ETAS MDA V8

Measure Data Analyzer

User Guide

www.etas.com

MDA V8 I User Guide R08 EN I 12.2025

Contents

1 Introduction

1.1 Intended Use

The ETAS Measure Data Analyzer (MDA) software allows you to evaluate ECU and vehicle measurement data in MDF (Measurement Data Format) format. MDA V8 enables quick processing even with large files and data volumes as regards the number of signals, measured values per signal, and channel groups.

The MDA V8 data analysis tool can be used for a variety of applications:

- Visualization and evaluation of ECU and vehicle measurement data
- Evaluation of extensive measurements
- Data extraction from measure files
- Conversion of MDF formats
- Adjustment of measurement grids

Working with MDA V8 offers many advantages:

- Intuitive use
- Fast processing with numerous signals and large measure files
- Fast zooming and scrolling in long measurement series with the oscilloscope
- Easy calculation of derived signals
- Consistent data management when applying a time offset to a measure file
- Simple comparison of relevant findings using cross-instrument synchronization
- Easier interpretation of the measurement data thanks to the direct connection to EHANDBOOK and its ability to clearly display the ECU software documentation

1.2 Target Group

This manual addresses personnel who wants to evaluate measure data, especially when working in the fields of automobile control unit development and calibration. To operate MDA, general knowledge of computer operation procedure is sufficient. For a reasonable interpretation of the data, understanding of the recorded signals and their meaning is required.

1.3 Data Protection

If the product contains functions that process personal data, legal requirements of data protection and data privacy laws shall be complied with by the customer. As the data controller, the customer usually designs subsequent processing. Therefore, he must check if the protective measures are sufficient.

1.4 Data and Information Security

To securely handle data in the context of this product, see the next sections about data and storage locations as well as technical and organizational measures.

1.4.1 Data and Storage Locations

The following sections give information about data and their respective storage locations for various use cases.

1.4.1.1 GPS Map

When using the GPS Map, GPS data points are not sent to the external data pro-vider (Omniscale GmbH), but processed and visualized internally within the tool. Particularly the fol-lowing personal data and/or data categories which can be traced to a specific individual is used for the purpose of visualization:

- Communication data: IP address

When using the GPS Map, particularly the following personal data and/or data cat-egories, that can be traced to a specific individual, is sent to the external map data provider (Omniscale GmbH) and used there for the purpose of providing the request map data, and for detecting and preventing malicious attacks on their infrastructure:

- Communication data: IP address

1.4.1.2 License Management

When using the ETAS License Manager in combination with user-based licenses that are managed on the FNP license server within the customer's network, the following data are stored for license management purposes:

Data

- Communication data: IP address
- User data: Windows user ID

Storage location

- FNP license server log files on the customer network

When using the ETAS License Manager in combination with host-based licenses that are provided as FNE machine-based licenses, the following data are stored for license management purposes:

Data

- Activation data: Activation ID
  - Used only for license activation, but not continuously during license usage

Storage location

- FNE trusted storage

  `C:\ProgramData\ETAS\FlexNet\fne\license\ts`

1.4.2 Technical and Organizational Measures

We recommend that your IT department takes appropriate technical and organizational measures, such as classic theft protection and access protection to hardware and software.

2 Installation

2.1 System Requirements

For the latest information on the needed system requirements, see the Release Notes in the Service Pack Installer located at `Installation File\Documentation\ReleaseNotes` or, after the MDA installation, at `%ProgramFiles%\ETAS\MDA8.7\Documentation\Readme`.

2.2 Installing the Software

You can install the MDA software from a DVD, a network drive, or using the INCA Service Pack Installer.

For more information, see the MDA Installation Guide or the INCA Installation Guide.

In the MDA Installation Guide you can find the following topics:

- How to install MDA as an administrator
- How to install the software using the INCA Service Pack Installer
- How to customize the network installation
- How to customize the support information

2.3 Licensing

A valid license is required to use the software. You can obtain a license in one of the following ways:

- from your tool coordinator
- via the self-service portal on the ETAS website at www.etas.com/support/licensing
- via the ETAS License Manager

To activate the license, you must enter the Activation ID that you received from ETAS during the ordering process.

For more information about ETAS license management, see the ETAS License Management FAQ or the ETAS License Manager help.

To open the ETAS License Manager help

The ETAS License Manager is available on your computer after the installation of any ETAS software.

1. From the Windows Start menu, select **E** > **ETAS** > **ETAS License Manager**.

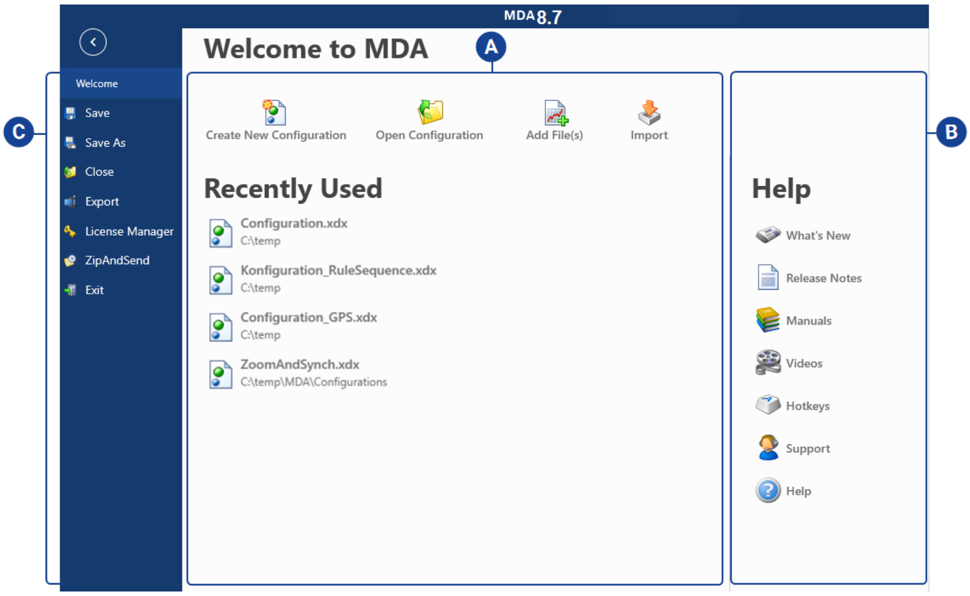
   The ETAS License Manager opens.

2. Click in the ETAS License Manager window and press F1.
   The ETAS License Manager help opens.

3 Basics

In MDA V8 the central view is the configuration which contains the layout and visualization (layers, instruments, and signals), and a reference to each measure file. You can customize the layout and content for the different views. MDA provides many useful functions such as zooming, scrolling, or synchronizing different views.

When you start MDA V8, the **Home** page is displayed. The Home page makes it easier for you to get started with the software and provides an overview of the main functionalities. To go back to the **Home** page, click the ribbon **Home**.



A

	Create New Configuration Creates a new configuration containing one default layer.
	Open Configuration Allows the selection of an existing configuration file in XDX format.
	Add Files Adds a measure file from your file system. For more information, see "To add a measure file" on page 43.
	Import Imports configuration contents for reuse into the active MDA V8 configuration from different file formats (e.g. XDX, XDA and XCS).. For more information, see "Importing an XDA Configuration" on page 40and subsequent chapters.
	Recently Used Contains a list with the recently used configurations.

B

An overview lists new functions and program properties, as well as all available PDF manuals and videos. For support, you find the direct access to the online help and the contact information of the ETAS hotline.

C

To manage the license required for using MDA V8, click **License Manager**.

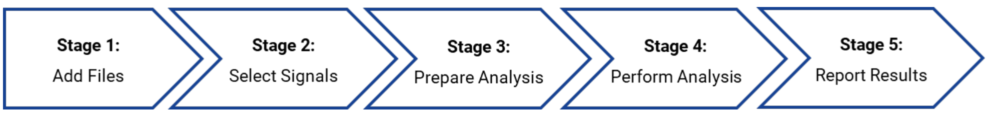
In case of issues, you can send with **ZipAndSend** a report of product issues. For more information, see "To send problem reports using ZipAndSend" on page 194.

The following section provides you an overview of the basic steps and windows in V8.7.

3.1 Getting to Know the Basic Workflow

To see how you can quickly get familiar with the basic handling of MDA V8, watch our video **Just start - Get quickly familiar with MDA V8**.

The following outlines the workflow of a typical MDA use case:



Stage 1: Add Files	To add measure files, select the desired target configuration. For more information, see "To add a measure file" on page 43
Stage 2: Select Signals	In the Variable Explorer, you can use filter and search options to find the variables (respectively signals) that you want to use in your configuration. Then, you assign the selected variables (respectively signals) to an instrument. For more information, see "To assign signals to a new instrument" on page 132.
Stage 3: Prepare Analysis	You can enhance and optimize the configuration, for example, defining a time offset (see "Defining a Time Offset for a Measure File" on page 47), creating calculated signals (see "Defining Calculated Signals" on page 173), and adding additional signals, instruments, or layers. For more information about the usage of instruments, see "Instruments" on page 64, and about the layers, see "Using Layers" on page 60.
Stage 4: Perform Analysis	To analyze the data in a configuration, for example, by creating calculated signals, or by synchronizing instruments. For more information, see "Calculated Signals" on page 172, respective "To synchronize instruments" on page 120. You can use the zoom function to navigate to a specific time segment and get the best representation of the data. Using cursors allows you to get more precise values at time stamps and synchronizing several instruments enables you to monitor them in parallel and identify correlations. For more information, see "Using Cursors" on page 80, respective "To synchronize instruments" on page 120.
Stage 5: Report Results	Document your findings, for example by printing the oscilloscope's view or by exporting only relevant measure data into a new file. For more information, see "Exporting and Converting Measure Data" on page 50.

3.2 Getting to Know the Windows

- **Configuration**

  The configuration is the working area in which you can analyze the measure files. Signals can be visualized in the instruments and analyzed. Additionally, multiple layers and multiple instruments can be invoked inside a single configuration file. This allows you to carry out multiple analysis simultaneously.

The following docking windows provide specific functionality for different aspects of the analysis.

- **Calculations**

  In this window, calculated signals and function instances can be created. The resulting signals can be used like ordinary signals for further analysis. For more information, see "Calculated Signals" on page 172 and "Functions" on page 139.

- **Configuration Manager**

  The Configuration Manager is a schematic representation of the working area which allows you to find and display efficiently all items of your configuration. The content of each configuration with its related layers, instruments, and signals is displayed in a tree view.

- **Display Name Rules**

  In this window, you can create and maintain rules to shorten long signal names to the relevant string. For more information, see "Defining Variable Name Display" on page 53.

- **File Explorer**

  The File Explorer displays the list of all configuration files that are opened in the present session of MDA. Additionally, details of the measure files associated with the configuration are also displayed. The active configuration is highlighted in bold.

- **Information Window**

  The Information window provides additional information about the selected object, like a configuration description, the measure file comment, or meta information for signals.

- **Instrument Box**

  The Instrument Box displays the list of instruments that can be used to visualize and analyze signals.

- **Notifications**

  In this window, you can find all warning and error messages. The newest message is always at the top. You can open the Notifications window by clicking on the message in the status bar.

- **Properties**

  In this window, you can set and maintain the appearance and behavior of the instrument properties.

  For each instrument property, the tooltip provides a detailed description of the behavior and the possible options.

- **Time Offset**

  In this window, you can align data from different measure files with regard to time.

- **Variable Explorer**

The Variable Explorer displays the list of variables available for analysis in the active configuration. For more information, see "Signal Selection" on page 125.

You can select and arrange all windows according to your needs. For more information, see "Customizing the Window Layout" on page 22.

To see how you can optimize the view by defining docking windows behavior and position, or basic instrument settings, watch our video **Optimizing the View**.

3.3 Undoing or Redoing Actions

You can reverse all changes that are stored in the configuration file. You cannot undo some actions, like for example:

- saving or closing a configuration
- creating and exporting a measure file
- undocking and docking windows

You can perform the following actions:

- "To undo an action" below
- "To redo an action" below

To undo an action

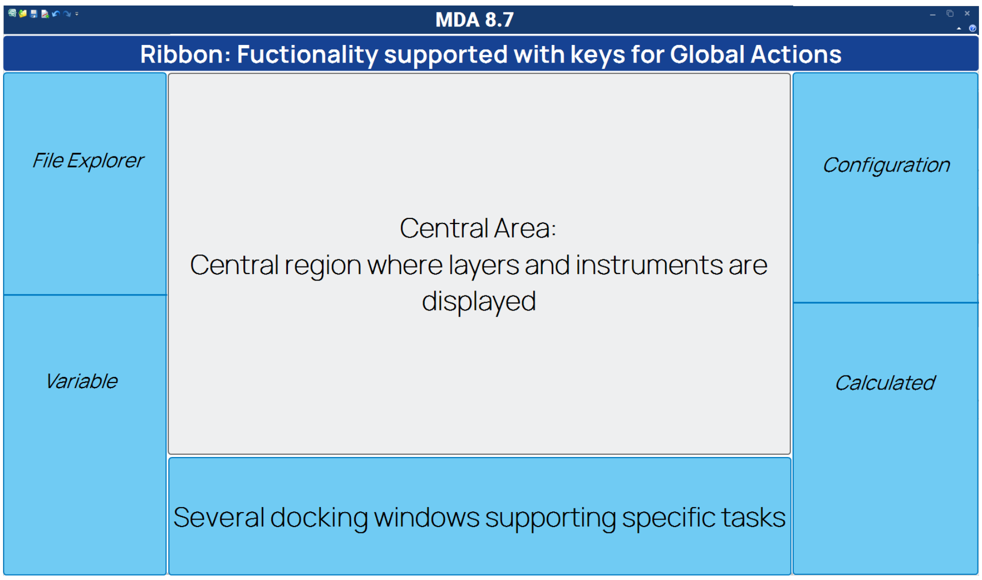
1. On the Quick Access Toolbar click or press Ctrl+Z.

To redo an action

1. On the Quick Access Toolbar click or press Ctrl+Y.

3.4 Operating MDA via Keyboard

The concept for operating MDA V8 via keyboard has three aspects:



Color	Description
	Within the central configuration area, where instruments are displayed, each instrument has its own set of keyboard operations defined. Navigation between configurations, layers, and instruments is done using CTRL+TAB.
	Around the central area, docking windows for specific tasks are available. Each docking window is opened with a specific keyboard combination. Within one docking window the navigation is done using TAB, plus some dedicated keys for important operations.
	On top of the application the ribbon offers to trigger global actions. For most important global actions specific keyboard combinations are supported.

You can perform the following actions:

- "To display the hotkey list" below
- "To open and close a docking window" below
- "To navigate between instruments, configurations, and layers" below

To display the hotkey list

To display a list of all hotkeys, press CTRL+F1. The list contains the performed action, the scope for which the key works and the keyboard combination.

By clicking on the column header, the list can be sorted accordingly. Not shown in the list are pure navigation keys, e.g. PAGE UP, ARROW LEFT or similar.

To open and close a docking window

1. To open a docking window use the appropriate hotkey for the window. For more information, see "Customizing the Window Layout" on page 22.
2. To jump to another docking window, press the hotkey of the desired window.
3. To close the active docking window, press SHIFT+ESC.

To navigate between instruments, configurations, and layers

You can use the keyboard to quickly navigate between the instruments, configurations and layers.

1. Press CTRL+TAB.

   The **Quick Switch** dialog appears with the focus on the selected element on top of the list.
2. To navigate between the columns, press the TAB key.

   To navigate backwards, press SHIFT+TAB. Alternatively, you can use the ARROW RIGHT and ARROW LEFT keys.
3. To navigate within a column, use CTRL+TAB. Alternatively, you can use the ARROW UP or ARROW DOWN key.

4. To select the instrument, the configuration, or the layer, press the ENTER or the SPACE key.

   The selected element is now in the focus.

3.5 User Settings

Generally, the last used setting for signals, instruments, and window positions is stored and automatically reused when e.g. restarting the software or when creating another item of the same type. For example, if you hide the time slider and change the background color in an oscilloscope, all oscilloscopes that you create later will have the same appearance. Only a few exceptions exist. For example, the axis range needs to be set and saved as favorite axis range explicitly. For more information, see "To adjust the axis range manually" on page 76.

The user settings are stored per Windows user in the `settings.user` file which is located in:

`%LocalAppData%\ETAS\MDA\<MDA version>`

When closing MDA, the application creates in the named folder a file `settings_8.x.x.x.user`. Wherein x defines the used MDA V8 version.

When starting MDA again, the available settings are loaded. If for the current MDA version no user settings file exists, the latest user settings of an older version are loaded.

You can copy and paste this file into the user folder of another colleague.
To deploy default settings to new users of V8.7 the `settings.user` file can be added to the following folder:

`%programdata%\ETAS\MDA\DefaultSettings`

When a user starts V8.7 for the very first time and no user-specific user settings files of former MDA V8 versions are available, the default settings are loaded as initial settings. These are saved as user-specific settings when closing MDA V8.

The following table shows all settings that are persisted in the `settings.user` file.

Category	Setting
Bar Chart Instruments	– Signal unit, imbalance unit and imbalance factor – Visibility, naming and decimals in instrument's summary area – Optionally additional instrument settings, e.g. limits
Color	– Color theme of the user interface
Event List	– Number of decimals for time column – "Show/hide" setting for columns of unit, device, and raster – Width of time column
Folder	– Folders for configurations, measure files, exported measure files, and xda import
Format	– Formats for measure files to be added or exported
Histogram	– Class definitions, incl. number of buckets, interval size and center of first bucket
Instruments	– "Show/auto-hide" setting for the instrument header – "Show/hide/auto-hide" setting for the time slider
Language	– Language of the user interface
Oscilloscope	– Background color – Cursor modes (anchored/not anchored or sample-wise/-time-based, or show/hide sample values as cursor tooltips) – "Show/hide" setting for the cursor tooltips, toolbar, grid lines and signal list – Signal list columns and their order – Last selected image format for screen saving
Position	– Position of the quick access toolbar, docking windows, and the MDA main window (incl. size)
Scatter Plot	– Background color – "Show/hide" setting for the toolbar – Last selected image format for screen saving

Category	Setting
Signal	– Decimals for value representation – Favorite axis range – Sample connection (oscilloscope) – Signal line color (oscilloscope) resp. sample color (scatter plot) – Signal line width (oscilloscope) – Sample markers (oscilloscope) – Data representation – Treat as Boolean (oscilloscope)
Statistical Data	– Order and Width of columns – "Show/hide" setting for columns
Status	– "Show/hide" setting for the ribbon – "Show/hide/auto-hide" setting for windows
Table	– Filling option for empty cells – Number of decimals for time column – "Show/hide" setting for rows of unit, device, and raster – Width of time column
Variable Explorer	– Column settings ("show/hide" setting, order, width)

3.5.1 Setting the Language of the User Interface

You can choose for V8.7 one of five possible languages. Once the language is set in the tool, then V8.7 is independent from the language set in the registry entry (`Lang.exe`).

To change the language

1. On the Ribbon, select the **View** tab.
2. In the **Language** drop-down menu, select the language.
3. To display V8.7 in the selected language, restart the program.

3.5.2 Setting the Colors of the User Interface

You can choose for V8.7 one of three color modes.

To change the Colors of the User Interface

1. On the Ribbon, select the **View** tab.
2. In the **Light Colors** drop-down menu, select the preferred color.
   With the settings 'Dark Corlors' and 'Light Colors' the appearance of MDA's UI will be changed permanently. In the mode 'System Default Colors' MDA will follow the definition in the Windows settings.
3. To display V8.7 in the selected color mode, restart the program.

3.5.3 Customizing the Window Layout

By default, the windows of the File Explorer, the Variable Explorer, and the Time Offset are docked on the left side. The windows of the Instrument Box, the Calculations, the Configuration Manager, and the Display Name Rules are docked on the right side, and the Information Window is located at the bottom. You can change the default position and behavior of any of these windows. To see how you can optimize the view by defining docking windows behavior and position, or basic instrument settings, watch our video **Optimizing the View**.

You can perform the following actions:

- "To show and hide a window" below
- "To autohide a window" below
- "To undock a window" below
- "To dock a window" on the next page
- "To restore the default window layout" on the next page
- "To hide and unhide the ribbon" on the next page
- "To show the quick access toolbar below the ribbon" on page 24

To show and hide a window

1. On the Ribbon, select the **View** tab.
2. In the **Show/Hide** drop-down menu, select the window that you want to display.
3. To hide an open window, click ✖.

To autohide a window

1. In the Toolbar of the window, click .

   Depending on its previous position, the window is shown as a tab at the left or right edge of V8.7.
2. When you hover over the title, the window temporarily displays again until you move the mouse off the window.
3. To stop autohiding the window, click .

To undock a window

To display a window floating above the other V8.7 windows, do one of the following:

1. Click the title bar of the window.
2. While keeping the mouse button pressed, move the window to another position.

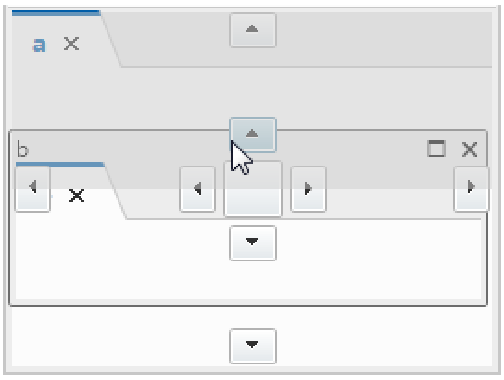
or

1. Right-click the title bar of the window.
2. In the context menu, click **Float**.

To dock a window

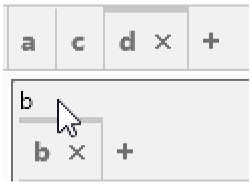
1. Drag the title bar of a floating window to a new position.
2. Do one of the following:
   - Move the cursor to one of the graphical elements.

     The future position of the window is displayed in light gray.

   - Move the cursor to the title bar of another window.

     The future position of the window is displayed in light gray.

3. If the position is correct, release the mouse button.

To restore the default window layout

1. On the ribbon, select the **View** tab.
2. Click **Restore Default Layout**.
3. Restart MDA.

➩ The default layout is restored. This includes the positions and sizes of the of docking windows.

To hide and unhide the ribbon

1. Do one of the following:
   - Click the arrow on the right side of the Ribbon.

   - Double-click one of the ribbon labels, i.e. **File**, **Configuration**, **View**, or **Help**.

     The ribbon is no longer displayed.
2. To show the hidden ribbon, redo one of the actions described under step 1.

To show the quick access toolbar below the ribbon

1. To show the quick access toolbar below the ribbon, click the arrow.

2. Select **Show below the Ribbon**.
3. To relocate the quick access toolbar again, click the arrow and select **Show above the Ribbon**.

3.6 Supported File Formats by MDA V8

For measure files, the following file formats are supported:

- "Binary file formats" below
- "ASCII-based textual formats" on the next page
- "Excel file formats" on page 27
- "Supported File Formats by MDA V8" above

Binary file formats

MDA V8 can read and write all versions of the MDF (Measurement Data Format) file format:

- MDF format V3.x `*.dat`
- ASAM MDF V4 format (`*.mdf`, `*.mf4`)

These formats offer efficient and high performance storage of huge amounts of measurement data. Especially ASAM MDF V4 supports the indexation (see "Indication of File Index Status" on page 49) and compression of measure data in line with the ASAM (Association for Standardization of Automation and Measuring Systems) standard. If the MDF file contains CAN bus trace data see "Loading Bus Trace Files (BLF, ASCII, MDF)" on page 56. The file format allows storage of raw measurement values and all necessary meta information for its interpretation (see "Measure File Comment and Other Meta Information" on page 48).

For a measure file in MDF format, MDA calculates an additional signal named $DateTime from the start of recording. It can be used like other recorded event signals and allows to read the absolute date and time information, e.g. in the cursor tooltip of the oscilloscope or in the column of a table instrument. Note that if the recording was interrupted by a Pause event, the date and time information after the pause event will be incorrect.

MDA V8 additionally supports only specific format dialects of MATLAB®.

ASCII-based textual formats

ASCII-based file formats are usable when exchanging signal data between tools which have no other common file format. Such files contain for the signals only physical data, and are not defined to include as much meta information as MDF files. Moreover, when using textual file formats the tool performance is not as high as with MDF files.

MDA V8 supports a variety of textual measure file formats. There are basically two different format variants for handling multiple rates also known as data groups.

- "Multirate file formats (DXL and INCA dialect)" below
- "Single rate file formats" below

Multirate file formats (DXL and INCA dialect)

DXL (ASCII Multi Rate V4.0)

The advantage of a multirate file format is that it contains only real data, i.e. only data that has actually been recorded, and no interpolated data. The first column contains all recorded time stamps (combined from all available measured rasters). The second column contains all recorded time stamps for a specific raster, e.g. 100 ms. The next columns show signal specific recorded values. The values correspond to the time stamps in the second column. If no value has been recorded for a signal at a specific time stamp, an empty cell is shown.

Two DXL formats are supported by V8.7. These differ in the handling of enumeration signals. In DXL format (ASCII Multi-Rate V4.0) the verbal strings are stored in the file. The format DXL INCA dialect corresponds to the format created in INCA as `ETASGroupAscii (ASCII (Multirate-Write only)`. In this format, for enumerations their numerical values are stored, more precisely their decimal values before the verbal conversion has been applied.

Single rate file formats

In contrast to multirate file formats, all single rate file formats allow to have one time channel only, which can be a merged time channel (based on all time stamps from all rates) or, optionally, defined by an equidistant rate. As a consequence, such files contain typically (constantly) interpolated data.

V8.7 provides an interface to define such file formats. The delivery of MDA includes already some file formats.

- **DIA**

  DIA stands for "Diagra File format". Besides signal names and data also units are given.
- **MRF**

MRF stands for "Measure data refiller format", which was introduced as a first format with refilled (i.e. interpolated) data in an early version of MDA. The format is characterized by an index in the first column which acts as a row counter. Additionally, MRF includes meta information such as device and unit for the exported signals.

– **PEMS CSV** recorded with Portable Emission Measurement Systems (PEMS)

  CSV stands for "Comma-separated values". Ideally, this should be a CSV file according to Real Driving Emissions (RDE) standard for measurement file format of PEMS.
  To support in MDA your own measurement files in CSV format follow the description below for defining INI files for ASCII formats in general.

– **TSV**

  TSV stands for "Tab-separated values". It is reduced to the minimum. The first row contains pure signal names, any following row item stamp and the corresponding values.

For different ASCII format files, i.e. with different file extensions, separate INI files can be created, and will be supported after a restart of MDA. To support differently structured variants of the same file extension, one INI file can be created. If such a combined INI file is applied to interpret a given ASCII file, the first defined structure will be used, which allows to load the file. Therefore it is important to define in the INI file first the more detailed variants, and the most general one at the end.

To define a specific ASCII-based textual format, do the following:

1. Navigate to the following folder:

   `%ProgramData%\ETAS\MDA\8.x\CorePlugins\Etas.TargetAccess.Targets.MeasureFile.Formats.AsciiConfigurable\Examples`

   There you find the files:

   - `exampleAsciiFormat.ini`

     Contains the information how the file must be structured. Note that only the information as described in this file is available when writing a new format. Other contents will not be generated.

   - `exampleAsciiFile.exampleExtension`

     Provides an example of a measure file for the above mentioned INI file.

2. Save the INI file for the new file extension in folder. The contents in the subfolder /Examples will be ignored by MDA. Ensure that for each file extension only one INI file is available. Also file extensions for formats supported by V8.7 directly must not be used again. Otherwise, conflicts will be caused that might lead to errors.

Excel file formats

Loading the contents of Excel files in MDA opens the various analysis capabilities for data in XLS, XLSX or XLSM file format. A similar approach as for supporting customer-specific files in ASCII format is used. Customers can define their own INI file describing the structure of the Excel file. Within one INI file, multiple structures can be given, and MDA will use the first one which matches. If an Excel file includes multiple sheets, each sheet is loaded, and the sheet name is handled as the signal's raster information. Sheets that cannot be interpreted are skipped.

The folder for customer specific Excel file format description is: `%ProgramData%\ETAS\MDA\8.x\CorePlugins\Etas.TargetAccess.Targets.MeasureFile.Formats.Excel`

An example INI file including a description how to define its contents can be found in the subfolder /Examples. The contents of the subfolder will be ignored by MDA.

Bus Trace files

Besides measure files, MDA also supports so-called Trace files from CAN Bus and LIN Bus. For more details see the chapter "Loading Bus Trace Files (BLF, ASCII, MDF)" on page 56

3.7 MDA V8 Add-Ons

You can expand the functional scope of MDA V8 with several add-ons.

3.7.1 Command Line Tools

- **MdfConvert.exe**

  `MdfConvert.exe` allows to convert files from one measure file format into another. Additionally, it can be used to extract only a specific time range or a subset of signals from the original file including an argument for resampling.

  By means of a LAB file you can define which signals shall be exported.

  With a LAB file in V1.3 format, additional filtering for the combination of signal name AND device name is possible.

- **MdfExtract.exe**

  `MdfExtract.exe` can be used in combination with `MdfConvert.exe` and allows you to extract Events from a specific time range of the original MDF V4.x file, and transfer them into the MDF V4.x target file.

- **MdfCombine.exe**

  `MdfCombine.exe` allows to merge multiple measure files into one combined measure file. Thereby signals having the same name and setup (device, raster, data type, etc.) but from separate files, result in one combined signal. `MdfCombine.exe` is limited to source files having the same file format.

If you use the `merge` option, you can combine the source files in a chronological order while the `append` option allows you to define the order in which the source files must be combined.

To see how you can merge multiple measure files into one combined measure file, watch our video **Merging of Measure Files**.

- **Mdf4Indexing.exe**

  `Mdf4Indexing.exe` allows to add an ASAM standard conform index to an existing measurement file in MDF V4 format. Indexing is beneficial for a faster drawing of signal curves in MDA's oscilloscope.

For more information about the individual applications, enter in the Windows DOS console the argument `--help` for the respective command line tool, e.g. `mdfconvert --help`.

3.7.2 Supporting Bus Trace Files (BLF, ASCII, MDF)

3.7.2.1 Loading Bus Trace Files

MDA supports the loading of Bus Trace files, namely CAN Bus Trace files and LIN Bus Trace files. See the following paragraphs for detailed information about how to load both Bus Trace file types.

An additional add-on is required to support Bus Trace files for the CAN or LIN Bus. It enables to load trace files into MDA. In combination with a description file signals can be interpreted from the trace data.

The INCA CAN Trace add-on includes a valid license for the CAN and LIN Bus trace functionality in MDA.

Supported protocols are:

- for CAN: CAN 2.0, CAN FD, J1939,
- for LIN: LIN 2.x

For more information, see "Loading Bus Trace Files (BLF, ASCII, MDF)" on page 56.

3.7.3 Video Instrument

In the Video instrument, you can display the video file recorded with the INCA video add-on and especially for analysis in relation to other measurements. To use the Video instrument, you need a valid license provided with the INCA Video-Integration add-on. For more information, see "Video" on page 106.

3.7.4 Connecting to an ODS Server

The increasing number of measure files requires an intelligent and reliable storage, which should allow to retrieve specific signals from different origins without loading the files. ASAM ODS standard defines such an approach which can be realized as an ODS database. ASAM ODS support is a customer-specific MDA V8 add-on. It is available on demand, and needs to be adapted for the communication and the structure of the ODS database.

Note

The supported database format is limited to ODS 6.

3.8 Interoperability with ETAS Products

3.8.1 Launching MDA from INCA

When working in INCA, you can directly access MDA V8.7 to analyze the measurement recording.

To launch V8.7 from INCA

To establish a connection, MDA V8.7 and INCA (V7.2.2 or higher) must be installed on the same machine.

1. As a prerequisite, you have preselected MDA V8.7 as default MDA version in INCA under **User Options** > **General** tab.
2. Launch V8.7, e.g., by doing one of the following:
   - After having finished the measurement recording in the Experiment Environment of INCA, click .
   - While the recording is still running in INCA, you can take a snapshot, also by clicking . The recording must be in the file format `*.mdf4`.

     Please note that the snapshot recording requires a combination of MDA V8.4.1 and INCA V7.3.0 or higher.

     For further details about the settings for snapshot recording, please see the INCA User Documentation.
   - In the INCA main window, click SHIFT + . Then, select the measure files that you want to use in MDA V8.7.
3. Perform your analysis in MDA V8.7.
4. With INCA V7.2.14 (or higher) and MDA V8.3.3 (or higher), if MDA is already open, and exactly one measure file is assigned to the active configuration, a measure file replacement will take place. In case more than one file is assigned to the active configuration, a dialog appears to decide for an add or a replace operation. For more information, see "To

use the "Add or Replace Files" dialog" on page 43.

If INCA provides besides the measure file also an XDA file, and MDA V8 has no or only an empty configuration, MDA V8 imports the XDA file and the measure file from INCA.

3.8.2 Connecting MDA to EHANDBOOK-NAVIGATOR

To analyze system behaviors, you can combine the measure data displayed in V8.7 with the description of the ECU functionality as provided in EHANDBOOK-NAVIGATOR.

You can perform the following actions:

- "To connect to EHANDBOOK-NAVIGATOR" below
- "To disconnect from EHANDBOOK-NAVIGATON" below
- "To send the cursor time to EHANDBOOK-NAVIGATOR" on page 83
- "To pass the signal name to EHANDBOOK-NAVIGATOR" on page 137

To connect to EHANDBOOK-NAVIGATOR

To establish a connection, MDA V8.7 and EHANDBOOK-NAVIGATOR (V6.1 or higher) must be installed on the same machine.

1. Click .
2. Select a EHANDBOOK file.
3. Select a measure file.

   All measure files are listed that are currently in the active configuration. You can synchronize only one measure file at a time.
4. Click **Connect**.

⇨ The connection is established. In the File Explorer, the selected measure file is marked as synchronized.

You can also connect V8.7 to a running EHANDBOOK-NAVIGATOR session. This requires EHANDBOOK-NAVIGATOR V8.0 or higher.

If you replace the synchronized measure file in the MDA configuration, EHANDBOOK-NAVIGATOR is automatically informed. If you delete the synchronized measure file in the configuration, the connection to EHANDBOOK-NAVIGATOR is disconnected.

To disconnect from EHANDBOOK-NAVIGATON

1. Click .

⇨ The measure file is no longer synchronized.

3.9 Finding Out More

On the **Home** page and on the **Help** tab of the ribbon, you find the following items that help to get more information about V8.7:

What's New	Overview of new functions and program properties
Release Notes	Description of prerequisites and known limitations of the program
Manuals	Access to all available PDF manuals
Videos	Access to all available MDA V8 feature videos
Hotkeys	Overview of all keyboard hotkeys
Support	Contact information of the ETAS hotline and optionally customer-specific support information
Info	Version information and Safety Advice
Help	Access to the online help

You can access the information materials also from the **Windows Start Menu** › **ETAS V8.7** › **Manuals** or open the folder directly in Windows Explorer under `C:/Program Files/ETAS/MDA8.x/Documentation`.

4 Configuration Creation

In order to analyze measure data, a configuration is needed. You can create a new configuration or open an existing configuration file.

4.1 Maintaining Configurations

When starting V8.7, an empty default configuration is created automatically. This configuration is set to active and measure files can be added directly.

> **Note**
>
> Starting with MDA V8.7.6 configurations from newer MDA versions can be loaded at least partially. The used MDA version will inform you that an import of the supported objects from the newer configuration is done. The original (newer) configuration will remain unchanged. With the next MDA releases the compatibility import will be extended continuously.

4.1.1 Creating, Saving, and Closing Configurations

You can perform the following actions:

- "To create a new configuration" below
- "To load a configuration" below
- "To load a recently used configuration" on the next page
- "To select the active configuration" on the next page
- "To save a configuration" on page 34
- "To close a configuration" on page 34

To create a new configuration

1. On the **Configuration** tab of the ribbon, click .

   The new configuration is created containing one default layer. By default, the name of the new configuration is set to "Configuration". If the name is already in use, it is extended by an increasing number. The name of the new configuration is displayed in the header of the V8.7 window.

2. Continue with adding measure files to the newly created configuration. For more information, see "To add a measure file" on page 43.

To load a configuration

To see how you can add a measure file, save and open a configuration, or add configuration comments, watch our video **How to Use a Configuration**.

1. On the **Configuration** tab of the ribbon, click .
2. Select an existing configuration file in XDX format.

⇨ The configuration file is opened.

If the configuration file is invalid or the version is not compatible, an error message is displayed.

To accelerate the usage of a configuration, V8.7 performs an automatic search for all files referenced in the configuration. In a first step, MDA tries to load the original file from the referenced absolute path. In a second step, it searches for a file with the same name in the folder from where the configuration is opened. Finally, the automatic search is extended to the subfolders. The automatic search results are shown in a dialog. Use the checkbox to decide whether to replace or reject a proposed measure file. If no file is found, the file will be listed in the Configuration Manager with as a "missing'" file .

MDA automatically saves at regular intervals the loaded configuration to avoid data loss in case of an accidental closure or crash. After the restart, you can choose whether the original or the backup configuration shall be loaded. If MDA is closed in a controlled manner, all backup configurations are cleaned up. The file extension for the backup configuration is XDX.TMP, which is saved in the same directory as the original configuration. If the configuration has not yet been saved, it is saved under Windows Documents.

To load a recently used configuration

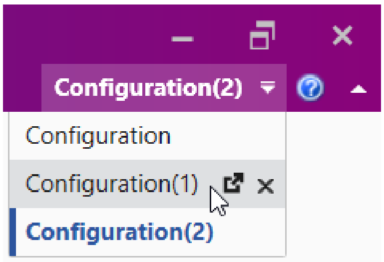
1. On the **Configuration** tab of the ribbon, click the drop-down menu **Open**.
2. A list with the recently used configurations is displayed.

   Select one of the entries and the configuration file is opened. If the configuration file cannot be opened, an error message is displayed.

To select the active configuration

If you have opened several configurations, you can select the active one.

1. Do one of the following:
   - Click the drop-down menu at the right side in the blue area of the application header, and select one of the listed configurations.

     The currently active configuration is marked with a bar and listed in bold blue font.
   - In the File Explorer, select one of the configurations.

     The selected configuration is active. This means that the information of all other windows refers now to this configuration. You can also select one of the layers of the active configuration. For more information, see "To switch to a specific layer" on page 62.

- Press CTRL+TAB to open the Quick Switch window. By using the arrow keys you can navigate horizontally and vertically. Select one of the configurations loaded already in MDA, and confirm the selection.

To save a configuration

To see how you can add a measure file, save and open a configuration, or add configuration comments, watch our video **How to Use a Configuration**.

1. If a configuration contains unsaved changes, an asterisk is displayed in front of the configuration name. To save the changes, select one of the following entries on the **Configuration** tab of the ribbon:
   - To save the configuration under its existing name, click .
   - To save the configuration under a new name, select **Save As** in the drop-down menu below .

