BOSCH

WinDarab V7

Manual

Version 1.0 13/03/2019

Table of Contents

Desktop 88

Oscilloscope 91

General

WinDarab enables you to analyse and display logged data recorded by BOSCH Motronic®.
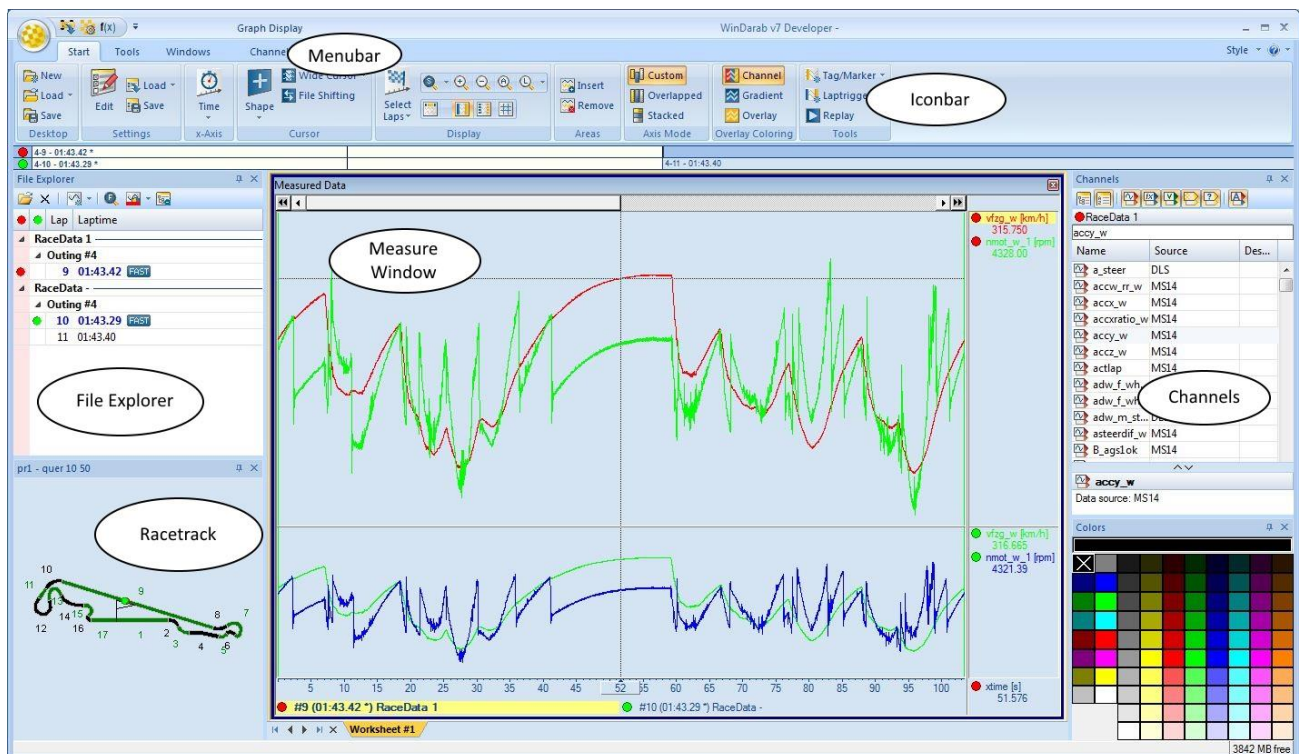
System Requirements

Hardware	Minimum	Recommended
	• Pentium 800 MHz processor • 128 MB RAM • 800 x 600, 256 colors video resolution • Video card with 3D hardware accelerator support for	• Pentium 1 GHz processor • over 256 MB RAM • 1024 x 768, 24-Bit color video resolution • Video card with 3D hardware accelerator support for OpenGL

OpenGL • Mouse	• 3-Button Mouse

Operating System and Software	MS Windows XP, Vista, 7
Known hardware problems	During development of the 3D functions we got several problems with certain graphic chips. Our developers tried to work solve these problems but still, we have known problems concerning certain chip sets and their drivers:
	Chipset S3 Savage IX/VX The Windows XP drivers are very slow if the mouse cursor is inside of a 3D drawing area. Additionally the driver causes sometimes blue screens.
	Our developers made best experience with graphic chips like ATI Radeon or Nvidia.

Symbols

The user manual employs the following symbols:

This symbol means that an action with the **right mouse button** should be performed at this time. The *action* itself is highlighted in bold and italicized. The **object** is highlighted in bold font.
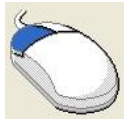
This symbol means that an action with the **left mouse button** should be performed at this time. The *action* itself is highlighted in bold and italicized. The **object** is highlighted in bold font.

This symbol means that an action with the **left and right mouse (or middle) button** should be performed at this time. The *action* itself is highlighted in bold and italicized. The **object** is highlighted in bold font.

This symbol means that input should be provided with the **keyboard**. The *action* itself is highlighted in bold and italicized. The **object** is highlighted in bold font.

This symbol means that something **special** must be noted.

Copyright / License Agreement

We thank you for choosing a Bosch Motorsport product. Please remember that it is the user's responsibility to comply with all applicable copyright laws.

Bosch Engineering GmbH

Motorsport

Robert-Bosch-Allee 1

74232 Abstatt

Germany

Liability

Glossary

Area	Refers to the horizontal division of a Oscilloscope in which channel signals are displayed.
Base Directory	A Base Directory represents a link to a particular folder on the hard drive (--> network). WinDarab saves data in sub-folders within a specified base directory.
Base Directory for Calculating Differences	Logged Data Directory that provides channel values (for the second channel) for calculating a difference.
Bit Area	See Area
CAN	**C**ontroller **A**rea **N**etwork
CanCard	Most commonly used hardware for data recording. A more recent device is CardMemory.
Channels	Separate data signals within a recording.
Conditions	A condition is a linked set of mathematical criteria that must be met in a specific sample in order for that data sample to be included in an evaluation.
Configuration	A particular arrangement of channels, which can be loaded and displayed in a Oscilloscope.
Darab Explorer	Darab Explorer is used in WinDarab to manage Racetrack and Logged Data Files.
Data Logger	Data Logging Device for Measurement Channels
Differences	The Differences Channel is a special channel that calculates values for the difference resulting from a comparison of the same channel in two different Logged Data Files.
DLS	**D**ata **L**ogging **S**ystem (consists of C55 Data Logger and MSI Devices)
DOS Format	DOS Darab, WinDarab's precursor, uses a data format that cannot be directly processed by WinDarab.
ECU	**E**ngine **C**ontrol **U**nit
Grid	Grid lines which can be displayed in a Oscilloscope.

Logged Data File   A Logged Data File contains all data recorded in a vehicle.

Racetrack or Racecourse   A Racetrack or Racecourse is the location where vehicle data recording has taken place. WinDarab assumes that the selected racetrack corresponds to the racetrack where data was actually recorded.

Reader   See CanCard

Supported Devices in WinDarab

WinDarab supports the following devices from Bosch Motorsport.

Driver Display Units (DDUs)

The display types from Bosch Motorsport are referred to as DDU (**D**river **D**isplay **U**nit).

DDU8

(TFT high contrast color display, up to 12 user-defined pages, internal data logging, sensor interface and CAN In-/Output)

Data Logger

Data Loggers supported by WinDarab:

C 40

(Used for data recording and storage of data transferred via CAN-bus from ECUs MS 3.x and MS 4.x)

C 40 Plus

(Used for data recording and storage of data transferred via CAN-bus from ECUs MS 3.x and MS 4.x. In addition, up to 15 sensors can be connected)

C 55

(Data Logger with Firewire interface)

C 60

(Data Logger with Ethernet interface)

Program Start

After starting WinDarab the user interface appears in which the last edited desktop (see chapter Desktop) or new desktop is opened. A new desktop will open if WinDarab is started the first time.

