displayed. Both may be changed in the **Export properties** dialog.

How to proceed

1.   You may decide to export all channels of the (later) specified type or just those contained in the Channel list.
     Refer to Building a channel list on page 246.
2.   Then open the Export dialog, e.g. with 📥.
3.   Decide whether you want to export data concerning the whole project or certain tests from this project only.
4.   Select whether all types of data, e.g. Snapshot data, Fast data etc. is to be exported.
5.   Choose the export format, destination and file name: **Export properties**. Depending on the export format, additional settings are possible.
     See also File formats and format options on page 247.
6.   Decide which export options to use.
     See also Available export options on page 247.
7.   Click on **Start export** to start the process.
     Depending on the number of data the export may take a while. The progress bars at the bottom of the dialog indicate the status of the export. After the export a log

file is available stating the operations done and whether the export of a channel specified caused problems or errors and was successfully or not.

☞ The file EXPORT.LOG is created in "Documents/HBM/MDExport" and over-written by the next export. If selected, it also contains a list of all channels *not* exported (**Log missing channel names**).

5.4.1 Building a channel list

How to proceed

● Mark in the upper part of the window either the Devices entry of a test or a single device. This displays the relevant channels in the lower part of the window.

● Open the **Channel list** window with 📋 or **Channel -> Channel list**.

● Mark the required channel(s) from your project (lower part of the window) and drag these to the **Channel list** window.

🔍 allows you to search for specific channel names using part of the name: Use **%** or ***** as wild cards for several characters and **_** or **?** as wild cards for single characters. You may use wild cards more than once in the search string as in **%nnel??04%**. Without any wild card character, an exact match is searched. Check the Search result section for channels found.

● You may also import channels contained in a MC Definition List (🔲), a MC Setup database file (🔲) or an Excel file (🔲).

● Use ➕🔲 to include the information about zero values acquired either for storage channels (Zero_Tag slow DAQ) or for snapshot channels (TagZero). However, the channels contain only references. The zero values themselves are stored in file ZERO.BIN in the MD Server project. The number shown here is the index under which the zero value and time point can be found in file ZERO.BIN.

● Use ➕🔲 to include time channels.

● If more channels are added by these functions than required, mark and delete unwanted channels with ✏️.

● You may rearrange channels: mark the required channels in the left column (number column), then use the context menu to cut and paste these channels. A red dotted line shows the insertion position.

☞ Close the **Channel list** window with **Accept** if all required channels are selected.

5.4.2 Available export options

Export "Stored data" into one file: If the data has been saved to multiple files by MD Server (one file per channel) you may use this option to again combine all files into one. Otherwise one file per channel is saved.

Concatenate split test files: This option saves all data into one file if several tests have been made producing several data files.

Export empty channels: Even channels in overflow are exported if this option is activated. The default setting is not to export these channels. Overflow values can be defined for non-HBM devices and HBM devices (e.g. MGCplus, QuantumX) separately, see Null and Overflow values on page 247.

Log missing channel names: All channels are listed which could not be exported because they were not there (available in this project). This might e.g. happen if you import a channel list from an older project.

5.4.3 File formats and format options

☞ We recommend to use **catman** or **ASCII + channel information** as format, because here all traceability data is stored as well. The same applies to **Microsoft Excel**, if **Include channel information** is activated on the **Excel** tab.

For most data formats you can choose which type of data is to be included and how **Null values** and **Overflow values** shall be treated: Use the **General** tab to define your settings. A **File comment** will be used in the catman and ASCII formats only.

If other than the default file names (MD Server file names) for the respective export data (Stored data, Snapshot data etc.) are required, specify these on the **General** tab.

Null and Overflow values

A **Null value** is a user defined overflow value for a non-HBM device. For example, if a MOOG channel sends -55555 as overflow, use this field to define that value.

An **Overflow value** is for HBM devices like MGCplus or QuantumX. If an HBM device signals an overflow, the value specified here is stored in the database. This makes it easier to find those values because they do not depend on scaling and resulting maximum amplifier output.

In default channels with these values are not exported. Activate **Export empty channels** in the **Data export** dialog to include those channels.

The (binary) catman standard format (∗.BIN)

Files in the **catman standard format** can only be read in again by catman itself. However, the speed of the export is substantially higher than in other formats. In addition the channel information (acquisition parameters of a channel) are also saved.

ASCII and ASCII + channel information (∗.ASC)

⚠️ When exporting values in one of these formats, the currently set number formatting for each channel in the *Database editor view* is used. The default format is with three decimal places. For very small values, this could lead to a loss of data. Increase the number of post decimal places in this case. A time channel that contains absolute time values (serial date-time format) must have a date-time formatting. A change can only be carried out in the **Database editor**.

Example

With a sample rate of 9600 measurements per second, then in the setting with three decimal places, the first five values for the time channel are saved with 0.000 because the value is less than 0.0005. From the sixth value there is then 0.001, etc. Either use more decimal places or one of the scientific formats with an exponent.

Data exported in this form is present as ASCII text file and can be read directly by numerous other programs. The data of a channel is located in a column with the columns being separated from one another by a freely selected **ASCII separator,** e.g. 🔳 (tab key) or semicolon. Each row is terminated by CRLF (Carriage Return/Linefeed, standard for DOS and Windows). The first rows of the file contain, if required (**+ channel information**), acquisition parameters (traceability data).