     If you want to use the configuration as a template for measure file analysis, see "Configuration Template" on page 38.
2. Navigate to the location where the configuration shall be stored.
3. Enter the configuration file name.

   The complete configuration is saved. This means, that the current size and position of all instrument windows is saved as well as the current state of the objects (e.g. enabled or disabled synchronization of instruments).

   After saving, the full path name of the configuration is displayed in the header of the MDA V8.7 window.

   If you try to save a configuration created with an older version than MDA V8.7, an information window is displayed.

   If you overwrite the file with MDA V8.7, it cannot be used with an older version anymore. To keep the original configuration, select **Save As** in the information window and save the file under a new name.

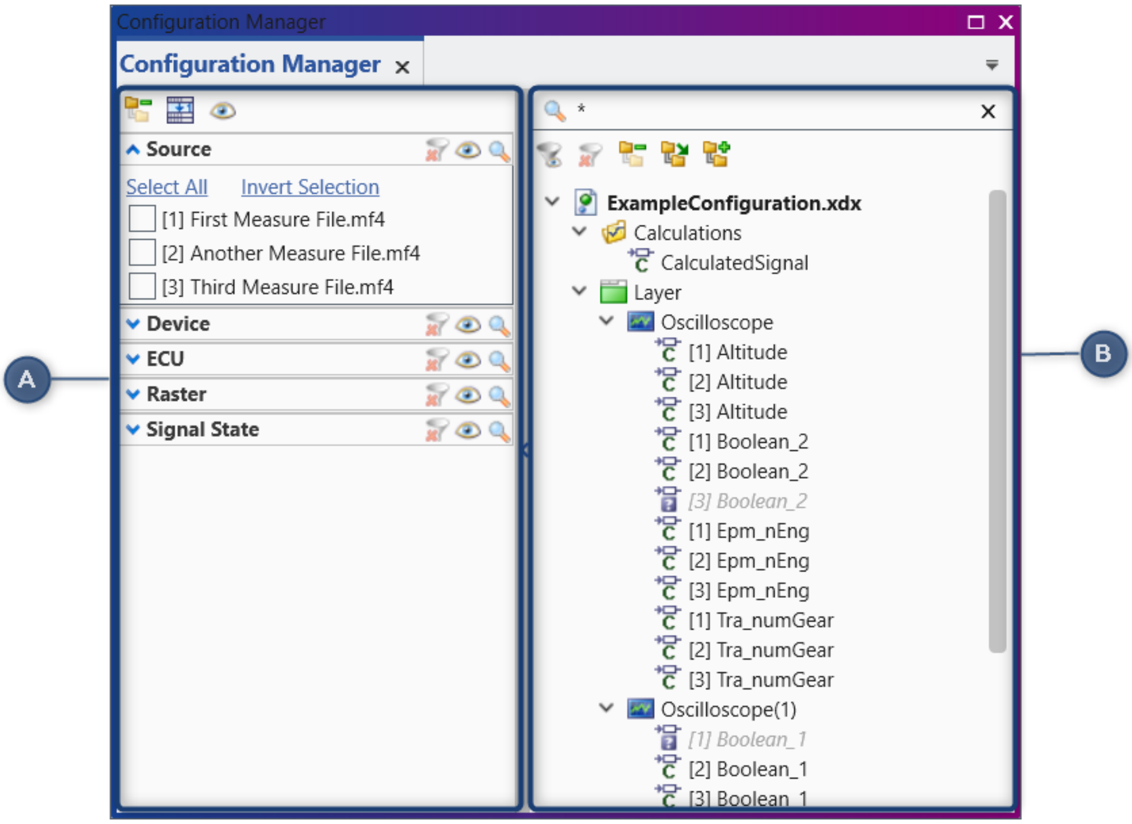
To close a configuration

1. Do one of the following:
   - Click the drop-down menu next to the configuration tab. Move the cursor to a configuration in the list and click .
   - On the **Configuration** tab, right-click the configuration name and select the **Close** entry.
   - On the **Configuration** tab of the ribbon, click .
2. Save the changes or close the configuration without saving.

4.1.2 Searching and filtering within the Configuration

You can perform searches in different areas of the program. If you want to search for a signal being available in the measure file, perform your search in the Variable Explorer. For more information, see "To search for signals" on page 128. If you perform a search in the Configuration Manager, the search will be conducted for signals already being in use and assigned to an instrument.

The Configuration Manager is divided into two areas:



No.	Description
A	**Filter** Entries in the configuration tree view can be filtered by the filter categories. Open or close the area for the filters by clicking on the splitter, or by clicking on the funnel symbol just above the tree. Enlarge or reduce the width of the area by dragging the splitter.
B	**Tree view of the configuration** Hierarchical view of the configuration objects. It includes nodes for calculations, layers and instruments. Eventually signals will be listed.

To adapt the hierarchical view to your needs, the tree elements can be expanded and collapsed:

	**Collapse All Categories** Closes all categories to the layer level.
	**Expand to Instruments** Expands the tree view to the level of instruments.
	**Expand All** Expands the tree view to the level of signals.
	**Accordion** Allows to see only one category at a time. If you activate the accordion mode, the chosen category expands, while any other one is collapsed.
	**Enable / Disable Filters** Applies the selected filters. When you disable the filters, the original selection is maintained, even if you clear all filters using the funnel icon above in the variables list.
	**Clear Filters** Deletes all selected entries in one category or all filters in the Variable Explorer.
	**Search Items** Searches items per category or in the variables list.

In the Configuration Manager you can perform the following action:

- "To search for objects used in the configuration" below
- "To filter the configuration's tree view" on the next page
- "To reset filters" on the next page

To search for objects used in the configuration

1. Open the **Configuration Manager**.
2. Bring the focus into the search box .
3. Enter the search string.

For search queries, take the following rules into account:

- The search is not case-sensitive; it finds data, even if the capitalization in the text is different from the capitalization in the search term.
- You can use the ? and * characters as wildcards in your search string.
- By default, the search string is appended to the "*" wildcard in the search box . If you want to search for data starting with a specific string, delete the wildcard.

The search is initiated directly after entering the first character. The matching search string is highlighted. The result of the search can be a layer, an instrument, or a signal as long as the name matches the entered search string. If you want to bring the respective layer or instrument into the visible area, double-click the desired object.

If you change the data, e.g. rename or delete an instrument, the search result is updated automatically.

To filter the configuration's tree view

To see the desired signals in the tree view list, check within a category the corresponding entries.

If checking several entries within one category these will be combined logically using OR.

Filters defined in different categories and the search box will be combined using a logical AND.

You can disable all filters in a category, the original selection is maintained, but has no effect on the tree view, even if you clear filters using the icon.

To reset filters

– **Clear all filters**

  To clear all filters in categories and search box , click in the area of the variables list.

  Disabled filters in categories remain unchanged.

– **Filters in a category**

  i. To clear the filters in one category, click .

  ii. To deactivate the selected filters, click .

     The original selection is maintained, even if you clear the filters using the icon.

4.1.3 Exporting a Configuration

With the **Export** function you can easily create one zipped file with all relevant objects, namely the configuration and its assigned files. It allows you to provide in a fast way a ready-to-use analysis to other users.

To export a configuration and its files

1. On the **Configuration** tab of the ribbon, click .

   If the configuration is not already saved, you are forced to save it to continue with the export process.

2. Select the location where you want to store the export file.

   MDA V8 combines into the zipped export file:

   - Configuration (XDX)
   - Files (measure files, LAB files, CDF files)

For AFF files, linked BLF and CAN description files are collected.

⇨ The export file gets automatically zipped (the file extension is `*.zdx`) and the export process is visible in the overall status of the progress bar.

> **Note**
>
> FMU files are not exported with the configuration. In this case, you need to provide the FMU file associated with the exported configuration or create a new ZIP file containing both the FMU file and the export file.

To export only a subset of instruments or with just an extract of the measure file, do the following:

1. Create a copy of the configuration that you want to export.
2. Reduce its contents as desired, optionally replace the original measure file by an extracted one.
3. Start the export as described above.

<u>To export a configuration in XDA format for older MDA versions</u>

MDA V8.7 allows to export specific configuration contents into an XDA file format to be used in older MDA versions.

1. On the **Configuration** tab of the ribbon, click .

   If the configuration is not already saved, you are forced to save it to continue with the export process.
2. Select **Export as XDA file** and the location where you want to store the export file.

⇨ The export file can now be reused in older MDA versions.

> **Note**
>
> For technical reasons only the instruments oscilloscope, scatter plot and table are supported. Other instruments, calculated signals, time offsets and layer information are excluded from the export to XDA format.

4.1.4 Configuration Template

MDA V8 allows to load the same configuration in XDX format only once. However, if you want to use a specific configuration as basis for the analysis of different files, you can save it as a configuration template in XDT format.

You can open the same configuration template several times and thereby conduct the analysis of multiple files in parallel. For a better performance, the configuration template is opened without loading the original measure file. To work with the template, it is sufficient to add the desired files and to assign them accordingly in the Add or Replace dialog.

You can perform the following actions:

- "To create a configuration template" below
- "To load a configuration template" below
- "To edit a configuration template" below

To create a configuration template

1. In MDA, switch to the desired configuration to make it active.
2. On the **Configuration** tab of the ribbon, select **Save As Template** in the drop-down menu below .

   You can use the suggested file name or enter a new one. Existing XDX configuration files are not overwritten by the XDT template file.

⇨ The configuration is saved as a template in XDT format.

To load a configuration template

1. On the **Configuration** tab of the ribbon, click .
2. Select the configuration template.
3. Add the new measure file(s) which shall be analyzed.

   In the Add or Replace dialog assign the new files accordingly.

To edit a configuration template

1. On the **Configuration** tab of the ribbon, click .
2. Select the configuration template that you want to edit. Optionally, assign new measure file(s) to it.
3. After the changes, on the **Configuration** tab of the ribbon, select **Save As Template** in the drop-down menu below .

4.1.5 Importing an XDX Configuration

You can reuse contents from an XDX file by importing them into MDA V8. Supported contents are:

- referenced files
- layers and instruments
- calculated signals
- function instances
- display name rules

1. On the **Configuration** tab of the ribbon, click .
2. Select an existing configuration in the XDX format.
3. In the **Import** dialog, select the content that you want to import.

⇨ All selected objects will be imported into the currently active configuration.
If just one file is available in the target configuration, and no file is selected in the import dialog, then the input signals of the imported calculated signals are mapped to the available file.

If more than one or no measure file is available, the original file is shown as entry in the File Explorer, but as a missing file. Calculated signals for which the input signals are not available are indicated with .

Files that are already loaded in the target configuration are skipped. If another content with the same name already exists, an increment is automatically appended to the name of the imported content. The references of the respective file will stay the same.

Time offsets for measure files are ignored.

> **Note**
>
> If you are importing a function instance that uses an FMU for calculation, you need to check that the FMU file is available on the target computer.

4.1.6 Importing an XDA Configuration

In INCA and MDA V7.x, configurations are created in XDA format. To reuse these existing configurations, simply import the XDA file into an V8.7 configuration. The original XDA file will not be changed by V8.7.

1. On the **Configuration** tab of the ribbon, click .
2. Select an existing configuration in XDA format.
3. If V8.7 cannot find the measure file, you can do one of the following:
   - Select the measure file from your file system. The measure file that was previously used in the XDA file is mapped with the newly selected measure file. All signals that are contained in the new measure file are displayed. Only signals that are not contained in the new measure file are indicated as 'no-match' signals.
   - Continue the import without selecting a measure file. All signals are indicated as 'no-match' signals. If you add a measure file later, the **Add or Replace Files** dialog opens. For more information, see "To use the "Add or Replace Files" dialog" on page 43.

For XDA files from INCA V7.x, a new layer is created for the complete imported content. For XDA files from MDA V7.x, a new layer is created for each instrument. V8.7 imports as much contents (instruments, signals and settings, calculated signals, Find criteria as calculated signals, and references to measure files) as are described in the XDA file, and supported in MDA V8. If a content could not be imported, this is indicated by a warning message in the status bar at the bottom of the V8.7 window. When clicking onto it, more details are given.

For more information about the import of calculated signals see "Importing Calculated Signals via XCS Export File" on the next page and "Import Calculated Signals from XDA Files: Differences between MDA V7 and MDA V8 " on page 197.

4.1.7 Importing a ZDX Configuration

1. On the **Configuration** tab of the ribbon, click .
2. Define the location for the extraction of the ZDX file.

⇨ MDA V8 automatically extracts the ZDX file and opens the configuration file.

Thereby it loads all files from the folders created during the extraction process. Eventually, MDA saves the opened configuration with the new path information for the assigned files.

> **Note**
>
> Although the export file is an ordinary zipped file format, it is not recommended to extract it manually.
>
> During the import of the ZDX file, MDA V8 checks the configuration and updates the file paths for the new file locations. If you unzip the file manually, the update for the extracted configuration file is not done.

4.1.8 Importing Calculated Signals via XCS Export File

From INCA and MDA V7.x, calculated signals can be exported in an XCS file.

To reuse these calculated signals formula definitions simply import the XCS file into an MDA V8.7 configuration. The original XCS file will not be changed by MDA V8.7.

1. On the **Configuration** tab of the ribbon, click .
2. Select an existing calculated signals export file in an XCS format.
3. In V8.7 a dialog appears where you can select the measure file to be used for the input signals of the imported formula definitions.

   You can choose between two options:

   - Select the measure file from your file system:

     The formula definitions are mapped with the selected measure file. Only signals that cannot be clearly identified are indicated as 'no-match' signals.
   - Without selecting a measure file:

     All calculated signals are imported but linked to a missing file as shown in the File Explorer. By replacing the missing file by a measure file, the 'no-match' status of the input signals of the calculated signals can be resolved.

If a calculated signal with the same name already exists, the import from an XCS file cannot be conducted, which is indicated by a warning message in the status bar at the bottom of the MDA V8.7 window. When clicking onto it, more details are given.

To overcome the failed import, rename the existing calculated signal and start the import procedure again.

For detailed information how calculated signals are imported, see "Import Calculated Signals from XDA Files: Differences between MDA V7 and MDA V8 " on page 197.

4.1.9 Adding Configuration Comments

You can enter additional information for the configuration. For example, you can explain how this configuration should be used by others.

To know how to add a comment for a specific layer only, see "To add a comment to a layer" on page 62.

1. In the Configuration Manager, right-click the configuration and select **About Configuration**. Alternatively, press CTRL+I.

   The Information Window opens.

2. Enter your comment (up to 10,000 characters).

   Note that if you select a measure file in the File Explorer or a signal in the Configuration Manager before pressing the shortcut keys, then the metadata of the measure file or signal is displayed. For more information, see "Measure File Comment and Other Meta Information" on page 48 and "Displaying Signal Information" on page 135.

   If a configuration comment was entered and the Information window does no longer have the focus, an icon is shown at the configuration node in the Configuration Manager.

4.2 Maintaining Measure Files

The measure file provides the data you want to display in a configuration.

4.2.1 Adding, Replacing, and Removing Measure Files

For each added measure file, an identifier is displayed in front of the file name in the File Explorer. The identifier helps you to identify the source file of a signal. This identifier is always the lowest number that is not yet used for another file in the configuration. If you replace a measure file, the identifier remains the same as before.

To see how you can replace a measure file, watch our video **Replacing Measure Files**.

You can perform the following actions:

- "To add a measure file" on the next page
- "To use the "Add or Replace Files" dialog" on the next page
- "To replace a measure file" on page 44
- "To delete a measure file" on page 45

- "To delete 'no-match' signals" on page 45
- "To map devices" on page 46
- "To assign signals with the same name from another file" on page 132

To add a measure file

To see how you can add a measure file, save and open a configuration, or add configuration comments, watch our video **How to Use a Configuration**.

1. Select the configuration to which you want to add the measure file. If no configuration is already open, it is created automatically when adding a measure file.
2. Do one of the following:
   - If the respective measure file extension was linked to MDA V8 (using **Open with** in Windows Explorer), just double-click the desired measure file in your file system.
   - Drag and drop one or more measure files from your file system onto the main window of V8.7.
   - On the **Configuration** tab of the ribbon, click . Select the measure files from your file system and click **Ok**.
   - In the File Explorer, right-click the configuration name and select **Add Measure File(s)**. Select the measure files from your file system and click **Ok**.

     You can add the same measure file only once. Otherwise, a message is displayed in the status bar at the bottom of the V8.7 window.

     If measure files have been deleted earlier and the configuration thus contains 'no-match' signals, a dialogue opens that allows you to either add or replace the selected measure files. For more information, see "To use the "Add or Replace Files" dialog" below.

To use the "Add or Replace Files" dialog

1. The dialog opens if the following prerequisites are fulfilled:
   - Measure files have been deleted earlier. The configuration thus contains 'no-match' signals.
   - You have performed the steps described in "To add a measure file" above.

     In the table, all newly selected measure files are displayed in the column at the left. In the column at the right column, all deleted measure files from which there are still some signals in use ('no-match' signals) are displayed. If available, all used measure files are also displayed in the column at the right.
2. To replace a missing or used measure file, drag and drop a newly added file from the left into the center of the row for the file shown in the right column.

3. To undo the replacement, click the undo icon on the right side of the file shown in the center column.

   The cell is cleared and the file is added to the list in the column on the left side again.

4. Click **Ok**.

   This results in all files shown in the column at the right to be added as new files to the configuration. The files listed in the center are used to replace the adjacent file at the right and the signals referencing to it.

<u>To replace a measure file</u>

To see how you can replace a measure file, watch our video **Replacing Measure Files**.

To resolve the 'no-match' situation, see "To replace a signal" on page 134.

1. Do one of the following:
   - In the File Explorer, select a single measure file. On the **Configuration** tab of the ribbon, click .
   - In the File Explorer, hover over a measure file and click .
   - Right-click a measure file and select **Replace Measure File**.
   - Drag the new measure file from the Windows File Explorer onto the measure file in the File Explorer that you want to replace. When the Replace icon appears, you can drop the measure file.

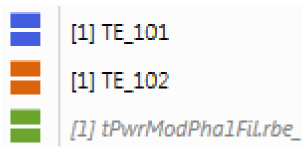
2. Select the new measure file in your file system.

   In the File Explorer, the selected measure file is displayed instead of the previous measure file.

   If a file is already assigned to the present configuration, it cannot be used to replace another file.

   If signals from the previous measure file had been assigned to an instrument, they are displayed in the instrument as follows:

   - Signal names that are contained in the new measure file are displayed in black color as before. This means the signals could be mapped and are displayed with the data from the new measure file.
   - Signal names, that are not contained in the new measure file or cannot be mapped clearly, are displayed in gray color. This status is called 'no-match'. For 'no-match' signals no measure data can be displayed.

     If you have assigned a specific data representation to a signal before replacing the measure file and the desired representation is not supported for the signal in the new measure file, a warning icon is displayed. You can switch to the available data representation. For more information about the oscilloscope setting, see "To change the data representation of a signal" on page 86. For more information about the table setting, see "To change the data representation of a signal" on page 96.

To delete a measure file

Deleting a measure file in MDA does not delete the actual measure file, but removes only its reference to the configuration.

1. In the File Explorer, select the measure files that you want to delete.
2. Do one of the following:
   - On the **Measure Files** tab of the ribbon, click .
   - Right-click the highlighted files and select **Remove Measure File(s)**. If you want to delete only a single measure file, hover over it and click .

     All selected measure files are deleted from the File Explorer. Signals of which the measure file is deleted are still displayed but in gray color and in italics. This status is called 'no-match'.
3. If you add the same measure file again, the signal names are displayed as before the removal indicating that the signals have been restored.

To delete 'no-match' signals

You can clean-up your configuration from signals in 'no-match' state. This is possible in the Configuration Manager for layers or instruments or for the whole configuration.

1. In the Configuration Manager, do one of the following:
   - Right-click the configuration where you want to delete all 'no-match' signals.
   - Select one or multiple layers or instruments where you want to delete all 'no-match' signals.

     Note

     A mixed selection of layers and instruments is not possible.
2. Select **Remove No-Match Signals**.

⇨ An information of the removal is shown in the status bar.

If a file was removed from a configuration, usually 'no-match' signals will remain.

1. In the File Explorer select the entry for the removed file.
2. Right-click and select "Remove No-Match Signals".

This causes all signals from this file to be removed from any instrument. Calculated signals need to be cleaned-up manually.

⇨ An information about the removal operation will be shown in the status bar.

As soon as there are no more signals referencing to the removed file, its entry disappears in the File Explorer.

Alternatively, you can replace a single signal as described in "To replace a signal" on page 134.

To map devices

For information how signals are clearly identified by meta information like ECU, device etc., see the chapter "Signal Selection" on page 125.

In case signals from multiple devices are available in a measure file, it is not always clear which ECU/device combination from the new measure file shall be mapped to the ECU/device combination referenced from a signal used in the configuration. The Device Mapping dialog appears if you start a file replacement or if you copy content from a configuration to a target configuration. In these cases, it is not possible for MDA to automatically resolve the device mapping unambiguously. The dialog allows you to remap the ECU/device combinations.

On the left side of the dialog you can see the list of the ECU/device combinations available in the new measure file, on the right side the combinations of the used signals from the old file.

To map the devices, do the following:

1. Drag an entry from the column **(ECUs) Devices from replacement file** and drop it in the center column of the desired target row.
   MDA tries to assign automatically clear combinations. If the ECU/device information is identical, this is called a 'perfect match'. It is indicated by the lock symbol  , and cannot be changed or removed.
   You can map the same entry to more than one ECU/devices in the current file. This is a possibility to merge signals on a single ECU/devices that formerly were assinged to different ECU/devices.

   or

   Activate the checkbox Map **(ECU/) device assignments automatically** and MDA will map all signals with unique names.
2. Click **OK**.

To reuse a configuration with signals that have the same name but originate from different devices, it is necessary to perform the manual device mapping.

If 'no-match' signals remain after the mapping, you can assign the signals manually (see "To replace a signal" on page 134) or delete them (see "To delete 'no-match' signals" on the previous page).

4.2.2 Defining one Color per File

You can define the color for a file in the File Explorer. This color is used in the oscilloscope and in the GPS instrument when drawing the signal curves of the signals from this file. This allows you to easily identify signals with the same name but from different measure files.

You can perform the following actions:

- "To assign a color to a file" below
- "To clear the color assignment to a file" below

To assign a color to a file

1. In the File Explorer, click the color icon in front of the file name.

   or

   In the File Explorer, right-click the file and select **Set Color**.
2. Select the color that you want to assign to the file and confirm with **OK**.

⇨ The color change affects all signals belonging to the file and overrules the individual signal color.

The assigned color is saved in the configuration and applied the next time the configuration is loaded.

> **Note**
>
> The mechanism is not applied for calculated signals. You can manually assign a color to each calculated signal.

To clear the color assignment to a file

1. In the File Explorer, click the color icon in front of the file name.

   or

   In the File Explorer, right-click the file and select **Set Color**.
2. Click **Clear Measure File Color** and confirm with **OK**.

> **Note**
>
> If you change the color of a signal in the oscilloscope or of the track in the GPS map instrument, the "one color per file" mode is deactivated and the "one color per signal" mode is reactivated.

1. In the corresponding instrument in the configuration, click colored square of the signal in the **Style** column.
2. Select **Clear Measure File Color** and confirm with **OK**.

⇨ The color assignment is deleted.

4.2.3 Defining a Time Offset for a Measure File

Working with several measure files usually requires to align the time line of the different files such that the actual data can be compared.

To see how you can align the timebase of different measure files or apply a time offset to individual signals, watch our video **Using the Time Offset**.

You can perform the following actions:

- "To apply a time offset by shifting a signal curve" below
- "To apply a time offset in the Time Offset window" below

To apply a time offset by shifting a signal curve

1. In the oscilloscope, select the signal that you want to shift. You can define later whether it is a representative for a file or it shall be shifted individually.
2. Press SHIFT and drag the signal curve to the desired position.
3. Confirm one of the following options:
   - **Apply Offset to File**

     The time offset is applied to the whole measure file, i. e. it affects all signals of the file. The numerical offset value is displayed in the Time Offset window.
   - **Apply Offset to Signal**

     For the shifted signal, a calculated signal will be created, which gets the shift applied as time offset in the **Output Options** of the Calculated Signal window. Subsequently, the calculated signal with the time offset will be used to replace the input signal everywhere in the configuration, i.e. in all instruments and when it is used as input of a calculated signal.

   Note that for a calculated signal the individual time offset and the file time offset of its input signal(s) are cumulated.

To apply a time offset in the Time Offset window

1. In the **Time Offset** window, the available measure files with their identifiers are listed. Enter the required time shift in seconds for the respective file. Positive or negative values as well as decimals are allowed.

   To apply a time offset for individual signals, see "Defining Calculated Signals" on page 173.
2. Click **Apply**.

   For all signals from the same file, the samples are shifted accordingly. The shifted data is also used for calculated signals. Note that if you replace the measure file with another one, the defined time offset remains still the same.

4.2.4 Measure File Comment and Other Meta Information

For each measure file used in a configuration, you can display its metadata, e.g. the file comment and information about when, by whom, and for which project the file was created. Depending on the actual file format, some of the described possibilities might not be supported.

To see how you can get more information about a measure file or a signal, watch our video **Displaying Meta Information**.

You can perform the following actions:

- "To display comment and other meta information" below
- "To edit comment and other meta information" below

To display comment and other meta information

1. Do one of the following:
   - In the File Explorer, hover over a measure file.

     The metadata is displayed as a tooltip.
   - In the File Explorer, select the file and press CTRL+I.

     The metadata is displayed in the **Information Window**. You can select the content of the table and copy it to the clipboard.
2. If you select another measure file, the metadata is updated automatically in the Information window.

To edit comment and other meta information

1. In the File Explorer right-click the measure file and select **About Measure File**.

   or

   Press CTRL+I.
2. To edit the comment or meta information, click  .

   You can edit the following fields:
   - User
   - Company
   - Vehicle
   - Project
   - Default and User Comment.

   The original comment or meta information remains in the measure file. It can be found if the measure file is opened with an editor tool. To delete completely the original comment or meta information and make it anonymous, perform subsequently an export of the measure data into a new measure file.

   Note that editing is only possible if MDA V8 gets a write access on the file and only works with MDF3 and MDF4 files. This function does not work for measure files created with snapshot recording.
3. To store all changes, click **Save** or press `Enter`.

4.2.5 Indication of File Index Status

For faster data display when zooming and scrolling in the oscilloscope, a standard indexed measure file is beneficial.

The standard index is supported for MDF file format V3.3 and V4.x. The indexing can be set in the INCA user options. For more information, see the INCA documentation. If a new measurement file in MDF V4.x format is exported from MDA V8 standard indexing is conducted automatically.

In the File Explorer of MDA V8, the icon indicates if the measure file format supports the standard index, but is not yet indexed.

The status of the standard indexing is also listed in the measure file tooltip and Information window.

4.3 Exporting and Converting Measure Data

V8.7 supports all versions of MDF files. MDF files can be read and written. You can either export a complete measure file or a subset of signals. Exporting measure date, i.e. writing new files, is limited to the capabilities of the target format. Therefore a validation step is required before exporting. To speed up the process, validation is done on demand only. When you have made all settings, click **Validate**. If the validation was successful, you can continue with the Export process by clicking **Export**.

To see how you can export a measure file for a selection of signals, or how to convert a measure file into a new file format, watch our video **Exporting Signals and Files**.

With MDA V8 the additional add-ons `MdfCombine.exe` and `MDFConvert.exe` are delivered. For more information, see "Command Line Tools" on page 27. The header of MDF 4.x files created with MDA shows which MCD Core version has been used.

You can perform the following actions:

- "To export measure data" below
- "To define the output raster for exported signals" on the next page
- "To convert measure data into another file format" on page 52
- "To compress measure files" on page 52
- "To cancel an export" on page 52
- "To check the export status" on page 52
- "To see the exported file in Windows Explorer" on page 52
- "To use the exported file directly in MDA" on page 52

To export measure data

To see how you can export a measure file for a selection of signals, or how to convert a measure file into a new file format, watch our video **Exporting Signals and Files**.

You have the following choices to export measure data:

- If you want to export a complete measure file, e.g. into another file format or just for a specific time range, select the measure file in the File Explorer.
- If you want to export a subset of signals from the configuration, select the signals in the Variable Explorer.

– In case you want to export a subset of signals for a specific time range only, the export from the instrument level is recommended. Then the assigned signals and the visible time range are used as default values. In the export dialog it is still possible to adapt the time range or to include all signals in the export file.

  Note that exporting from an instrument is currently only supported for oscilloscopes, scatter plots, and tables. In oscilloscopes, signals for which you have hidden the signal curve are also exported.

1. Do one of the following:
   - To open the export dialog from the File Explorer or Variable Explorer, right-click and choose **Export Measure Data**.
   - To open the export dialog from an instrument, click the export icon in the toolbar of the instrument  .

     The Export Measure Data window displays the total number of signals that will be exported. When exporting from an instrument, you can additionally choose to export all signals from all sources.
2. Define the start and end time.

   If you have opened the export dialog from an instrument, it displays the time values of the currently visible time range of the instrument by default.
3. To choose a folder path and file name, click **Browse**.

   V8.7 checks whether a file with the same name already exists. If this is the case, an increment is automatically appended to the export file name. If the export file name is manually changed by the user to a file that already exists, a warning message appears.
4. Click **Validate**.
5. Click **Export**.

> **Note**
>
> When exporting a measure file, the attachments are deleted.

To define the output raster for exported signals

1. To open the **Export Measure Data** window, perform step 1 and 2 as described in section "To export measure data" on the previous page.
2. Select the checkbox **Output Raster** and define the desired raster value.
3. Click **Export**.

   This raster will be used for all exported signals.

If no sample exists for a time stamp of the new output raster, the last available sample before this time stamp is used.

To convert measure data into another file format

1. To open the **Export Measure Data** window, perform step 1 and 2 as described in section "To export measure data" on page 50.
2. If multiple file extensions are possible, select the desired extension from the **File Format** drop-down menu.

   The new file format is automatically applied in the **File Name** field.
3. Click **Export**.

To compress measure files

Depending on the chosen target MDF format, MDA and `MdfConvert.exe` automatically select the compression method. No compression is applied to files in format MDF V4.0. For MDF V4.1, the only defined compression method is deflate. For MDF V4.3, compression method ZSTD is used, because it has a better compression ratio compared to deflate and LZ4, while at the same time a high compression and decompression speed is given.

To cancel an export

1. In the bottom right corner, click  .
2. Click the red icon next to the running export.

   A warning symbol is shown to indicate that the export was canceled.

To check the export status

1. In the bottom right corner, click  .

   All individual exports are listed. The status is marked as follows:
   - Blue: The export is in progress.
   - Red: The export has failed.
   - Yellow: The export was canceled.
   - Green: The export is finished.

   The export icon shows the overall status of the export progress.
2. If you want to reduce this list, you can delete entries for exports that are not in progress. Right-click an export in the list and choose one of the following:
   - **Remove finished element** to delete this specific export
   - **Remove all finished elements** to clear the complete list of finished exports.

To see the exported file in Windows Explorer

1. Right-click an export in the list.
2. Select the option **Open File in Windows Explorer**.

To use the exported file directly in MDA

1. After the export is completed, drag and drop the file from the exported list into the configuration..

The files can be added or can be used to replace an already loaded file directly in MDA.

4.4 Defining Variable Name Display

As variable names can be very long, and thus hard to read, MDA V8.7 allows to shorten them based on user-definable rules. Each rule consists of a combination of sub-rules and the group of variables to which it shall be applied. Each sub-rule shrinks the variable name left or right from user-defined separator or character sequence. Multiple sub-rules are applied step-wise, i.e. the shrinked name from a former sub-rule is used as input for the subsequent sub-rule. All defined rules are listed in the block **Rule Sequence**. The individual rule are applied subsequently in the listed order. If the display name of a variable is modified by a rule, the variable is excluded from all following rule sets. This means that per variable only one rule will be used.

> **Note**
>
> Per signal only one rule is applied.
>
> The rule affects the Display Name only, but not the Name, Display Identifier, or Symbol Link as shown in the Variable Explorer or Information Window.

For more information about what is displayed from the variable name, see "Defining Display Name in the Application" on page 125.

You can perform the following actions:

.

- "To add a rule" below
- "To define the affected variables" on the next page
- "Defining Variable Name Display" above
- "Defining Variable Name Display" above
- "To reorder rules within the Rule Sequence" on page 55
- "To delete a rule" on page 55

To add a rule

1. To add a rule, click the ✚ icon in the **Rule Sequence** toolbar.
2. Assign a name to the rule.

   By default the name is set to "Rule". If the name is already in use, it is extended by an increasing number.
3. To add a new sub-rule, click the ✚ icon under **Define sub-rules**.

   A new sub-rule block appears.
4. Select the direction from which the variable name shall be interpreted when applying the present sub-rule.

5. Select in **Action** which segment of the variable name shall be hidden respectively remain visible.
6. Define the **Separator**, e.g. a dot, underscore, slash, or similar, which shall be used for segmentation of the variable name.
   It is also possible to enter letters or numbers. You can use a single character or a string.
7. The **Number** defines how often the separator must appear until the action is applied.
   The selected direction applies.
8. Select the checkbox **Trim leading and trailing** to delete from the beginning and from the end respectively the undesired separators.
9. Click **Save** to store all sub-rules and the group of variables to be effected.

   After saving, all affected variable names are updated in the active configuration. To apply the sub-rules to a specific group of variables, see "To define the affected variables" below.

   A * indicates if a sub-rule set has unsaved changes.

To define the affected variables

1. Click the **Apply Sub-Rules to Variables where** button to select the group of variables that you want to apply the sub-rule to. You can define the group of variables based on the variable's name, the ECU or device information, its association to a function or a group.
   By default the selection is set to **All**. In this case, the text field is disabled.
   By using another of the available options, the string entered in the text field defines to which variable names the defined sub-rule will be applied.

To adapt the example for test the sub-rule set

To see the effect of the defined sub-rules for a variable name, you can modify the example string.

1. Copy the variable name to be used as test example from e.g. the Variable Explorer, Configuration Manager, or an instrument.
2. Paste it into the field **Example to test**.

   The example string is the input for the first rule. The result of a rule is shown at the bottom of each sub-rule block.
   This intermediate result is used as input for the next sub-rule. The final result, i.e. after applying all sub-rules, is shown under the example string field. If the result string would be empty after applying the sub-rules, the original variable name is used without modifications.

To reorder sub-rules within the rule

1. Click a rule in the **Rule Sequence** list.
   All sub-rules contained in the rule are shown in the editing area.
2. Click a rule-set block in the list.

3. Drag and drop the selected sub-rule block to the new position.
   The new position is indicated by a black line.

To reorder rules within the Rule Sequence

Only the first rule which modifies a variable name is applied. All subsequent ones will be ignored. Therefore you may need to adapt the order of the rules in the **Rule Sequence**.

1. Click a rule in the **Rule Sequence** list.
2. Drag and drop the selected rule to the new position.
   The new position is indicated by a black line.

To delete a rule

1. Click the rule in the **Rule Sequence** list.
2. Click ✖ in the **Rule Sequence** toolbar.

4.5 Handling of Special Files

4.5.1 Using Label Files (LAB)

V8.7 supports to read and write a selection of signals in a label file format (`*.lab`). Besides the signal names, a label file can optionally contain raster information or other meta information for a signal. Using LAB files to filter variables facilitates and accelerates the variable selection in INCA and MDA.

You can perform the following actions:

- "To add a LAB file" below
- "To write a LAB file" below

To add a LAB file

Label files can be handled like measure files. To know how you can add, replace, or delete them, see "Adding, Replacing, and Removing Measure Files" on page 42

Label files are listed in the Variable Explorer in an own category and can be used in the same way as the source category. If you filter based on a LAB file, only the signal name is used as criterion. Additional criteria can be applied by other filter categories. For more information, see "Filtering per categories" on page 129.

To write a LAB file

1. Open the Configuration Manager.
2. Right click an object node or select different object nodes for which you want to export the signals.
3. In the context menu click **Create Label File**.
4. In the **Save As** window select the format.

You can choose between V1.0 (only signal names) and V1.1 (signal names and raster information).

5. Click **Save**.

or

Wherever MDA allows you to create a new measure file, you can select `*.lab` as target format.

MDA supports two variants for LAB file format V1.3. The usual LAB V1.3 file format adds the device information additionally to the signal name and raster information. When loading this LAB file as a filter in INCA's Variable Selection dialog, only ordinary measurement signals will be listed. The LAB file variant "V1.3 INCA dialect" includes adapted device information for calibration variables and so-called #MeasureCals. With this LAB file variant not only measurements signals but also calibration variables and signals derived from calibration variables will be listed in INCA's Variable Selection dialog.

To check the export progress, see "To check the export status" on page 52 and "To see the exported file in Windows Explorer" on page 52. Alternatively, you can trigger an export (see "Exporting and Converting Measure Data" on page 50) and select in the export dialog one of the available export formats `LabFile`.

4.5.2 Loading Bus Trace Files (BLF, ASCII, MDF)

For prerequisites about using this functionality, see "Supporting Bus Trace Files (BLF, ASCII, MDF)" on page 28.

The steps to load CAN and LIN Bus Trace files are basically the same.

1. In the **Configuration** ribbon, click **Add Bus Trace** .
2. MDA will check whether a valid license for the Bus Trace functionality is available.
3. In the dialogue "Enter Bus Trace information":
   - Select a Bus Trace file (mandatory for CAN and LIN).
   - Select a description file (optional for CAN and LIN).
   - If a description file was selected, define the Bus ID or the Bus cluster.
   - Optionally adapt the proposed name for the AFF entry.
4. Click **Save and Add**.

⇨ The input files are combined to an AFF (Associated File Format) file shown as entry in the File Explorer.

By means of the description file the Trace File contents are interpreted to derive signals. These can be used as ordinary signals from measure files.

4.5.2.1 CAN Bus Specifics

A description file can be in DBC or ARXML format. In case of a DBC file the CAN Bus ID needs to be set. In case of an ARXML file the CAN Cluster can be selected from the drop-down menu.

Even without a description file three basic signals will be shown for the CAN AFF file in the Variable Explorer, namely the Bus ID, the Frame ID and the payload.

> **Note**
>
> If no data is shown for the interpreted signals check whether the CAN Bus ID was selected properly.

4.5.2.2 LIN Bus Specifics

A description file in LDF format is optional. If used, defining a Bus ID is mandatory.

Even without a description file three basic signals will be shown for the LIN AFF file in the Variable Explorer, namely the Bus ID, the Frame ID, and the payload.

> **Note**
>
> The LIN Bus ID is typically "1". But if no data is shown for the interpreted signals, another ID should be tried.

4.5.3 Extracting Measure File Attachments

In INCA you have the option to attach some files to an MDF file.