WinDarab offers a structured user interface with several components:
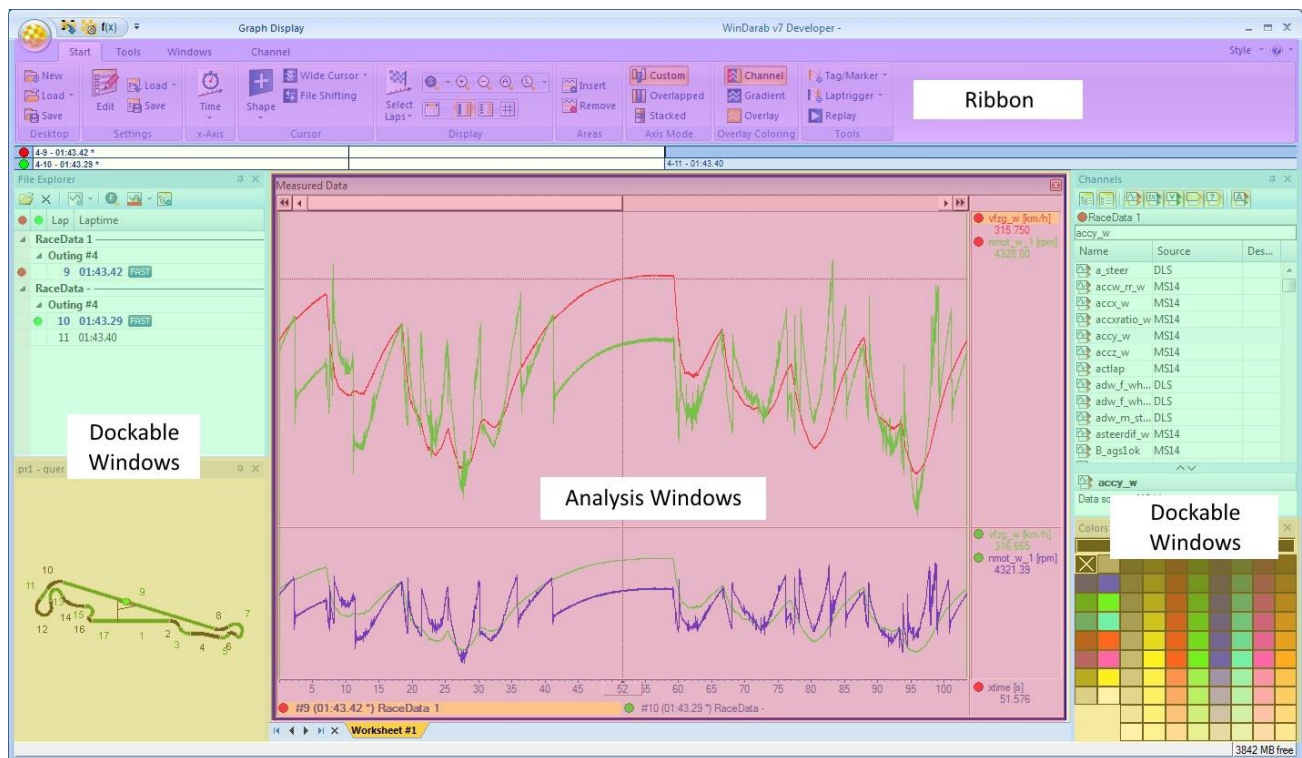
This is a possible view of composition of the WinDarab components. You can configure the WinDarab components so that the view complies with your operation methods.

Further description of the different components for the shown arrangement:
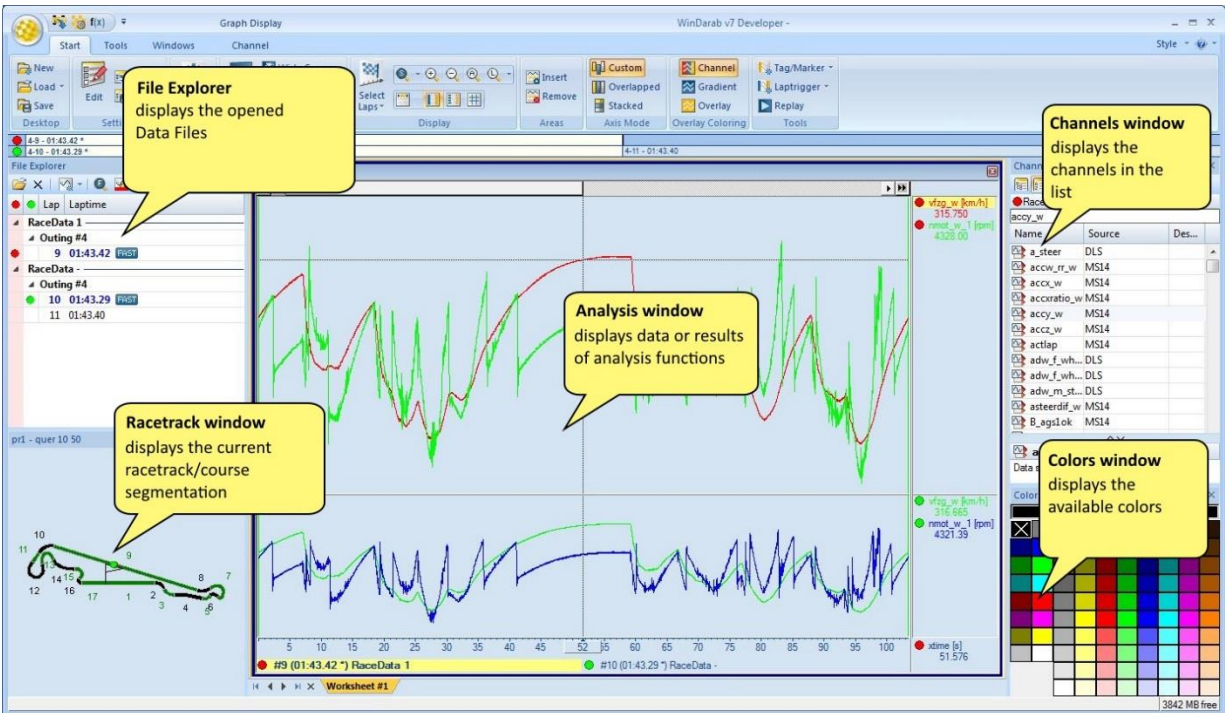
File Explorer window displayed upper left contains the list of opened logger data files.

Racetrack window placed bottom left shows the current racetrack and course segmentation.

Channels window arranged upper right displays a list of measurement, mathematical, internal or temporary channels according to the users' choice.

Color window arranged bottom right allows you to pick a color for an element (for instance a channel in an analysis window).

Oscilloscope, Analysis Window, Instrument Panel resides in the middle of WinDarab and represents data or results of the calculations that are performed.

Important Tips

Keyboard   Right next to some menu items you will also see keystroke combinations that allow you to perform functions from your keyboard. Using your keyboard rather than the mouse to execute frequently used functions saves a great deal of time.

Context Menu   The context menu of an element (window, channel etc.) contains a list of frequently used and useful functions. It is opened by clicking with the right mouse button on the element. In case the element has a zoom function, the context menu is opened by using the middle mouse button, or the left and right mouse buttons.

Changing the Style

You can change the style of your WinDarab7 by clicking on **Style** in the top right of the window and then chose a style from the dropdown-box.