⚠️ The traceability data (channel information) can only be restored from a file saved in format **ASCII + channel information**, if the *parameter IDs* have been saved additionally (appended).

☞ Please note that export in the ASCII format is slower than the binary export due to the formatting processes.

MS Excel (∗.XLS, ∗.XLSB)

⚠️ Export in this format is only possible if MS Excel is installed.

Files in this format can be directly read by MS Excel. When exporting, you may use several options, just click on the **Excel** tab. The channel information contains the traceability data. If these are transferred, it is useful to set the start row for values to 35 or higher,

because not every channel might have the same amount of traceability information, e.g. the time channel has no scaling etc.

☞ The entry for **Data starts at...** is always evaluated, even if no channel information is transferred.
If data is stored with **Channels into rows**, the corresponding number of columns is used. If more than 256 columns are necessary, the export routine creates the required number of worksheets and uses those for consecutive columns.

⚠️ With MS Excel 97 to 2003 the export is restricted to 65,000 values, because these worksheets cannot have more than 65,536 rows. If more data is present in the channel, they are ignored or the import is terminated with an error message. For MS Excel 2007 or higher (default setting) even 1,000,000 values are possible. Make sure the format **Excel workbook (∗.xlsx)** or **Excel binary workbook (∗.xlsb)** is the default setting for saving in Excel in this case. In both cases a binary workbook is saved (extension xlsb).

Further information can be found at https://www.microsoft.com/.

NI DIAdem (∗.DAT)

Files exported in this format can be directly read in by the data analysis software DIAdem® from National Instruments Engineering GmbH & Co KG. A header file and a data file are produced when exporting data in this format. The header file can be directly interpreted by DIAdem® and the data file is in the format REAL64.

☞ You may import files in this format into catman Professional again.

Further information can be found at https://www.ni.com/.

FlexPro (∗.FPD)

Files in this format can be directly processed by the data analysis software FlexPro®.
Further information can be found at https://www.weisang.com/.

CAEMAX RemusLAB (∗.RMS)

Files in this format can be directly processed by the data analysis software CAEMAX Remus.
Further information on CAEMAX can be found at https://www.caemax.de/en/.

MTS RPC III (∗.RSP)

Files in this format can be processed directly from the data analysis software RPC® III from the MTS® Systems Corporation. As the format does not allow individual channel lengths, so-called groups each with 2048 values are always saved. If the channels to be exported do not contain a multiple of this value, the missing values will be filled with zeroes. You should therefore try to acquire or export a multiple of 2048 values. As integer values with 16 Bit must always be saved in this format, catman Enterprise uses the minimum and maximum of the respective channels and scales all values accordingly.

Further information on MTS Systems Corporation can be found at https://www.mts.com/.

Vector MDF 3 (∗.MDF)

MDF (**M**easurement **D**ata **F**ormat) is a binary file format for measurement data which was developed in 1991 by Vector together with Robert Bosch GmbH. In the meantime, the MDF format is widely used for measurement data in the automotive industry. Refer also to MDF 4.

Further information on Vector Informatik GmbH can be found at https://www.vector.com/.

ASAM MDF 4 (MDF 4.1, ∗.MDF)

MDF 4 (**M**easurement **D**ata **F**ormat) is a further development of the MDF 3 format and in this version is an ASAM standard (Association for Standardization of Automation and Measuring Systems). With this development there is no 2 GB restriction as with MBF 3 (theoretically 2^{64} bytes, i.e. 10^{10} TB per file can now be written). In addition the format has far-reaching possibilities of saving metadata. Also, special functions of MDF 4.1, such as data compression or Quick Preview, are not supported by catman Enterprise.

Further information on ASAM can be found at https://www.asam.net/.

nSoft (∗.DAC, ∗.MDF)

catman Enterprise supports the export of Database channels for the evaluation and analysis package nSoft® from nCode. catman Enterprise can produce two different nSoft® formats:

Time series format (∗.DAC)

An equidistant measurement series is written to a file, i.e. one Database channel produces one file. The file name is created with the name provided by you, the channel name (appended with an underscore) and the file extension DAC.

X/Y pairs format (∗.MDF)

Two channels are written to one file, one as the x-axis, e.g. displacement, and the other as the y-axis, e.g. force. The file extension is MDF.

Further information on nSoft and nCode can be found at https://www.ncode.com.

EDASWin

Files in this format can be directly processed by the data analysis software EDAS Win®.

Further information can be found under https://www.mh-gmbh.de/.

nCode s3t

Files in this format can be directly processed by the software GlyphWorks and nCode DesignLife.

GlyphWorks is a software program for the analysis of test data; it processes a large amount of data and offers a graphical, process-orientated user interface. You can create a workflow for the analysis using drag & drop of analysis modules.

nCode DesignLife is a program for lifetime analysis. nCode DesignLife uses FEM results and operational strength tests for the lifetime assessment.

Further information on HBM's nCode software can be found at https://www.hb-mprenscia.com/.

5.4.4 Save/Load export layout

Save your export settings if you want to use the same or similar settings again: **Export -> Save export layout**. The Channel list is saved together with all export settings. You may still modify settings after loading an export layout ( ) or **Export -> Load export layout**).

The last used settings are always retained for the next export as *LastAutoExportlayout* and are also used for menu item **Data export** in MD Server.

INDEX

Z

A00871 17 E00 02