In V8.7 you can extract these files to easily recreate the Experiment Environment. The measure file containing the attachments is displayed in the File Explorer with a child node.

> **Note**
>
> INCA will use the original name of the attachment.

You can perform the following actions:

- "To extract single attachments" below
- "To extract all attachments" on the next page

To extract single attachments

1. In the File Explorer, right-click the attachment that you want to extract.
2. In the context menu, select **Extract Attachment(s)**.

   The Windows Explorer dialog appears.

3. Save the attachment in the desired folder.

   The extraction process is visible in the overall status of the export progress.

To extract all attachments

1. In the File Explorer, right-click the measure file.
2. In the context menu, select **Extract Attachment(s)**.

   The Windows Explorer dialog appears.
3. Save the attachments in the desired folder.

   The extracted files are individual files.

   The extraction process is visible in the overall status of the export progress.

   Note

   When exporting a measure file, the attachments are deleted.

4.5.4 Using Calibration Data Exchange Files (CDF)

V8.7 allows to add CDF files to a configuration.

You can perform the following actions:

- "To add a CDF file" below
- "To use CDF files for the Lookup Table functions" below
- "To use a VALUE from an CDF file as constant" on the next page

To add a CDF file

CDF files can be handled in the same manner as measure files. To see how you can add, replace, or delete them, see "Adding, Replacing, and Removing Measure Files" on page 42. Parameters (i.e. curves or maps) provided via a CDF file are listed in the Variable Explorer.

Note

Axis values of curves and maps used as input for Lookup Tables must have monotonous axis points. If V8.7 founds an inconsistency, an error or warning icon appears.

If CDF files have been deleted earlier and the configuration thus contains calculated signals with missing input signals, a dialogue opens that allows you to either add or replace the selected CDF files. For more information, see "To replace a measure file" on page 44.

To use CDF files for the Lookup Table functions

Curves and maps from CDF files can be used as input signals for the functions `Lookup table (curve)` and `Lookup table (map)` in the Calculated Signals window.

The needed inputs for these functions are:

- the curve or map
- a measured signal
- the interpolation mode (constant or linear)

To use a VALUE from an CDF file as constant

A scalar parameter 'VALUE' from a CDF file can be used in MDA as a constant. This allows to display in an oscilloscope a horizontal line throughout the whole time range.
In a table instrument an entry for the constant VALUE is shown at the very first time stamp of a measurement files assigned to the configuration.
A scalar value can also be used a constant in a calculated signal. Optionally a fixed time raster must be assigned, if no other signal is used in the formula.

To update or modify the values of the curves and maps, use an external editor. Afterwards, you can replace the CDF file by using the replacement function in the File Explorer. For more information on how to replace a file, see "To use the "Add or Replace Files" dialog" on page 43.

To see how you can replace a measure file, watch our video **Replacing Measure Files**.

5 Layers and Instruments

5.1 Layers

To organize the configuration, you can distribute the data on several layers. Each layer can have multiple instruments. The Configuration Manager helps you to keep an overview of all existing layers and instruments. Additionally, you can search for the relevant objects in your configuration. For more information, see "Searching and filtering within the Configuration" on page 34.

5.1.1 Using Layers

Layers help you to organize your configuration. Each configuration can have multiple layers where analysis is run individually.

Calculated signals are included as an extra layer in the configuration.

You can perform the following actions:

- "To create a new layer" below
- "To color a layer" on the next page
- "To duplicate a layer" on the next page
- "To rename a layer" on the next page
- "To reorder layers" on page 62
- "To switch to a specific layer" on page 62
- "To add a comment to a layer" on page 62
- "To delete a single layer" on page 63
- "To delete several layers" on page 63

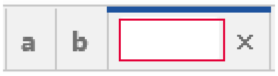
To create a new layer

To see how you can import an MDA V7 configuration and manage the instrument, watch our video **Import and Layer Handling**.

1. Do one of the following:
   - On the right of the layer tabs, click the plus symbol.

     Alternatively, you can right-click one of the layer tabs and select **Add**.
   - In the Configuration Manager, right-click a configuration and select **Add Layer**.

     A new layer is added to the current configuration.
2. Enter a name for the layer. The name can consist of up to 256 characters. If the name is not valid, it is displayed with a red frame. Refer to the tooltip for further information.



If you do not enter a name, the name is set to "Layer" by default. If the name is already in use, it is extended by an increasing number.

To color a layer

To facilitate the differentiation of the layer tabs, the background color of a layer tab can be defined.

1. Right-click the layer tab.
2. In the context menu click **Properties**.
3. Select the desired background color.

Only the lower area of a layer tab will be colored and the offered colors are limited, both to ensure a good readability of the layer name.

To duplicate a layer

Within the active configuration, do the following:

1. On the layer tab, right-click and select **Duplicate**.

   The duplicated layer appears right from the active layer. The name is made unique by adding " (1) ".

In between different configurations, do the following:

1. In the Configuration Manager, right-click one or more layers and choose **Copy**.
2. If you want to copy the layers into another configuration, select this configuration. For more information, see "To select the active configuration" on page 33.
3. In the Configuration Manager, right-click the configuration and choose **Paste**.

   The layers are added. If the name is already in use, it is extended by an increasing number. If you have selected a configuration in which the measure file is not available, the signals of these newly added layers are indicated as 'no-match' signals. To avoid a 'no-match' situation, an automatic signal mapping is performed in case the "Copy" source and the "Paste" target contain both exactly one measure file.

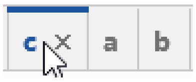
To rename a layer

1. Do one of the following:
   - On the layer tab, double-click the layer name.
   - On the layer tab, right-click the layer name and select **Rename**.
   - In the Configuration Manager, right-click the layer name and select **Rename**.
2. Enter the new name. The name can consist of up to 256 characters. If the name is not valid, it is displayed with a red frame.

To reorder layers

1. Move a layer by dragging its tab header to the new position within the current configuration. You cannot drag a layer to another configuration.

   If you move the layer to the left or right border, the tabs scroll in the respective direction. After scrolling, the new position is indicated.

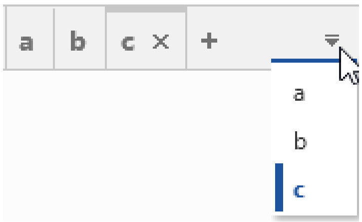
2. Release the mouse button.

   The tab is displayed at the new position.

To switch to a specific layer

1. Do one of the following:
   - On the right of the layer tabs, click the drop-down menu.

     A list with all layers of the same tab arrangement is displayed in alphabetical order. If you click an entry of this list, the selected layer is displayed in the foreground of the configuration.

   - To scroll the layer tabs to the left or right, use the arrow symbols next to the drop-down menu.

     Alternatively, you can use the mouse wheel for scrolling.

To add a comment to a layer

You can enter additional information for a layer. For example, you can describe and document for what a specific layer is meant.

1. Select a layer tab or a layer entry In the Configuration Manager, right-click the layer and select **About Layer**. Alternatively, press CTRL+I.

   The Information Window opens.

2. Enter your comment (up to 10,000 characters).

   When the focus is no longer on the Information window, a symbol  appears on the layer itself and in the Configuration Manager.

   To add comments to a configuration, see "Adding Configuration Comments" on page 42

To delete a single layer

1. Do one of the following:
   - On the layer tab of the active layer, click ✖.
   - On the layer tab, right-click the layer name and select **Remove**.
   - In the Configuration Manager, right-click the layer name and select **Remove**.

To delete several layers

1. Right-click a layer tab.
2. Choose one of the following entries:
   - **Remove All Layers But This**
   - **Remove All Layers**

   If all existing layers have been deleted, a new default layer is created.

5.1.2 Displaying Previews

The preview allows you to quickly navigate through your configuration and display the instrument that is relevant for you. If the display in an instrument is currently scrolling, the preview shows this movement as well. Constraints of your graphics card can have an impact on the performance of this behavior.

You can perform the following actions:

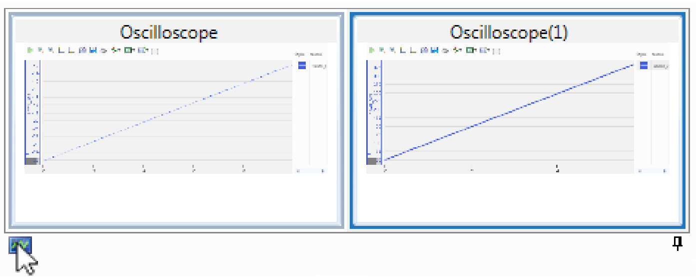
– "To display the preview via the layer taskbar" below

– "To hide and unhide the layer taskbar" on the next page

To display the preview via the layer taskbar

In the layer taskbar at the bottom of the layer, all available instruments of the same type are represented by one icon for the type. Only instruments on the currently selected layer are displayed.

1. If the layer taskbar is currently hidden, unhide it. For more information, see "To hide and unhide the layer taskbar" on the next page.
2. On the layer taskbar, hover over the icon of one of the instrument types.



   For all instruments of this instrument type, a small preview is displayed. If more instruments exist than can be displayed in the small preview, arrows at the left and the right border appear which allow you to navigate to the relevant instrument. The mouse wheel can be used for navigation as well.

3. Hover the cursor over the preview of one of the instruments.

   A large preview is shown in actual size above the small preview.
4. To move the instrument into the foreground of the configuration, click its small or enlarged preview.

To hide and unhide the layer taskbar

1. At the bottom of the configuration window, click 🖈.

   For all layers of the configuration, the layer taskbar is set to auto-hide mode.
2. To unhide the layer taskbar, hover over the gray line and click ⊸.

5.2 Instruments

The following instruments are supported:

- **Absolute Value Bar Chart**

  The Absolute Value Bar Chart provides a bar graph showing the signal values and highlights signal value beyond user defined limits.
- **Delta Bar Chart**

  The Delta Bar Chart allows to visualize for multiple signals the differences of the individual signal values to the average value of all assigned signals for one point in time. Two auxiliary lines for the minimum resp. maximum values make the overview easier to read.
- **Event List**

  In the event list, you can find certain events for analysis by evaluating a search condition. This instrument is recommended for:
  - Finding all time stamps at which the value of a signal changes
  - Displaying a complete list of EVENT entries from an MDF file, like calibration activities, pause events, or comments
- **GPS Map**

  The GPS tracks are displayed in a map view. This instrument is recommended for comparing and analyzing geographic data in relation to other measured signals.
- **Histogram**

  Graphical representation of the measured values of a signal over a definable period of time.time. This instrument is recommended for:
  - Graphical representation of the frequency distribution
- **Oscilloscope**

  The measurement data are displayed as a graph. This instrument is recommended for:

- Displaying numeric signals, especially periodic signals and signals with a large amplitude
- Getting an overview of one or more signals over the complete time range of one or more measure files
- Comparing two signals over time

– **Scatter Plot**

  The values of two signals are displayed as distribution of data samples along the orthogonal value axes of the two signals. This instrument is recommended for:

  - Detecting a correlation of signals
  - Getting an overview of the sample distribution

– **Signal Distribution Chart**

  The Signal Distribution Chart allows to quickly see the distribution of the values of multiple signals for a given point in time.

– **Sortable List**

  The Sortable List allows a quick evaluation of many comparable signals. It is suited for identifying at one point in time the signals with the highest or lowest deviation from the average value of all assigned signals.

– **Statistical Data**

  Statistical properties of numeric and enumeration signals are displayed, such as the average, minimum, maximum and standard deviation. This instrument is recommended for:

  - Analyzing the statistical properties of signals, which can give insight into a signal's character and quality
  - Comparing multiple signals

– **Table**

  The measurement data are displayed in a table sorted by the time stamps of the signals' samples. This instrument can be used for:

  - Displaying non-numeric signals, as well as numeric signals
  - Inspecting the precise value of a signal for a specific time stamp

– **Video**

  In the video instrument you can display the video file recorded with the INCA Video-Integration add-on. This instrument is recommended for combining a visual observation with the recorded measure data, especially for analyzing in a synchronized mode with other instruments.

  To use the Video instrument, you need a valid license provided with the INCA Video-Integration add-on.

5.2.1 Maintaining Instruments

You can perform the following actions:

- "To create an instrument filled with signals" below
- "To create an empty instrument" below
- "To duplicate an instrument" below
- "To move an instrument to another layer" on the next page
- "To rename an instrument" on the next page
- "To delete an instrument" on the next page
- "To change instrument properties" on the next page

To create an instrument filled with signals

1. As a prerequisite, you have added a measure file. For more information, see "To add a measure file" on page 43.

   To know how you can assign a color to a file, see "Defining one Color per File" on page 46.
2. Drag and drop signals from the Variable Explorer or the Configuration Manager onto a layer or layer tab of the configuration. For more information, see "To assign signals to a new instrument" on page 132.

To create an empty instrument

1. Do one of the following:
   - Right-click an empty area on the layer or on the name of the layer tab and select **Add Instrument**. The available instrument types are listed.
   - From the Instrument Box, drag and drop the instrument type that you want to create onto a layer of the configuration. When you hover over a non-active layer tab, the respective layer is selected.
   - In the Configuration Manager, right-click a configuration or a layer and select **Add Instrument**. If you right-click a configuration, the instrument is added to the active layer.

   On the layer, the new instrument is displayed in the foreground and highlighted. In the Configuration Manager, the name of this instrument is displayed in bold.
2. If the configuration does not contain a measure file, the instrument is shown without time information in the time slider. Continue with adding a measure file. For more information, see "To add a measure file" on page 43.

To duplicate an instrument

To see how you can import an MDA V7 configuration and manage the instrument, watch our video **Import and Layer Handling**.

To duplicate a complete layer, see "To duplicate a layer" on page 61.

1. In the Configuration Manager, right-click one or more instruments and choose **Copy**.
2. If you want to copy the instruments into another configuration, select this configuration. For more information, see "To select the active configuration" on page 33.

3. In the Configuration Manager, right-click a layer and choose **Paste**.

   The instruments are added to this layer. If the name is already in use, it is extended by an increasing number.

   After the paste operation, MDA tries to conduct an automatic mapping of the signals from the pasted object and the available signals in the target configuration. In case an automatic mapping is not possible, a file mapping and optionally a device mapping dialog might open. Eventually, signals from the pasted instrument might still not be mapped to signals of the measurement file in the target configuration. Then the unresolved signals will end up in a 'no-match' state. For more information, see "To use the "Add or Replace Files" dialog" on page 43 and "To map devices" on page 46.

To move an instrument to another layer

To see how you can import an MDA V7 configuration and manage the instrument, watch our video **Import and Layer Handling**.

1. In the Configuration Manager, select one or more instruments.
2. Drag and drop the instruments to another layer. For more information about adding layers, see "To create a new layer" on page 60.

To rename an instrument

1. Do one of the following:
   - Right-click the title bar of the instrument in the instrument window.
   - Right-click the instrument in the Configuration Manager.
2. Select **Rename**.
3. Enter the new name. If the name is not valid, it is displayed with a red frame. Refer to the tooltip for further information.

To delete an instrument

1. Do one of the following:
   - On the title bar of the instrument window, click the **Remove** icon. This icon is also available in the preview of the instrument. For more information, see "To display the preview via the layer taskbar" on page 63.
   - In the Configuration Manager, right-click the instrument and select **Remove**.

To change instrument properties

Instrument properties are stored in your user settings. For more information, see "User Settings" on page 19.

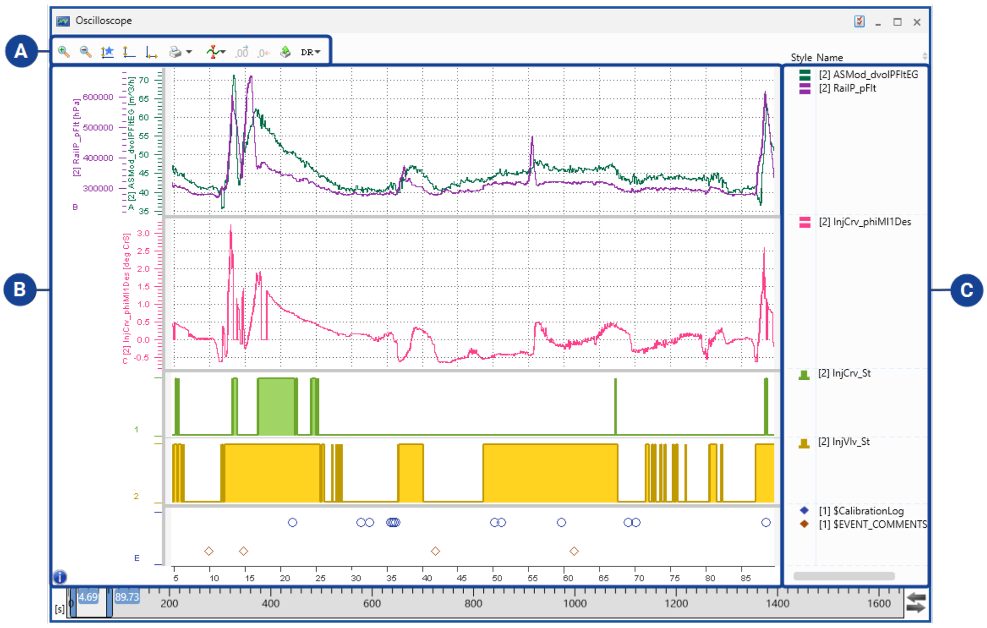
To define the instrument properties, click or use the context menu **Properties**.

For each instrument property, the tooltip provides a detailed description of the properties and their possible options.

5.2.2 Oscilloscope

The oscilloscope can graphically display the course of signals over time.

The oscilloscope is divided into the following areas:



No.	Description
A	**Toolbar** Here several icons for specific functionalities are shown. For a detailed description of the toolbar functions, see "Oscilloscope's Toolbar" below.
B	**Strip area** The signals in the oscilloscope can be distributed onto strips for a better overview. For more information, see "Using Strips" on page 72.
C	**Signal list** The signal list shows the signal information and cursor values. For more information, see "Adjusting the Signal List" on the next page.

5.2.2.1 Oscilloscope's Toolbar

The toolbar provides you with the following functions:

	Zooming functionality
	Adjustment of axes
	Copy, Save or Print
	Cursor options

.00 .0	Number of decimals for signal values
	Export measure data functionality
DR▾	Data representation

The complete set of oscilloscope configuration possibilities is available in the Properties docking window. There the tooltips provide a detailed description of the properties and their possible options.

In the following, only the more advanced functions are explained in further detail.

For more information about synchronizing, scrolling, and zooming, see "Time Navigation and Synchronization" on page 117.

5.2.2.2 Adjusting the Signal List

The signal list on the right side of the oscilloscope shows the signal information, e.g. signal names and optionally meta information (i.e. ECU, device, unit, raster) and cursor values.

You can perform the following actions:

- "To show or hide the signal list" below
- "To define the appearance of the signal list" below
- "To show or hide columns" on the next page
- "To reorder columns" on the next page
- "To reorder signals in the signal list" on the next page
- "To change the number of decimals" on the next page

To show or hide the signal list

To see how you can use strips and define the signals list settings of the oscilloscope, watch our video **Oscilloscope - Defining Strips and Signal List**.

Use the tab **Instrument** in the Properties window.

To define the appearance of the signal list

1. Do one of the following:
   - To show all columns of the signal list, you can adjust the width of the signal list by double-clicking on the border between the graph and the signal list.
   - To show the full content of a column in the signal list, you can adjust the width of the column by double-clicking on the splitter between the current column and the next column on the right side.

To show or hide columns

1. Use the tab **Instrument** in the Properties window.
2. In the **Signal List Columns**, click .
3. To show or hide a column, select or clear the checkbox of the respective column name in the list.

To reorder columns

1. Move a column by dragging its column header to the new position within the table header.

   A line between the columns appears to indicate the new position.
2. Release the mouse button.

To reorder signals in the signal list

1. Do one of the following:
   - To sort the signals in ascending order per column, click one of the column headers in the signal list. If you click the same column header again, the signals are sorted in descending order.
   - To move one or more signals to a specific position in the signal list, drag and drop the selected signals to this position. The new position is indicated by a blue line. When ordering the signals manually, the sorting mechanism gets inactive.

To know how you can copy the signal name and other meta information to other applications, see "Reusing the Signal Name in Other Applications" on page 136.

To change the number of decimals

1. In the signal list, select one or more signals (except enumerations or Boolean signals).
2. In the toolbar, click one of the following icons:
   - To show more decimal places, click .
   - To show fewer decimal places, click .

   The number of decimals is adapted accordingly for all cursor values in the signal list. The cursor values are also adapted within the cursor's tooltips.

5.2.2.3 Zooming

To adapt the value or time axis ranges for the best representation of your data, MDA provides several zoom possibilities.

You can perform the following actions:

- "To define the zoom area" on the next page
- "To show the full time range" on the next page
- "To zoom to fit signals" on the next page

To define the zoom area

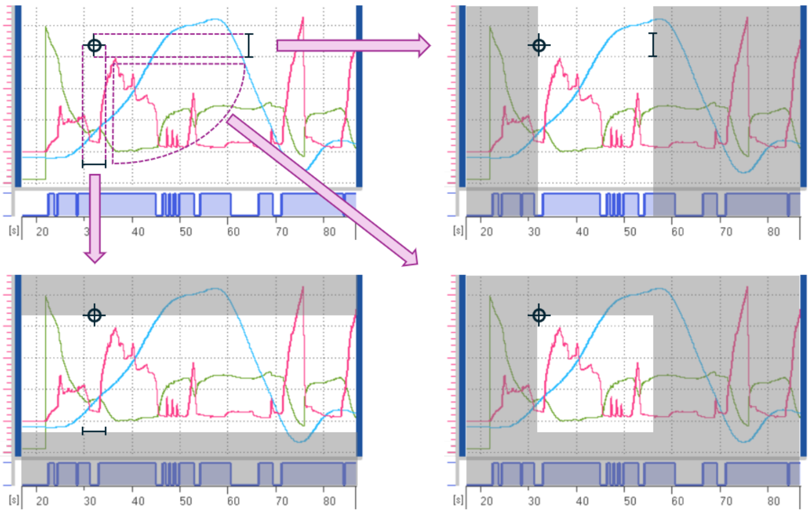
To see how you can zoom, synchronize, and scroll in instruments, watch our video **Navigating in Instruments**.

1. Press either CTRL and the left mouse button or just the right mouse button.
2. While keeping the button pressed, move the cursor along a value axis or along the time axis.

   Depending on the mouse movement, the zooming is performed as follows:

   - Time zoom (movement in horizontal corridor)
   - Value axis zoom (movement in vertical corridor)
   - Time zoom and value zoom (movement outside the corridors)

   The selected zoom area is highlighted.



   If you zoom in deeply, sample markers are displayed. To change the appearance of the sample markers, see "To define the appearance of the signal" on page 84.

To show the full time range

If the time range is not completely displayed in the oscilloscope, the time axes can be adapted to show the full time range.

1. Click .

   Alternatively, you can perform this operation by using the time slider. For more information, see "To show the full time range of the measure file" on page 122.

To zoom to fit signals

If a signal is not completely displayed in the oscilloscope strip, the value axis can be adapted to show the complete value range. Do one of the following:

1. To adapt selected signals, right-click the signals in the signal list or right-click the signal directly in the strip.
2. In the context menu, click **Zoom to fit Signal(s)**.

or

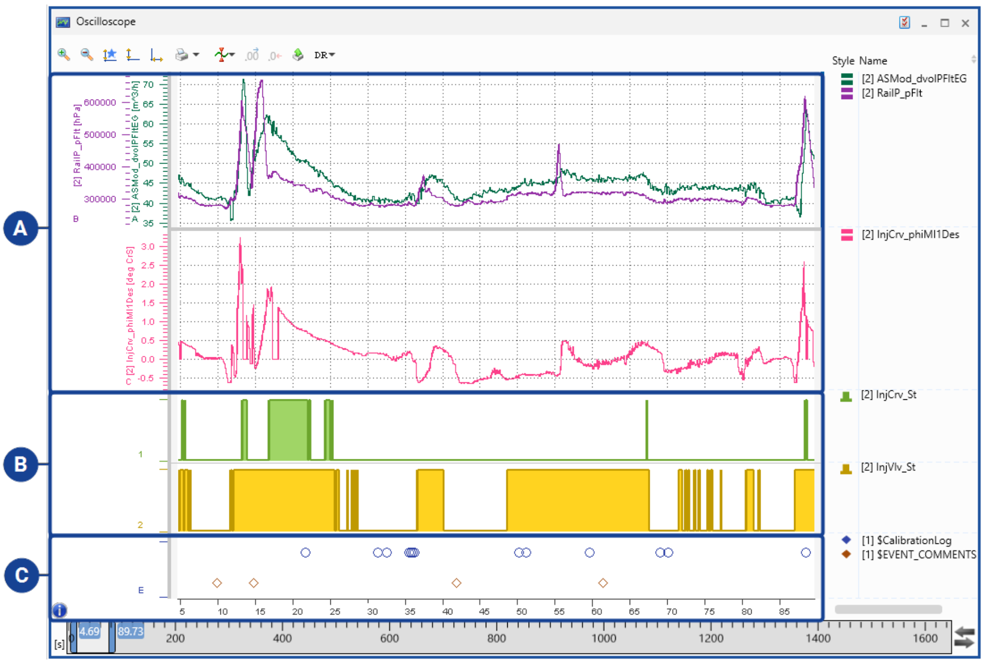
1. To adapt all signals of a selected strip or value axis specifically, right-click the strip or value axis.
2. In the context menu, click **Zoom to fit Signal(s)**.

or

1. To adapt all signals in all strips, click .

5.2.2.4 Using Strips

The signals in the oscilloscope can be distributed onto analog and Boolean strips for a better overview. After the signal selection, an analog strip, the required number of Boolean strips, and optionally one Event strip are added in an oscilloscope corresponding to the selected signals.



No.	Signal Type	Position	No. of Strips	Height	Axes	Signal Configuration
A	Analog	at the top	any	customizible per strip	scalable	color, line width, marker, connection type
B	Boolean	in the middle	any	customizible but for all Booleans the same	fix	color, marker, connection only steps
C	Event	at the bottom	one	customizible	none	color, marker, connection only samples

To see how you can use strips and define the signals list settings of the oscilloscope, watch our video **Oscilloscope - Defining Strips and Signal List**.

You can perform the following actions:

- "To add another strip" below
- "To reorder strips" below
- "To move analog signals to a common or individual strip" on the next page
- "To delete a strip" on the next page

To add another strip

1. Open the context menu of an existing strip.
2. Select **Add Strip**.

   Depending from the strip the action was triggered, an analog or a Boolean strip is added. The new strip is displayed below the current strip.

   This is not the case for the Event strip, as only one Event strip is available for all events.

   When you add signals to the oscilloscope, the selection wheel appears, which allows to create a new strip for the signals. For more information, see "To assign signals using the selection wheel" on page 75.

To reorder strips

1. Click the strip.

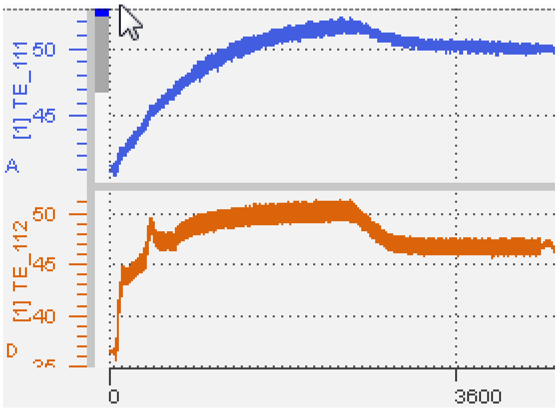
   A blue bar occurs on the left and right edge of the strip.
2. To move the selected strip, do one of the following:
   - To move the strip up, press ALT+PAGE UP.

     To move the strip down, press ALT+PAGE DOWN.

   or

- Click one of those bars and pull it up or down to the desired position.

  The new position of the floating strip is indicated by a small blue rectangle.



This action is not possible for the Event strip, as there is only one event strip always located at the bottom of the strip area.

To move analog signals to a common or individual strip

To use the selection wheel for this operation, see "To assign signals using the selection wheel" on the next page.

Alternatively, use the context menu as follows:

1. To merge signals to one strip, right-click the desired signals from different strips.
2. Click **Move to New Strip**.

or

1. To move signals to separate strips, right-click the desired signals from one strip or different strips.
2. Click **Move to Individual Strip(s)**.

To delete a strip

1. Open the context menu of the strip that you want to delete.
2. Select **Delete Strip**.

Note that a strip is also deleted when the last signal of a strip is deleted.

5.2.2.5 Using Axes

You can perform the following actions:

– "To assign signals using the selection wheel" on the next page
– "To use common axes" on the next page
– "To use individual axes" on page 76
– "To scroll the value range of axes" on page 76
– "To scroll the time axis" on page 76
– "To zoom the value range of axes" on page 76
– "To adjust the axis range manually" on page 76

- "To predefine an axis range for a signal" on page 77
- "To set the default behavior for signal-to-axis assignment" on page 78
- "To change the axis name" on page 78
- "To define the representation of figures at the value axis" on page 77
- "To delete axes" on page 79

To assign signals using the selection wheel

To assign signals to an oscilloscope, you can use the selection wheel. It appears when signals are assigned to an oscilloscope or moved within an oscilloscope via drag and drop.

1. Drag the signals into the analog graphical area of the oscilloscope or to the desired position in the signal list. After a short delay the selection wheel appears.
2. Within the selection wheel, drop the signals onto one of the options:

	One common axis for all signals Assign all signals to one axis.
	Shared axis per unit Assign signals to an axis having the same unit.
	Individual axes per signal Assign each signal to an individual axis.
	Take-over axis assignment Take over axis assignment of source oscilloscope. (Option is only provided if the signals derive from an existing oscilloscope.)
	New strip Create a new strip below the current strip. The selection wheel expands to the above described options, and the signals will be assigned to a new strip using the chosen option.

Note that an axis can be shared only for matching data types and the same data representation. 'No-match' signals get an own axis, as the original unit is not known.

Boolean and Event signals will be assigned to the corresponding strips independently from the drop position.

To use common axes

To see how you can modify in an oscilloscope signal and axes settings, watch our video **Oscilloscope - Settings for Signal and Axes**.

As long as the data type of signals fits, several signals can share the same value axis. To use common axes:

1. While pressing the CTRL or SHIFT key, select your desired signals in the signal list.
2. Right-click your selection and click **Use common axes**.

   Alternatively, drag and drop the desired signals on one axis. Note that the signal is only added to this axis if it has same data representation.

To use individual axes

If several signals share the same axis, the signals can be assigned to individual axes again.

1. Do one of the following:
   - Right-click the shared axis.
   - Right-click the desired signal(s) in the signal list.
2. Click **Use Individual Axes**.

To scroll the value range of axes

1. Bring the focus to the axis that you want to scroll.
2. Do one of the following:
   - Use the mouse wheel or the left mouse button to scroll the measurement scale up and down.
   - Alternatively, click the strip and drag the measurement scale of all axes up and down.
   - Use the keyboard and press ARROW UP or ARROW DOWN to scroll the measurement scale up and down.

To scroll the time axis

To quickly navigate to a specific time stamp, you can use the time slider. For more information, see "Time Navigation and Synchronization" on page 117. Alternatively, you can scroll by using the keyboard:

1. To move left, press the PAGE UP key.
2. To move right, press the PAGE DOWN key.
3. To navigate to the start of the time range, press the HOME key.
4. To navigate to the end of the time range, press the END key.

To zoom the value range of axes

1. Hover the cursor over the axis that you want to zoom.
2. Do one of the following:
   - Press CTRL and use the mouse wheel to zoom out or in.
   - Press CTRL and the left mouse button. To zoom out or in, move the cursor up or down.
   - Use the keyboard and press CTRL+ARROW UP or CTRL+ARROW DOWN to zoom in or out.

To adjust the axis range manually

To see how you can modify in an oscilloscope signal and axes settings, watch our video **Oscilloscope - Settings for Signal and Axes**.

In analog strips, you can define the value range. In Boolean strips, the value range of the axes is fixed (from 0 - 1) and cannot be changed.

To adjust the axis range as needed, do the following:

1. Click the tab **Axes** in the Properties window.
2. To define the minimum and the maximum value of an axis, do one of the following:
   - For analog signals, enter the desired values in the input fields.
   - For discrete signals, select the values from the drop-down menu.

   Note that if you have changed the data representation of a signal into the hexadecimal or implementation values, the minimum and maximum values are limited by the data type. If you enter a value below the minimum or above the maximum, the value is automatically set to the minimum or maximum value of the data type.

To define the axis range so that the samples are all in the visible range, use the **Zoom to fit** function.

To predefine an axis range for a signal

You can predefine an axis range for signals displayed in an analog strip. A predefined axis range is named "favorite axis range" and allows you to call up the predefined setting by clicking . Additionally, the favorite axis range is used when a signal is assigned newly to an oscilloscope or scatter plot instrument as default axis range.

To predefine the axis range, do the following:

1. To set the values as favorite axis range, click the star in the **Set as Favorite** column.

   If the values are shown in bold and blue, the current axis range matches to the favorite axis range already.

   To change the favorite axis range, define another axis range and click again the star in the **Set as Favorite** column. You cannot delete a once defined favorite axis range. You can only overwrite it with new values.

   For shared axes the minimum and maximum range is calculated based on the minimum and maximum values of the individual signals.

   The favorite axis range cannot be set if you have changed the data representation of a signal into the hexadecimal or decimal values.
2. To apply the favorite axis range, do one of the following:
   - To apply the favorite axis range to all axes, click .
   - To apply the favorite axis range to a selected axis, strip, or signal, right-click this item and choose **Apply Favorite Axis Range** from the context menu.

To define the representation of figures at the value axis

The setting of the **Scale mode** offers three different representations of the axis values.

It can be defined in the **Scale mode** drop-down menu.

Scale Mode	Description
Normal	Each value displayed on the axis is always written out as a normal number, e.g., 12,345,678 or 0.000001.
Scientific	Each value is always displayed using exponential notation, e.g., 8 E+2 (instead of 800) or 1 E-1 (instead of 0.1).
Automatic	If the upper axis value is larger than 10,000,000, a scientific representation will be shown, and also if the range is only from 0 to 0.0001. For other value ranges normal representation is done.

To set the default behavior for signal-to-axis assignment

To reduce the configuration effort, you can define the default behavior for signal-to-axis assignments by using the Properties window.

1. Click the tab **Instrument** in the Properties window.
2. Select one of the following entries from the **Signals Distribution** menu:
   - **Per unit one axis**
   - **One common axis**
   - **Individual axes**

   The setting is applied when adding signals with the INSERT key or in the Configuration Manager. It is used for newly added signals from the Variable Explorer or from other instruments, given the signal can be assigned to a shared axis.

   In case a signal is added by drag and drop, the selection wheel appears. For more information, see "To assign signals using the selection wheel" on page 75.

   Note that an axis can be shared only for matching data types and the same data representation. 'No-match' signals get an own axis, as the original unit is not known.

   If the signals are dropped from another oscilloscope, the axis assignment from the source oscilloscope remains unchanged.

To change the axis name

1. Do one of the following:
   - In the instrument, click .

   or

   - Select the instrument and press ALT+ENTER.

   or

   - In the instrument, right-click and select **Properties** in the context menu.

2. Select the **Axes** tab.
3. In the column **Set Name**, select the checkbox to enter the new axis name.

   The new name is automatically applied when you leave the cell.

To delete axes

1. Right-click a value axis.
2. Click **Delete Axis**.

➪ The axis is deleted and the signals are deleted from the view.

5.2.2.6 Basic Navigation

To navigate within the oscilloscope, you can perform the following actions:

- "To navigate from axis to strip to signal list" below
- "To navigate between the strips" below
- "To navigate between the axes" below
- "To navigate between the signals in the signal list" below
- "To navigate to the time slider" below

For more information about the use of the keyboard, see "Operating MDA via Keyboard" on page 17.

To navigate from axis to strip to signal list

To quickly navigate horizontally within the oscilloscope use the TAB key and backwards press SHIFT+TAB.

To navigate between the strips

1. Bring the focus into the graphical area.
2. To move upwards, press CTRL+PAGE UP.
3. To move downwards, press CTRL+PAGE DOWN.

To navigate between the axes

1. Bring the focus to the axis area.
2. To move to the right, press the ARROW RIGHT key.
3. To move to the left, press the ARROW LEFT key.

To navigate between the signals in the signal list

1. Bring the focus to the signal list.
2. To move upwards, press the ARROW UP key.
3. To move downwards, press the ARROW DOWN key.

To navigate to the time slider

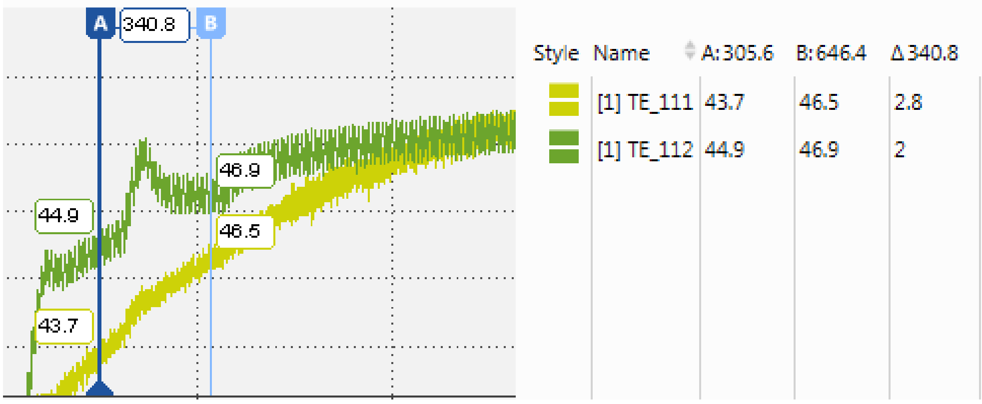
To navigate to the time slider and to enable entering a time value, press CTRL+B.

5.2.2.7 Using Cursors

Cursors are used to get more precise values at certain time stamps. Moreover cursors enable to determine differences between samples values and in view of time. The oscilloscope shows directly in the graph the values of the signals at the time stamp of the cursors as well as the difference on the y-axis between the cursors.

For performance reasons, V8.7 initially shows a signal value based on the minimum value of the file index. As long as index data is used, a rounding symbol appears in front of the tooltips and in the cursor's column in the signal list. The rounding symbol automatically disappears as soon as the cursor is not moved anymore, then the exact values are loaded in the oscilloscope.

All cursor and sample values are displayed as separate columns in the signal list. Also, the value differences of the cursors are displayed. To avoid a change in size of the signal list when adding and removing cursors, the signal list size has to be adapted manually by moving the splitter bar between oscilloscope area and signal list.