License

If you want to update your WinDarab7-License, you can do this by clicking on **WinDarab-Button/ Tools/ Update license ...**
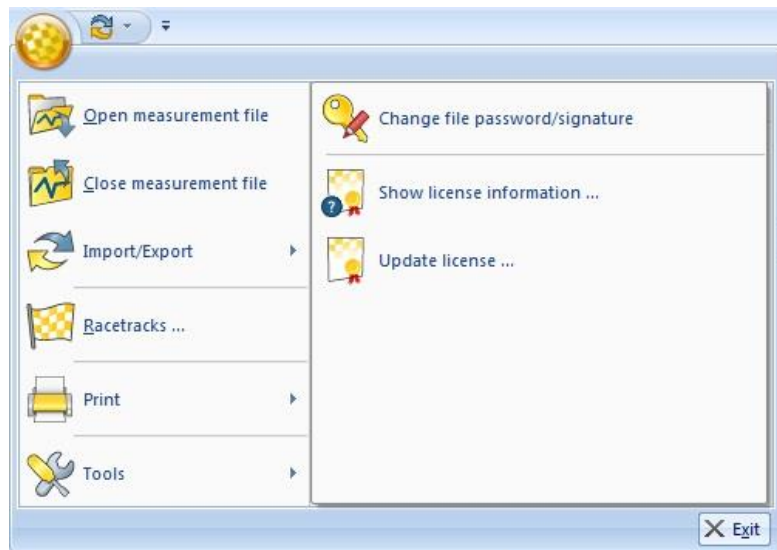
From here on just follow the instructions. There are two ways to install a new WinDarab7 license.

1. You can insert a license file (.lic)
2. You can insert a license number. For this option you have to be connected to the internet

To check your license information, just click on **WinDarab-Button/ Tools/ Show license information**

Darab Ribbon

WinDarab menu and toolbars are organized in a Ribbon. The Ribbon is designed to help you quickly find commands and perform tasks. The commands are grouped logical together and placed in a tab. Darab Ribbon consists at least the following tabs:

Start

The tool tabs on the **Start** tab depend on which analysis window is currently active. If no analysis window is opened the **Start** tab contains the following tool tabs (see chapter Analysis and Oscilloscope)

- Desktop
- Settings

When at least one analysis window is opened the analysis window specific tool tabs will be displayed right of the **Settings** tool tab.

For example here are the tool tabs of the oscilloscope:
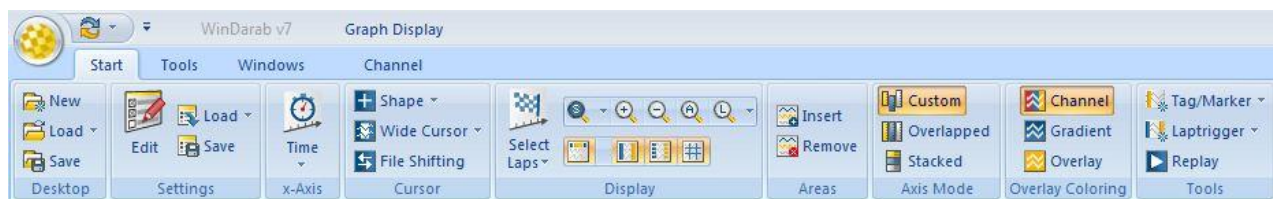
Desktop

The tool tab **Desktop** provides tools for creating, loading and storing a desktop.

Start/Desktop/ New   This item will destroy all worksheets and create a worksheet with a single oscilloscope.

See chapter [Desktop](#) for a detailed description.

**Start/Desktop/
Load**

Open the **Load desktop** dialog box and select the desktop you want to load.
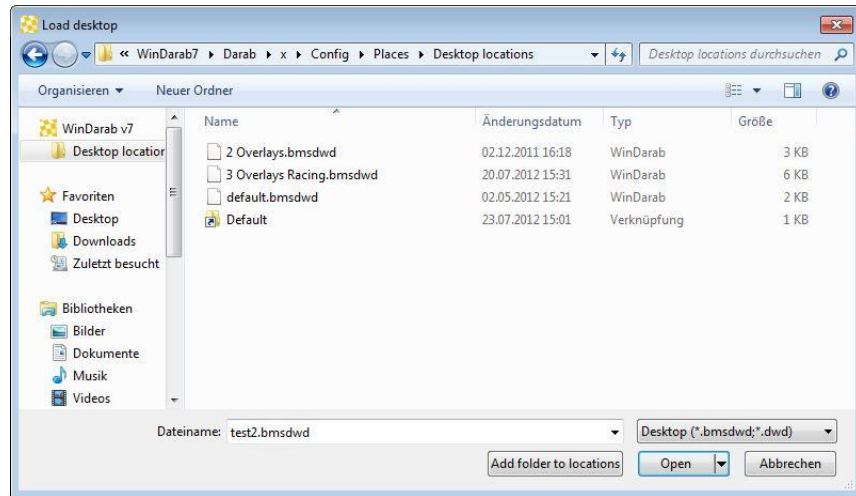
Select the Desktop file

Click on the **Open** button

**Start/ Desktop/
Load
(Dropdown)**

You can also open a recently opened desktop by clicking on the dropdown-button right to the **Load**-button and chose a Desktop-file from the list.

**Start/Desktop/
Save**

Open the **Save desktop** dialog box to save the current desktop.
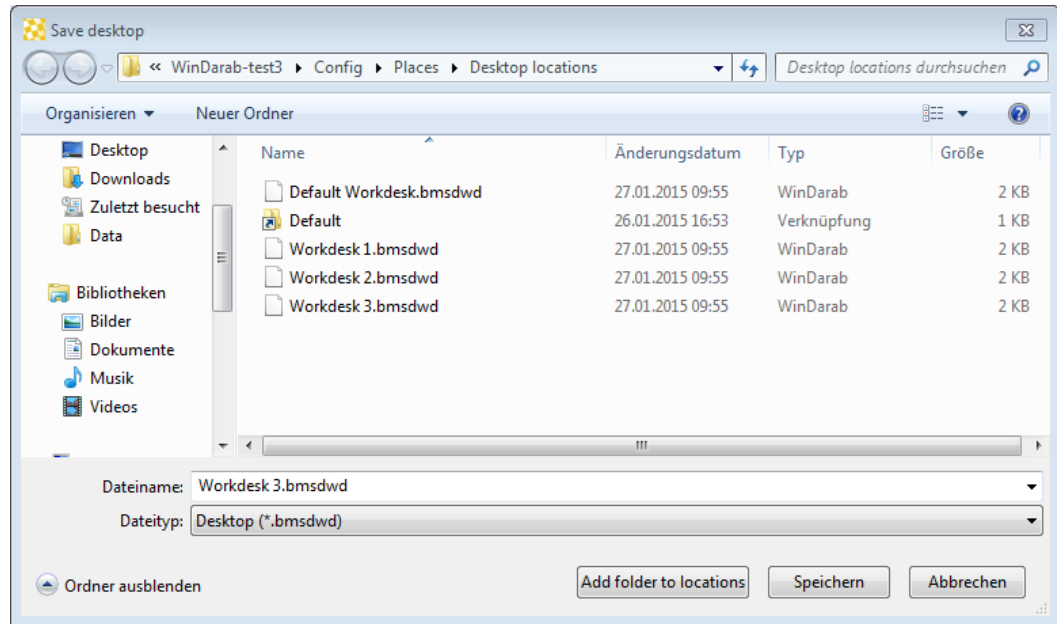
1. Navigate to the folder you want to store the current desktop
2. Enter the desktop name in the text field Filename
3. Click on the Save button

Settings

The tool tab **Settings** provides tools for editing, loading and storing of Analysis windows and oscilloscopes settings.

The **Settings** tools:

Start/ Settings/ Edit – edit settings of an analysis window or oscilloscope

Start/ Settings/ Load - load settings of an analysis window, oscilloscope or instrument panel

Start/ Settings/ Save - save settings of an analysis window, oscilloscope or instrument panel

They are described in the chapters Oscilloscope, Analysis, Telemetry.