You can perform the following actions:

- "To show and hide cursors" on the next page
- "To delete cursors" on the next page
- "To move a cursor sample-wise" on the next page
- "To anchor a cursor to a position in the visible range" on page 82
- "To switch the synchronization cursor" on page 82
- "To hide and unhide the cursor's tooltips" on page 82
- "To copy the signal values at cursor's position" on page 82
- "To change the number of decimals of the cursor's tooltips" on page 83
- "To send the cursor time to EHANDBOOK-NAVIGATOR" on page 83
- "To move the cursor to a specific value" on page 83

To see how you can define the cursor's behavior, watch our video **Oscilloscope - Using Cursors**.

To show and hide cursors

1. To show cursor A, select **Show/Hide Cursors** in the cursors' drop-down menu .
2. To show cursor B, repeat step 1.

   The active cursor is indicated by the dark blue background color of the cursor label.
3. If you want to use the other cursor as active cursor, press Ctrl+1.
4. Click the icon again. This has the following effect:
   - If both cursors have been in the visible area, they are hidden now.
   - If one of the cursors has been outside the visible area, it is displayed now.

To delete cursors

1. To delete a specific cursor, select it and use delete in the context menu.
2. To delete all cursors, even if not in the visible range, use Ctrl+Alt+R.
3. **Note**

   If instrument synchronization is active, always one cursor must remain.

To move a cursor sample-wise

By default, the cursors' movement mode is time-based. You can change the movement mode to navigate a cursor from sample to sample. The selected mode is valid for all cursors in the oscilloscope.

1. Select **Switch to Sample-wise Navigation** in the cursors' drop-down menu or use the tab **Instrument** in the Properties window.

   The cursor label changes into a round shape A.

   Depending on the location of the cursor, the cursor's line is shown as follows:
   - If the cursor is exactly on an existing sample, the cursor's line is solid.
   - If the cursor is in-between two samples, the cursor's line is dashed.
2. To move a cursor to another sample, hover over the blue line and drag it to the new position. Alternatively, press Arrow Left or Arrow Right (the shortcut keys apply to the active cursor).

   The cursor jumps to the closest sample of the current time stamp. For navigation, only samples of the currently active signals are considered. If no signal is selected in the signal list, all samples from all signals are taken into account.

   If you move the cursor to a position where no sample is available, the tooltip next to the cursor label is marked in red indicating that the cursor cannot be moved in this direction any further.

To anchor a cursor to a position in the visible range

By default, cursors have a fixed time stamp and a variable screen position. When scrolling or zooming, the cursor moves with the signal curve. This means that the cursor can move outside the visible range. To keep the cursor in the visible range, you can anchor it.

1. Hover over the cursor label at the top of the cursor (**A** or **B**).

   The character changes into an anchor symbol .

2. Click the anchor symbol.

   The cursor is fixed to the screen position. The cursor label shows the anchor symbol permanently.

3. If you want to switch back to the non-anchored mode, click the cursor label again.

To switch the synchronization cursor

When synchronizing instruments, a synchronization cursor automatically appears in the oscilloscope. By default, the active cursor within the visible range is used as synchronization cursor. For more information, see "Synchronization time stamp" on page 119. To see how you can zoom, synchronize, and scroll in instruments, watch our video **Navigating in Instruments**.

The synchronization cursor is marked by a synchronization symbol at the bottom of the cursor line .

To switch the synchronization cursor, click the line of the other cursor. Alternatively, you can press CTRL+1.

The cursor becomes active and is used as synchronization cursor. Thus, the time stamp that is used for indicating the synchronization changes in all other instruments. For example, in other oscilloscopes the synchronization cursor is also switched accordingly.

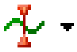
If you have selected the sample-wise navigation mode, the instrument in which you have performed the last action provides this sample information. In an oscilloscope, this is indicated by the dark blue background color of the cursor label. All other synchronized oscilloscopes are shown with transparent cursor labels (A).

To hide and unhide the cursor's tooltips

1. Click or use the tab **Instrument** in the Properties window.
2. To hide the tooltips, select **Hide Cursor's Values Tooltips**.
3. To unhide them again, click and select the respective entry.

To copy the signal values at cursor's position

To copy the signal values from a specific point in time to the clipboard, do the following:

1. Show the cursor by selecting **Show/Hide Cursors** in the cursors' drop-down menu  .
2. Move the cursor to the position of interest.
3. Right-click the desired signal in the signal list and select **Copy contents**.

   The content of the visible columns and columns header are copied in the clipboard.

To change the number of decimals of the cursor's tooltips

The number of decimals of the cursor tooltip is defined by the number of decimals for the respective signal. For more information, see "To change the number of decimals" on page 70.

To send the cursor time to EHANDBOOK-NAVIGATOR

If V8.7 is connected to EHANDBOOK-NAVIGATOR, the cursor time is automatically sent to EHANDBOOK-NAVIGATOR each time you change the cursor’s position. For more information on how to establish a connection to EHANDBOOK-NAVIGATOR, see "Connecting MDA to EHANDBOOK-NAVIGATOR" on page 30.

To move the cursor to a specific value

The time position for a cursor can be entered directly after a mouse click or using CTRL+SHIFT+B. As a result the cursor will move to the entered point in time. To access the time position field of the other cursor use again CTRL+SHIFT+B or **TAB** as long as the time position field is in edit mode.

5.2.2.8 Adjusting Signals

In the **Style** column, the type of each signal is shown.

	Icon for discrete signals
	Icon for continuous signals
	Icon for Boolean signals
	Icon for Event signals

You can perform the following actions:

- "To define the appearance of the signal" on the next page
- "To show or hide a signal curve" on the next page
- "To treat a signal as a Boolean or Analog signal" on page 85
- "To return to the default behavior for strip selection of a signal" on page 85
- "To change the data representation of a signal" on page 86
- "To delete a signal from the oscilloscope" on page 86

To define the appearance of the signal

To see how you can modify in an oscilloscope signal and axes settings, watch our video **Oscilloscope - Settings for Signal and Axes**.

1. Within the **Style** column, click the colored square of the corresponding signal. You can multi-select several signals and adapt their settings at the same time. If a specific signal cannot take over the selected setting, this setting remains unchanged.

   A pop-up for the signal settings opens.
2. Do one of the following:
   - You can select a color from the default colors. To define and select another color, click **More Colors**.

     To know how you can assign a color to a specific file, see "Defining one Color per File" on page 46.
   - You can change the appearance of the symbol by which the individual samples of the signal are represented. Select the preferred sample marker from the **Marker Symbol** drop-down menu. This setting is only visible after zooming in when the marker symbols are displayed.
   - For analog signals, you can adapt the line width. Click the drop-down menu and choose from five different sizes.
   - For analog signals, you can define the connection line between the samples. The calculation of the signal is not affected by this graphical setting. In the **Sample Connection** drop-down menu, you can choose a direct connection line between the samples, a step-wise connection line, or no visible connection line. This last setting is only visible after zooming in when the marker symbols are displayed.
   - You can change the data representation of the signal via the **Data Representation** drop-down menu. For more information, see "To change the data representation of a signal" on page 86.
3. Click outside the pop-up to hide it again.

To show or hide a signal curve

To see how you can modify in an oscilloscope signal and axes settings, watch our video **Oscilloscope - Settings for Signal and Axes**.

1. To hide the signal curve of one or more signals, select the signals in the graph or in the signal list.
2. In the context menu, select **Show/Hide Signals Curve**.

   The signal curve is hidden. The name of the signal is still displayed in the signal list.
3. To show the signal curve again, repeat step 1 and 2.

To show or hide all signal curves of a measure file

To quickly achieve a clearer representation in an oscilloscope, all signals from the same measure file can be hidden.

1. To hide all signal curves of one measure file, select the signals in the graph or in the signal list.
2. In the context menu, select **Show/Hide All Signal Curves of Measure File**.

   The signal curves are hidden.
3. To show the signal curves again, repeat step 1 and 2.

This procedure has the advantage that also for hidden signals the cursor columns in the signal list will still show the signals' values.

To show a signal curve on top

1. Select the desired signal in the signal list.
2. In the context menu, select **Enable/Disable Signal shown on top**.

   The signal's curve will stand out as long as no signal is actively selected in the strip. The signal name will appear in a bolder font in the signal list. Per strip only one signal can have this property.

To treat a signal as a Boolean or Analog signal

To see how you can use strips and define the signals list settings of the oscilloscope, watch our video **Oscilloscope - Defining Strips and Signal List**.

1. In the signal list, select one or more signals in an analog strip.
2. In the context menu, select **Treat As Boolean/Analog Signal**.

   Each signal is moved into a new Boolean strip. The area between 0 and the signal curve is filled automatically to enable a better visibility. The icon in the column **Style** remains the same as before.

   By the same context menu entry, a signal in a Boolean strip can be moved to an analog strip.

To return to the default behavior for strip selection of a signal

It can happen that you assigned a signal by accident to a wrong strip. For example by selecting an Event signal and using the option **Treat As Boolean/Analog Signal**. To return to the default strip selection when adding the signal to an oscilloscope again, proceed as follows:

1. Select the signal in the signal list of the oscilloscope
2. Shift the signal per drag & drop into an anolog strip

or

1. Select the option “Treat as boolean/analog signal” in the signal context menu.
2. Remove the signal from the analog strip

These steps cause that MDA deletes its wrong default strip setting for this signal. The next time the signal is added again to an oscilloscope, MDA adds it to the default strip defined by the data type of the signal.

To change the data representation of a signal

1. In the signal list, select the signals for which you want to change the data representation.
2. In the toolbar, click DR▾.
3. Select one of the following entries from the **Data Representation** drop-down menu:
   - **Physical Values**
   - **Hexadecimal (memory) Values**
   - **Decimal (RAW) Values**
   - Logarithmic view (Physical Values)

     Only value axes for physical values can be set to a logarithmic scale. The time axis remains always with the equidistant scale. Enumerations also cannot be assigned to a logarithmic scale.

   All selected signal values including cursor and axis values are shown in the selected data representation. The unit information is adapted accordingly. If the signal has been on a common axis before, it is moved to an individual axis with the respective data representation.

   For any kind of signals having float data type, a dialog appears. Select one of the following bits for the data representation:
   - 8 bit
   - 16 bit
   - 32 bit
   - Do not cast

     In this case, the shown hexadecimal or binary representation is for the Float value according to the IEEE-754 standard.

To delete a signal from the oscilloscope

1. In the signal list, select one or more signals.
2. In the context menu, select **Remove Signal(s)**.

5.2.2.9 Creating a Bound in a Strip

1. Define a new calculated signal.
2. Enter a value in the **Formula Definition** field.
3. Define in the section **Output Options** a fixed rate with any value, for performance reasons preferably slow (e. g. 10 sec).
4. Drag and drop the calculated signal on the same axis as the signal for which the bound shall be used.

5.2.3 Scatter Plot

In the scatter plot, one signal is displayed on the x-axis and another signal is displayed on the y-axis. If you assign multiple signals to the y-axis, each signal is displayed in its individual strip.

The toolbar provides you with the following functions:

	Adjustment of axes
	Copy, Save or Print
	Cursor options
	Bound options
	Export measure data functionality

The complete set of configuration possibilities for this instrument is available in the **Properties** docking window. There, the tooltips provide a detailed description of the properties and their possible options.

1. Do one of the following:
   - In the instrument, click .

   or

   - Select the instrument and press ALT+ENTER.

   or

   - In the instrument, right-click and select **Properties** in the context menu.

In the following, only the more advanced functions are explained in further detail.

For more information about synchronizing, scrolling, and zooming, see "Time Navigation and Synchronization" on page 117.

5.2.3.1 Adjusting Signals

You can perform the following actions:

To delete a signal

1. Do one of the following:
   - If only one strip exists and you want to delete its signal, right-click the y-axis.
   - If you want to delete the signal of the x-axis, right-click the x-axis.
2. Select **Remove Signal**.

5.2.3.2 Zooming

To see how you can use the possibilities of the Scatter Plot instrument, watch our video **Using the Scatter Plot**.

You can perform the following actions:

- "To define the zoom area" below
- "To show the complete value range" below

To define the zoom area

1. Press either CTRL and the left mouse button or just the right mouse button.
2. While keeping the button pressed, move the cursor horizontally or vertically.

⇨ The selected zoom area is highlighted.

To show the complete value range

If you perform the following steps, only the signal values are considered that are part of the currently visible time range in the time slider. If all samples of the signal from the whole measure file shall be considered, you must additionally adapt the time range. For more information, see "To show the full time range of the measure file" on page 122.

1. If you want to show the complete value range for all strips, click .

or

1. If you want to show the complete value range of a specific signal, right-click its axis.
2. Select **Zoom to Complete Value Range**.

or

1. If you want to show the complete value range of both signals in a strip, right-click its axis.
2. Select **Zoom Both Signals to Complete Value Range**.

5.2.3.3 Using Strips

You can perform the following actions:

- "To reorder strips" below
- "To delete a strip" on the next page

To reorder strips

1. Click the strip.

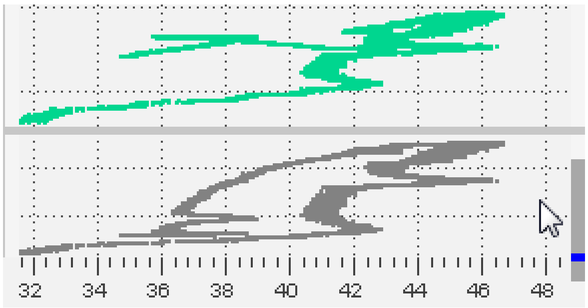
   A blue bar occurs on the left and right edge of the strip.
2. To move the selected strip, do one of the following:
   - To move the strip up, press ALT+PAGE UP.

     To move the strip down, press ALT+PAGE DOWN.

   or

- Click one of those bars and pull it up or down to the desired position.

  The new position of the floating strip is indicated by a small blue rectangle.



To delete a strip

1. Open the context menu of the strip that you want to delete.
2. Select **Delete strip**.

5.2.3.4 Using Axes

You can perform the following actions:

- "To use a y-axis as x-axis" below
- "To scroll the value range of axes" below
- "To zoom the value range of axes" below
- "To adjust the axis range or set a favorite axis range" on the next page
- "To define a figure representation at a value axis" on the next page
- "To change the axis name" on the next page
- "To assign multiple signals to the same y-axis" on page 91

To use a y-axis as x-axis

1. Right-click the y-axis that you want to use as x-axis.
2. Select **Use as X-axis**.

⇨ The x-axis and y-axis including their ranges are swapped.

To scroll the value range of axes

1. Hover the cursor over the axis that you want to scroll.
2. Use the mouse wheel or the left mouse button to scroll the measurement scale up and down.

   Alternatively, click the strip and drag the measurement scale of all axes up and down.

To zoom the value range of axes

1. Hover the cursor over the axis that you want to zoom.
2. Do one of the following:
   - Press CTRL and use the mouse wheel to zoom out or in.
   - Press CTRL and the left mouse button. To zoom out or in, move the cursor up or down.

To adjust the axis range or set a favorite axis range

1. Click the tab **Axes** in the Properties window.
2. To define the minimum and the maximum value of an axis, do one of the following:
   - For analog signals, enter the desired values in the input fields.
   - For discrete signals, select the values from the drop-down menu.
3. To set the values as favorite axis range, click the star in the **Set as Favorite** column. When the values are shown in bold and blue, the current axis range matches to the favorite axis range already.
4. To apply the favorite axis range, do one of the following:
   - To apply the favorite axis range to all axes, click .
   - To apply the favorite axis range to a selected axis or strip, right-click this item and choose **Apply Favorite Axis Range**.

To define a figure representation at a value axis

The setting of the **Scale mode** offers three different representations of the axis values.

It can be defined in the **Scale mode** drop-down menu.

Scale Mode	Description
Normal	Each value displayed on the axis is always written out as a normal number, e.g., 12,345,678 or 0.000001.
Scientific	Each value is always displayed using exponential notation, e.g., 8 E+2 (instead of 800) or 1 E-1 (instead of 0.1).
Automatic	If the upper axis value is larger than 10,000,000, a scientific representation will be shown, and also if the range is only from 0 to 0.0001. For other value ranges normal representation is done.

To change the axis name

1. Do one of the following:
   - In the instrument, click .

   or

   - Select the instrument and press ALT+ENTER.

   or

   - In the instrument, right-click and select **Properties** in the context menu.
2. Select the **Axes** tab.
3. In the column **Use Custom Label**, select the checkbox to enter the new axis name.

   The new name is automatically applied when you leave the cell.

To assign multiple signals to the same y-axis

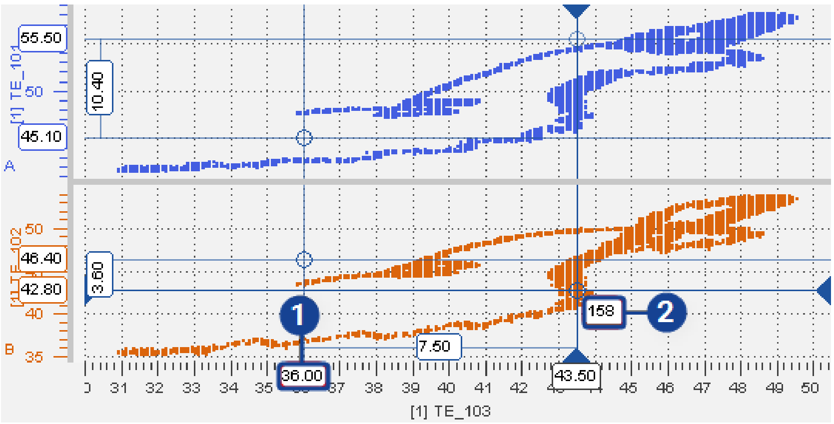
1. Select the signal(s) you want to add to a scatter plot instrument.
2. Drag and drop the selected signal(s) to the y-axis of the Scatter Plot.

➪ The signal(s) will be added to the same strip if these can share the axis used as drop target. If the signals cannot be added to a shared axis a new strip will be created (e.g. for enumeration signals). New strips will also be created if the signals are dropped into the graphical area of the scatter plot.

The color for the samples of a signal can be defined by selecting the desired color for the signal in an oscilloscope, and afterwards adding the signal to the scatter plot.

5.2.3.5 Using Cursors

In the scatter plot, two values of the signal are marked by cursors. The cross-hair indicates the exact position of each cursor. If several strips exist, two horizontal lines per strip are displayed.



No.	Description
1	Tooltip at the value axis displaying the signal value
2	Tooltip at the cross-hair of the cursors displaying the time stamp

To see how you can use the possibilities of the Scatter Plot instrument, watch our video **Using the Scatter Plot**.

To show and hide cursors

1. To show the cursors, click .

   Alternatively, press CTRL+R.

   As in the oscilloscope first one cursor is created, another click will create a second cursor.

2. If you want to move a cursor to another sample, hover on one of the cross-hairs and drag it to the new position.

Alternatively, press the following keys:

- To switch between the cursors, press Alt+Arrow Left/ Alt+Arrow Right.
- To change the cursor position, press Arrow Left/ Arrow Right/ Arrow Up/ Arrow Down.

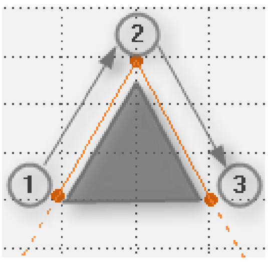
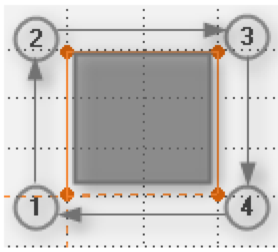
3. When clicking the cursor icon again, this has the following effect:
   - If both cursors have been in the visible area, they are hidden now.
   - If one of the cursors has been outside the visible area, it is displayed now.

> **Note**
>
> If instrument synchronisation is active, always one cursor must remain.

If the instrument synchronization is active, also the cursors will be synchronized. The currently active instrument defines how many cursors will be shown. A cursor movement in the active instrument causes a cursor movement in the synchronized instruments. However, a cursor in the scatter plot will move only, if for the time defined by the cursor position another sample is available in the scatter plot.

5.2.3.6 Using Bounds

You can use a bound to define a region in the scatter plot. A bound can either be an extrapolated line or closed to a polygon depending on how you set the endpoints of the bound.

Extrapolated bound:	Closed bound:
	

You can perform the following actions:

- "To add a bound" below
- "To close or extrapolate a bound" on the next page
- "To delete a single bound" on the next page
- "To delete all bounds" on page 94

To add a bound

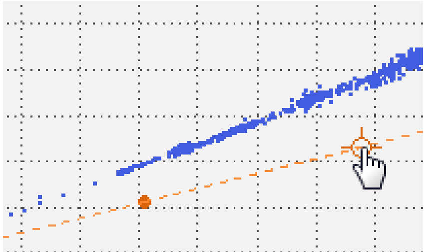
To add a bound, you must have assigned signals to the scatter plot first.

1. Click .
2. Select **Add Bound** from the drop-down menu.

   A hand symbol with a cross-hair appears.
3. Move the cross-hair to the first endpoint of the bound and click.

   The endpoint is fixed.
4. Move the cross-hair to the next endpoint.

   As a preview, a connection line between the fixed and the current endpoint is shown.

   
5. If the position is correct, click.

   The endpoint and the connection line are fixed.
6. Repeat the steps 4 and 5 until you have defined all endpoints.

   Note that you cannot create a bound with intersected lines.
7. To correct the last fixed endpoint, press BACKSPACE and move the cross-hair to a new endpoint.
8. To delete the complete preview with all fixed endpoints, press ESC.
9. To complete the bound, do one of the following:
   - Press RETURN.

     The ends of the bound are extrapolated to infinity. If these extrapolated lines are intersected, the bound is automatically closed to a polygon.
   - Add the last endpoint onto the coordinate of the first endpoint.

     The bound is closed to a polygon.

To close or extrapolate a bound

1. To change a closed bound into a extrapolated bound or vice versa, right-click the connection line of the bound.
2. Do one of the following:
   - For a closed bound, select **Extrapolate Bound**.
   - For an extrapolated bound, select **Close Bound**.

To delete a single bound

1. Click the connection line of the bound.
2. Click .
3. Select **Delete Bound** from the drop-down menu.

   Alternatively, press DEL.

To delete all bounds

1. Click .
2. Select **Delete All Bounds** from the drop-down menu.

5.2.4 Table

The table can display different kinds of signals including enumeration (VTab), event, and string signals.

The table shows enumeration signals as follows: If a string value is available, it is displayed. If the enumeration signal has a default value, the table displays the default value for all numerical values for which there is no associated string value. If the enumeration signal has no default value and if there is no string value defined for a numerical value, "n/a" is displayed.

In an MDF measure file, single samples can be marked with an "invalid" flag. In the table instrument, these invalid samples are marked by a red exclamation mark.

You can perform the following actions:

– "To define the table properties" below
– "To scroll the visible time range" on the next page
– "To navigate to the start or end of the time range" on the next page
– "To move the synchronization time stamp" on the next page
– "To show or hide header rows" on the next page
– "To reorder columns" on the next page
– "To fill empty cells" on page 96
– "To change the number of decimals" on page 96
– "To change the data representation of a signal" on page 96
– "To delete signals" on page 96
– "To filter data rows" on page 97
– "Table" above

To see how you can use the possibilities of the Table instrument, watch our video **Using the Table**.

To define the table properties

The complete set of configuration possibilities for this instrument is available in the **Properties** docking window. There, the tooltips provide a detailed description of the properties and their possible options.

1. Do one of the following:
   - In the instrument, click .

   or

   - Select the instrument and press ALT+ENTER.

or

- In the instrument, right-click and select **Properties** in the context menu.

To scroll the visible time range

To quickly navigate to a specific time stamp, you can use the time slider. For more information, see "Time Navigation and Synchronization" on page 117. Alternatively, you can scroll by using the keyboard:

1. To move upwards, press the PAGE UP or CURSOR UP key.
2. To move downwards, press the PAGE DOWN or CURSOR DOWN key.

To navigate to the start or end of the time range

1. To navigate to the start of the time range, press the HOME key.
2. To navigate to the end of the time range, press the END key.

To move the synchronization time stamp

In synchronized mode, the synchronization time stamp is highlighted in the second row in blue. If the synchronization time stamp is located between the first two rows, a blue line is displayed between these rows. For more information, see "Synchronizing Instruments" on page 118.

1. To move the synchronization time stamp, double-click a row.

or

2. Use the keyboard:
   - To select a row above the current synchronization time stamp, press ALT+CURSOR UP.
   - To select a row below the current synchronization time stamp, press ALT+CURSOR DOWN.

To show or hide header rows

To distinguish signals that have the same name but are coming from different devices or different rasters, you can display the device, raster, or unit.

1. In the toolbar, click  or use the tab **Instrument** in the Properties window.

   A list of the default header rows appears.
2. To show or hide a header row, select or clear the checkbox of the respective name in the list.

To reorder columns

1. Move a column by dragging its column header to the new position within the table header.

   A line between the columns appears together with highlighted areas on the left and on the right of the line.
2. Select the left or right highlighted area to place the column in front of or behind this line.
3. Release the mouse button.

To fill empty cells

1. If the value of a signal is not available, you can display its interpolated value. In the table toolbar, click .

   The empty cells are filled with the last available sample (so-called "step-mode" or "constant interpolation"). These values are shown in gray italic font.
2. To undo this action, click the icon again.

To change the number of decimals

1. Mark the columns for which you want to change the number of decimal places.
2. In the toolbar, click one of the following icons:
   - To show more decimal places, click .00.
   - To show fewer decimal places, click .0←.

   The decimals for the time values can be adapted in the same manner.

To change the data representation of a signal

1. In the table, select one or more signals.
2. In the toolbar, click DR▾.
3. Select one of the following entries from the **Data Representation** drop-down menu:
   - **Physical Values**
   - **Hexadecimal (memory) Values**
   - **Binary (memory) Values**
   - **Decimal (RAW) Values**

   All selected signal values are shown in the selected data representation. The unit information is adapted according to the selected data representation.

   Note that if the device name of the signal ends with **#MeasureCal**, you must additionally select which data type shall be used to display the values. Select one of the following:
   - 8 bit
   - 16 bit
   - 32 bit
   - Do not cast

     In this case, the shown hexadecimal or binary representation for the Float value is according to the IEEE-754 standard.

To delete signals

1. Select one or more signals. Do one of the following:
   - In the instrument, mark one or more columns.
   - In the Configuration Manager, mark one or more signals.

2. In the context menu, select **Remove**.

<u>To filter data rows</u>

To reduce the shown amount of data to what is relevant for you, you can define a column filter per column. Only those rows (i.e. time stamps) for which the defined column filter conditions are fulfilled will be listed.

1. Click the funnel icon of the column.
2. Select from the conditions offered.

   Only one condition can be set per column.
3. Confirm the defined condition.
4. If needed, repeat the steps for other columns. Multiple column filters will be logically AND combined.

   Merely the rows for which all filter conditions are met will be displayed.
5. To filter enumeration signals, do the following:
   i. Create a second column with the enumeration signal.
   ii. Switch the data representation of the column to decimal.
   iii. Create the filter definition as defined above.

   You can then see the data rows with the respective verbal values of the enumeration signal.

Column filters can be removed individually in the filter definition window, or per instrument using the icon.

5.2.5 Statistical Data

This window displays the statistical properties of numeric signals or enumeration signals, such as the average, minimum, maximum, standard deviation and median value. If you add a signal with a different format, an error icon is displayed. The statistical values are calculated based on the time range that is selected in the time slider. If the time range changes, the statistical values are recalculated automatically. In an MDF V4.x measure file, single samples can be marked with an "invalid" flag. In the statistics instrument, an exclamation mark indicates whether in the selected time range an invalid sample exists. Invalid samples are excluded from the statistical calculations. If the assigned signal changes, the content also updates automatically. For example, the signal changes in the following cases:

- The assigned signal is a calculated signal and you modify the calculation formula. For more information, see "Defining Calculated Signals" on page 173.
- You replace the measure file that contains the signal or other signals that are used for calculating the assigned signal. For more information, see "To replace a measure file" on page 44.
- You define a time offset. For more information, see "Defining a Time Offset for a Measure File" on page 47.

You can perform the following actions:

- "To scroll the list of assigned signals" below
- "To reorder columns" below
- "To reorder signals in the signal list" below
- "To show or hide columns" below
- "To increase and decrease the number of decimals" on the next page
- "To copy signal name and other meta information" on the next page
- "To delete signals" on the next page

To scroll the list of assigned signals

To quickly navigate to a specific signal, you can use the scrollbar on the right side of the instrument. Alternatively, you can scroll by using the keyboard:

1. To move upwards, press the CURSOR UP key.
2. To move downwards, press the CURSOR DOWN key.

To reorder columns

1. Move a column by dragging its column header to the new position within the table header.

   A line between the columns appears together with highlighted areas on the left and on the right of the line.
2. Select the left or right highlighted area to place the column in front of or behind this line.
3. Release the mouse button.

To reorder signals in the signal list

1. To move one or more signals to a specific position in the signal list, drag and drop the selected signals to this position.

The new position is indicated by a blue line.

To show or hide columns

Each statistical function is shown in a separate column. You can select which statistical functions shall be displayed.

To distinguish signals that have the same name but are coming from different devices or different rasters, you can display the device, raster, or unit column.

1. In the toolbar, click  or use the tab **Instrument** in the Properties window.

   A list of all columns appears.
2. To show or hide a column, select or clear the checkbox of the respective column name in the list.

To increase and decrease the number of decimals

1. Mark the columns for which you want to change the number of decimal places.
2. In the toolbar, click one of the following icons:
   - To show more decimal places, click .
   - To show fewer decimal places, click .

To copy signal name and other meta information

To know how you can copy the signal name and other meta information to other applications, see "Reusing the Signal Name in Other Applications" on page 136.

To delete signals

1. Select one or more signals. Do one of the following:
   - In the instrument, mark one or more rows.
   - In the Configuration Manager, mark one or more signals.
2. In the context menu, select **Remove**.

5.2.6 Histogram

The histogram allows to graphically display the results of a simple classification of the samples of one signal as vertical bars. For the classification the numeric value of the sample is used. Therefore only numeric scalar data types and Status String Ref signals are supported. The samples to be classified are defined by the time range set in the histogram's time slider bar. A change of the time range automatically triggers a recalculation.
The result of the classification is represented by the height of the vertical bar of each bucket. Additionally a figure above the bar indicates the number of samples within the bucket.

The tool tip of a bar lists the absolute number of samples, the relative share of its samples to all samples for the defined time range, the signal name and the file name.

You can perform the following actions:

– "To define the Histogram properties" below
– "To define the number of buckets and their value ranges" on the next page
– "To scroll or zoom the time range" on the next page
– "To delete the assigned signal" on the next page
– "To replace the assigned signal" on the next page

To define the Histogram properties

The complete set of configuration possibilities for this instrument is available in the **Properties** docking window. There, the tooltips provide a detailed description of the properties and their possible options.

1. Do one of the following:
   - In the instrument, click .

   or
   - Select the instrument and press ALT+ENTER.

   or
   - In the instrument, right-click and select **Properties** in the context menu.

To define the number of buckets and their value ranges

1. Open the instrument Properties window as described above.
2. In the **Number of Buckets** field, enter the desired number of classes for the histogram.
3. In the **Interval Size** field, enter the value for the width used for each bucket.
4. In the **Center of First Bucket** field, define the value which shall be used as center of the first bucket.

⇨ All changes are applied immediately.

The number of buckets and the interval size define the overall value range for the histogram.

If there are samples with a value outside the defined overall value range, optionally additional buckets on the left and right of the defined bucket will be shown. If there are samples that have a value that cannot be classified into the defined buckets, a new bucket for uncountable samples ("NaN") will be shown automatically. Examples are samples with an invalid flag, not a number (NaN), infinity (+INF, -INF) and for Status String Ref signals samples with a verbal value.

To scroll or zoom the time range

This can be done by using the time slider bar.

For more information about synchronizing, scrolling, and zooming, see "Time Navigation and Synchronization" on page 117.

To delete the assigned signal

Open the Configuration Manager, mark the signal assigned to the histogram, and delete it via the context menu.

To replace the assigned signal

Just add the desired signal to the instrument by means of drag & drop or press the INSERT key.

5.2.7 Event List

The Event List shows all value changes of the assigned signals. Events can be shown for Boolean signals as well as for numeric, enumerations, and string signals. String signals are, for example, user comments during recording. For

Boolean and numeric signals a rising respective falling edge icon is shown depending on an increasing respective decreasing physical value. For string and enumeration signals, only string values with an event icon are shown.

If the assigned signal changes, the content updates automatically. For example, the signal changes in the following cases:

- The assigned signal is a calculated signal and you modify the calculation formula. For more information, see "Defining Calculated Signals" on page 173.
- You replace the measure file that contains the signal or other signals that are used for calculating the assigned signal. For more information, see "To replace a measure file" on page 44.
- You define a time offset. For more information, see "Defining a Time Off-set for a Measure File" on page 47.

To see how you can use the Event List to navigate quickly between specific events and display these in an oscilloscope, watch our video **Finding Events**.

You can perform the following actions:

- "To define the Event List properties" below
- "To scroll the visible time range" below
- "To navigate to the start or end of the time range" on the next page
- "To move the synchronization time stamp" on the next page
- "To show or hide header rows" on the next page
- "To reorder columns" on the next page
- "To change the decimals of the time column" on the next page
- "To delete a signal" on page 103
- "To filter a signal/ signals" on page 103

To define the Event List properties

The complete set of configuration possibilities for this instrument is available in the **Properties** docking window. There, the tooltips provide a detailed description of the properties and their possible options.

1. Do one of the following:
   - In the instrument, click .

   or

   - Select the instrument and press ALT+ENTER.

   or

   - In the instrument, right-click and select **Properties** in the context menu.

To scroll the visible time range

To quickly navigate to a specific time stamp, you can use the time slider. For more information, see "Time Navigation and Synchronization" on page 117. Alternatively, you can scroll by using the keyboard:

1. To move upwards, press the PAGE UP or CURSOR UP key.
2. To move downwards, press the PAGE DOWN or CURSOR DOWN key.

To navigate to the start or end of the time range

1. To navigate to the start of the time range, press the HOME key.
2. To navigate to the end of the time range, press the END key.

To move the synchronization time stamp

In synchronized mode, the synchronization time stamp is highlighted in the second row in blue. If the synchronization time stamp is located between the first two rows, a blue line is displayed between these rows. For more information, see "Synchronizing Instruments" on page 118.

1. To move the synchronization time stamp, double-click a row.

   or

2. Use the keyboard:
   i. To select a row above the current synchronization time stamp, press ALT+CURSOR UP.
   ii. To select a row below the current synchronization time stamp, press ALT+CURSOR DOWN.

To show or hide header rows

To distinguish signals that have the same name but are coming from different devices or different rasters, you can display the device, raster, or unit.

1. In the toolbar, click or use the tab **Instrument** in the Properties Window.

   A list of the default header rows appears.

2. To show or hide a header row, select or clear the checkbox of the respective name in the list.

To reorder columns

1. Move a column by dragging its column header to the new position within the table header.

   A line between the columns appears to indicate the new position.

2. Release the mouse button.

To change the decimals of the time column

1. Mark the time column for which you want to change the number of decimal places.
2. In the toolbar, click one of the following icons:
   - To show more decimal places, click .
   - To show fewer decimal places, click .

<u>To delete a signal</u>

1. Right-click the signal name in the column header of the event list.
2. Select **Remove Signal**.

   Alternatively, select the signal name and press DEL.

<u>To filter a signal/ signals</u>

From the amount of information, you can filter out the data that is relevant to you. Conditions can be defined that the values must then fulfill, the rows with the time stamps are thinned out and only those which fulfill these conditions are displayed.

1. Click the column.
2. Select from the conditions offered.

   Only one condition can be set per column.
3. Click **Clear all** or **Clear** to re-set all or merely one filter in a specific column.

> **Note**
>
> This only works out on rows that have a numerical value. You can display the signal again and filter it by numerical value. If there are several filters, a logical link with "and" is feasible. Merely the time rows for which both conditions are met remain.
>
> Signals including textual information cannot directly be filtered so far. No selection list is provided.

When filtering, the table always uses the interpolated mode. This means that cells for which the relevant signal has not provided a value are filled with the last available value.

5.2.8 GPS Map

The GPS map view displays GPS tracks in a map that are constituted of the measure signals latitude and longitude. It is very useful if you want to put geographic data providing street and terrain information in relation to other measured signals. Thus, any abnormal behavior in the engine module under test can be analyzed in a better way if the GPS data like longitude and latitude are considered during the offline analysis.

Additionally, you can add an indication signal, for example the velocity, either as a Boolean signal or as an analog signal. In case of a Boolean signal, the color of the track changes depending on the state of the Boolean signal. In case of an analog signal, the track is displayed with a color gradient.

MDA uses a license-free open-source street map.

To see how you can create a GPS Map instrument, use the zoom and scroll functions, and synchronize the GPS Map with other instruments, watch our video **Using the GPS Map**.

Since V8.7, if the GPS Map instrument displays only a blue field instead of the expected map and a warning error appears, the download is blocked by the Windows Defender Firewall. To load the required map content, the following releases in the customer firewall are needed and should generally be unblocked by the customer IT department:

- Release of port 443 (over HTTPS)
- URL: maps.omniscale.net.

You can perform the following actions:

- "To define the GPS Map properties" below
- "To add a signal" below
- "GPS Map" on the previous page
- "To define the color of a track based on an indication signal" on the next page
- "To add an Event signal" on the next page
- "To zoom" on the next page
- "To zoom to show the complete track" on page 106
- "To select a specific time range of the track" on page 106
- "To show cursors" on page 106
- "To delete a signal" on page 106

For more information about synchronizing instruments, see "Synchronizing Instruments" on page 118.

To define the GPS Map properties

The complete set of configuration possibilities for this instrument is available in the **Properties** docking window. There, the tooltips provide a detailed description of the properties and their possible options.

1. Do one of the following:
   - In the instrument, click .

   or

   - Select the instrument and press ALT+ENTER.

   or

   - In the instrument, right-click and select **Properties** in the context menu.

To add a signal

1. Select a latitude and a longitude signal from the Variable Explorer or the Configuration Manager.
2. Drag and drop the signals onto a layer or layer tab of the configuration and select the GPS Map instrument. For more information, see "To assign signals to a new instrument" on page 132.

   The map view appears and the track is displayed.

V8.7 tries to assign automatically the signals to the correct coordinates by checking for "long" respective "lat" in the signal names. If the automatic signal assignment is incorrect, switch their assignment by clicking on .

To define the color of a track

1. Click the color icon of the respective track.
2. Select the desired color.

To know how you can assign a color to a specific file, see "Defining one Color per File" on page 46.

To define the color of a track based on an indication signal

To see sections of the track which fulfill a specific condition, you can color the track based on an indication signal.

The color of the indication signal will overrule the track color and a potential color definition for the measurement file.