Tools

The **Tools** tab contains:

- Mathematics
- Settings
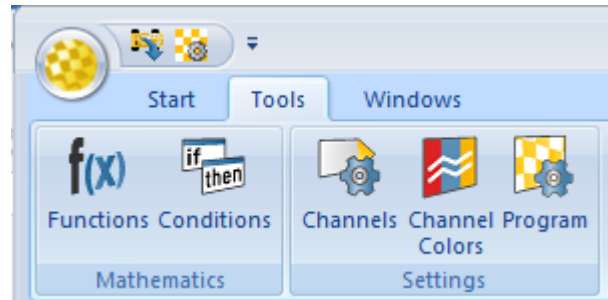
Mathematics

The tool tab **Mathematics** provides tools for managing mathematical functions and conditions. This will help you to solve a series of mathematical and analytical problems.

The **Mathematics** tools:

Tools/ Mathematics/ Functions – allow you to define filters, lookup tables and mathematical functions

Tools/ Mathematics/ Conditions – allow you to define conditions to exclude certain data from consideration during data analysis

They are described in detail in the chapters Mathematical Channels and Conditions.

Settings

The tool tab **Settings** provides the possibilities to manage channel settings, channel colors and general WinDarab settings.

The **Settings** tools

Tools/ Settings/ Channels– allow you to define basic settings for each channel in WinDarab

Tools/ Settings/ Channel Colors – allow you to define the color of each channel in each overlay

Tools/ Settings/ Program – allow you to make basic WinDarab settings

They are described in detail in the chapters Channel Settings and Settings.

Windows

The Windows tab contains the following tool tabs

- Dockable windows
- Worksheet
- Worksheet controls
- All controls
- Current control
- Desktop layout
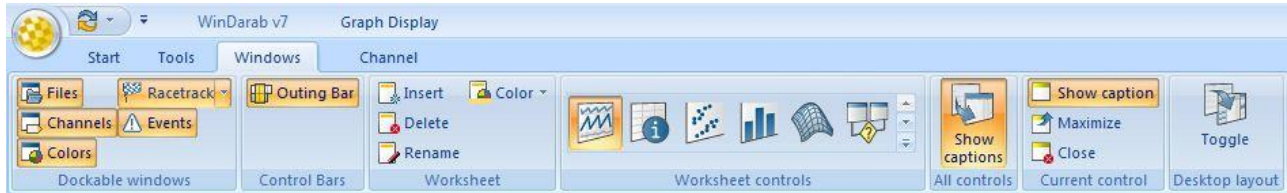
Dockable Window

WinDarab manages the following dockable window:

- Files – see chapter File Explorer Window
- Channels
- Colors
- Racetrack – see chapter Racetrack
- Events – see chapter Events

The tool tab Dockable Windows allows you to open or close a dockable window.

Files
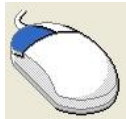
Click with the left mouse button on **Windows/ Dockable/ Files** to *open* the **File Explorer** window.

Events
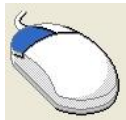
Click with the left mouse button on **Windows/ Dockable/ Events** to *open* an **Events** window.

Racetrack
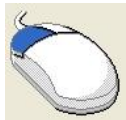
Click with the left mouse button on **Windows/ Dockable/ Racetrack** to *open* a **Racetrack** window.

The Racetrack window has three different display states. You can select one by clicking on the Racetrack-Drop down button and selecting an item.
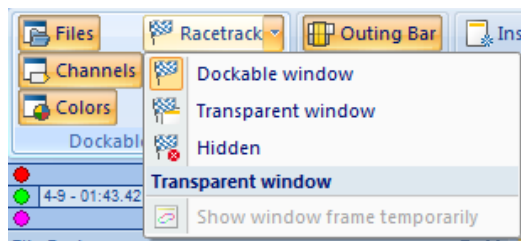
Dockable window	**Dockable window** is the standard option. In this state, your Racetrack window is docked wherever you chose and can be handled like explained in The Racetrack Window.
Transparent window	The **transparent window** is activated when you click on it or when you drag the docking racetrack window and drop it somewhere without docking it. As soon as the window loses the focus, its background disappears and all you see is the racetrack.
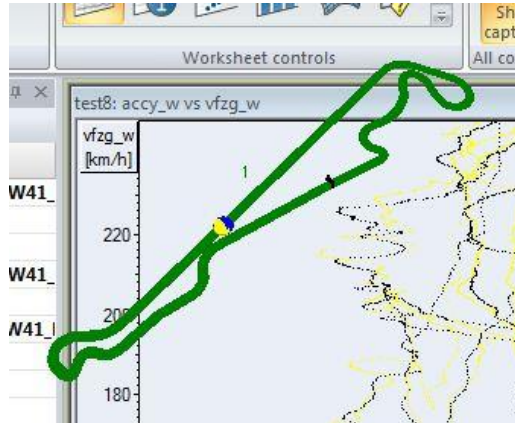
You can move the racetrack by **STRG-Key** + left click on it and drag it to another location.

Alternatively, you can move your cursor over the racetrack and stay there for one second to display the background again. Then the racetrack can be moved by draging the window.

Hidden	The last option is to hide the Racetrack window completely. You can quickly switch through all the states by clicking on **Windows/ Dockable windows/ Racetrack**.

Further Racetrack information can be found at The Racetrack Window.

Channels

Press **F3** key to *open* the **Channels Window**.

Click with the left mouse button on **Windows/ Dockable/ Channels** to *open* the **Channels** window.
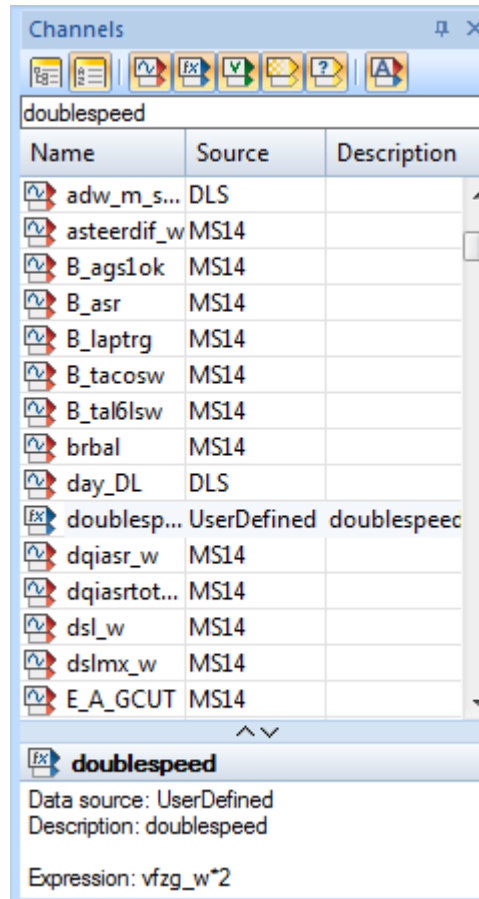
The **Channels** window displays a list of WinDarab channels. WinDarab channels are divided into four channel lists:

- Measurements channels
- Math channels
- Internal channels
- Temporary channels

The **Channels** window toolbar allows you to select which channel list should be currently displayed.

The **Channels** window provides you the possibility to drag & drop a channel to an oscilloscope, analysis window or instrument panel.

The channel list offers also a channel search function. You can find a channel in the channel list by entering the name of the channel in the text field under the window tool bar.

Channel Info   The **channel info** is a small region under the channel list. In this region you see **detailed information** about the currently selected channel like the sample rate, the data source or a selected math channel's expression.

You can resize the area by dragging the border. You can also max-/minimize it by double clicking the border.