1. Create a track in a GPS map instrument as described under "To add a signal" on the previous page.
2. Select a third signal and add it to the entry in the track list of the GPS map instrument.
3. Depending on the type of signal, the color coding can be defined in the track list of the GPS map instrument.

   Two distinct colors can be defined. This leads to a change of color or to a color gradient of the track for Boolean or analog signals, respectively.

   Note

   Only Boolean and analog signals can be used as indication signals, but not for example enumeration signals.

To add an Event signal

When adding event signals to a GPS map, the location where an event occurred is indicated by .

If several tracks are displayed, for a unique identification the color of the icon has the same color as the track. The color of the flash inside the icon represents the event type (e.g. pause event, comment event, calibration activity, etc.).

To zoom

Do one of the following:

1. To zoom in, click .
2. To zoom out, click .

or

1. Use the mouse wheel to zoom in or out.

To zoom to show the complete track

1. Click [icon], and the map will be zoomed that the track is completely visible.

To select a specific time range of the track

1. To see which track was driven during a specific time range, the Time Slider is used. For more information, see "Time Navigation and Synchronization" on page 117.

   When only a sub-time range is selected, this track is shown in dark blue, while the track outside this time range is indicated in light blue.

To show cursors

In synchronized mode, cursors are shown as these are given by the master instrument. Cursors can be moved directly in the GPS Map instrument or follow the cursor's movement in the master instrument.

To delete a signal

- Open the Configuration Manager and delete the respective signal from the GPS Map view.

  or

- In the GPS Map instrument, right-click the signal and select **Remove Track(s)**.

  or

- In the GPS Map instrument, click the signal and press the **Del** key.

5.2.9 Video

In the Video instrument, the video signal `VIDEO_TIMECODE` contains only a series of time stamps. The device name of the video signal is used as link to the actual video file on the hard disk named <measurefile>_<device_name>.mp4.

Note

The video file names must not be renamed as they are referenced by the corresponding MDF file (`*.mf4`).

The time stamps in the `VIDEO_TIMECODE` are used for synchronization.

You can perform the following actions:

- "To define the Video properties" below
- "To display a video signal" on the next page
- "To scroll and synchronize" on the next page

To define the Video properties

The complete set of configuration possibilities for this instrument is available in the **Properties** docking window. There, the tooltips provide a detailed description of the properties and their possible options.

1. Do one of the following:
   - In the instrument, click ☑.

   or

   - Select the instrument and press ALT+ENTER.

   or

   - In the instrument, right-click and select **Properties** in the context menu.

To display a video signal

1. In the Variable Explorer, select the signal `VIDEO_TIMECODE` from the corresponding device which refers to the camera name.
2. Drag the video signal onto a layer or layer tab of the configuration and select the Video instrument.

   If a signal is already assigned, the new video signal replaces the old one. It is not possible to assign multiple signals. For more information, see "Assigning Signals to Instruments" on page 132.
3. To start the video, click **Play** and to stop it, click **Pause**.

To scroll and synchronize

To quickly navigate to a specific time stamp in the video, or to synchronize the Video instrument with other instruments, see "Time Navigation and Synchronization" on page 117.

5.2.10 Bar Chart Instruments

For evaluating and comparing several similar signals for one point in time, such as the cell voltages of an electric vehicle battery system.

The display can be customized to the specific physical quantities (such as voltages, temperatures, pressures, etc.) using the settings in the instrument properties.

Using the bar chart instruments, you have the following options:

- To get a graphical overview of all signal values of the individual signals. The overview can be based on the absolute signal values (Absolute Value Bar Chart) or the deviations from the mean (Delta Bar Chart).
- A sortable tabular overview (sortable list) so that you can easily identify the signals with the highest or lowest values or deviations.
- A classification of the signals into groups with comparable signal values (Signal Distribution List).
- User-definable additional signals
  - Per drag and drop other relevant signals can be added to the summary area. In the Properties window the displayed name, the decimals and the unit can be adapted. Additionally, added signals can be removed from the summary area in the Configuration Manager.

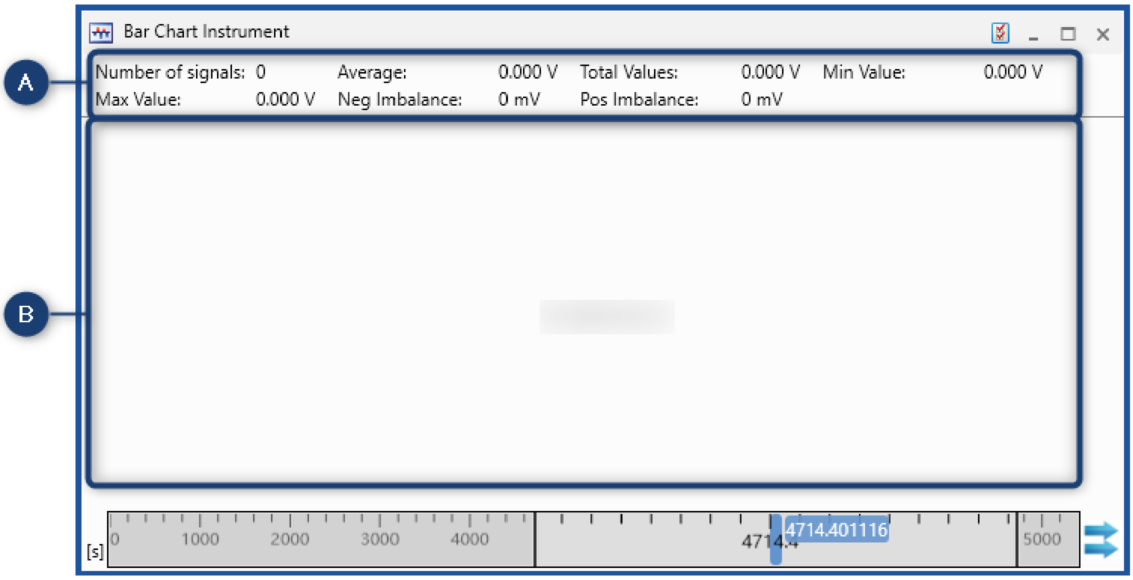
All bar chart instruments always show values for one specific point in time.

All bar chart instruments support instrument synchronization possibilities of MDA.

Note

A signal can only be assigned once to each bar chart instruments.

The bar chart instruments contain the following:



No.	Description
A	Summary Area
B	Specific instrument view

Summary Area

In the Properties window you can customize the summary area and define and define the following items.

By default, the Summary Area A shows the following information:

Number of Signals

Displays the number of signals assigned to the instrument.

Average

Displays the average value of all signals assigned to the instrument.

Total

Displays the total value of all signals assigned to the instrument.

Minimum

Displays the minimum value of all signals assigned to the instrument.

Maximum

Displays the maximum value of all signals assigned to the instrument.

Negative Deviation

Displays the highest negative deviation from the average value of all signals assigned to the instrument.

Positive Deviation

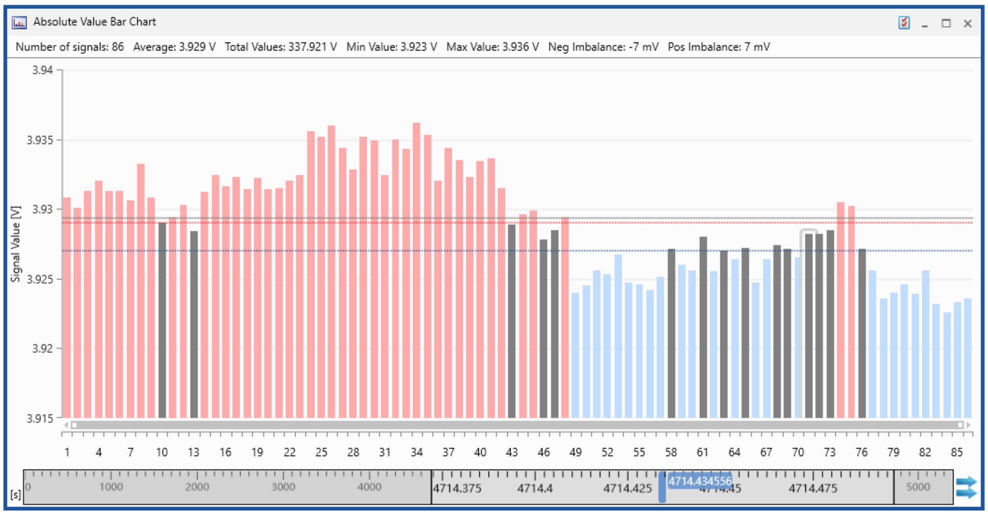
Displays the highest positive deviation from the average value of all signals assigned to the instrument.

In the Properties window you can customize the summary area and define which of the entries listed above you want to see and, if desired, rename it accordingly.

- Additional signals can be assigned to the summary area of a bar chart instrument via the function "Drag and drop".
- The displayed name of the new entry can be defined in the instrument properties.
- Both recorded as well as calculated signals can be assigned.
- The entry can be moved in the Configuration Manager.

5.2.10.1 Absolute Value Bar Chart

The Absolute Value Bar Chart allows you to get an overview of the signal values and and to identify the signals which exceed a lower or upper limit at the given point in time.
Each signal value is represented by a vertical bar. The signals are sorted alpha-betically by the signal names.



Signal values

The graph shows the signal values of all signals as vertical bars. The height of each bar represents the value of the specific signal. Signals with a value exceeding the lower or upper limit are highlighted.

Bar color	Description
	Signals exceeding the lower limit
	Signals within the limits
	Signals exceeding the upper limit

Limits

Upper and lower limits are shown as horizontal lines.

Line color	Description
	Upper limit
	Lower limit

You can perform the following actions:

- "To define the Absolute Value Bar Chart properties" below
- "To assign signals" below
- "To replace signals" below
- "To delete a signal" on the next page

To define the Absolute Value Bar Chart properties

The complete set of configuration possibilities for this instrument is available in the **Properties** docking window. There, the tooltips provide a detailed description of the properties and their possible options.

1. Do one of the following:
   - In the instrument, click .

   or

   - Select the instrument and press ALT+ENTER.

   or

   - In the instrument, right-click and select **Properties** in the context menu.

To assign signals

To see how you can select the relevant signals and assign them to the desired instrument, watch our video **Selecting Signals**.

To know how you can add signals to a new or existing instrument, see "Assigning Signals to Instruments" on page 132.

Note: Each signal can be added only once.

To replace signals

For more information, see "To replace a signal" on page 134.

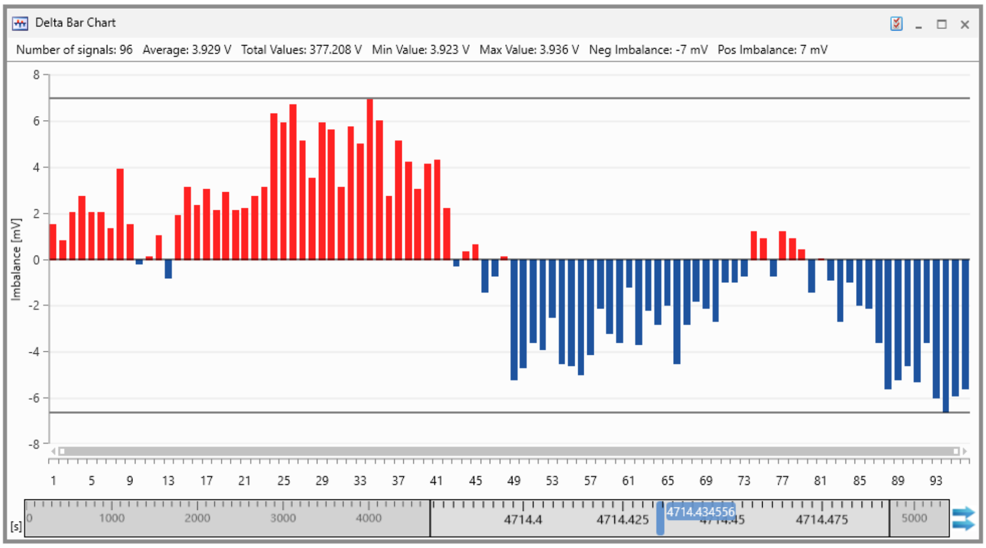
To delete a signal

1. Right-click the row with the signal that you want to delete.
2. Select **Remove Signal(s)**.

⇨ Only one signal can be deleted in this way. To remove multiple signals open the Configuration Manager, multi-select the undesired signals and delete these.

5.2.10.2 Delta Bar Chart

The Delta Bar Chart instrument provides a quick overview of the deviation from the average value of many signals in parallel. The deviation of each signal is shown as vertical bar wherein the zero line represents the average of all signal values for the defined point in time. Additional auxiliary lines for the minimum and maximum deviation are displayed to facilitate the overview.



You can perform the following actions:

– "To define the Delta Bar Chart properties" below
– "To assign signals" on the next page
– "To zoom the view" on the next page
– "To show bar details" on the next page
– "To delete a signal" on the next page

To define the Delta Bar Chart properties

The complete set of configuration possibilities for this instrument is available in the **Properties** docking window. There, the tooltips provide a detailed description of the properties and their possible options.

1. Do one of the following:
   - In the instrument, click ☑.

   or

   - Select the instrument and press ALT+ENTER.

or

- In the instrument, right-click and select **Properties** in the context menu.

To assign signals

To see how you can select the relevant signals and assign them to the desired instrument, watch our video **Selecting Signals**.

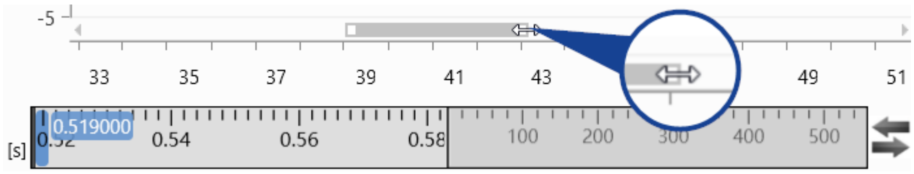
To know how you can add signals to a new or existing instrument, see "Assigning Signals to Instruments" on page 132.

Note: Each signal can be added only once.

To zoom the view

To enlarge the view for some adjacent imbalance columns, do the following:

1. Grab the left or right end of the scrollbar below the imbalance columns and drag the end towards the centre of the scrollbar.



2. In zoomed mode, you can scroll the bar to view other adjacent cells.
3. Double-click the end of the scroll bar to return it to the maximum position.

To show bar details

To view meta information for a particular column, simply move the mouse over it. The tooltip shows more details such as the position ID, the imbalance value, the signal name, and its measure file.

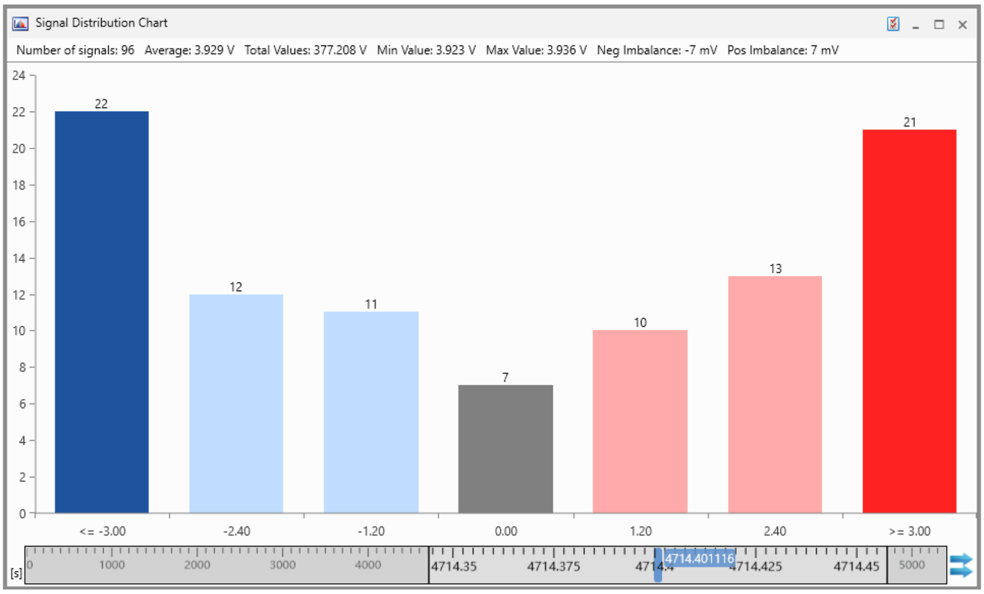
To delete a signal

1. Right-click the signal bar, if you want to delete a signal.
2. Select **Remove Signal(s)**.

➪ Only one signal can be deleted in this way. To remove multiple signals open the Configuration Manager, multi-select the undesired signals and delete these.

5.2.10.3 Signal Distribution Chart

The Signal Distribution Chart is an instrument for statistical analysis. The chart is calculated based on the signal values at a given point in time. In the chart, the height of each bar represents the number of signals that fall within the defined value range of the respective bucket.



Bar color	Description
	Signals with a value below the lowest bucket
	Signals with a negative deviation from the average
	Signals with a value around the average
	Signals with a positive deviation from the average
	Signals with a value above the highest bucket

You can perform the following actions:

- "To define the Signal Distribution Chart properties" below
- "To assign signals" on the next page
- "To define the number of buckets and their value ranges" on the next page
- "To identify the signals of a bucket" on the next page
- "To move or copy signals from the Signal Distribution Chart to another Instrument" on the next page
- "To delete a signal" on page 115

To define the Signal Distribution Chart properties

The complete set of configuration possibilities for this instrument is available in the **Properties** docking window. There, the tooltips provide a detailed description of the properties and their possible options.

1. Do one of the following:
   - In the instrument, click .

   or
   - Select the instrument and press ALT+ENTER.

   or
   - In the instrument, right-click and select **Properties** in the context menu.

To assign signals

To see how you can select the relevant signals and assign them to the desired instrument, watch our video **Selecting Signals**.

To know how you can add signals to a new or existing instrument, see "Assigning Signals to Instruments" on page 132.

Note: Each signal can be added only once.

To define the number of buckets and their value ranges

1. Open the instrument Properties window as described above.
2. In the **Number of Buckets**, select the number of buckets from the drop-down menu. Only odd numbers are available.
3. In the **Interval Size [mV]** field, enter the value for the interval size.

⇨ All changes are applied immediately.

The central bucket is always aligned with the imbalance value 0. The number of buckets and the interval size define the overall value range for the histogram. The number of signals with lower or higher imbalance values are shown as additional columns in darker colors on the left and right.

To identify the signals of a bucket

Hover over the respective bucket. The tooltip displays the list of signals that fall within the value range of the bucket.

To move or copy signals from the Signal Distribution Chart to another Instrument

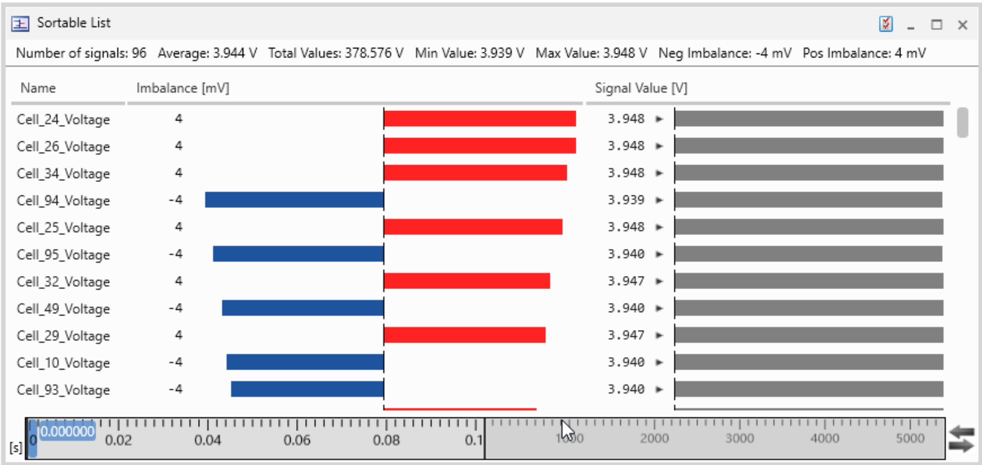
1. Select the bucket from which you want to move or copy the signals to another instrument.
2. Do one of the following:
   - "To move or copy signals from the Signal Distribution Chart to another Instrument" above another instrument.
   - To copy the signals press the CTRL key first before dropping the selected signals onto another instrument.

<u>To delete a signal</u>

1. Click the Configuration Manager window.
2. In the Signal Distribution Chart instrument select the signals that you want to delete.
3. Press **Del**.

5.2.10.4 Sortable List

The Sortable List allows to identify quickly the signals with the highest deviation from the average value or the highest resp. lowest absolute value.



Imbalance [mV]

Displays the imbalance of the signals. The imbalance bars displayed for each signal are colored blue or red.

Bar color	Description
	Negative deviation from the average value of all signals.
	Positive deviation from the average value of all signals.

Signal Value [Unit]

Displays the absolute values of the signals.

Trend indicator	Description
	The value of the subsequent sample is lower than the current sample.
	The value of the subsequent sample is the same as the current sample.
	The value of the subsequent sample is higher than the current sample.

You can perform the following actions:

- "Sortable List" on the previous page
- "To reorder columns" below
- "To show or hide columns" below
- "To assign signals" below
- "To replace signals" below
- "To sort signals" below
- "To delete a signal" on the next page

To define the Sortable List properties

The complete set of configuration possibilities for this instrument is available in the **Properties** docking window. There, the tooltips provide a detailed description of the properties and their possible options.

1. Do one of the following:
   - In the instrument, click .

   or

   - Select the instrument and press ALT+ENTER.

   or

   - In the instrument, right-click and select **Properties** in the context menu.

To reorder columns

1. Move a column by dragging its column header to the new position within the table header.
2. Release the mouse button.

To show or hide columns

1. Right-click a column.

   A list of the default columns appears.
2. To show or hide a column, select or clear the checkbox of the respective name in the list.

To assign signals

To see how you can select the relevant signals and assign them to the desired instrument, watch our video **Selecting Signals**.

To know how you can add signals to a new or existing instrument, see "Assigning Signals to Instruments" on page 132.

Note: Each signal can be added only once.

To replace signals

For more information, see "To replace a signal" on page 134.

To sort signals

If you click one of the column headers, you can sort the table contents by this column in increasing order. With another click on the same column header, the sorting order is inverted.

<u>To delete a signal</u>

1. Right-click the row with the signal that you want to delete.
2. Select **Remove Signal**.

5.2.11 Time Navigation and Synchronization

When having loaded signals and their data from a measure file, it is usually necessary to navigate to a specific time segment. This can easily be done by zooming and scrolling through the entire time range. To keep the overview even in case of many signals, it is often useful to distribute data to different instruments. But sometimes these data need to be monitored in parallel to identify root causes for unexpected observations or to document correlations. Then, a simple synchronization method for combined zooming and scrolling in different instruments is needed. All these zooming, scrolling, and synchronization activities can be done with the time slider in each instrument.

The time slider displays the complete time range of all measure files assigned to the current configuration. If you add a new measure file into the configuration, the minimum and maximum time sample of the time slider are automatically updated.

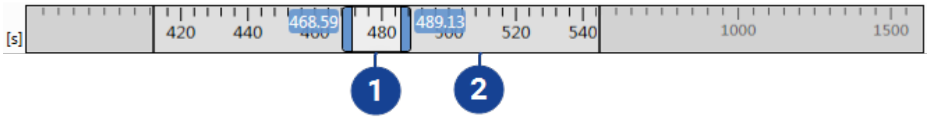
The time slider differs depending on the instrument type and its corresponding scale. The instrument types can be classified as follows:

Instrument Type	Variable Scale	Fixed Scale
Absolute Value Bar Chart	-	x
Delta Bar Chart	-	x
Event List	-	x
GPS Map	x	-
Histogram	x	-
Oscilloscope	x	-
Scatter Plot	x	-
Signal Distribution Chart	-	x
Sortable List	-	x
Statistical Data	x	-
Table	-	x
Video	-	x

Time slider in instruments with variable scale

The time slider can display any arbitrarily chosen section of the complete time range. Thus, the time slider can be used for zooming, scrolling, and synchronization. The exact start and end time of the displayed time range is shown in tooltips. The number of decimals shown in the tooltips depends on the zooming level. To change the values in the tooltips, see "Zooming with the Time Slider" on page 121.

To get a sufficient size and an accurate reporting of the relative position of the currently visible time range at a high amount of zoom, the time slider automatically switches into the magnifier mode.



1	Currently visible time range
2	Magnified section of the time slider (displayed after zooming in)

Time slider in instruments with fixed scale

If an instrument just display a fixed scale, the time slider supports only scrolling and synchronization. The missing zooming capability of the instrument is represented in the time slider by a blue line. This blue line marks the time stamp currently shown in the instrument. To enter an exact value for the time range, see "To enter an exact value for the time range" on page 121.

Possible actions depending on the instrument type

Time Slider Action	Variable Scale	Fixed Scale
"To synchronize instruments" on page 120	x	x
"To scroll the time range" on page 120	x	x
"To perform a fast scrolling" on page 121	x	x
"To zoom the visible time range" on page 122	x	-
"To perform a fast zooming" on page 122	x	-
"To show the full time range of the measure file" on page 122	x	-
"To enter an exact value for the time range" on page 121	x	x

5.2.11.1 Synchronizing Instruments

If several instruments exist in a configuration, you can synchronize them. Instruments can be distinguished into instruments with variable scale and fixed scale. For more information, see "Time Navigation and Synchronization" on the previous page.

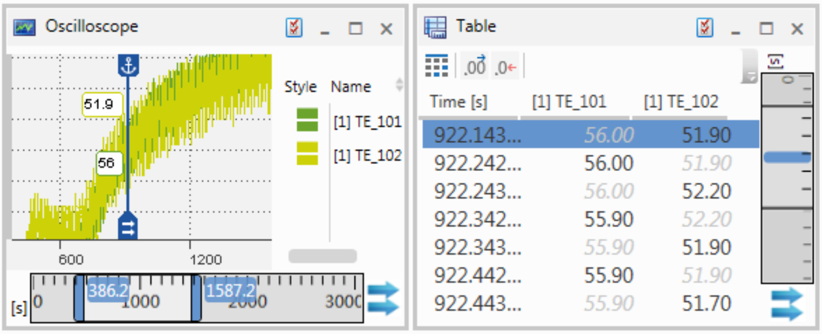
Synchronization of different instrument types

If you want to synchronize these two different types of instruments with each other, the following recommendation applies: Start the synchronization from an instrument with variable scale. Then, the zooming level of each synchronized instrument with variable scale is adapted to this first instrument.

If you start the synchronization from an instrument with fixed scale, the current time stamp is used. All cursors in instruments with variable scale are deleted and one new synchronization cursor is created. The zooming level of each synchronized instrument with variable scale remains the same as before.

Synchronization time stamp

A synchronization time stamp indicates at which time stamp an instrument is synchronized with other instruments.



In the oscilloscope, this time stamp is shown by a synchronization cursor that automatically appears when synchronizing instruments. The active cursor within the visible range is used as synchronization cursor. If no cursor exists or the cursors are outside the visible range, a new cursor is created. The synchronization cursor cannot be deleted. In case the synchronization cursor is anchored, it remains always in the visible range and helps you to see the same point in time in e.g. a table or event list and the oscilloscope. If the synchronization cursor is not anchored, it is moved with the time line during scrolling or zooming. This means, it can be outside the currently visible time range. But the time stamp of the synchronization cursor is still used to synchronize with other instruments. If you stop synchronizing, the synchronization cursor remains in the oscilloscope. For more information, see "To switch the synchronization cursor" on page 82.

In instruments with fixed scale, the synchronization time stamp is highlighted in the second row in blue. If the synchronization time stamp is located between the first two rows, a blue line is displayed between these rows. You can move the synchronization time stamp to another row in the table. For more information, see "To move the synchronization time stamp" on page 95. When you stop the synchronization, the color of the highlighted row or line changes from blue to gray to indicate the current time of the time slider within the measure data.

To synchronize instruments

1. Click .

   All instruments of the configuration are synchronized. Scrolling can be performed at any instrument in synchronized mode.
2. To stop the synchronization, click .

To Pause individual instruments

1. Click below the synchronization arrows.

   A paused state is indicated by a red pause icon . And the synchronization symbol is shown in grey. In paused state the instrument is excluded from any synchronization activities. This means any changes of the time range or the cursors positions have no effect, neither from the paused instrument to the synchronization group nor vice versa. The paused state of an instrument is kept independently from whether the synchronization is active or not.
2. To end the paused state, click again the pause icon .

5.2.11.2 Navigating with the Time Slider

You can perform the following actions:

- "To scroll the time range" below
- "To perform a fast scrolling" on the next page
- "To scroll slowly line by line" on the next page
- "To enter an exact value for the time range" on the next page

To scroll the time range

For scrolling, you can use the mouse wheel. Alternatively, you can do the following:

1. Hover the cursor over the time slider.
2. When the cursor changes into a hand symbol, drag this area to the desired position.

or

1. Click the scale (outside the currently visible time range for instruments with variable scale).
2. The time slider scrolls by one page.

or

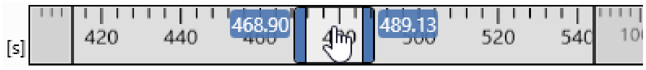
1. To move left, press the PAGE UP key. To move right, press the PAGE DOWN key.

   The time slider scrolls by one page.

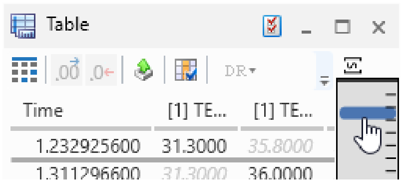
2. To navigate to the beginning of the time range, press the Home key.
3. To navigate to the end of the time range, press the End key.

To perform a fast scrolling

1. Do one of the following:
   - In instruments with variable scale (e.g. oscilloscope and scatter plot), the magnified section of the time range must be displayed (see "To zoom the visible time range" on the next page). Hover the cursor over the currently visible time range.

   - In instruments with fixed scale (e.g. table), hover over the current time stamp that is displayed as a blue line.

2. While keeping the mouse button pressed, move the cursor to the desired position.

   The faster you move the cursor, the faster is the scrolling performed.

To scroll slowly line by line

Use the keyboard and press the Arrow Up, Arrow Down, Arrow Left, or Arrow Right keys.

To enter an exact value for the time range

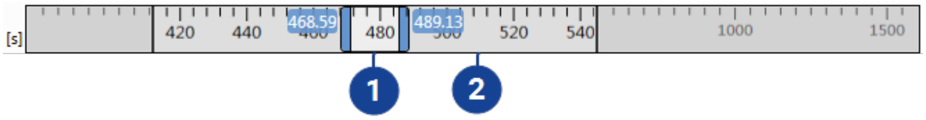
1. Click one of the tooltips in the time slider showing the start and end time of the displayed time range. Alternatively, press Ctrl+B.
2. Enter the value for the time range.
3. Do one of the following:
   - To apply the new value to the time range, press Enter.
   - To apply the new value to the time range and to directly jump to the other tooltip, press Tab or Ctrl+B.

   If the value is invalid, the edit box is marked with a red border. If you hover over the edit box, an error message is displayed.

5.2.11.3 Zooming with the Time Slider

You can perform the following actions in instruments with variable scale:

– "To zoom the visible time range" on the next page
– "To perform a fast zooming" on the next page
– "To show the full time range of the measure file" on the next page



(1)	Currently visible time range
(2)	Magnified section of the time slider (displayed after zooming in)

<u>To zoom the visible time range</u>

1. Move the cursor to the left or right edge of the currently visible time range (1).

   The cursor changes into a double arrow.
2. If you want to perform a symmetric zoom operation, press CTRL.
3. Enlarge or reduce the visible time range by dragging the double arrow.

   When you zoom in very deep, the complete time range cannot be displayed anymore. An additional area appears showing a magnified section of the time range (2).

<u>To perform a fast zooming</u>

1. The magnified section of the time range must be displayed (see "To zoom the visible time range" above). Move the cursor to the left or right edge of the currently visible time range (1).

   The cursor changes into a double arrow.
2. If you want to perform a symmetric zoom operation, press CTRL.
3. While keeping the mouse button pressed, move the cursor to the desired position.

   The faster you move the cursor, the faster is the zooming performed.

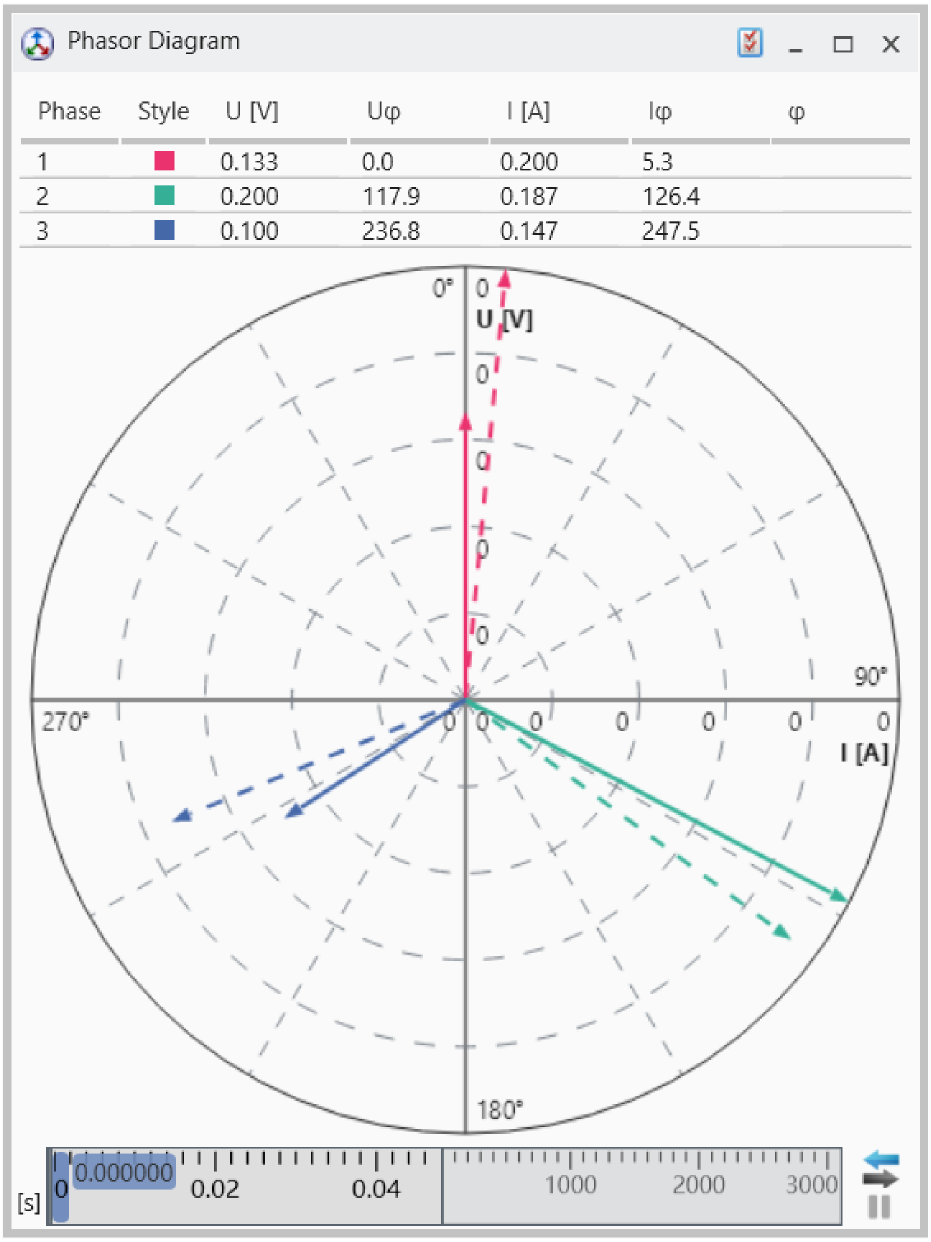
<u>To show the full time range of the measure file</u>

1. Move the cursor to the left or right edge of the currently visible time range (1).

   The cursor changes into a double arrow.
2. Double-click.

   The time range is expanded to the left or right boundary respectively.
3. If you press CTRL in advance, the time range is expanded for both boundaries simultaneously.

5.2.12 Phasor Diagram



A phasor diagram is used to visualize and analyze AC (alternating current) quantities such as voltage, current, and phase angle. It displays these quantities as phasors, i.e. rotating vectors that represent waveforms in a simplified, static form.

It helps to easily see the magnitudes of the assigned signals and the differences between the phases. The different phases are indicated in predefined colors such as red, green, and blue.

The voltage phasors are represented as solid lines, while the current phasors are shown by dashed lines. The length of a phasor represents the amplitude of the voltage or current signal. The angle between the voltage and the current phasor represents the phase shift between both quantities in each phase.

You can perform the following actions:

- "To add signals" below
- "To replace signals" below
- "To remove signals" below

To add signals

1. Drag and drop each of the input signals individually into the table above the diagram.
2. Ensure that the signals are assigned correctly in view of physical quantities and phases.

To replace signals

To replace an already assigned signal

1. drag and drop another one into the cell in the table.

To remove signals

Signals can only be removed via the Configuration Manager.

1. Expand the view for the respective phasor diagram to see the assigned signals.
2. Select the signal(s) to be deleted and remove these by means of the entry in the context menu.

6 Signal Selection

In the Variable Explorer, you can see the contents available in the measure files. Before a recording in INCA is done, variables are selected in a defined raster. By definition, a variable becomes a signal if a raster is assigned and latest if measure data is available. Thus in the context of MDA, only signals are available.

In a configuration, each signal is uniquely identified by its signal name in combination with multiple meta information. According to ASAM MDF V4 standard these are the ECU, the device, the recording raster, the ECU raster, and the device raster. In addition, MDA also uses the path and the name of the measure file.

The device information can be set by the user in INCA's Hardware Configuration (HWC) and is visible in several places in the Experiment Environment. The ECU information is taken from the A2L file. It is not visible in INCA but is added to the MDF V4 file as meta information for the signal. MDF V3.x files do not support ECU information and only one type of raster information. When loading an MDF V3 file or another file format that does not support all of the mentioned meta information, MDA will use the string “NULL” for the missing information. This allows to handle the missing meta information gracefully when replacing measure files.

6.1 Defining Display Name in the Application

V8.7 allows to define which of the following potential variable name shall be used as Display Name:

- **Name**
- **Display Identifier**
- **Symbol Link**

To select the Display Name, do the following:

1. In the Variable Explorer, click .
2. From the drop-down menu, select the Display Name.

   The selected name is shown in the **Display Name** column.

V8.7 ensures that the Display Name is unique, if necessary the name is extended by a number in round brackets. Additionally a warning icon is shown.

If the Display Identifier or the Symbol Link is not available, the Name is used as Display Name.

The Display Name is used application-wide, i.e. it is shown in instruments and in calculated signals. It is also used when exporting a measure file using a file format which only supports a single name field (e.g. ASCII).

Only the **Display Name** column is used for the search in the Variable Explorer.

6.2 Defining the View in the Variable Explorer

You can perform the following actions:

- "To freeze columns" below
- "To show or hide columns" below
- "To reorder columns" below

To freeze columns

1. To freeze columns, click .

   On first usage, the first two columns are in this frozen area. A vertical, gray line separates the frozen area in the table from the unfrozen area. Columns on the left side of the line remain visible when you use the horizontal scrollbar.
2. To enhance or reduce the frozen area, drag any of the columns into or from this area.
3. To unfreeze the complete table, click the icon again.

To show or hide columns

1. In the toolbar, click .

   A list of the default columns appears.
2. To show the complete list of columns, click the **More columns** drop-down menu.
3. To show or hide a column, select or clear the checkbox of the respective column name in the list.

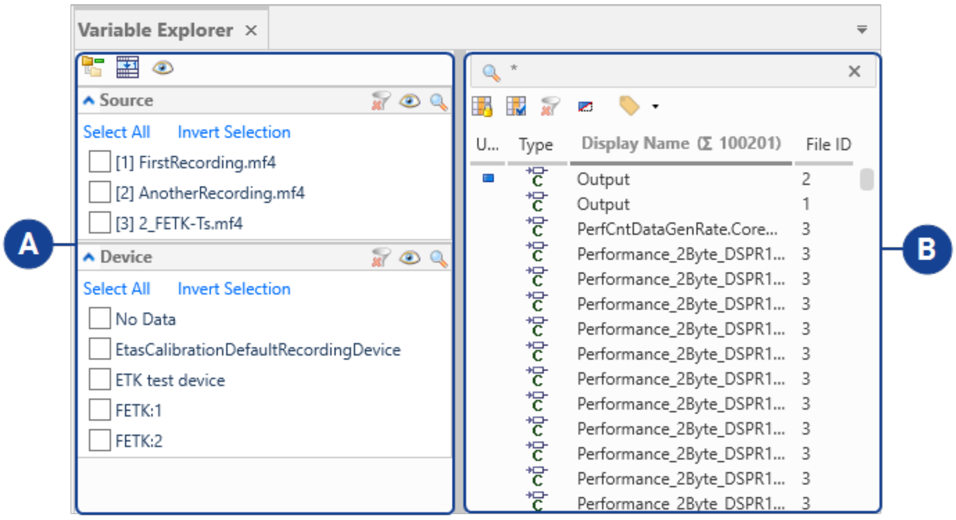
To reorder columns

1. Move a column by dragging its column header to the new position within the table header.

   A line between the columns appears to indicate the new position.
2. Release the mouse button.

6.3 Sorting and Filtering

The Variable Explorer is divided into two areas:



No.	Description
A	**Filters** Entries from several categories (e.g. Source, Device, etc.) can be selected. They define the variables shown in the variables list of the Variable Explorer. Open or close the area for the filters by clicking on the splitter. Enlarge or reduce the width of the area by dragging the splitter.
B	**Variables list** List of all variables matching the search and filter criteria. From here variables can be selected to be used for various operations, like display in an instrument, export into a measure file, or usage as input for a calculated signal.

To find the desired signals, the Variable Explorer offers different functionalities:

	Collapse All Categories Closes all categories simultaneously.
	Accordion Allows to see only one category at a time. If you activate the accordion mode, the chosen category expands, while any other one is collapsed.
	Enable / Disable Filters Applies the selected filters. When you disable the filters, the original selection is maintained, even if you clear all filters using the funnel icon above in the variables list.
	Clear Filters Deletes all selected entries in one category or all filters in the Variable Explorer.

	Search Items Searches items per category or in the variables list.
	Freeze Columns For more information, see "To freeze columns" on page 126.
	Show / Hide Columns For more information, see "To show or hide columns" on page 126.
	Used / Unused Signals For more information, see "To filter used/unused signals" on page 130.
	Change Display Name For more information, see "Defining Display Name in the Application" on page 125.

You can perform the following actions:

- "To search for signals" below
- "To filter signals" on the next page
- "To reset filters" on page 130
- "To sort signals per column in the Variable Explorer" on page 130

To search for signals

If you perform a search in the Variable Explorer, only the **Display Name** column is used for the search. The search is executed for all signals that are currently listed in the Variable Explorer.

If you perform a search in a category, only entries matching the search criteria are shown in the category list.

If you perform a search in the Configuration Manager, the search is executed for signals already being in use and assigned to an instrument. For more information, see "Searching and filtering within the Configuration" on page 34.

Do the following:

1. Bring the focus to the search box window.
2. Enter your search string.

For search queries, take the following rules into account:

- The search is not case-sensitive; it finds data, even if the capitalization in the text is different from the capitalization in the search term.
- You can use the ? and * characters as wildcards in your search string.
- By default, the search string is appended to the "*" wildcard in the search box. If you want to search for data starting with a specific string, position the cursor in front of this wildcard.
- If you have additionally used filters, a logical AND is applied between the search and the filters of the other columns.

The matching search string is highlighted in the **Display Name** column. If this column is currently not displayed, unhide the column. For more information, see "To show or hide columns" on page 126.

To filter signals

In the Variable Explorer, there are temporarily different locations to define filters. Depending on the availability of metadata in a measure file, you have more or less options to filter the variables list.

To see how you can select the relevant signals and assign them to the desired instrument, watch our video **Selecting Signals**.

You can perform the following actions:

— **Filtering in columns**

1. In the header of the column that you want to filter, click .

   A list with the possible filter criteria for this column is displayed.

   In the **Raster** column, the filter for variables that have been recorded in several rasters works as follows:

   - If variables were recorded in INCA in several rasters and are stored into one single file, the variables can be filtered by the combined raster only (combined per + symbol) and not by the different individual rasters.
   - If variables were recorded in several files and exported into one common file, the variables are shown as list separated by a semicolon and can be filtered per raster. The filter option allows you only to select a single value (e.g. "0.1"). The variable is shown if one of its rasters is used for filtering.

2. Clear the checkboxes of those entries that you do not want to be displayed in the list. When checking several filters within the list, a logical OR is applied.
3. Click **Apply**.

   The Variable Explorer table displays the rows which match the defined filter criteria.

4. You can repeat the steps 1-3 for other columns.

   A logical AND is applied between the filters of the different columns.

   All column filters possibilities will be integrated in the filters per categories.

— **Filtering per categories**

  To see the desired signals in the variables list, check within one category the corresponding entries.

  You can disable all filters in a category. The original selection is maintained, but has no effect on the variables list, even if you clear filters using the icon.

Filters possibilities in categories, columns and search box can be applied in combination (using a logical AND). In the header of the **Display Name** column, the number of matching signals out of the total number of signals is shown.

To filter used/unused signals

In the Variable Explorer toolbar, click .

There are three different states of this button:

	Shows all used signals, i.e. signals assigned to an instrument or used for a calculated signal
	Shows all unused signals, i.e. signals not assigned to an instrument or not used for a calculated signal
	Shows all used and unused signals

To reset filters

You can perform the following actions:

- **Clear all filters**

  To clear all filters in categories, columns, and search box , click in the area of the variables list.

  Disabled filters in categories remain unchanged.
- **Filters in columns**
  i. To reset filters of a specific column, click .
  ii. In the drop-down list, select the **Select all** checkbox.
- **Filters in a category**
  i. To clear the filters in one category, click .
  ii. To deactivate the selected filters, click .

  The original selection is maintained, even if you clear the filters using the icon.

To sort signals per column in the Variable Explorer

In the Variable Explorer, the **Display Name** column is sorted by default. If you want to sort another column, click the header of the respective column. The currently sorted column is indicated by a blue color label and underline. The sorting is not case-sensitive. If the entries start with a number, they are ordered alphanumerically.

6.4 Extracting Bits from a Signal or Elements from an Array

MDA supports the extraction of individual bits from analog signals and the extraction of individual or all elements from an array signal.

For each selected bit or extracted array element, a calculated signal is created The name of the calculated signals consists of the original signal name and the bit number, respectively the array element ID. The calculated signals are available in the Variable Explorer and in the Calculated Signals Editor. You can use these signals in instruments and other calculated signals. For more information, see "Assigning Signals to Instruments" on the next page and "Defining Calculated Signals" on page 173.

To delete extracted bit signals, see "To delete a calculated signal" on page 176.

6.4.1 Extracting Bits from a Signal

Some signals can be interpreted as a combination of 1-bit signals. For example, a byte signal contains eight different, independent status bits. Each bit represents one specific status information. For such signals, rather the individual bit signals are relevant.

To see how you can extract individual bits from a signal and rename them, watch our video **Extracting Bits from a Signal**.

To extract these from the complete signal block, proceed as follows:

1. In the Variable Explorer, select the signal from which the individual bits shall be extracted.
2. In the context menu, select **Generate Bit Signals**.
3. Select one or more bits.

   The number of checkboxes depends on the data type of the selected variable.
4. Click **Generate**.

6.4.2 Extracting Elements from a Array

1. In the Variable Explorer, select the array signal from which the elements shall be extracted.
2. In the context menu, select **Extract Array Elements**.
3. Select one, more or all elements.
   - Select one element via clicking on one individual element.
   - Click **Select all** to choose all elements available.
   - You can also define the array size manually by clicking on individual elements to make your own choice.
4. Click **OK** or **Clear Selection**.

   For each extracted array element the same conversion formula is applied.

The conversion formula defined for the array in total is valid for all elements of the array. Therefore, when extracting an element of an array, the individual element gets the same conversion formula.

6.5 Assigning Signals to Instruments

To display the actual measure data of a signal, the signal must be assigned to an instrument. For more information, see "To add a measure file" on page 43.

You can perform the following actions:

- "To assign signals with the same name from another file" below
- "To assign signals to a new instrument" below
- "To assign signals to multiple instruments" on the next page
- "To assign signals to an existing instrument" on the next page
- "To move or copy signals from one instrument to another instrument" on page 134
- "To replace a signal" on page 134
- "Assigning Signals to Instruments" above

To quickly assign signals to an existing oscilloscope, the selection wheel can be used. For more information, see "To assign signals using the selection wheel" on page 75.

To assign signals to a new instrument

To see how you can select the relevant signals and assign them to the desired instrument, watch our video **Selecting Signals**.

1. In the Variable Explorer, press the CTRL or SHIFT key and mark all signals that you want to use for analysis.
2. Drag and drop the selected signals to an empty area of the layer or onto the layer tab in the currently active configuration. It is not possible to drag signals to another configuration.

   The context menu of the layer opens containing an item for each available instrument type.
3. Select the instrument type that you want to create.

   On the layer, the new instrument is displayed in the foreground and highlighted. In the Configuration Manager, the name of this instrument is displayed in bold.

   In the Variable Explorer, you can identify all signals that are assigned to an instrument in the Used column. For more information, see "To filter used/unused signals" on page 130.

To assign signals with the same name from another file

1. In the Configuration Manager, select one or more oscilloscopes and open the context menu. Alternatively right-click a layer or the configuration node.
2. In the context menu, select **Select file for comparison**.
3. In the File Explorer,

1. Select one or more files
2. In the context menu, select **Use as File(s) for Comparison**

⇨ Signals from the new files will be added to the target oscilloscopes. For each existing signal the respective signal from the new files is added, but only once per target instrument. If a signal is missing in the new file, a placeholder in the 'no-match' state will be created.
Currently, the functionality is limited to oscilloscopes.

To assign signals to multiple instruments

1. In the Variable Explorer, press the CTRL or SHIFT key and mark all signals that you want to use for analysis.
2. Right-click and select **Add Signals to Instruments**.

   A dialog appears.
3. In the dialog, select the layers and the instruments where you want to add the signals and confirm with **OK**.

⇨ The signals are added to the default positions for the signal types in each of the selected instruments.

To assign signals to an existing instrument

1. In the Variable Explorer, press the CTRL or SHIFT key and mark all signals that you want to use for analysis.
2. Using the mouse:
   i. Drag and drop the selected signals to an existing instrument in the configuration or to the respective entry in the Configuration Manager.
3. Using the keyboard:
   i. In the instrument, press the INSERT key.

      A pop-up appears.

   ii. Type at least one letter of the signal name.

      A list of signals appears.

      The next time the pop-up appears, it displays the previously selected signal name in a marked state.

   iii. To move to the desired signal, use the ARROW UP or ARROW DOWN key.
   iv. To add a single signal to the instrument, press ENTER.

   or

      To add several signals to the instrument, press the SPACE key and continue to select or search.

To move or copy signals from one instrument to another instrument

1. In the instrument from where you want to move or copy the signals, select the signals.
2. Do one of the following:
   - To move the signals, drag the selected signals and drop them onto another instrument.
   - To copy the signals press the CTRL key first before dropping the selected signals onto another instrument.

   The same operations are also supported in the Configuration Manager.

   If you drag signals to an instrument of the same instrument type, all settings that you have already defined for a signal in the first instrument persist.

   In case the target instrument is located on another layer, use the layer tabs to navigate to the desired layer.

   To create a new instrument, drag and drop the selected signals onto an empty space of the layer or onto the layer tab.

   When you move or copy a signal, MDA checks whether the target instrument supports the current data representation. If this is not the case, the physical data representation is used instead. For more information about data representation in the oscilloscope, see "To change the data representation of a signal" on page 86, and in the table, see "To change the data representation of a signal" on page 96.

To replace a signal

If you need to replace a signal by another signal, for example to resolve a 'no-match' status, or to replace all signals with the same name in one step, do the following:

1. In the instrument or Configuration Manager, right-click the signal that you want to replace.
2. In the context menu, select **Replace Signal**.

   A dialog appears.
3. In the drop-down menu **Replace Signal**, select which signal you want to replace:
   - this signal/ECU/Device combination
   - all signals with the given name
4. In the drop-down menu, select where you want to replace the signal:
   - in an instrument (just in the currently active instrument from which you triggered the **Replace Signal** operation)
   - **All instruments and calculations** (which includes all instruments and input signals for calculated signals)
5. In the search box, enter the signal name that you want to use for the replacement.

6. Click the signal name or press the **Enter** key.

   The result of the signal replacement is shown in the status bar.

   The signal properties are maintained in all instruments and in the calculated signal.

6.6 Displaying Signal Information

You can perform the following actions:

- "To identify the origin of a signal" below
- "To display errors and warnings of a signal" below
- "To display the metadata of an assigned signal" on the next page

To identify the origin of a signal

1. You can identify the origin of an assigned signal via the identifier of the measure file. Do one of the following:
   - In the Variable Explorer, make the **File ID** column visible. For more information, see "To show or hide columns" on page 126.
   - In the Configuration Manager, expand the tree view until the signals are shown. The identifier of the measure file is displayed in front of the signal name, and the tooltip shows the file name.
   - Bring the instrument in which the signal is used into the foreground. The identifier of the measure file is displayed in front of the signal name, and the tooltip shows the file name.

   For calculated signals, a square root symbol is shown as file identifier. If the measure file has been deleted from the configuration, a "?" is displayed instead of the identifier.

   If you have replaced the measure file and the signal is not part of the new file, the former identifier still remains visible but the signal is indicated as a 'no-match' signal.

To display errors and warnings of a signal

1. Do one of the following:
   - In the Variable Explorer, make the **Error** column visible. For more information, see "To show or hide columns" on page 126. If a signal contains an error or warning, a corresponding icon is displayed in this column.
   - In the Configuration Manager, expand the tree view until the signals are shown. If an assigned signal contains an error or warning, a corresponding icon is displayed next to the signal name.
   - Bring the instrument in which the signal is used into the foreground. If an assigned signal contains an error or warning, a corresponding icon is displayed next to the signal name.
2. To display the complete list of all errors and warnings for the selected signal, hover over the error or warning icon.

To display the metadata of an assigned signal

To see how you can get more information about a measure file or a signal, watch our video **Displaying Meta Information**.

1. In the Variable Explorer, Configuration Manager or in an instrument, select a signal.
2. Press CTRL+I.

   All meta information which is available for the corresponding signal is displayed in the **Information Window**. You can select the content of the table and copy it to the clipboard.
3. If you select another signal, the metadata is updated automatically in the Information window.

6.7 Reusing the Signal Name in Other Applications

You can perform the following actions:

- "To copy the signal name to the clipboard" below
- "To copy signal name and meta information to the clipboard" below
- "Reusing the Signal Name in Other Applications" above
- "To pass the signal name to EHANDBOOK-NAVIGATOR" on the next page

To create a file with the names of multiple signals, see "Using Label Files (LAB)" on page 55.

To copy the signal name to the clipboard

1. In the Configuration Manager, Variable Explorer, or in an instrument, select a signal. If you select several signals, the last selected signal is used.
2. Press CTRL+C.

⇨ The signal name is copied to the clipboard in plain text without file ID.

To copy signal name and meta information to the clipboard

1. Open the Information window for the respective signal (e.g. by pressing CTRL+I in an instrument or the Configuration Manager).
2. Select the rows with the information to be copied.
3. Press CTRL+C.

⇨ The row name and the signal information are copied to the clipboard, and can be pasted into other applications as text.

To copy information for multiple signals

To copy the information for multiple signals, so that you can reuse the information in another application, do the following:

1. In the Variable Explorer select one or multiple signals.
2. Right-click to open the context menu and select **Copy Contents**.

⇨ For the chosen signals all contents of the active columns and the column headers are copied.

The **Copy Contents** function is available in the oscilloscope's signal list and in the statistical data instrument, too.

To pass the signal name to EHANDBOOK-NAVIGATOR

You can display additional information for a selected signal in EHANDBOOK-NAVIGATOR. To pass the signal name to EHANDBOOK-NAVIGATOR, the connection must be established. For more information, see "Connecting MDA to EHANDBOOK-NAVIGATOR" on page 30.

1. In an instrument, select a signal.
2. In the context menu, select **Open Signal Documentation**.

⇨ In EHANDBOOK-NAVIGATOR, the available signal information is displayed.

7 Calculations

Calculations are helpful in displaying and analyzing measure data. MDA supports two types of calculations:

- **Functions**: use of predefined calculations based on Functional Mock-up Units (FMUs, for example created with ASCMO)

  Some FMUs require that the input signals provide a value at exactly time stamp 0, which is typically not the case for usual measurement files with recordings in different acquisition rates. MDA handles the input signals in a special manner to enable the usage of such FMU models:

  Basically, all input signals are shifted as one group so that for at least one input signal a value can be given at time stamp = 0. The outputs of the FMU model are shifted in the opposite direction and are then displayed in MDA.

  There is a warning icon shown for all FMU output signals to indicate whether there was such a time shift operation applied in the background.

- **Calculated Signals**: use of mathematical operators for defining individual formulas

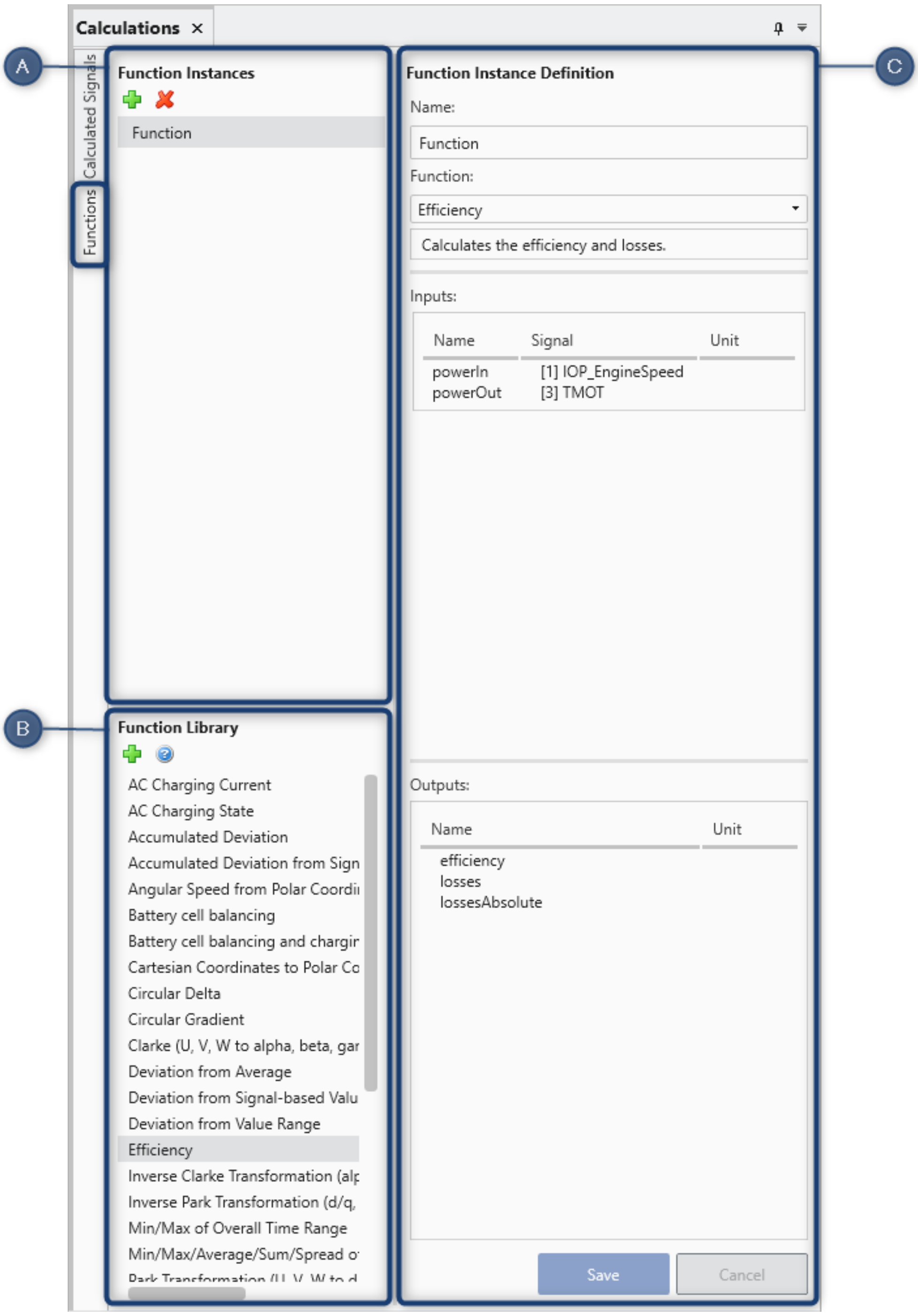
After creating a calculation, the calculation results are displayed in the Variable Explorer. They can be selected and used like other measure signals.

7.1 Functions

The Functions tab allows users to use complex calculations based on ASCMO Functional Mock-up Units (FMU) in an easy way.

The usage of an FMU is comfortable as you only have to assign the needed input signals. The calculated outputs are available in the Variable Explorer and can be used like normal signals.

The Functions editor is divided into the following areas:



No.	Description
A	**List of all instances** As soon as you have created your own function instance, by selecting a predefined function and assigning inputs to it, the instance is displayed in the area **Function Instances**.
B	**List of all functions** In the **Function Library**, the list of all available predefined functions and loaded FMUs is shown.
C	**Fields to define an instance** To define an instance, select from the drop-down-menu one of the available predefined functions. To map measure signals to the inputs of the function, drag the measure signals onto the **Inputs** block. The **Outputs** block shows you which calculations the chosen pre-defined function will provide.

7.1.1 Functions delivered with MDA

7.1.1.1 AC Charging Current

Determines the maximum charging current derived from the duty cycle of the CP (Communication Pilot) signal, according to IEC.

Input(s)

Name	Description	Type
dutyCycle	Duty cycle of electrical signal of the CP line between EV and EVSE	Analog Signal

Output(s)

Name	Description	Unit	Type
currentRaw	Charging current	A	Analog Signal
currentRounded	Charging current rounded to full Ampere	A	Analog Signal
dutyCyclePercent	Duty cycle converted to percent	%	Analog Signal

7.1.1.2 AC Charging State

Determines the charging state derived from the voltage of the CP (Communication Pilot) signal, according to IEC 61851-1.

Input(s)

Name	Description	Unit	Type
CP Voltage	Electrical signal of the CP line between EV and EVSE	V	Analog Signal

Output(s)

Name	Description	Unit	Type
CPPeakVoltage	Peak value of the input signal over the last 100 msec	V	Analog Signal
EVConnectedBit	EV connection status in Boolean representation		Logical Signal
EVConnectedText	EV connection status in textual representation		Textual Signal
EVSEConnectedBit	EVSE connection status in Boolean representation		Logical Signal
EVSEConnectedText	EVSE connection status in textual representation		Textual Signal
readyForChargingBit	Charging readiness status in Boolean representation		Logical Signal
readyForChargingText	Charging readiness status in textual representation		Textual Signal
readyWithVentilationBit	Charging readiness status in Boolean representation		Logical Signal
readyWithVentilationText	Charging readiness status in textual representation		Textual Signal
stateAlpha	Charging status in short textual representation		Textual Signal
stateNum	Charging status in numerical representation		Analog Signal
stateText	Charging status in textual representation		Textual Signal

7.1.1.3 Accumulated Deviation

Evaluates the values of input signals with lower and upper limits and identifies exceedance situations.

Considers the time range starting from the beginning of the measurement to the current point in time.

Input(s)

Name	Description	Type
inputs	Arbitrary number of input signals	Multiple Signals
lowerLimit	Lower limit of range	Analog Signal
upperLimit	Upper limit of range	Analog Signal

Output(s)

Name	Description	Type
inputArray	Array of input signals	Array Signal
upperExcdBoolArray	Array indicating if upper exceedance occurred since the beginning of the measurement (bool)	Array Signal
lowerExcdBoolArray	Array indicating if lower exceedance occurred since the beginning of the measurement (bool)	Array Signal
upperExcdTimeArray	Durations of upper exceedances	Array Signal
upperExcdAccArray	Integrated values of upper exceedances	Array Signal
upperExcdSquareArray	Integrated square values of upper exceedances	Array Signal
lowerExcdTimeArray	Durations of lower exceedances	Array Signal
lowerExcdAccArray	Integrated values of lower exceedances	Array Signal
lowerExcdSquareArray	Integrated square values of lower exceedances	Array Signal
upperExcdCount	Count of signals with upper exceedance	Analog Signal
lowerExcdCount	Count of signals with lower exceedance	Analog Signal

Name	Description	Type
maxUpperExcd	Maximum upper exceedance	Analog Signal
maxLowerExcd	Maximum lower exceedance	Analog Signal
maxIDUpperExcd	ID of signal with maximum upper exceedance (0-based)	Analog Signal
maxIDLowerExcd	ID of signal with maximum lower exceedance (0-based)	Analog Signal
totalUpperExcd	Total upper exceedance	Analog Signal
totalLowerExcd	Total lower exceedance	Analog Signal

Notes

Input signals are sorted in natural order on save. Time channel of array outputs is the combination of all input signals' rasters.

7.1.1.4 Accumulated Deviation from Signal-based Value Range

Evaluates the values of an array of input signals with signal-based lower and upper limits and identifies exceedance situations.

Input(s)

Name	Description	Type
inputArray	Array signal including an arbitrary number of input signals	Array Signal
lowerLimit	Lower limit of range	Analog Signal
upperLimit	Upper limit of range	Analog Signal

Output(s)

Name	Description	Type
upperExcdBoolArray	Array indicating if upper exceedance occurred since the beginning of the measurement (bool)	Array Signal
lowerExcdBoolArray	Array indicating if lower exceedance occurred since the beginning of the measurement (bool)	Array Signal
upperExcdTimeArray	Durations of upper exceedances	Array Signal
upperExcdAccArray	Integrated values of upper exceedances	Array Signal
upperExcdSquareArray	Integrated square values of upper exceedances	Array Signal
lowerExcdTimeArray	Durations of lower exceedances	Array Signal
lowerExcdAccArray	Integrated values of lower exceedances	Array Signal
lowerExcdSquareArray	Integrated square values of lower exceedances	Array Signal
upperExcdCount	Count of signals with upper exceedance	Analog Signal
lowerExcdCount	Count of signals with lower exceedance	Analog Signal
maxUpperExcd	Maximum upper exceedance	Analog Signal
maxLowerExcd	Maximum lower exceedance	Analog Signal
maxIDUpperExcd	ID of signal with maximum upper exceedance (0-based)	Analog Signal
maxIDLowerExcd	ID of signal with maximum lower exceedance (0-based)	Analog Signal
totalUpperExcd	Total upper exceedance	Analog Signal
totalLowerExcd	Total lower exceedance	Analog Signal

7.1.1.5 Angular Speed from Polar Coordinates

Calculates the turning rate from the angle of the vector.

Input(s)

Name	Description	Unit	Type
angle	Angle of the vector in polar coordinates	rad	Analog Signal
length	Length of the vector		Analog Signal
threshold	Minimum length of the vector to trigger the calculation		Analog Signal

Output(s)

Name	Description	Type
angularSpeed	Angular Speed	Analog Signal

Note

The length of the vector must exceed the threshold.

7.1.1.6 Battery cell balancing

Analyzes the battery cell balancing and determines the current, charge, power and energy dissipation at the balancing resistors. Data is provided for individual cells and for the battery pack.

Input(s)

Name	Description	Unit	Type
cellVoltages	Voltage signals of multiple battery cells as array	V	Array Signal
balancingInd	Balancing indicator of multiple battery cells as array. 0 = inactive, 1 = active.		Array Signal (of logicals)
resistor	Value of the balancing resistor.	Ohm	Analog Signal

Output(s)

Name	Description	Unit	Type
balCount	Number of cells in the battery pack currently balancing.		Analog Signal
balPowerTotal	Total power dissipation at the balancing resistors during balancing for the battery pack.	W	Analog Signal
balEnergyTotal	Total energy dissipation at the balancing resistors during balancing for the battery pack.	Wh	Analog Signal
balChargeTotal	Total charge dissipation at the balancing resistors during balancing for the battery pack.	Ah	Analog Signal
balTimeArray	Accumulated duration of balancing for each cell.	h	Array Signal
balCurrentArray	Current at the balancing resistor during balancing for each cell.	A	Array Signal
balChargeArray	Charge dissipation in the balancing resistor during balancing for each cell.	Ah	Array Signal
balPowerArray	Power dissipation in the balancing resistor during balancing for each cell.	W	Array Signal
balEnergyArray	Energy dissipation in the balancing resistor during balancing for each cell.	Wh	Array Signal

Notes

Inputs are arrays and probably require conversion with "Signal to Array". Time channel of array outputs is the combination of all input signals' rasters.

7.1.1.7 Battery cell balancing and charging

Analyzes the battery cell balancing and determines the current, charge, power and energy dissipation at the balancing resistors and at the battery cells. Data is provided for individual cells and for the battery pack.

Input(s)

Name	Description	Unit	Type
cellVoltages	Voltage signals of multiple battery cells as array	V	Array Signal
balancingInd	Balancing indicator of multiple battery cells as array. 0 = inactive, 1 = active.		Array Signal (of logicals)
resistor	Value of the balancing resistor.	Ohm	Analog Signal
packCurrent	Battery pack current	A	Analog Signal

Output(s)

Name	Description	Unit	Type
balCount	Number of cells in the battery pack currently balancing.		Analog Signal
balPowerTotal	Total power dissipation at the balancing resistors during balancing for the battery pack.	W	Analog Signal
balEnergyTotal	Total energy dissipation at the balancing resistors during balancing for the battery pack.	Wh	Analog Signal
balChargeTotal	Total charge dissipation at the balancing resistors during balancing for the battery pack.	Ah	Analog Signal
balTimeArray	Accumulated duration of balancing for each cell.	h	Array Signal
balCurrentArray	Current at the balancing resistor during balancing for each cell.	A	Array Signal
balChargeArray	Charge dissipation in the balancing resistor during balancing for each cell.	Ah	Array Signal
balPowerArray	Power dissipation in the balancing resistor during balancing for each cell.	W	Array Signal
balEnergyArray	Energy dissipation in the balancing resistor during balancing for each cell.	Wh	Array Signal
chrgPowerTotal	Total power applied to the battery cells during balancing for the battery pack.	W	Analog Signal
chrgEnergyTotal	Total energy applied to the battery cells during balancing for the battery pack.	Wh	Analog Signal
chrgChargeTotal	Total charge applied to the battery cells during balancing for the battery pack.	Ah	Analog Signal

Name	Description	Unit	Type
chrgCurrentArray	Current applied to the battery cells during balancing for each cell.	A	Analog Signal
chrgPowerArray	Power applied to the battery cells during balancing for each cell.	W	Analog Signal
chrgEnergyArray	Energy applied to the battery cells during balancing for each cell.	Wh	Analog Signal
chrgChargeArray	Charge applied to the battery cells during balancing for each cell.	Ah	Analog Signal

Notes

Inputs are arrays and probably require conversion with "Signal to Array".

Time channel of array outputs is the combination of all input signals' rasters.

7.1.1.8 Cartesian Coordinates to Polar Coordinates

Calculates the polar coordinates from Cartesian coordinates

Input(s)

Name	Description	Type
x	x-coordinate	Analog Signal
y	y-coordinate	Analog Signal

Output(s)

Name	Description	Unit	Type
angle	Angle of the vector in polar coordinates	rad	Analog Signal
length	Length of the vector		Analog Signal

7.1.1.9 Circular Delta

Calculates the difference of adjacent samples of an input signal ("delta"), and considers periods (e.g. angle).

Input(s)

Name	Description	Type
input	Input Signal	Analog Signal
period	Period	Constant Value

Output(s)

Name	Description	Type
circularDelta	The delta of adjacent samples considering the period.	Analog Signal

7.1.1.10 Circular Gradient

Calculates the value difference divided by the time difference of adjacent samples of the input signal ("gradient"), and considers periods (e.g. angle).

Input(s)

Name	Description	Type
input	Input signal	Analog Signal
period	Period	Constant Value

Output(s)

Name	Description	Type
circularGradient	The gradient of adjacent samples considering the period.	Analog Signal

7.1.1.11 Clarke Transformation

Clarke Transformation (U, V, W to alpha, beta, gamma)

Converts the signals of a three phase system into two axis stationary system.

Input(s)

Name	Description	Type
U	U	Analog Signal
V	V	Analog Signal
W	W	Analog Signal

Output(s)

Name	Description	Type
alpha	alpha	Analog Signal
beta	beta	Analog Signal
gamma	gamma	Analog Signal

7.1.1.12 Deviation from Average

Calculates the deviation of the current signal value from the average for an abitrary number of signals.

Input(s)

Name	Description	Type
inputs	Arbitrary number of input signals	Multiple Signals

Output(s)

Name	Description	Type
average	Average value of input signals	Analog Signal
deviationArray	Array of deviations from the average for each input signal	Array Signal
IDMax	ID of the array element with the maximum value (0-based)	Analog Signal
IDMin	ID of the array element with the minimum value (0-based)	Analog Signal
inputArray	Array of the input signals	Array Signal
maximum	Maximum value of input signals	Analog Signal
maxNegDeviation	Maximum negative deviation from the average	Analog Signal
maxPosDeviation	Maximum positive deviation from the average	Analog Signal
minimum	Minimum value of input signals	Analog Signal
spread	Spread between minimum and maximum of input signals	Analog Signal
sum	Sum of input signals	Analog Signal

7.1.1.13 Deviation from Signal-based Value Range

Evaluates for an array of input signals whether the current signal value exceeds the lower or upper limit defined by analog signals.

Input(s)

Name	Description	Type
inputArray	Array signals including an arbitrary number of input signals	Array Signal
lowerLimit	Lower limit of range	Analog Signal
upperLimit	Upper limit of range	Analog Signal

Output(s)

Name	Description	Type
aboveLimitBoolArray	Array indicating which cells are above upper limit	Array Signal
aboveLimitCount	Number of signals above upper limit	Analog Signal
average	Average value of input signals	Analog Signal
belowLimitBoolArray	Array indicating which cells are below lower limit	Array Signal
belowLimitCount	Number of signals below lower limit	Analog Signal
inputArray	Array of the input signals	Array Signal
lowerExcdArray	Array indicating the difference to lower limit (for signals which are below lower limit, otherwise 0)	Array Signal
maximum	Maximum value of input signals	Analog Signal
min	Minimum value of input signals	Analog Signal
sum	Sum of input signals	Analog Signal
upperExcdArray	Array indicating the difference to upper limit (for signals which are above upper limit, otherwise 0)	Array Signal

7.1.1.14 Deviation from Value Range

Evaluates whether the current signal value exceeds the user-defined lower or upper limits for an arbitrary number of signals

Input(s)

Name	Description	Type
inputs	Arbitrary number of input signals	Multiple Signals
upperLimit	Upper limit of range	Constant Value
lowerLimit	Lower limit of range	Constant Value

Output(s)

Name	Description	Type
aboveLimitBoolArray	Array indicating which cells are above upper limit	Array Signal
aboveLimitCount	Number of signals above upper limit	Analog Signal
average	Average value of input signals	Analog Signal
belowLimitBoolArray	Array indicating which cells are below lower limit	Array Signal
belowLimitCount	Number of signals below lower limit	Analog Signal
inputArray	Array of the input signals	Array Signal
lowerExcdArray	Array indicating the difference to lower limit (for signals which are below lower limit, oth-erwise 0)	Array Signal
maximum	Maximum value of input signals	Analog Signal

Name	Description	Type
min	Minimum value of input signals	Analog Signal
sum	Sum of input signals	Analog Signal
upperExcdArray	Array indicating the difference to upper limit (for signals which are above upper limit, otherwise 0)	Array Signal

Notes

Input signals are sorted in natural order on save. Time channel of array outputs is the combination of all input signal rasters.

7.1.1.15 Efficiency

Calculates the efficiency and losses

Input(s)

Name	Description	Type
powerIn	Power In	Analog Signal
powerOut	Power Out	Analog Signal

Output(s)

Name	Description	Type
efficiency	Efficiency = powerOut / powerIn	Analog Signal
losses	Losses = 1 - (powerOut / powerIn)	Analog Signal
lossesAbsolute	lossesAbsolute = powerIn - powerOut	Analog Signal

7.1.1.16 Inverse Clarke Transformation

Inverse Clarke Transformation (alpha, beta, gamma to U, V, W)

Converts the signals of a two axis stationary system into a three phase system.

Input(s)

Name	Description	Type
alpha	alpha	Analog Signal
beta	beta	Analog Signal
gamma	gamma	Analog Signal

Output(s)

Name	Description	Type
U	U	Analog Signal
V	V	Analog Signal
W	W	Analog Signal

7.1.1.17 Inverse Park Transformation

Inverse Park Transformation (d/q, theta, gamma to U, V, W)

Converts the signals of a two axis rotational system into a three phase system.

Input(s)

Name	Description	Type
d	d	Analog Signal
gamma	gamma	Analog Signal
q	q	Analog Signal
theta	theta	Analog Signal

Output(s)

Name	Description	Type
U	U	Analog Signal
V	V	Analog Signal
W	W	Analog Signal

7.1.1.18 Min, Max, Average and Sum of several input signals

Calculates the minimum, maximum, average, sum and spread value from all given input signals.