Colors
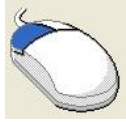
Click with the left mouse button on **Start/ Dockable/ Colors** to *open* a **Colors** window.
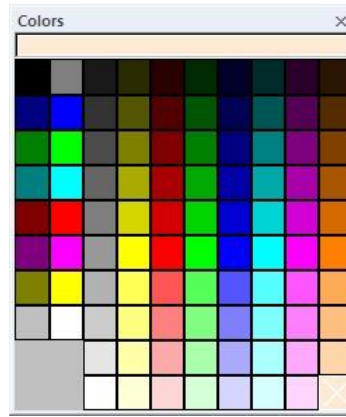
The **Colors** window provides you the possibility to change the color of a channel, overlay, or something else by drag & drop the color to the corresponding element.

Control Bars

The ribbon **Controll Bars** tab manages the Outing Bar

Click with the left mouse button on **Windows/ Control Bars/ Outing Bar** to *enable/disable* the **Outing Bar**.

Outing Bar

The **Outing Bar** gives you an Overview over the Overlays, their laps and the position to each other.

| ● | 1:34.05 | 28 - 01:33.48 | 29 - 01:33.22 | 30 - 01:33.14 | 31 - 01:33.54 | 32 - 01:33.29 | 33 - 01:32.88 | 34 - |
| ● | 2 | 1 - 02:03.10 < | 2 - 01:41.99 | 3 - 01:39.39 | 4 - 01:39.44 | 5 - 01:37.53 | 6 - 01:35.51 | 7 - |

The current visible area is marked yellow; the current cursor position is visualized with a dotted line. When the Area or the Cursor-position changes, the Outing Bar will update automatically.
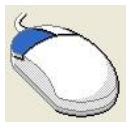
Click with the left mouse button on a lap outside the visible area, to *set the visible area and cursor-position* to the exact **position in the lap** where it was in the previous selected lap.
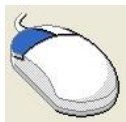
Click and drag the visible area in the **Outing Bar** to *move the visible area* in the Oscilloscope. The cursor-position moves with the visible area.
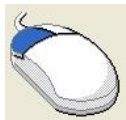
Click and drag the border of the visible area in the Outing Bar to *increase or decrease the Size* of the visible area (zoom out / zoom in).
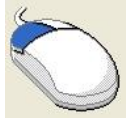
A doubleclick on a Lap *moves the visible area* to the size and the position of the **selected Lap**.

Worksheet

The ribbon **Worksheet** tab allows you to

- Insert a new worksheet
- Delete a worksheet
- Rename the worksheet label
- Change the color of the worksheet tab
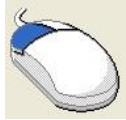
Click with the left mouse button on **Windows/ Worksheet/ Insert** to *create* a **new worksheet**.
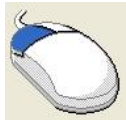
Click with the left mouse button on **Windows/ Worksheet/ Delete** to *delete* the **active worksheet**.

Click with the left mouse button on **Windows/ Worksheet/ Rename** to *rename* the **label** of the active worksheet.
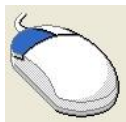
Click with the left mouse button on **Windows/ Worksheet/ Color** to *change* the **color** of the active worksheet tab.

Worksheet controls

WinDarab manages the following worksheet controls (windows):

- Oscilloscope – see chapter [Oscilloscope](#)
- Inform – see chapter [Inform](#)
- x/y-Plot – see chapter [x/y-Plot](#)
- Histogram – see chapter [Histogram](#)
- Distribution – see chapter [Distribution](#)
- FFT – see chapter [FFT](#)
- Flow Chart – see chapter [Flow Chart](#)
- Min/Max-Table – see [Table Min/Max](#)
- Outings report - see [Outings report](#)
- Lap Analysis – see chapter [Lap Analysis](#)
- Instrument Panel - see chapter [Instrument Panel](#)

The tool tab Worksheet controls allows you to open one or more worksheet controls.

A worksheet control could be an oscilloscope, analysis window, inform window or instrument panel.
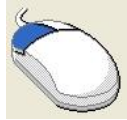
Click and drag a symbol with the left mouse button onto your workspace to open the corresponding worksheet control.

All controls

With the All controls tab, you can change the caption for all your worksheet controls.
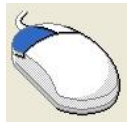
Click with the left mouse button on **Windows/ Worksheet controls/ Show captions** to *show/hide* the **caption** for all your worksheet controls.

When you hide the caption of a single window while your other windows still have the caption shown, the Show captions button will be unchecked. You can click on the Show captions button now to show all captions again or press the button two times to hide all captions.

Current control

The Current control tab allows you to adjust the current active control
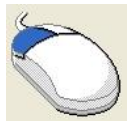
Click with the left mouse button on **Windows/ Current control/ Show caption** to *change the visibility* of your **current active control**.
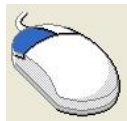
Click with the left mouse button on **Windows/ Current control/ Maximize** to *toggle* between the normal size and the maximum size of your **current active control**.
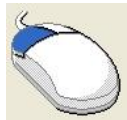
Click with the left mouse button on **Windows/ Current control/ Close** to *close* your **current active control**.

These functions can also be used by right-clicking on a windows caption-bar and selecting the item from the context-menu.

Desktop layout
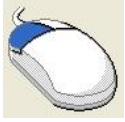
Click with the left mouse button on **Windows/ Desktop layout/ Toggle** to *toggle* between 2 different **desktop layouts**.

When you click on Toggle for the first time, the docks won't be visible. You can now adjust your workspace. Both workspaces will be saved when you close WinDarab 7, so they are still the same after the next start.

File Explorer Window

The File Explorer Window displays and manages all logger data files opened in the File Explorer (see chapter File Explorer). WinDarab supports the following logger files:

- WinDarabv7 Files
- WinDarabv6 Files
- WDServer Telemetry Files
- Linked Files
- MDF Files
- MSX Files

Open File Explorer Window
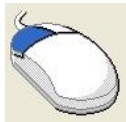
Select **Windows/ Dockable windows/ Files** from Darab Ribbon to *open* the **File Explorer Window**.
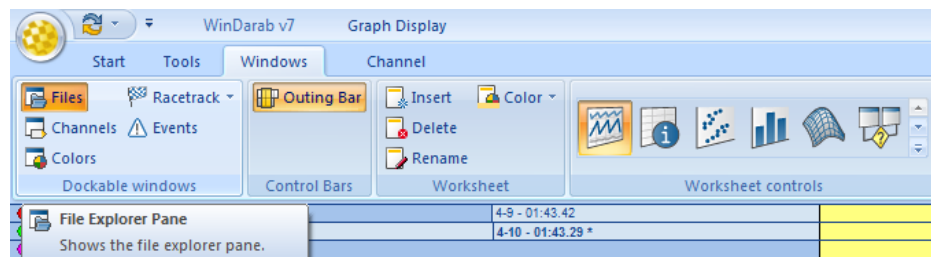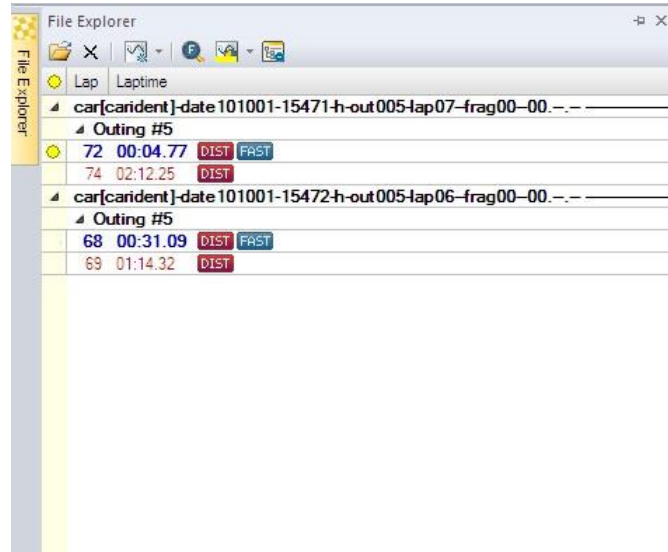
The File Explorer Window is a dockable window. It contains a toolbar and a WinDarab file list window.