Input(s)

Name	Description	Type
inputs	Arbitrary number of input signals	Multiple Signals

Output(s)

Name	Description	Type
average	Average of all input signals	Analog Signal
maximum	Maximum of all input signals	Analog Signal
minimum	Minimum of all input signals	Analog Signal
spread	Spread between minimum and maximum of input signals	Analog Signal
sum	Sum of all input signals	Analog Signal

Note

Minimum and maximum output signals represent the envelope curve in an oscilloscope.

7.1.1.19 Min and Max of Overall Time Range

Calculates the minimum and maximum value of all samples for the whole time range.

Input(s)

Name	Description	Type
input	Input signal	Analog Signal

Output(s)

Name	Description	Type
minimum	Minimum of all samples	Analog Signal
maximum	Maximum of all samples	Analog Signal

7.1.1.20 Park Transformation

Park Transformation (U, V, W to d/q)

Converts the signals of a three phase system into a two axis rotational system.

Input(s)

Name	Description	Type
theta	theta	Analog Signal
U	U	Analog Signal
V	V	Analog Signal
W	W	Analog Signal

Output(s)

Name	Description	Type
d	d	Analog Signal
gamma	gamma	Analog Signal
q	q	Analog Signal

7.1.1.21 PWM Analysis

Calculates the duty cycle, pulse width, period or frequency from a single input signal. The input signal is converted to a digital signal.

Input(s)

Name	Description	Unit	Type
input	Input signal		Analog Signal
signalThreshHigh	Threshold for the high state		Constant Value
signalThreshLow	Threshold for the low state		Constant Value
timeout	Time-out	s	Constant Value

Output(s)

Name	Desription	Unit	Type
dutyCycleHigh	Duty cycle based on the high phase of the input signal. Calculated signal is updated on positive slope of the input signal. Range 0...1.		Analog Signal
dutyCycleHighPercent	Duty cycle based on the high phase of the input signal. Calculated signal is updated on positive slope of the input signal. Range 0...100%.	%	Analog Signal
dutyCycleLow	Duty cycle based on the low phase of the input signal. Calculated signal is updated on negative slope of the input signal. Range 0...1.		Analog Signal
dutyCycleLowPercent	Duty cycle based on the low phase of the input signal. Calculated signal is updated on negative slope of the input signal. Range 0...100%.	%	Analog Signal
frequency	Frequency of the input signal. Considers the complete cycle. Calculated signal is updated on positive or negative slope of the input signal.	Hz	Analog Signal
frequencyNeg	Frequency of the input signal. Considers the complete cycle. Calculated signal is updated on negative slope of the input signal.	Hz	Analog Signal
frequencyPos	Frequency of the input signal. Considers the complete cycle. Calculated signal is updated on positive slope of the input signal.	Hz	Analog Signal

Name	Desription	Unit	Type
period	Duration of the signal period between two identical slopes of the input signal. Considers the complete cycle. Calculated signal is updated on positive or negative slope of the input signal.	s	Analog Signal
periodNeg	Duration of the signal period between two negative slopes of the input signal. Considers the complete cycle. Calculated signal is updated on negative slope of the input signal.	s	Analog Signal
periodPos	Duration of the signal period between two positive slopes of the input signal. Considers the complete cycle. Calculated signal is updated on positive slope of the input signal.	s	Analog Signal
pulseWidthHigh	Duration of the high period of the input signal. Calculated signal is updated on negative slope of the input signal.	s	Analog Signal
pulseWidthLow	Duration of the low period of the input signal. Calculated signal is updated on positive slope of the input signal.	s	Analog Signal
state	Signal state of the derived digital signal after applying the thresholds. Calculated signal is updated on positive or negative slope of the input signal.		Logical Signal

Notes

The threshold parameters are used to derive the 'high' and 'low' states of the digital signal. The output signals are updated whenever the state of the input signal changes, or after a timeout duration (specified in seconds).

7.1.1.22 Rolling Integral, Average, Minimum, Maximum, Sum (time based)

Calculates for the given time range the rolling values for integral, average, minimum, maximum and sum.

Input(s)

Name	Description	Unit	Type
input	Input signal		Analog Signal
timeRange	Defines the time window for the rolling calculation.	s	Constant Value

Output(s)

Name	Description	Type
average	Rolling average for the defined time range	Analog Signal
integral	Rolling integral for the defined time range	Analog Signal
maximum	Rolling maximum for the defined time range	Analog Signal
minimum	Rolling minimum for the defined time range	Analog Signal
sum	Rolling sum for the defined time range	Analog Signal

Notes

Calculation is based on the samples which are available in the time window defined by the time range and the current point in time. Time range must be given in seconds.

7.1.1.23 Rotation2D Transformation

Transforms rotational coordinates (v_x, v_y, angle) into stationary coordinates.

Input(s)

Name	Description	Type
angle	Angle of the two-dimensional rotation	Analog Signal
v_x	v_x (original x-coordinate)	Analog Signal
v_y	v_y (original y-coordinate)	Analog Signal

Output(s)

Name	Description	Type
r_x	r_x (rotated x-coordinate)	Analog Signal
r_y	r_y (rotated y-coordinate)	Analog Signal

7.1.1.24 Section-wise Integral, Average, Minimum, Maximum, Sum

Calculates the integral, average, minimum, maximum and sum of the samples in the time ranges for which the 'condition' is in state TRUE.

Input(s)

Name	Description	Type
condition	Condition for calculating the 'TRUE' state	Logical Signal
input	Input signal	Analog Signal

Output(s)

Name	Description	Type
average	Average value of the input signal for the time range in 'True' state.	Analog Signal
integral	Integral of the input signal for the time range in 'True' state.	Analog Signal
maximum	Maximum value of the input signal for the time range in 'True' state.	Analog Signal
minimum	Minimum value of the input signal for the time range in 'True' state.	Analog Signal
sum	Sum of the input signal for the time range in 'True' state.	Analog Signal

Note

The calculations are reset to 0 in time ranges for which the 'condition' is in state FALSE.

7.1.1.25 Signals To Array

Creates an array signal from provided inputs.

Input(s)

Name	Description	Type
inputs	Arbitrary number of input signals	Multiple Signals

Output(s)

Name	Description	Type
array	Array of the input signals	Array Signal

Notes

Input signals are sorted in natural order on save. Time channel of array outputs is the combination of all input signal rasters.

7.1.1.26 State of Charge (voltage and temperature based)

Calculates the state of charge (SoC) and depth of discharge (DoD) of the battery, based on the battery voltage and temperature in the relaxed state.

Input(s)

Name	Description	Unit	Type
current	Current	A	Analog Signal
SoCMap	State-of-Charge map using voltage (x-axis) and temperature (y-axis).		Map
temperature	Temperature signal for SoC map	°C	Analog Signal
voltage	Voltage signal for SoC map	V	Analog Signal
currentThreshPos	Threshold for the positive current	A	Constant Value
currentThreshNeg	Threshold for the negative current	A	Constant Value
relaxTime	Relaxation time is the minimum duration for which the current must stay within the thresholds to detect the relaxed state.	min	Constant Value

Output(s)

Name	Description	Unit	Type
DoD	Depth of discharge in relaxed state, else no-value is returned.		Analog Signal
DoD Percent	Depth of discharge in percent in relaxed state, else no-value is returned.	%	Analog Signal
relaxState	Indicates whether the battery is in relaxed state.		Logical Signal
SoC	State-of-Charge in relaxed state, else no-value is returned.		Analog Signal
SoCPercent	State-of-Charge in percent in relaxed state, else no value is returned.	%	Analog Signal

7.1.1.27 State of Charge (voltage based)

Calculates the state of charge (SoC) and depth of discharge (DoD) of the battery, based on the battery voltage in the relaxed state.

Input(s)

Name	Description	Unit	Type
current	Current		Analog Signal
SoCCurve	State-of-Charge Curve using voltage (x-axis).		Curve
voltage	Voltage signal for SoC Curve	V	Analog Signal
currentThreshPos	Threshold for the positive current	A	Constant Value
currentThreshNeg	Threshold for the negative current	A	Constant Value
relaxTime	Relaxation time is the minimum duration for which the current must stay within the thresholds to detect the relaxed state.	min	

Output(s)

Name	Description	Unit	Type
DoD	Depth of discharge in relaxed state, else no-value is returned.		Analog Signal
DoDPercent	Depth of discharge in percent in relaxed state, else no-value is returned.	%	Analog Signal
relaxState	Indicates whether the battery is in relaxed state.		Analog Signal
SoC	State-of-Charge in relaxed state, else no-value is returned.		Analog Signal
SoCPercent	State-of-Charge in percent in relaxed state, else no-value is returned.	%	Analog Signal

7.1.1.28 Thermal Energy Accumulation

Calculates the accumulated thermal energy in a component of the thermal system based on temperature difference, heat capacity and mass of the component.

Input(s)

Name	Description	Unit	Type
temperature	Temperature of the component	°C	Analog Signal
refTemp	Temperature of a reference point	°C	Analog Signal
heatCapacity	Specific heat capacity of the component	J/K	Constant Value

Output(s)

Name	Description	Unit	Type
energy	Accumulated thermal energy	J	Analog Signal
gradient	Change rate of the accumulated thermal energy of the component (equivalent to the thermal energy flow from/to the component including heat dissipation)	J/s	Analog Signal
tempDiff	Difference between the two input temperature signals	K	Analog Signal

7.1.1.29 Thermal Energy Flow (Heat Capacity as Constant)

Calculates the thermal energy flow in the thermal system based on temperature difference, flow and specific heat capacity of the coolant (constant).

Input(s)

Name	Description	Unit	Type
temperature	Temperature of the fluid	°C	Analog Signal
refTemp	Temperature of a reference point	°C	Analog Signal
heatCapacity	Specific heat capacity of the fluid	J/(kg*K)	Constant Value
coolantFlow	Coolant mass flow at a specific point.	kg/s	Analog Signal

Output(s)

Name	Description	Unit	Type
thermalFlow	Thermal energy flow derived from temperature difference and flow rate	J/s	Analog Signal
gradient	Change rate of the energy flow	J/s^2	Analog Signal
tempDiff	Difference between the two input temperature signals	K	Analog Signal

7.1.1.30 Thermal Energy Flow (Heat Capacity as Curve)

Calculates the thermal energy flow in the thermal system based on temperature difference, flow and specific heat capacity of the coolant (depending on temperature).

Input(s)

Name	Description	Unit	Type
coolantFlow	Coolant volume flow at a specific point.	l/s	Analog Signal
heatCapacity	Specific heat capacity curve of the coolant with temperature dependency		Curve
temperature	Temperature of the fluid	°C	Analog Signal
refTemp	Temperature of a reference point	°C	Analog Signal

Output(s)

Name	Description	Unit	Type
thermalFlow	Thermal energy flow derived from temperature difference and flow rate		Analog Signal
gradient	Change rate of the energy flow	J/s^2	Analog Signal
tempDiff	Difference between the two input temperature signals	K	Analog Signal

7.1.2 Managing Instances

You can perform the following actions:

- "To add an FMU to the Function Library" below
- "To see the origin of an FMU model" below
- "To define the FMU location" below
- "To delete the FMU file" below
- "To define an instance" on the next page
- "To edit an instance" on the next page
- "To copy and paste an instance" on the next page
- "To import an instance" on the next page
- "To export an instance" on the next page
- "To delete an instance" on page 172

To add an FMU to the Function Library

1. In the **Function Library** block, click the ✚ icon.
2. Select the desired file with the file extension `*.fmu`.
3. Click **Save**.

   MDA stores the FMU file in the folder `C:\ProgramData\ETAS\MDA\8.7\CorePlugins\Etas.TargetAccess.VirtualTarget`.

⇨ The FMU is loaded automatically.

The FMU file is a zipped file. It contains the model and a model description file. MDA takes the name of the model as defined in the model description file. This name may be different from the name in the FMU file. It is also possible that the same model name is used in the model description file of different FMU files.

To see the origin of an FMU model

1. Hover the mouse over an FMU entry in the Function Library.

⇨ A tooltip will show the name of the FMU file from which the FMU model was loaded.

To define the FMU location

1. In the Functions window, right-click the FMU model in the Function Library.
2. Click **Open File in Windows Explorer**.

You can also use this method to gain access to the FMU file, for example to modify its contents, or to copy the file and make it available to a colleague, etc.

To delete the FMU file

1. In the Functions window, right-click the FMU model in the Function Library.
2. Click **Open File in Windows Explorer**.

3. Select the FMU model and delete it.

To define an instance

1. In the **Function Instances** block, click the ✚ icon to create a new instance.
2. Select an entry from the **Function** drop-down menu.
3. For each input listed under **Inputs**, drag a measure signal onto the **Signal** field.

   For some specific functions, the number of input signals is not fixed. Then drag the desired input signals onto the ✚ field in the **Inputs** block.
4. Click **Save**.

   A * indicates if an instance has unsaved changes.

   The function instance is stored in the MDA configuration.

⇨ The outputs are available in the Variable Explorer.

   You can save an instance even if you do not assign signals to all inputs. In this case, the outputs cannot be calculated.

To edit an instance

1. Select the instance in the **Function Instances** list.
2. You can directly edit the function in the **Function Instance Definition** area.
3. Click **Save**.

To copy and paste an instance

1. In the **Function Instances** field, right-click the function instance that you want to copy.
2. Select **Copy**.
3. Paste the function instance in the **Function Instances** field.

   You can paste the function instance also into another configuration.

To import an instance

Function instances are part of the configuration and can be imported accordingly. For more information, see "Importing an XDX Configuration" on page 39

If you are importing a function instance that uses an FMU for calculation, you need to check that the FMU file is available on the target computer.

To export an instance

Function instances are part of the configuration and therefore no explicit export is required. In this case, you need to provide the FMU file associated with the exported configuration, or create a new ZIP file containing both the FMU file and the export file. To know how you can access the FMU file, see "To define the FMU location" on the previous page.

If you need a zipped configuration file, see "To export a configuration and its files" on page 37.

To delete an instance

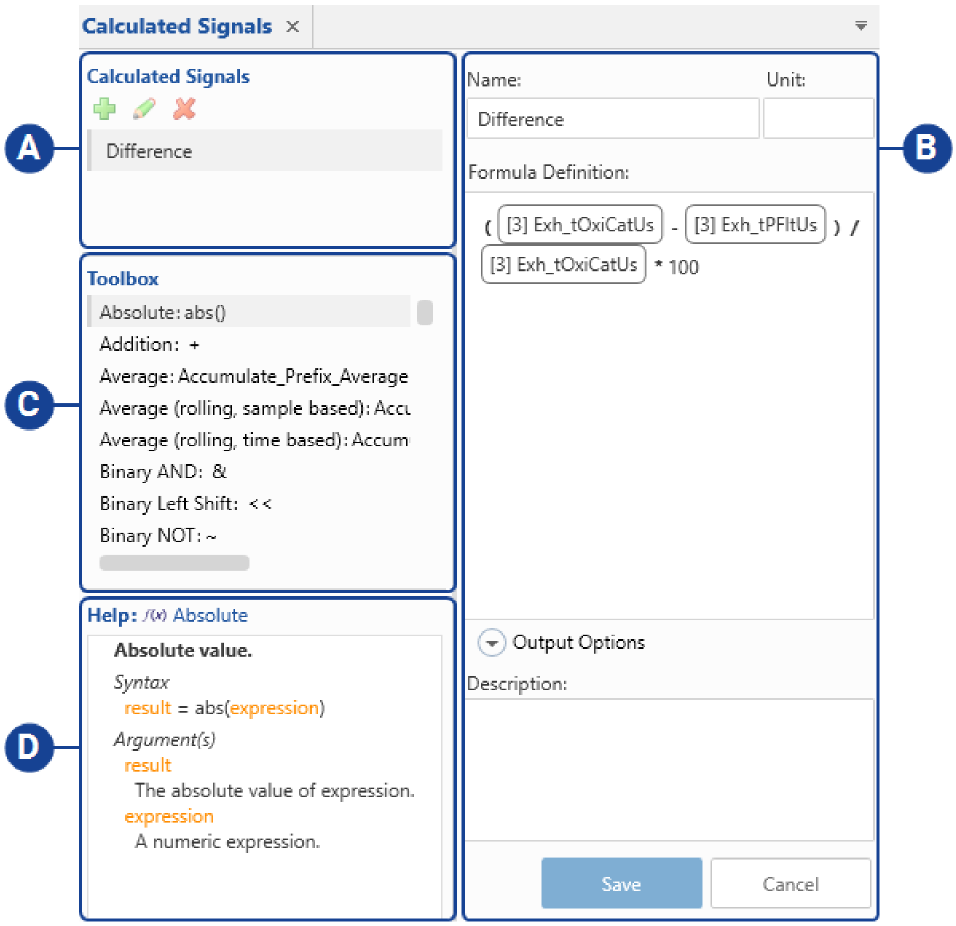
1. Click the instance in the Function Instances list.
2. Click ✘ in the Function Instances toolbar.

   Outputs from the deleted instance which are assigned to an instrument will appear in no-match state.

7.2 Calculated Signals

Calculated signals can be defined more flexibly, for example, to generate a different signal or a direct comparison with specific measured values. After you defined a calculated signal, it can be selected like any other measure signal in the Variable Explorer and assigned to an instrument. The inputs to calculated signals are always step interpolated, i.e. the last available value will be taken for calculation until there is a new value available.

The calculated signals editor is divided into the following areas:



No.	Description
A	**List of all calculated signals** As soon as you have created a calculated signal, it is displayed in this area. From here, you can perform the following actions: – To use a calculated signal, drag and drop it into an instrument of your configuration. – Select a calculated signal in the list to edit, duplicate, or delete it. For more information, see "Managing Calculated Signals" on page 175.
B	**Entry fields to define or edit a calculated signal** To define a calculated signal, drag and drop any input signal from the Variable Explorer or an instrument or an existing calculated signal into the formula definition field. To complete the formula, enter mathematical operators via the keyboard. For more information, see "Defining Calculated Signals" below.
C	**List of all mathematical operators** When defining a calculated signal, you can select a mathematical operator in this list and move it into the formula definition field on the right side of the editor. Alternatively, you can type the desired operation name or symbol directly into the formula definition field.
D	**More information about the mathematical operators** When you select a mathematical operator in the list above, this field displays additional information about the meaning and usage of the operator. The same detailed information is shown in the Formula Definition field, when hovering in edit mode onto the function name.

Note

Calculated signals imported from MDA V7 may behave slightly differently in MDA V8. For more information, see "Import Calculated Signals from XDA Files: Differences between MDA V7 and MDA V8 " on page 197.

7.2.1 Defining Calculated Signals

For a general overview of the window, see "Calculated Signals" on the previous page.

To define a calculated signal

The following steps describe the drag and drop usage. It is also possible to directly type into the formula editor field, and a drop-down list offers the available signals or mathematical operators.

To see how you can create and clone a calculated signal, watch our video **Creating Calculated Signals**.

1. If this is the first calculated signal to be defined, you can directly start with step 2. If you have already defined a calculated signal, click .

   By default, the name of the new calculated signal is set to "CalculatedSignal". If this name is already in use, it is extended by an increasing number.

2. In the **Name** field, you can enter another name instead of the default name.

   Note

   Valid characters are a-z, A-Z, 0-9, underscore, point, and square brackets. Square brackets must always occur in pairs. Other characters or blank space are not allowed.

3. Optionally, you can enter a unit in the **Unit** field.
4. Drag and drop one or more input signals into the **Formula Definition** field.

   Note that enumeration signals can be used as input signals only in combination with the following functions:

   - `Raw()` function for mathematical calculations using the decimal implementation value
   - `ToString()` function for comparing the strings of different enumeration signals.

   For each input signal, the file identifier and the short name of the signal are displayed. If several signals with the same short name exist, information about the device and/or the raster is additionally displayed to distinguish the signals.

   If an input signal contains an error or warning, a corresponding icon is displayed in front of the file identifier. Note that you can save a calculated signal even if an input signal contains an error.

5. Connect the signals with mathematical operators.

   You can drag a mathematical operator from the **Toolbox** field into the **Formula Definition** field. The **Help** field displays additional information about the meaning and usage of the operator.

   Tip: To quickly navigate through the formula, use the keyboard buttons.

6. Open the **Output Options** and define the following parameters:

   i. **Rate**: Define the rate in which a new sample of the calculated signal shall be created.

      - **Combined Raster (Merge Raster)** (by default): For every new sample of any input signal, a new sample for the calculated signal is created. This means, different input signal time rasters are merged.

- **Fixed**: The calculation of a new sample is done in a periodic raster independently from the rasters of the input signals.
- **Same as Signal**: The calculated signal has the same time stamps as the selected input signal. You can only select available input signals. Thus, for example, if the selected input signal has been deleted from the configuration, an error message is displayed.

ii. **Type**: Select the data type of the calculated signal. A time offset is applied to the result of calculation only. It can be a negative or positive value with up to 6 decimals. A time offset allows to shift individual signals and thus synchronizes their time line with other signals from the same measure file.

- **Automatic** (by default): The program tries to detect whether the result is a logical (Boolean) signal or not.
- **Boolean**
- **Double**

iii. **Time Offset** for the calculated signal.

7. Click **Save**.

   If the calculation is not valid, you cannot save the calculated signal. An error message is displayed and the exact location of the error is highlighted. After saving, the new calculated signal is available in the **Variable Explorer** and in the calculated signal list.

8. To use the calculated signal, drag and drop it onto an instrument. For calculated signals, a square root symbol is shown as file identifier.

> **Note**
>
> The copy and paste function is currently not supported in the **Formula Definition** field.

7.2.2 Managing Calculated Signals

For a general overview of the window, see "Calculated Signals" on page 172.

You can perform the following actions:

- "To edit a calculated signal" on the next page
- "To rename a calculated signal" on the next page
- "To delete a calculated signal" on the next page
- "To duplicate a calculated signal" on the next page

For more information on how to import calculated signals from an XCS file exported from INCA or MDA V7, see "Importing Calculated Signals via XCS Export File" on page 41.

To see how you can create and clone a calculated signal, watch our video **Creating Calculated Signals**.

To edit a calculated signal

1. In the calculated signal list on the left of the window, select a calculated signal.
2. Click .
3. Change the data. For more information about the possible values of the entry fields, see "To define a calculated signal" on page 173.

To rename a calculated signal

1. In the calculated signal list on the left of the window, select a calculated signal.
2. Click .
3. Enter a new name.

   Valid characters are a-z, A-Z, 0-9, underscore, point, and square brackets. Square brackets must always occur in pairs. Other characters or blank space are not allowed.

⇨ After saving, the new name is displayed in all instances where the calculated signal is used.

To delete a calculated signal

1. In the calculated signal list on the left of the window, select a calculated signal.
2. Click .

⇨ The calculated signal is deleted but the assignment of the calculated signal to instruments is kept.

To duplicate a calculated signal

1. Select a calculated signal in the list on the left of the window.
2. In the context menu, select **Clone calculated signal**.

⇨ A new calculated signal with the given data from the selected signal is filled in the form. The name is based on the selected signal.

7.2.3 Examples for Calculated Signals

7.2.3.1 Extracting Bits or Bit Fields from an Integer

To extract individual bits see "Extracting Bits from a Signal or Elements from an Array" on page 130.

You can perform the following actions:

– "To extract a single bit from an integer" on the next page
– "To extract a bit field from a signal" on the next page

<u>To extract a single bit from an integer</u>

1. Shift the integer such that the bit of interest is in the 0 position.
2. Apply a bit-wise AND with 1 to isolate the individual bit.

```
singleBit = (inputsignal >> BIT)& 1
```

<u>To extract a bit field from a signal</u>

1. Shift the input signal such that the least significant bit (LEAST_SIGNIFICANT_BIT) of the bit field that you want to extract is in the 0 position.
2. Isolate the NUMBER_OF_BITS of the bit field by applying a bit-wise AND with a bitmask.

```
bitfield = (inputsignal >> LEAST_SIGNIFICANT_BIT) & ~(~0
<< NUMBER_OF_BITS)
```

7.2.3.2 Calculating RMS

For alternating current wave forms (e.g. sine wave) there are several ways of characterizing the voltage. One is the RMS (root mean square) which is the equivalent DC voltage that would have the same heating effect on a resistor. The calculation of RMS involves taking the average voltage over some time range. To generate a low noise result it is important to align these time ranges with the cycles of the wave form. This can be achieved by the following calculations.

- An example sine wave (use periodic 100 ms raster):

```
voltage = sin(Master())
```

- Detecting the zero crossings (used to finish the integral):

```
positive = voltage > 0
windowDetect = positive && !State_Register(positive, !1)
```

- Also provide the detected signal delayed by one sample (used to restart the integral):

```
windowDetect2 = State_Register(windowDetect, !1)
```

- Remember start of window on each crossing:

```
windowStart = Latch(Master(), windowDetect2)
```

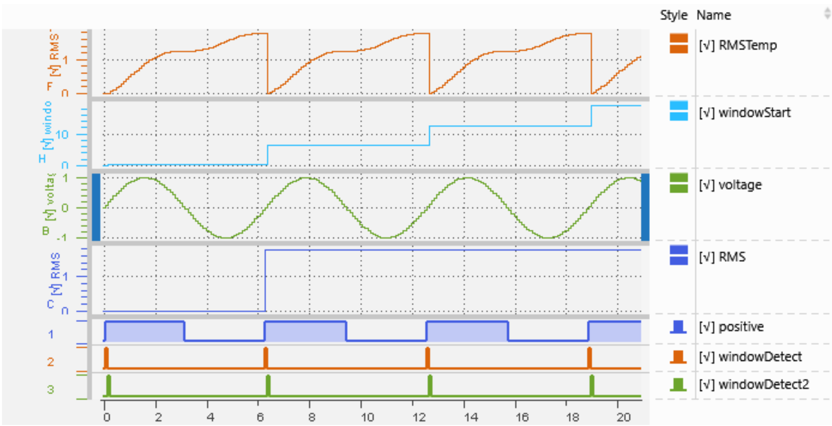
- Calculate the RMS since the last zero crossing:

```
RMSTemp = sqrt(Rolling_Accumulate_Integral(voltage ** 2,
windowStart))
```

- Hold the RMS value from the end of the last window:

```
RMS = Latch(RMSTemp, windowDetect)
```

Overall the calculated RMS will be one period delayed.



7.2.3.3 Using Enumeration Signals

As the physical value of an enumeration signal is a string, usual mathematical operations cannot be performed directly using the pure enumeration signal as input signal in formulas. To use the respective decimal value defined by the conversion formula instead of the string value, the Raw() function is provided. To compare the physical string values of two different enumeration signals, the function ToString() can be used.

- Obtaining the decimal value of the enumeration signal:

  `Raw(Enumeration)`

- For checking whether the value of an enumeration signal equals or exceeds a specific limit, the corresponding decimal value as given in the conversion formula of the enumeration signal is used for comparison:

  `Raw(Enumeration) >= [numeric decimal value]`

- Comparing the strings of two enumerations signals for being equal:

  `ToString(Enumeration_1) = ToString(Enumeration_2)`

- Counting the number of changes of an enumeration signal. First the positive and the negative edges are detected, then these are counted:

  `Edges = Gradient(Raw(Enumeration)) != 0`

  `Accumulate_Prefix_Sum(Edges)`

- Detecting the duration how long (in seconds) a Boolean enumeration signal stayed in state True:

  `Accumulate_Prefix_Integral(Raw(Boolean_Enumeration))`

  Provided that the enumeration definition is 0 = False, and 1 = True.

- Detecting the duration how long (in seconds) an enumeration signal stayed in a specific state, the decimal value of this state is detected and then integrated:

  `Accumulate_Prefix_Integral(Raw(Enumeration)= [Value of state])`

7.2.3.4 Applying Calculations to Specific Samples

There are different methods how a calculation can be limited to specific samples.

First, the use of the `If-Then-Else` function is a powerful way to define the true and the false case behavior.

Second, the `If-Then-Else` function also allows to exclude specific samples, so that these are skipped in the calculation.

- "If-Then-Else basics" below
- "If-Then-Else within another calculation" below
- "If-Then-Else to ignore samples" on the next page
- "If-Then-Else to ignore samples with Not a Number (NaN) state" on page 182

If-Then-Else basics

The `If-Then-Else` function requires a condition which results in `True` and `False`.

`Then` and `Else` represent the two behaviors namely for the True case (`Then`) and the False case (`Else`).

`Result = Condition ? True Case : False Case`

The three parts (`Condition`, `True case`, `False case`) in the `If-Then-Else` structure are expressions which can use the same input signals or different ones. The `True case` and `False case` can be complex formulas, single input signals, or constants.

EXAMPLE

You want to calculate the power of the engine, but only in case the car is driving uphill.

Uphill condition = `Gradient (Altitude) > 0`

True case = `EngineSpeed * Load`

False case = `0`

`Power_Uphill = Gradient (Altitude) > 0 ? EngineSpeed * Load [* Factor] : 0`

(Depending on the expected unit for `Power` and the units given for `EngineSpeed` and `Load`, an additional factor for unit conversion must be included.)

If-Then-Else within another calculation

If the `If-Then-Else` function is embedded in a more complex calculation, typically either the `True case` or the `False case` should be a neutral value for the overall calculation. The most simple neutral value is 0 for an addition or subtraction, while it is 1 for a multiplication or division.

EXAMPLE

1. You want to know the total amount of CO_2 emissions for the speed range of 40 to 80 km/h. By integrating the CO_2 emission gas flow (in g/s), you can calculate the amount.
   - Condition for speed range: `(Speed > 40) AND (Speed <= 80)`
   - True case: integration of CO_2 emission gas flow
   - False case: neutral value for integration, i.e. 0

   `CO2_Amount = Accumulate_Prefix_Integral ( ( (Speed > 40) AND (Speed <= 80) ) ? CO2_Emission : 0 )`

   `Accumulate_Prefix_Integral` is the name of the Integral function starting with the first sample.
2. You want to calculate the distance driven on a test trip in a speed range of 40 to 80 km/h only.

   The distance can be calculated as Integral of the Speed signal. Only the sections within the given speed range shall be considered.
   - Condition for speed range: `(Speed > 40) AND (Speed <= 80)`
   - True case: current `Speed` value is used for integral calculation
   - False case: a neutral value for the integral calculation, i.e. 0

   `Distance = [Factor *] Accumulate_Prefix_Integral ( ( (Speed > 40) AND (Speed <= 80) ) ? Speed : 0 )`

   Assuming that the unit for the speed is km/h, and the distance shall be in kilometers, you must apply a conversion factor = 1 / 3600.
3. You want to know for how long the speed was in the range of 40 to 80 km/h

   `Duration = Accumulate_Prefix_Integral ( (Speed > 40) AND (Speed <= 80) )`

   As the condition itself results already in a value of 1 or 0, a pure integral operation is sufficient, and the `If-Then-Else` function must not be used.

   By the mentioned formula the duration is given in seconds.

If-Then-Else to ignore samples

In all the above listed examples, the `If-Then-Else` function allows to calculate a result based on a specific condition. There is a calculation result for every sample, including those which do not fulfill the condition. By the appropriate selection of the neutral value, there is no effect for the calculation result, but still a value for every time stamp, i.e. every input sample. This can be seen, for example, by the curve drawn in an oscilloscope, which is a continuous line.

In some cases it is not so simple to find a neutral value for the calculation result. Then it is helpful if you can ignore samples for the calculation. This is the case in an average calculation where you have to ignore samples as there is not an appropriate neutral value.

To exclude samples completely, i.e. that there is no result of the calculation, you can use the `No Value` function. Actually, the `No Value` function does not delete samples, but sets a flag to the value that shall be ignored.

There are two ways of creating samples with a `No Value` flag.

EXAMPLE

1. You want to get statistical data for the speed range of 40 to 80km/h. To eliminate the samples with a different speed these get the `No Value` flag assigned.
   - Condition for speed range: `(Speed > 40) AND (Speed <= 80)`
   - True case: keep current `Speed` value as it is
   - False case: set the `No Value` flag for the sample

   ```
   Selected_Samples = (Speed > 40) AND (Speed <= 80) ?
   Speed : NoValue (0)
   ```

   When you add the `Selected_Samples` signal to a Statistical Data instrument, only samples in the defined speed range will be used as basis for the statistics.

   When you assign the signal to an oscilloscope, the curve is limited to the sections in which samples within the defined speed range exist.

   Note: The term `NoValue ( 0 )` means that a sample with value 0 and the `No Value` flag will be set.

or

   As a frequent use case is to ignore samples for a calculation, there is a separate function to assign the `No Value` state to samples.

   In contrast to the former definitions, you must define the condition in a way that it is clear which samples shall be excluded.

   - Condition for speed range to be excluded: `(Speed <= 40) Or (Speed > 80)`

     (In contrast to the True condition above)
   - Function to assign the `No Value` state for specific samples (outside 40 - 80 km/h):

     ```
     SetNoValueStatus ( Speed, ( (Speed <= 40) Or
     (Speed > 80) )
     ```

   This is equivalent to

`Selected_Samples = (Speed > 40) AND (Speed <= 80) ? Speed : NoValue (Speed)`

In this case, the `NoValue` (signal) causes that the original signal value of `Speed` is kept, but gets the `No Value` flag assigned.

This selection of samples for the speed signal can be used as input for e.g. an Average calculation (since start of the recording).

`Average_Speed = Accumulate_Prefix_Average ( Selected_Samples )`

2. The `No Value` function can also be used to suppress drawing samples in an oscilloscope, for example for the distance calculation mentioned above.

   `Interrupted_Distance_Curve = (Speed > 40) AND (Speed <= 80) ? Distance : NoValue (0)`

   With `Distance = [Factor *] Accumulate_Prefix_Integral ( ( (Speed > 40) AND (Speed <= 80) ) ? Speed : 0 )`

   Note: Although the calculation

   `Accumulate_Prefix_Integral ( ( (Speed > 40) AND (Speed <= 80) ) ? Speed : NoValue (0) )`

   would also show a curve with the same gaps as the `Interrupted_Distance_Curve` signal, the result would not be as you might expect it.

   This is an effect of the Integral function: If there is no sample (or a sample with `No Value` flag), the integral uses the last available sample value for the whole time range until the next sample is available.

If-Then-Else to ignore samples with Not a Number (NaN) state

In some cases your recorded signal includes already samples with `Not a Number` (NaN) value.

Typically such specific samples prevent a subsequent calculation, and again you need a method to exclude such samples from your calculation.

EXAMPLE

1. To eliminate a `NaN` sample, you need to detect first a `NaN` sample, and then assign the `No Value` flag instead.

   Condition for Not a Number: `InputSignal != InputSignal`

   As `NaN` prevents calculation for a sample, the condition is `true` if the input signal has at that point in time a `NaN` sample.

   You can directly replace the lengthy `If-Then-Else` function with the shorter `SetNoValueStatus` function:

```
InputSignal_without_NaN = SetNoValueStatus (InputSig-
nal, InputSignal != InputSignal)
```

You can use the `InputSignal_without_NaN` for calculations with history (like `Average`, `Minimum`, `Maximum`) and still get a result in which the `NaN` samples are excluded.

2. Excluding `NaN` samples from integral calculation

   As already mentioned above, the `No Value` state effects an integral calculated in an undesired manner.

   Therefore, to exclude `NaN` values from an integral calculation, you must use the `If-Then-Else` calculation, so that the `NaN` samples get the neutral value for the integral calculation, namely 0.

```
Integral_excl_NaN = Accumulate_Prefix_Integral (
InputSignal != InputSignal ? 0 : InputSignal)
```

7.2.4 Calculated Signals Details

7.2.4.1 Data Types

The following table gives an overview of the data types supported in calculated signals:

Name	Bits	Supported
Signed Integer	8, 16, 32, 64 bit [1]	Yes
Unsigned Integer	8, 16, 32, 64 bit [1]	Yes
IEEE Floating Point	32, 64 bit	Yes
Boolean		Yes
String		No[2]
Enumeration		No[2]
Mixed (combination of numeric and enumeration)		No[2]
Array		No
Event		No[2 3]

[1] Other bit sizes in input files are supported but the data is converted to the next higher available size.

[2] The type can be displayed but is not supported in calculated signals.

[3] Events are currently treated as strings.

Data Type Conversion

Values of one type can sometimes be converted to a different type according to the following table:

Conversion	Input Type	Output Type	Result
Convert_ ToBool(x)	Numeric	Boolean	True if x `!= 0`; false otherwise
Convert_ ToBool(x)	Boolean	Boolean	x
Convert_ ToDouble(x)	Numeric	64 bit float	The closest number to x representable as a 64 bit floating point number
Convert_ ToDouble(x)	Boolean	64 bit float	1 if x is true; 0 otherwise

Data Type Deduction

Input signals have a defined type as can be seen in the type column of the Variable Explorer. The types of intermediate results of a calculation formula are chosen automatically depending on the input types and the applied operations. This process is called type deduction.

Internally, a calculation operation is represented by multiple implementations each with specific types for the inputs and outputs of the operation: the type combinations. Type deduction works by choosing a type combination that matches the inputs. Inputs may be converted to similar larger types if no exact match can be found:

- Signed integer to larger signed integer
- Unsigned integer to larger unsigned integer
- Any integer to 64 bit float
- Boolean to any numeric

Examples:

- Bit operations exist for signed and unsigned integers of all sizes. Type deduction chooses the smallest size that is greater or equal than all inputs.
- Arithmetic operations (plus, times, …) only exist in double so type deduction will always convert inputs to double.

7.2.4.2 Formula Syntax

Formulas are entered in textual form. A formula consists of the following types of elements:

- Literals: a constant value directly part of the formula, e.g. 1
- Signals: a reference to an existing signal, represented by a box
- Operators: a non-alphanumeric sequence of characters representing a calculation operation e.g. +
- Functions: e.g. sin()

Literals

A literal is the textual representation of a constant in a formula. The following types of literals can be used:

Type	Example
Decimal integer number	123
Decimal floating point number	1.23
Hexadecimal number	0x1FA, 0x1fa
Binary number (e.g. bitmask)	0b1001010

- "Integer Numbers" below
- "Floating Point Numbers" below
- "Boolean" on the next page

Integer Numbers

Integers are usually specified as decimal numbers (base 10) using the digits from '0' through '9'. Alternative number systems (bases / radices) can be used by using one of the following prefixes:

Prefix	Base	Name
0b	2	Binary
0x	16	Hexadecimal

Binary only uses the digits '0' and '1'. Hexadecimal uses digits and letters 'A' through 'F'. The letters in the number are not case sensitive.

Examples:

```
17 = 0x11 = 0b10001
12 = 0xC = 0xc = 0b1100
```

Note: Integer literals are currently implicitly treated as floating point numbers.