File Explorer Window Toolbar
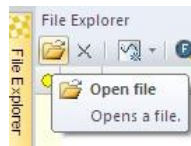
	Opens the **File Explorer** (see File Explorer)
	Close the selected file.
	Create an overlay to compare files/laps (see Overlay). Each overlay is displayed in a different color.
	Zoom and compare fastest laps of each overlay.
	Change the color of the selected overlay
	Reduce the lap list. The „outing #"- line under the file name in the file list will not be displayed.
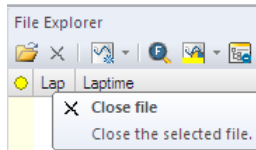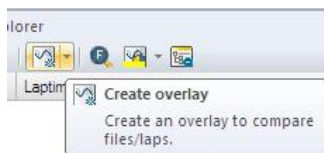
File Explorer Window File List

The file list contains a list of all opened files in the File Explorer.
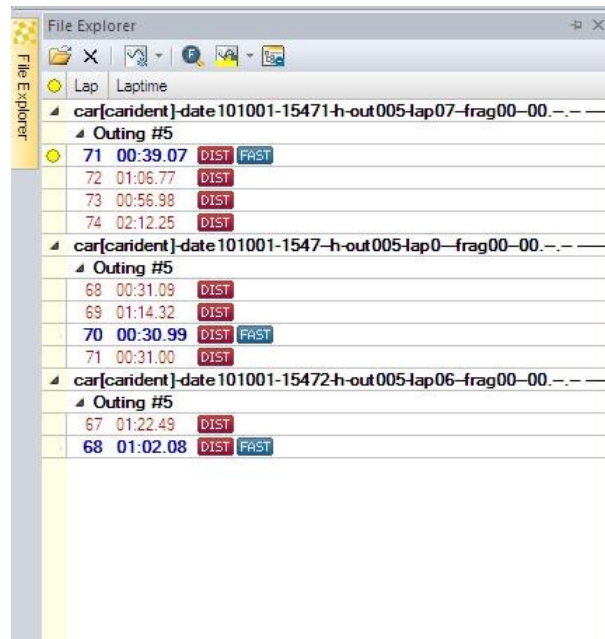
Under the file name the outing number is displayed. The laps are listed under the outing number. Each lap is indicated with a lap number and a lap time. The fastest lap is marked with the label ![FAST]. If an overlay is assigned to a lap it is displayed by a colored point before the lap number.

> If the menu point „**Reduce lap list**" is selected the line „**Outing#**" will be hidden.

File Explorer Window Context Menu

> After right-click on an object (file, lap) in the **File Explorer** window a **context menu** *opens*. The context menu contains the following options:
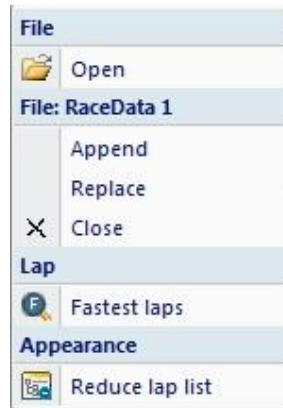
Open   Opens a new file.

Append   Opens a new file and appends it to the selected lap.

Replace   With this menu-item, you can replace the file by another file. After clicking on the item, the open-file-dialog opens, where you can select the file replacing the file in the file-explorer.

Close   You can use the close menu item to close this file.

Fastest laps   Zoom and compare fastest laps of each overlay.

Reduce lap list   Reduce the lap list. The „outing #"- line under the file name in the file list will not be displayed.

When the context-menu belongs to a lap, options appear to mark the lap as In-lap or Out-lap or to ignore it when the fastest lap is determined.
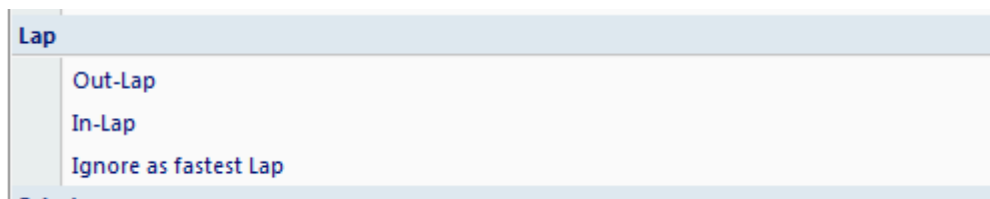
Overlay

Overlay provides the possibility to compare laps/files and to add a lap/file to an oscilloscope (see chapter Oscilloscope). Overlays are displayed as points in different colors.
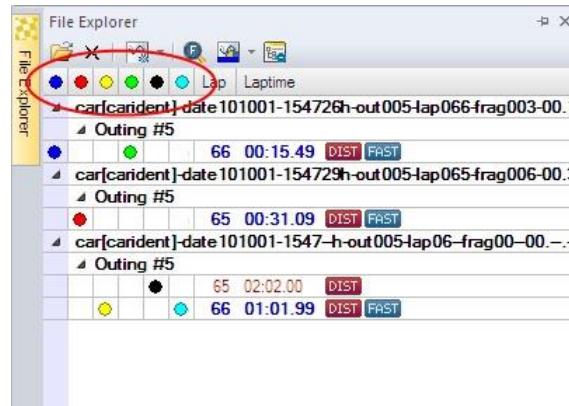
By default at least one overlay exists in the **File Explorer** Window. If you open a file in the **File Explorer** (see chapter File Explorer) the overlay is assigned to the fastest lap of the file.

Overlay Context Menu

Click with the right mouse button on one overlay point or the overlay row to ***open*** the **overlay context menu.**
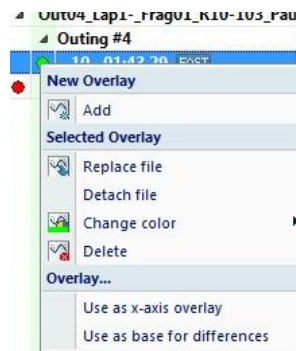
Add	Add an overlay in the overlay list under the **File Explorer** window toolbar. If at least one file is opened in the **File Explorer** window the overlay is assigned to the next longer lasting lap without overlay in the first file of the list. If each lap has an assigned overlay already all subsequent overlays will be assigned to the longest lap in the first file.
Replace file	Replace the file of the selected overlay. The File Explorer is opened and allows you to select the new file.
Detach file	Detach the file from the overlay. After this, the overlay has no file.
Change color	A color palette is opened that allows you to select the color of the selected overlay.
Delete	Select this menu point to delete the overlay.
Use as x-axis overlay	Select this option to use this overlay as base-overlay for the x-axis.
Use as base for differences	Select this option to use this overlay as the base-overlay for difference-channels.

Adding an overlay

There are the following methods to add an overlay:

1. From overlay context menu (see above)
2. From Add context menu in an Oscilloscope (see chapter Oscilloscope)
3. From **File Explorer Window** toolbar:
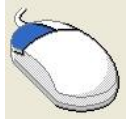
Select **Overlay/ Add** from **File Explorer Window**.
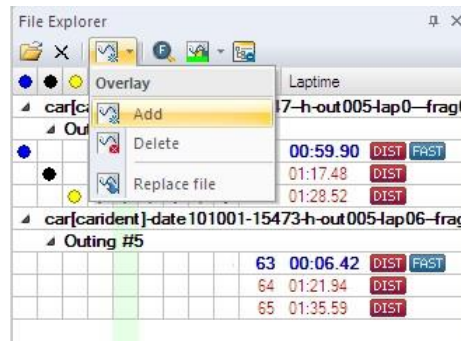
Delete an overlay

There are the following methods to delete an overlay:

1. From overlay context menu (see above)
2. From **File Explorer** Window toolbar:
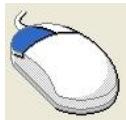
Select **Overlay/ Delete** from **File Explorer** Window.
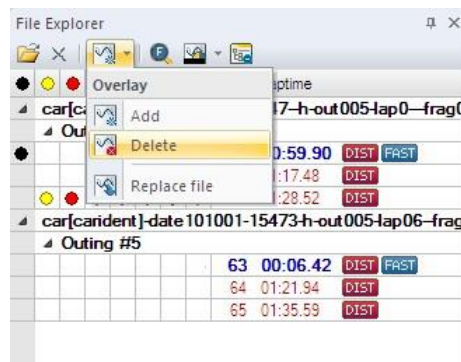
Assign Overlay to File/Lap

There are the following methods to assign an overlay to a file/lap:
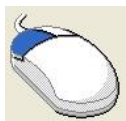
Click on the desired **overlay-field** of the lap, you want the overlay being ***assigned*** to. There will be a marker on this field to indicate, that this overlay is now assigned to this lap.

An overlay can only be drawn vertically.

Open the context menu of an overlay ⬤ in the **File Explorer** Window and select **Selected Overlay/ Replace file.** The **File Explorer** will be opened and you can select the file.

Select **Overlay/ Replace** file from **File Explorer** Window toolbar. The **File Explorer** will be displayed and you can select the file.

A file can also be detached from an overlay by selecting "Detach" from the context menu of the overlay.

Drag and Drop

Files can be opened by dragging them from your windows file system into the WinDarab FileExplorer. Depending on the location where you drop the file, the file can be added to the file-list, replace an other file or replace an overlay.

Adding a file	You can add a file to the file-list by dropping it anywhere on the list but an existing filename or overhead-column. Your files will be added to the list. If you add more than one file to the list, as default, they will be added as one file. To open more files as single files, press **Ctrl** while dropping the files.
Adding a file to a Filegroup	You can add one or more files to a filegroup by dropping them over an existing file. When you drag the files over an existing file, the target is highlighted to show that you can add the files.
Replacing a file	You can replace one or more files by dropping a file over an existing file while pressing the **Shift**-key. When you drag the files over an existing file, the target is highlighted to show that it can be replaced.
Replacing an overlay	Dropping a file on an overlay-column adds the file to the file-list and assigns the columns overlay-color to the new added file. If this removes the last overlay from a file, this file will be removed.

File System Monitoring

When a file is opened in the file explorer, the containing folder as well as all of his sub folders are monitored for new darab files. When a new darab file is created in a monitored folder, darab shows a balloon tip to let the user know that a new file was created.

In addition to this balloon tip, a region on the bottom side of the file explorer is created where all new files are listet.
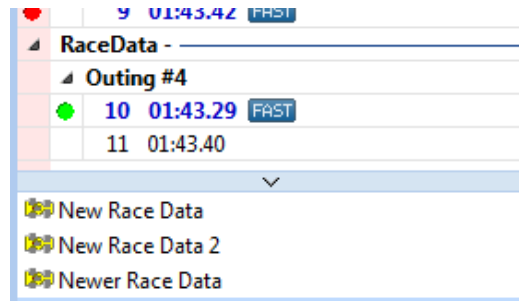
You can open these files by dragging them into the file explorer (see Drag and Drop) or by using the contextmenu.

> The separator between the new files and the file explorer can be dragged to reduce the size of the region. You also can minimize/maximize the region with a doubleclick on the separator.

File Explorer

The usability of WinDarab's File Explorer is much the same way as Windows' Windows Explorer.

File Explorer is used to manage all WinDarab supported files:

- WinDarabv7 files
- WinDarabv6 files
- WDServer Telemetry files
- Linked files
- MDF Files
- MSX files

> In addition to access files on existing disk drives you access them through locations. A location in fact represents the affiliation of files with a specific drive and folder. Using locations simplifies access and management of files. See chapter Locations

Open File

Open File Explorer

To open File Explorer there are the following methods:

From **File Explorer** Window

1. Open **File Explorer** Window if not opened.
2. Choose "**Windows/ Dockable Windows/ Files**" or press "**F3**" to open the **File Explorer** Window.
3. Choose **Open file** in the **File Explorer** Window toolbar.
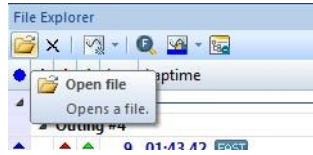
From **Darab Ribbon**

Choose **WinDarab-Button/ Open measurement file** in Darab ribbon
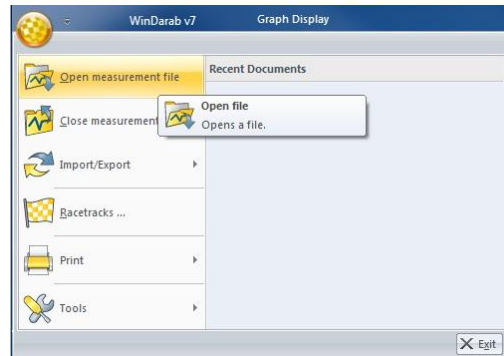
File Explorer
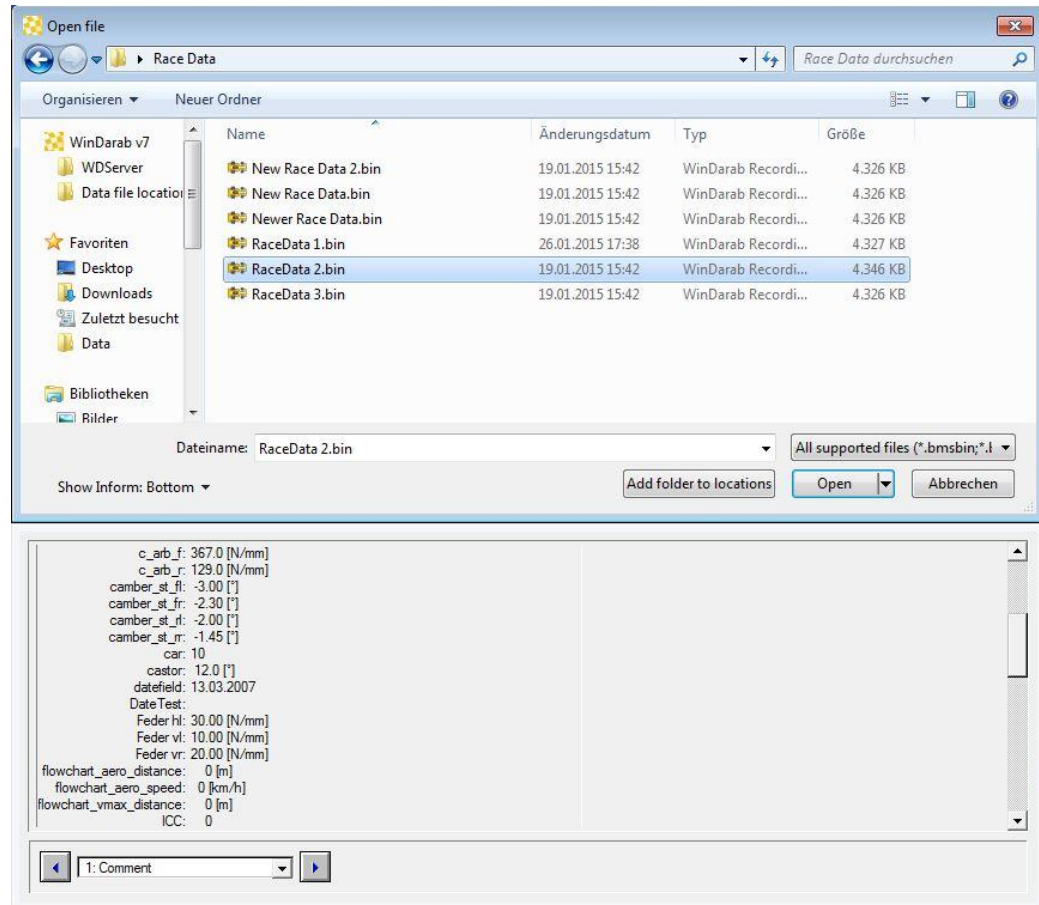
The File Explorer window is divided into:

1. **Open File** dialog box
2. **File Inform** window

The **Open File** dialog box allows you to browse through drives, folders, locations and select the file you want to open.