Floating Point Numbers

Floating point numbers use a '.' as decimal separator and optionally allow scientific notation. The general format is:

```
+/- integer '.' fraction 'e' +/- exponent
```

Notes:

- +/- is the character '-' or '+' indicating the sign and is optional.
- Integer, fraction and exponent are positive integers.
- Either integer or fraction can be left out.
- The exponent starting at the 'e' is optional.
- Spaces within the number are not allowed.

Examples:

- `2`
- `-1.5`
- `1e3 = 1000`
- `3.7e-1 = 0.37`

Boolean

True and false literals are currently not supported. As a workaround the not operation can be used to create Booleans:

Boolean	Workaround
false	!1
true	!0

Example:

When delaying a boolean signal by one sample, it is necessary to specify the initial value as a boolean:

```
State_Register(voltage > 0, !1)
```

Signals

A signal is a sequence of samples each having a value. The signal's values can be accessed in a formula by adding the signal to the formula. It will be displayed as a box with the name of the signal.

Implicitly each signal also has a time stamp for each sample. Some operations like the Integral use the time stamps to react to the passing of time. To explicitly access the time in a formula use the `Master()` function.

Examples:

- `Delta(signal)` calculates the difference of consecutive signal values.
- `Delta(Master())` calculates the difference of consecutive signal time stamps, e.g. `Delta(Master())+0*signal`.

Operators

Operators are a compact way of specifying frequently used calculation operations like addition or multiplication. When multiple operators are used the order in which they are evaluated needs to be defined. The order can be explicitly specified by using parentheses. If there are no parentheses, the order is determined implicitly using the precedence of the operators. Higher precedence operators are evaluated first, followed by lower precedence. Within the same precedence group, operations are evaluated from left to right or from right to left depending on the operator.

Examples:

- `a + b + c = (a + b) + c`
- `a + b * c = a + (b * c)`
- `- - a = -(-a)`
- `cond1? val1: cond2? val2: val3 = cond1? val1: (cond2? val2: val3)`

The following table shows the precedence of operators. The operators on the first line have the highest precedence. Operators on the same line have the same precedence and the evaluation direction is specified as left-to-right or right-to-left.

Operators	Arguments	Evaluation
- ~ !	unary	Right-to-left
**	binary	Left-to-right
* / %	binary	Left-to-right
+ -	binary	Left-to-right
< > <= >= ==	binary	Left-to-right
BIT_AND &	binary	Left-to-right
BIT_XOR ^	binary	Left-to-right
BIT_OR |	binary	Left-to-right
AND &&	binary	Left-to-right
XOR ^^	binary	Left-to-right
OR ||	binary	Left-to-right
?:	ternary	Right-to-left
,	binary	Left-to-right

For details of the operators, see the toolbox in the calculated signals editor.

7.2.4.3 Reduction

A reduction function is a function that takes a sequence of values and calculates a single result value, the reduction.

```
reduction = Reduce(value[1], …, value[n])
```

Examples:

- The sum of all values:
  `reduction = value[1] + … + value[n]`
- The number of samples:
  `reduction = n`
- The average of all values:
  `reduction = (value[1] + … + value[n]) / n`

A reduction behavior is a calculation operation, that internally uses a reduction function.

Example:

The rolling average operation applies the average at every sample position to the last length samples of the input to determine a new output sample.

```
output[i] = Average(input[i-length+1], …, input[i])
```

Here the reduction function "average" is used by the reduction behavior "rolling".

At the moment select combinations of behavior and function are made available as calculation operations.

The naming follows the pattern:

```
<Behavior>_<Function>
```

This means that for the rolling average the name is:

```
Accumulate_Rolling_Average
```

Ranges

A range is a time interval with a start and an end time. Ranges are used to represent subsets of samples to calculate a reduction function. A range includes all samples with time stamps greater than the start and less or equal the end, i.e. it is an interval that is open on the left and closed on the right.

Ranges can be used both as output of a calculation (e.g. Window_Signal) or as input of a calculation (Accumulate_Rolling). Ranges are not a separate data type but are encoded as a scalar signal where:

```
Value = start time
Time = end time
```

This means that the end time is implicit and cannot be selected as a separate signal. It can be accessed using the Master() function.

Examples:

- A range over the last 2 seconds can be created by
  `Master() - 2`
- Rolling average over the last 10 samples
  `State_Delay(Master(), 0, 10)`

Note: Due to the mapping of end time of the ranges to time stamps the end times have to be strictly monotonously increasing.

Reduction Behaviors

The following reduction behaviors can be used:

- "Accumulate_Rolling" below
- "Accumulate_Rolling(input, windowStart)" below
- "Window_Signal" on the next page
- "Window_Signal(input, limit)" on the next page
- "Accumulate_Prefix" on the next page
- "Accumulate_Reset" on page 191
- "Accumulate_Samples" on page 191

Accumulate_Rolling

Calculates the reduction over a moving window.

Syntax:

- ```
  result = Accumulate_Rolling_<reduction function>(input, range)
  ```

Arguments:

- T result: the reduction function applied to the given range
- T input: the signal to be reduced
- double range: a sequence of ranges

Note: T can be any type supported by the given reduction function.

Accumulate_Rolling(input, windowStart)

The Accumulate_Rolling behavior applies the reduction function to a moving window. The windowStart signal specifies a window (see "Ranges" on the previous page). For each range, the samples in that range from the input signal are reduced according to the reduction function and produce an output sample which has the same time stamp as the end of the range.

Examples:

- Rolling average over the last 2 seconds:
  ```
  Accumulate_Rolling_Average(input, Master()-2)
  ```
- Rolling average over the last 10 samples:
  ```
  Accumulate_Rolling_Average(input, State_Delay(Master(), 0, 10))
  ```

Notes:

- The memory usage of accumulate rolling behavior grows with the number of samples in the window. It is possible to use windowStart = 0, however each new sample will increase the memory. Depending on the input signal, this may lead to using significant amounts of memory.
- For correct functionality, the start times of the ranges must be monotonously increasing.
  ```

Window_Signal

Calculates a window with a given "size".

Syntax:

- `Window_Signal_<reduction function>(input, limit)`

Arguments:

- double result: ranges of input that have the given size of limit
- T input: the input signal that is reduced to determine the size of a window
- T limit: desired "size" of the calculated ranges

Note: T can be any type supported by the given reduction function.

Window_Signal(input, limit)

The Window_Signal behavior calculates for each input sample a range ending at that sample. The size and thus the start of the range is chosen such that when applying the reduction function to the values of the input in this range the reduction approximately equals the limit. More specific the smallest interval is chosen where the reduction is greater or equal the limit.

As an example, we can apply the Accumulate_Rolling to the result to see the actual accumulated values for the window.

```
result = Accumulate_Rolling_<function>(input, Window_Signal_
<function>(input, limit))
```

The result will be greater or equal the limit except at the start of the signal when there are not yet enough samples.

Example:

- Creating a moving window always containing at least 80 grams of CO_2 exhaust:

  `movingWindow = Window_Signal_Integral(CO2, 80)`

  The movingWindow can now be used to evaluate other signals normalized to the CO_2 exhaust.

Accumulate_Prefix

Calculates the reduction of a given signal from start to current sample.

Syntax:

- `result = Accumulate_Prefix_<reduction_function>(input)`

Arguments:

- T result: the reduction of signal from start to current sample
- T input : the signal to be reduced

Note: T can be any type supported by the given reduction function.

The Accumulate_Prefix reduction behavior accumulates the input samples with the given reduction function. The result is a signal with all the intermediate results, i.e. `result[i] = reduce(signal[1], …, signal[i])`.

Note: `result = Accumulate_Rolling(signal, -Infinity)` except that the memory usage is constant.

Accumulate_Reset

Calculates the reduction of a given signal from the last reset to the current sample.

Syntax:

- `result = Accumulate_Reset_<reduction_function>(input, reset)`

Arguments:

- T result: the reduction over signal since the last reset
- T input: the signal to be reduced
- bool reset: reduction is restarted when true

Note: T can be any type supported by the given reduction function.

The Accumulate_Reset reduction behavior accumulates the input samples with the given reduction function. The reduction is restarted when the reset input is true. The result is a signal with all the intermediate results, i.e. `result [i] = reduce(signal[k], …, signal[i])` where k is the index of the last time reset was true or 1 if it never was true.

Example: Accumulate_Reset_Maximum

Signal	Reset	Result
1	false	1
5	false	5
3	false	5
2	true	2

Accumulate_Samples

Calculates a rolling reduction over a given number of samples.

Syntax:

- `result = Accumulate_Samples_<reduction_function>(input, count)`

Arguments:

- T result: the reduction over the last count samples
- T input: the signal to reduce
- const int count: the number of samples to reduce

Note: T can be any type supported by the given reduction function.

The Accumulate_Samples function calculates the reduction over the last count samples before and including the current sample. At the beginning when there are less than count samples, all available samples are reduced.

Note: `Accumulate_RollingSample<R>(input, count) = Accumulate_Rolling<R>(input, State_Delay(Master(), -Infinity, count))`

Reduction Functions

A reduction function is a function that takes a sequence of values and calculates a single result value, the reduction.

```
reduction = Reduce(value[1], …, value[n])
```

In the border case n=0 the reduction function is applied to an empty sequence of values:

```
neutral = Reduce()
```

This defines a "neutral" element of the reduction.

A reduction function can be defined by repeatedly combining two values with a combine function. For example if we use the addition as the combine function we get the sum of the input values:

```
tmp[0] = 0
tmp[i] = tmp[i-1] + value[i]
reduction = tmp[n]
```

A reduction function can also be defined based on existing reduction functions.

Minimum

The Minimum reduction function returns the minimum of all input values:

```
combine(a, b) = min(a, b)
```

The Minimum is available for all numeric data types.

Maximum

The Maximum reduction function returns the maximum of all input values:

```
combine(a, b) = max(a, b)
```

The Maximum is available for all numeric data types.

Count

The reduction function returns the number of samples:

```
Count(values[1], …, values[n]) = n
```

Add

The Add reduction function returns the sum of all input values:

```
combine(a, b) = a + b
```

Average

The Average reduction function calculates the average over the input values. This is simply the sum of the samples divided by the number of the samples:

```
Average(values) = Add(values) / Count(values)
```

Integral

The Integral reduction function calculates the area under the signal curve from the time of the first sample to the time of the last sample selected. It assumes step interpolation, i.e. it is the sum over the rectangles extending to the right of each sample:

```
r_i = s_i * (t_i+1 - t_i)
```

Here s_i is the value and t_i is the time of the sample at index i. The rectangle of the last sample is not included in the sum as it extends past the end time of the range.

8 Troubleshooting

8.1 Access to the Online Help

The "Internet Options" settings can differ from machine to machine. The browser's behavior can also be affected by the permissions level of the user or any antivirus software installed on the machine. Therefore, problems with Internet Explorer can occur when trying to open the online help.

Try to open the online help with another browser, e.g., Mozilla Firefox. The HTM files are located in the following folders:

- `%ProgramFiles%\ETAS\MDA8.x\Documentation\Help`
- `%ProgramFiles%\ETAS\MDA8.x\Documentation\Glossary`

Alternatively, use this PDF document as reference. It contains all the instructions that are also available via the online help.

8.2 Support Function in Case of System Errors

While developing V8.7, the functional safety of the program was of utmost importance. To support us in finding the root cause and fixing the issue, provide the log files to ETAS. These files do not include customer data and all information sent are handled confidentially.

If a critical system error occurs, an exception dialog appears. You can perform the following actions:

- Click **Close MDA**.

  V8.7 is closed without sending any information.
- Click **Report and Close**.

  The latest ten log files are zipped. A new, filled out e-mail form is opened in your default e-mail client, the log files attached.

> **Note**
>
> Let us know, which steps you were about to perform with V8.7 before the error occurred.

To send problem reports using ZipAndSend

If you want to send a problem report later or in case V8.7 cannot be started anymore:

1. In the Windows Start menu, select **E › ETAS › V8.7 › ZipAndSend**.
2. Click **Create Report**.

   V8.7 generates automatically a report file and opens a new, filled out e-mail form in your default e-mail client, the report files attached.

Provide us the information what you have done in MDA or with your computer before the problem happened.

If you want to send a problem report in case V8.7 can be started:

1. Select the ribbon **Home**.
2. Click **ZipAndSend**.
3. Click **Create Report**.

When using the **ZipAndSend** functionality for reporting product defects, the log files included in the ZIP file may contain file paths on your system, i.e. the full path to a measure file used in V8.7. If you use personal data in your file paths (e.g. your user ID) and you do not want this data to be sent, you have to delete it manually from the log files.

9 Contact Information

Technical Support

For details of your local sales office as well as your local technical support team and product hotlines, take a look at the ETAS website:

www.etas.com/hotlines

ETAS offers trainings for its products:

www.etas.com/academy

ETAS Headquarters

ETAS GmbH

Borsigstraße 24	Phone:	+49 711 3423-0
70469 Stuttgart	Fax:	+49 711 3423-2106
Germany	Internet:	www.etas.com

10 Appendix

10.1 Import Calculated Signals from XDA Files: Differences between MDA V7 and MDA V8

Since MDA V8.3.3 it is possible to import calculated signals from XDA files. Because of several reasons, the results of the calculated signals in MDA V8 may be different to the ones in MDA V7. This document lists the differences in detail.

- MDA V7 and MDA V8 use different calculation engines for evaluating formulas: MDA V7 uses a Perl interpreter, while MDA V8 uses a C-like calculation engine. This may lead to different results (e.g. due to resolution). Internally MDA V8 uses different data types (e.g. int8, uint8, int16, boolean, double…) automatically, based on the calculation result (type deduction).
- For calculated signals, MDA V8 supports the types Boolean, Double and Automatic. MDA V7 in addition supports different integer types (uint16, sint32 …). During the import, the mode Automatic is applied for all Integer types. If you have selected an integer type in MDA V7 to ensure an integer result, MDA 8 may deliver float values.
- MDA V7 uses the decimal() function automatically for binary operations (e.g. in the function Binary_AND). MDA V8 will emulate this behavior by calling the Raw() function. If you use a calculated signal, MDA V8 does not do this and will use the physical value instead.
- When using the modulo (%) operator, MDA V7 uses the decimal value. MDA V8 uses the physical value.
- User-defined Perl scripts, which define new functions for calculated signals, are not supported. Calculated signals, which use such functions, are imported, but their formula shows an error.
- If you reference a signal in the raster settings, which is not part of the current formula, MDA V8 uses “Combined Rasters (Merge Rasters)”.
- Verbal conversion settings are not supported by MDA V8 and are ignored.
- Signals of type "Limit monitoring" are imported. However, the verbal conversion settings (and the associated messages) are ignored. The resulting signal type is set to Boolean (not string) in MDA V8. The signals are deleted from the oscilloscopes during the import. Please assign the calculated signal manually to a Boolean strip in an oscilloscope. You can also assign it to an Event List instrument. This will give a similar view than the Event strip in MDA V7.
- The functions TableMap1 and TableMap2 are not supported and show errors in MDA V8.

10.1.1 Constants

MDA V7	Migrated formula in MDA V8
BIRTHDAY	Not supported in MDA V8
DATE	Not supported in MDA V8
E	2.71828182845905
EPOCH	Not supported in MDA V8
G	9.80665
LOG2_E	1.44269504088896
LOG10_E	0.434294481903252
LOG_2	0.693147180559945
LOG_10	2.30258509299405
PI	3.14159265358979
PI_DIV_2	1.5707963267949
PI_DIV_4	0.785398163397448
ONE_DIV_PI	0.318309886183791
ONE_DIV_SQRT_2	0.707106781186548
SEC_PER_DAY	86400.0
SEC_PER_HOUR	3600.0
SEC_PER_MIN	60.0
SQRT_2	1.4142135623731
TWO_DIV_PI	0.636619772367581
TWO_DIV_SQRT_PI	1.12837916709551
TWO_PI	6.28318530717959

10.1.2 Standard Operations

Calculation results of imported calculated signals with standard operations have the following known issues:

- MDA V7 and MDA V8 have different values for the rint/RoundInt function. The rint function in MDA V7 has a specific "Asymmetric round half up" rounding convention, while MDA V8 has a "Symmetric round half up".

- In MDA V7 the Shift (>>, <<) and binary operations (&, I, ^) use the raw value of a signal. MDA V8 will emulate this by calling the Raw() function. However, this emulation does not work properly for nested calculated signals, i.e. when the argument is a calculated signal which has a measure signal inside.
- Calculated signals of integer type in MDA V7 will be converted into double type in MDA V8. This leads to differences in handling of arithmetical overflow. E.g. the value '-1' in MDA V7 is represented as '4294967295.00' which is ((uint32)'-1'), and in MDA V8 is real '-1.00'.

10.1.3 Operations of type "Single Bit"

What	MDA V7	Migrated formula in MDA V8	Comment
Single Bit	double((long(rint(signal)) >> shift_value) & and_value)	If a measure signal is used: **Raw(signal) >> shift_value & and_value** If a value is used: **value >> shift_value & and_value**	All measure signal arguments are wrapped by the Raw function.

10.1.4 Operations of type "Bitmask"

What	MDA V7	Migrated formula in MDA V8	Comment
Bitmask	double((long(rint(signal)) >> shift_value) & and_value)	If a measure signal is used: **Raw(signal) >> shift_value & and_value** If a value is used: **value >> shift_value & and_value**	All measure signal arguments are wrapped by the Raw function.

10.1.5 Operations of type "Limit Monitoring"

MDA V7 allows to define calculated signals which monitor if one or more signals exceed a predefined value. For each limit to monitor, a condition is defined, together with a message to be shown if the limit is not met.

For MDA V8 such an operation is migrated into a calculated signal of type Boolean. The associated message will not be migrated.

Example:

The XDA file for a signal contains three calculation rules:

1	MyLimitMonitor1?1 = ${'C:_Data\INCA-NG_Sample Files\Coldstart2.dat:DG0:CG0:N10'} >1000
2	MyLimitMonitor1?2 = ${'C:_Data\INCA-NG_Sample Files\Coldstart2.dat:DG0:CG0:N10'} <100
3	MyLimitMonitor = ${'MyLimitMonitor1?1'} + ${'MyLimitMonitor1?2'}

These formulas are converted into one calculated signal in MDA V8:

Name	MyLimitMonitor
Formula	(N10 > 1000) || (N10 < 100)
Type	Booelan

10.2 Behavior of Calculated Signals Depending on the Status Flag

In an MDF measure file, for sample additionally two flags are available: One for indication whether a value is available at a certain time stamp, and another which indicates whether a recorded value is valid or not.

Each of the flags can be True or False, therefore the following four combinations are possible:

Has a value	Value is valid	Description	Example
TRUE	TRUE	Regular sample	
FALSE	TRUE	No sample at this point	Time stamps before the first sample of the signal is available
TRUE	FALSE	Error sample with value	
FALSE	FALSE	Error sample with no value	Integer division by 0

For calculated signals the states of the flags of input signals are considered for the calculation result.

This can be summarized as follows:

- `No value` cases
  - The meaning is the same as if the sample (including time stamp) were missing completely.
  - The advantage is that with this flag a `No value` of one signal can be combined with other signals in the same group which still have a sample at the same point.
  - The result of a calculation on `No value` will also be marked as `No value` (unless the interpolation with other signals causes there to be a sample).
  - The state of stateful operations (like integral) will not update.
- `Error` cases (both with and without value)
  - If any input sample has an error (i.e. is invalid), the error is propagated to the calculation result.
  - The only exception is if the input sample does not matter, e.g. `true? 3: error` results in 3.
  - Errors are also propagated into the state of stateful operations.
  - Thus accumulating operations (like e.g. Integral) will stay on error as long as the error sample is part of the accumulation range.

10.3 Custom Operations

What	**MDA V7**	**Migrated formula in MDA V8**	**Comment**
Average	Average (signal)	Accumulate_Prefix_Average (signal)	Average since measure start.
AND	BinaryAND (signal, mask)	If a measure signal is used: **Raw(signal) & mask** If a value is used: **notsignal & mask**	All measure signal arguments are wrapped by the Raw function. Time range: From start
OR	BinaryOR (signal, mask)	If a measure signal is used: **Raw(signal) I mask** If a value is used: **notsignal I mask**	All measure signal arguments are wrapped by the Raw function. Time range: From start
XOR	BinaryXOR (signal, mask)	If a measure signal is used: **Raw(signal) ^ mask** If a value is used: **notsignal ^ mask**	All measure signal arguments are wrapped by the Raw function. Time range: From start
Const	Const (value)	value	The dialogue in MDA V7 restricts the raster selection to the fixed time raster. Time range: From start

What	MDA V7	Migrated formula in MDA V8	Comment
CountTimeLevel	CountTimeLevel (time, signal, value)	Accumulate_Prefix_Integral ((input = level) ? 1 : 0)	Time range: From start Counts how many times a signal value has been reached.
CountTime LevelTo Tolerance	CountTimeLevel ToTolerance (time, signal, min, max)	Accumulate_Prefix_Integral ((min < = signal) && (signal < = max) ? 1 : 0)	Time range: From start Sums up the time as long as a signal is within a certain tolerance level (`min, max`).
Debounce	Debounce (time, signal, risingDelay, fallingDelay)	Debounce (signal, risingDelay, fallingDelay)	Calculates a debounced version of `signal`. Time range defined by the edges of the input signals The calculation results might be different because Debounce in MDA V8 generates no edge if the input signal is non-zero since measurement start. MDA V8 assumes that the previous (unknown) sample has the same value as the first available sample. MDA V8 does not have an initial edge and the previous value is initialized as `NoValue`.
Delta	Delta (signal, count)	signal - State_Delay (signal, NoValue (0), count)	Delta over the last count samples: `signal(k) - signal(k - count)` Time range defined by sample parameter `count`. Signals of the type enumeration (VTAB) are not supported.

What	MDA V7	Migrated formula in MDA V8	Comment
DeltaT	DeltaT (time, signal)	Delta (master()) + 0*Raw (signal)	`0*signal` is required for applying the raster settings based on an input signal. Time range between the current and the previous measured value. Calculates `time(k) - time(k-1)`. V8.7 assumes the first value as `NoValue`.
Gradient	Gradient (time, signal, count)	(signal - State_Delay (signal, NoValue (0), count)) / (Master() - State_Delay (Master(), 0, count))	First derivative of the last `count` samples. Time range defined by sample parameter `count`. Signals of the type enumeration (VTAB) are not supported.
Integral	Integral (time, signal)	Accumulate_Prefix_Integral (signal)	Integral of `signal` since measure start.
LevelReached Count	LevelReachedCount (signal,level)	Accumulate_Prefix_Sum ((State_ Register (signal != level, !0) && (signal = level)) ? 1 : 0)	Counts how many times a signal value has been reached. Time range: From start
LowPassFilter_ ASCET_lib	LowPassFilter (time, signal, filterTime, startInput)	Filter_LowPass1 (signal, 1 / (2*PI*-filterTime))	Time range: From start
Maximum	Maximum (signal)	Accumulate_Prefix_Maximum (signal)	Maximum value of signal since measure start.

What	MDA V7	Migrated formula in MDA V8	Comment
MaximumOf 2Inputs	MaximumOf2Inputs (signal1, signal2)	Relation_Maximum (signal1, signal2)	Maximum of `signal1` and `signal2`. Time range: From start
MaxReset	MaxReset (input_signal, reset_signal)	Accumulate_Reset_Maximum (signal, reset › State_Register (reset, 0))	Calculates the maximum value of `input_signal`. The maximum value is reset at each positive edge of `reset_signal`. Time range: from start or the last positive edge of the `reset_signal`.
Minimum	Minimum (signal)	Accumulate_Prefix_Minimum (signal)	Minimum value of signal since measure start.
MinimumOf 2signals	MinimumOf2Signals (signal1,signal2)	Relation_Minimum (signal1, signal2)	Minimum of `signal1` and `signal2`.
MinReset	MinReset(input_signal, reset_signal)	Accumulate_Reset_Minimum (signal, reset › State_Register(reset, 0))	Calculates the minimum value of `input_signal`. The minimum value is reset at each positive edge of `reset_signal`. Time range: From start or the last positive edge of the `reset_signal`.

What	MDA V7	Migrated formula in MDA V8	Comment
PhaseShift	PhaseShift (signal, 0, count)	Delay (signal, 0, count)	Returns a previous value: `signal(k-count)` Time range defined by sample parameter `count`. MDA V8 returns `n/a` as long as not all values are defined.
Pulse11	Pulse11 (time, signal, duration)	Debounce (time, signal, 0, duration)	Time range defined by the parameter `duration`. Detects pulses given by the argument `duration`. MDA V8 assumes that the previous (unknown) values are the same as the first available sample.
RollingAverage	RollingAverage (signal, count)	Accumulate_Samples_Average (signal, count)	Calculates the average of signal over the number of samples defined by `count`. Time range defined by sample parameter `count`.
RSFlipFlop	RSFlipFlop (set_input, reset_input)	State_RSFlipFlop (set_input, reset_input)	Time range: From start RSFlipFlop with positive logic. Both parameters must have the same measure rate.

What	MDA V7	Migrated formula in MDA V8	Comment
SampleCounter	SampleCounter (signal)	Count (signal)	Time range: From start Returns the number of samples of the signal.
SumTotal	SumTotal (signal)	Accumulate_Prefix_Sum (signal)	SumTotal since measure start.
Threshold1	Threshold1 (l, u, s, a, b)	((l <= s) && (s <= u)) ? a : b	Threshold computation Time range: From start
Threshold2	Threshold2 (l, u, s, x)	(s < l) ? l : (s > u) ? u : x	Threshold computation Time range: From start
Threshold3	Threshold3 (l, u, s, a, b)	((l <= s) && (s <= u)) ? a : b	Threshold computation Time range: From start
Threshold4	Threshold4 (l, u, s, x)	Latch (x, (l <= s) && (s <= u))	Threshold computation Time range: From start
TriggerTrue1	TriggerTrue1 (signal)	signal && State_Register (!signal, !1)	Detects when `signal` is going from false to true. Time range between the current and the previous measured value.
Weighted Counter	WeightedCounter (signal, low, high, factor)	factor * Accumulate_Prefix_Sum ((low <= signal) && (signal <= high) && (signal = signal))	Counts the number of samples for which the signal is between `low` and `high`, and weights this number with `factor`. Time range: From start

10.4 Using Command Line Arguments

V8.7 only supports a basic set of command line arguments.

Command Line Arguments	Function
`mda.exe -help`	Displays the command line arguments.
`mda.exe -restoreLayout`	Restores the default docking windows layout.
`mda.exe -openConfig:"<XDX File Path>"`	Loads a configuration from a file.
`mda.exe -addMf:"<Measure File Path>"`	Adds measure files to the active configuration. If no configuration is open, a new one is created.
`mda.exe -addFile`	Adds files to the active configuration. Supported are any kind of supported file formats (e.g. LAB files). If no configuration is open, a new one is created.
`mda.exe -addOrReplaceMf:"<Measure File Path>"`	Adds a new measure file to or replaces an existing one in the active configuration. If no configuration is open, a new one is created.
`mda.exe -import:"<File Path>"`	Opens and imports supported file formats.
`mda.exe -importXDA:"<XDA File Path>"`	Opens and imports an XDA configuration file. Alternatively, a ZDX configuration file can be opened.

Command Line Arguments	Function
`mdfextract.exe`	Exports only signals of data type Event from an MDF V4.x file into another MDF V4.x file. Can be used as an argument of `mdf-convert.exe` and includes Event signals into a target export file. The file is located at `%Program Files%\ETAS\MDA.x.x.x\McdCore`
`mdfconvert.exe`	Converts measure data into another file format located at `%Program Files%\ETAS\MDA.x.x.x\McdCore` Can be used in combination with `mdfextract.exe` to include Event signals from an MDF V4.x file.
`mdfcombine.exe`	Combines multiple measure files into one measure file. The file is located at `%Program Files%\ETAS\MDA.x.x.x\McdCore` To see how you can merge multiple measure files into one combined measure file, watch our video **Merging of Measure Files**.
`mdf4indexing.exe`	To add an ASAM standard-conform index to an existing measurement file in MDF V4 format. Indexing is beneficial for a faster drawing of signal curves in MDA's oscilloscope.

11 Glossary

A

A2L file

An A2L file is a standard file format specified by the ASAM MCD-2MC (ASAP2) workgroup. It describes the interfaces, measurements, and parameters of an Electronic Control Unit (ECU).

Absolute Value Bar Chart

The Absolute Value Bar Chart allows you to get an overview of the signal values and and to identify the signals which exceed a lower or upper limit at the given point in time.Each signal value is represented by a vertical bar. The signals are sorted alphabetically by the signal names.

AFF file

An AFF file stands for Associated File Format. It is shown in the File Explorer as a combined entry for a bus trace file and a bus description file.

Analog signal

As opposed to a discrete signal: a signal which can take any value out of a value range.

Anchored

Setting by which a cursor in the oscilloscope has a variable time stamp and a fixed screen position, i.e. the cursor remains always in the visible area.

Array

Data type consisting of any number of identical individual elements. The conversion formula for the array applies equally to each of the elements.

ASCII file

ASCII (American Standard Code for Information Interchange) stands for a textual measure file format. It contains one timeline and for each signal at every time stamp a value, if needed an interpolated one.

B

BLF File

The BLF format (Binary Logging Format) is a binary file format. It is often used to record data from automotive Bus systems like CAN, LIN and Ethernet.

Boolean signal

As opposed to analog signal: a discrete signal for which exactly two specific values are defined.

Bound

Connection line used to define a region in a scatter plot.

C

Calculated signal

Virtual signal resulting from the mathematical and/or logical combination of one or more input signals and/or constants.

Calculations

View used for creating and maintaining calculated signals and function instances.

Calibration variable

Variable type which can be modified by a user or an algorithm. These variables will be used within a control system to define a certain behavior.

CAN FD

CAN with Flexible Data rate. CAN FD is based on the CAN protocol as specified in ISO 11898-1. CAN FD is able to achieve an effective data-rate by allowing longer data fields.

CDF file

CDF stands for Calibration Data Format (ASAM standard). It stores calibration parameters of different data types, the physical values and unit. CDF files is an XML-based format, which can be easily validated, edited, imported and exported by calibration tools and XML editors.

Configuration

Item that combines data from a data source, meta data (descriptions and comments) and visualization (layers, instruments, variable selection, property settings).

Configuration Manager

Hierarchical view of a configuration and its elements, also providing management functions such as copy, paste, delete, create new, rename.

CSV file

A comma-separated values (CSV) file is a delimited text file that uses a comma to separate values. Each line of the file is a data record. Each record consists of one or more fields, separated by commas.

Cursor

Part in the graphical view of the oscilloscope to analyze y values, differences to other cursors, and time values.

Curve

A curve is the graphical representation of two physical quantities. The curve is displayed as a line in a two-dimensional orthogonal coordinate system.

D

DAT file

This file is a binary file format for measurement data (MDF). This format is used to store, exchange and analyze measure data for automotive system development purposes. MDF files contain so-called meta-information like user, company, project, file comments etc. Typically for MDF format version V3.x the file extension *.dat is used. For MDF format version V4.x, the file extensions *.mdf or *.mf4 are used.

Data representation

Physical, binary or hexadecimal representation of a signal.

Delta Bar Chart

The Delta Bar Chart instrument provides a quick overview of the deviation from the average value of many signals in parallel. The deviation of each signal is shown as vertical bar wherein the zero line represents the average of all signal values for the defined point in time. Additional ... for the minimum and maximum deviation are displayed to facilitate the overview.

Discrete signal

As opposed to analog signal: a signal for which only a defined number of specific values is possible.

Display Name Rules

View used for creating and maintaining rules for shorten long variable names.

DXL file

DXL stands for a textual measure file format. It contains only real data, i.e. only data that has actually been recorded, and no interpolated data.

E

Endpoint

Point by which the shape of a bound in the scatter plot is defined.

Enumeration

Data type which is based on a verbal conversion formula (i.e. certain value ranges are mapped to certain output strings).

Event

A point in time at which a change or a specific situation happened. Events can be recorded in a measure file (e.g. a pause event, a comment event, a calibration activity, etc.), or are detected during analysis phase by defining a condition which turns true or false.

Event List

Instrument type that is used to find certain events for analysis by evaluating a search condition.

F

File compression

Used to reduce the size of MDF measurement files. MDA compresses according to the compression methods defined in the ASAM standard for MDF V4.x file formats.

File Explorer

View listing all opened containers (e.g. MDA configurations) and the assigned files. This view is used for managing file assignments.

File formats

Measure file formats can be textual and binary. Textual formats are: ASCII, CSV, DXL, TSV, MRF. Binary formats are: DAT, MDF4. Label file fomat includes purely variable names and some meta information: LAB

File index

ASAM file reduction for MDF V4 (According to ASAM MDF V4 standard also known as 'File reduction'.). Indexed measurement files enable signal curves to be drawn more quickly in the oscilloscope because only the significantly smaller amount of indexed data needs to be read out rather than the individual measured values. The indexing process itself ensures that no outliers (, i.e. individual extreme values) are lost in the process.

FMU file

FMU stands for Functional Mock-up Unit (FMI standard). The file contains a model that can be used as predefined calculation model for a function instance.

Functions

In the Functions docking window predefined calculations can be used. By assigning measure signals to the expected inputs, the outputs are calculated and can be used for further analysis.

G

GPS Map

Instrument type that is used to display GPS data (i.e. GPS tracks constituted of the signals latitude and longitude) in a map.

H

Histogram

The histogram allows to graphically display the results of a simple classification of the samples from one signal as vertical bars.. For the classification the numeric value of the sample is used. Therefore only numeric scalar data types are supported.

I

Information Window

View providing additional information, e.g. on results of actions performed by the software (errors, warnings, logging details).

Instrument

Widget used for visualizing or editing data (e.g. oscilloscope or table).

Instrument Box

View listing all instruments that are available for use in the software.

Interpolation

Constructing intermediate values between measured sample points.

L

Label file

Text file containing a selection of signals, and optionally raster information, that can be used as a filter in an INCA Experiment Environment.

Layer

Tabs used for displaying instruments.

Layer preview

Icon for each instrument type at the bottom of the layer that allows to quickly select the instrument to be shown in the foreground.

M

Magnifier mode

Mode of the time slider in which the scale around the slider is magnified.

Map

A map is the graphical representation of a characteristic that depends on two physical quantities. The map is either displayed as a set of curves or as a surface in a three-dimensional orthogonal coordinate system.

Marker

Icon that shows the particular position in the track.

MDF file

This file is a binary file format for measurement data (MDF). This format is used to store, exchange and analyze measure data for automotive system development purposes. MDF files contain so-called meta-information like user, company, project, file comments etc. Typically for MDF format version V3.x the file extension *.dat is used. For MDF format version V4.x, the file extensions *.mdf or *.mf4 are used.

Measure file

File containing measure data (independent from the actual file format, e.g. MDF).

Measure variable

Placeholder for a variable which can usually not be influenced directly (i.e. calibrated); often used for showing the effects of calibration activities or environmental conditions. The placeholder does not include the actual samples.

MRF file

MRF stands for "Measure data refiller format". It is a textual (ASCII based) file format. In the data block, each line starts with an index (row counter) followed by the time stamp. The time line can be a merged raster from all signals rasters or, optionally, a resampled raster. At every time stamp a signal value is given, if needed with an interpolated value, which is the reason for "refiller" in the file format name.

N

No-Match status

Status of signals that are not contained in a measure file, e.g., after replacing the measure file. In this case, no measure data can be displayed.

O

Oscilloscope

Instrument type for displaying graphically the course of signals over time.

P

Phasor Diagram

A phasor instrument is used to visualize and analyze AC (alternating current) quantities such as voltage, current and phase angle. It displays these quantities as phasors, i.e. rotating vectors that represent waveforms in a simplified, static form.

Properties

View used for setting and maintaining the appearance and behavior of the instrument and axes properties.

Q

Quick access toolbar

Toolbar that contains a set of commonly used commands. By default, it is located in the upper left-hand corner.

R

Raster

Time interval between the individual samples, given in ms.

Ribbon

Set of toolbars located on different tabs.

S

Sample

Individual value of a signal measured at a certain time stamp.

Sample marker

Graphical elements which appear when individual measured value can be differentiated in an oscilloscope.

Scatter Plot

Instrument type displaying values for typically two variables for a set of data. The data is displayed as a collection of not connected points in order to detect correlation of signals or value distribution.

Selection wheel

Circle with different segments, which appears in drag and drop situations and allows to select one of various options, for example axis assignments when dropping signals in the oscilloscope.

Signal

Measure variable including the measured data (samples); usually stored in a measure file.

Signal curve

Line in the oscilloscope displaying the course of sample values.

Signal Distribution Chart

The Signal Distribution Chart is an instrument for statistical analysis. The chart is calculated based on the signal values at a given point in time. In the chart, the height of each bar represents the number of signals that fall within the defined value range of the respective bucket.

Signal list

List of signals that are assigned to an oscilloscope (later on maybe also for other instruments). The signal list is a separate widget showing signal values (e.g. values at the cursors) and some meta information. It can also be used for showing/hiding the signal.

Sortable List

The Sortable List allows to identify quickly the signals with the highest deviation from the average value or the highest resp. lowest absolute value.

Statistical Data

Instrument type displaying the statistical properties of numeric signals (e.g. average, minimum, maximum, and standard deviation)

String signal

String signal is, for example, a user comment during recording.

Strip

Part within an oscilloscope which its own y axes and signal assignment. All strips of an oscilloscope share the same x axis.

Synchronization cursor

Indicator used for marking the common time stamp when synchronizing different instruments.

T

Table

Instrument type that is used to inspect the precise value of a signal for a specific time stamp.

Time Axis

Horizontal scale in the coordinate system of an oscilloscope in yt representation which is used to determine the value of a point in the dimension of time.

Time Offset

View used for aligning data from different measure files with regard to time.

Time Slider

Control used for navigating on the time axis, including zooming functionality.

Title Bar

Header of an instrument showing the name of that instrument.

Toolbar

Section which provides access to software function via icons.

Tooltip

Information on a concrete object (e.g. a measure file or an icon) that is shown when hovering over that object.

Track

Path travelled by a vehicle shown as a route in the map.

TSV file

A tab-separated values (TSV) file is a delimited text file that uses a tabulator to separate values. Each line of the file is a data record. Each record consists of one or more fields, separated by tabulators.

V

Variable

Generic term for measure and calibration variables as well as calculated signals.

Variable Explorer

View listing all variables and signals that are available from the data sources. This list is used for selection and assignment to an instrument or as input for an activity (e.g. Calculated Signal, trigger condition, export...).

Video

Instrument type that is used to display the measure files recorded with the INCA Video-Integration add-on. It is recommended for combining a visual observation with the recorded data, and for analyzing in a synchronized mode the file data in relation to other measurements.

W

Working area

Part of the MDA V8 main window where visualization and analysis takes place (i.e. layers with instruments and signals).

Index

T

U

V

W

X

Z