This area displays sub-folders and files contained in the selected folder.

To select an item you can either use your keyboard or your mouse. Whenever possible a preview of the selected file will be displayed.

If the selected file has a password protection act as shown in Open Password-protected Files.
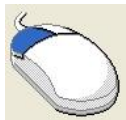
To *select and activate* an **item** click with the mouse on the item. Press Enter on the keyboard or double click with the mouse to *open* the **item**. Your subsequent steps depend on the type of the file you have selected.

You can *select* more than one **file** by using the **Ctrl-** or **Shift-key**. Whatever action you have chosen (e.g. open or delete) will be performed for each file and in the selection order.

The combo box "Show Inform" allows you to choose whether the **File Inform** window to should be shown or not.

Folders can be created, renamed and deleted in the **Open File** dialog box in the same way as regular windows folders in the Windows Explorer.

Open the selected file(s) by pressing the **"Open"-button**.
If there is more than one file selected, the files are opened as one file. You can also select **"Open as single files"** in the "**Open"-button**'s dropdown list to open all selected files as single files.

If you use the **Shift-key** while pressing the **"Open"** button, the **"open linked files"-dialog** appears. In this dialog you can change the order of the linked files, delete single files from the list or save the list as a predefined filegroup in the same folder as the used files are saved in. After that, you can open this exact filegroup by opening the generated filegroup-file.
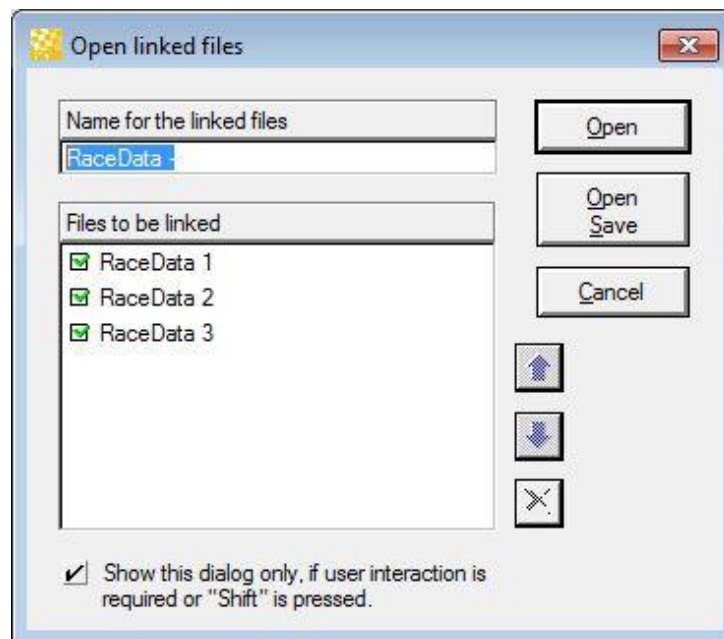
File Inform

File Info is divided to 3 groups:

1. Comments
2. Statusblock labels
3. CardMemory Info

To select one item navigate with the arrows.

These enables you to view the following file information:

Comments	Configuration Tools	Display configuration tool information stored in the WinDarab file.
	Comment field	Display the vehicle settings.
	Comment	Display the logger data specific comments.
Statusblock label	Statusblock label	Display the available status block label
CardMemory Info	Logbook	Display memory card log book
	Measinfo	Display memory card measurement info
	Sections	Display memory card sections

Locations

You can save data in any folder and on any disk drive you choose. In addition it is possible to create locations. WinDarab defines the special folder **Data file location** as container for all locations. A location actually is a link to a specific folder which enables you to save files in, or open files from, different locations quickly – without having to navigate the entire drive and folder structure.

Add folder to locations

Follow the steps below to add a folder to the **Data file locations** folder:

Open the **File Explorer**

Choose the folder you want to define as location.

Click on the button Add folder to locations

A link to the selected folder is generated and added to the **Data file locations** folder.

A location can be deleted or renamed in the same way as regular windows link to a folder in the Windows Explorer.

Racetracks

Racetracks are managed by **Darab Explorer – Racetrack**.

Following processing items are provided by the **Darab Explorer – Racetrack** and described in detail in the next sections:

- Creating a Racetrack
- Deleting a Racetrack
- Renaming a Racetrack
- Editing a Racetrack
- Selecting Racetrack's Segmentations

To open the **Darab Explorer – Racetrack** there are the following methods:

From **Racetrack Window**:

Open **Racetrack Window** if not opened.

Racetrack Window   Choose "**Windows/ Dockable windows/ Racetrack**".

Open context menu in the **Racetrack Window**.

     Click on the **Racetrack Window** to *open* the **context menu**.
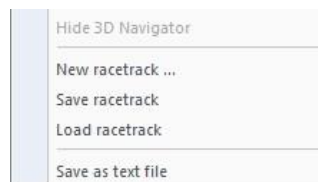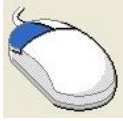
Choose **Load racetrack** in the context menu.

From **Darab** ribbon:

Choose **WinDarab-Button/ Racetracks** in Darab ribbon

WinDarab-Button/
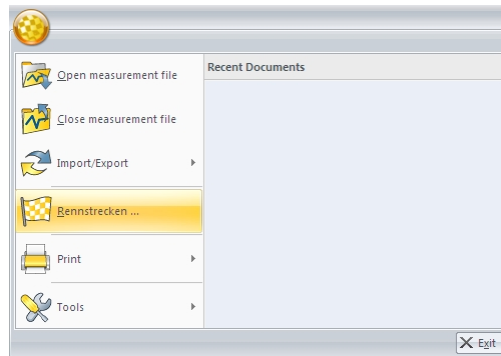
Racetracks

Darab Explorer - Racetrack
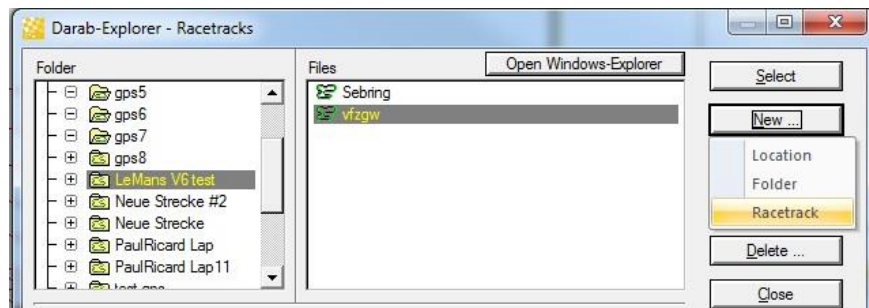
In addition to a course layout Racetrack files contain information about the length of the course as well as its breakdown into course segments. Segmented racetracks are used by a number of analytic functions in WinDarab.

Single racetrack files can contain multiple segmentations. Because of this Darab shows racetrack files as folders and the different segmentations as files.

Creating a Racetrack

To create a new racetrack, follow these steps:

1. Open **Darab Explorer - Racetrack**
2. Click on the **New** button.
3. Select **Racetrack** from the menu.
4. The **Racetrack** dialog box will appear. You are now able to create a new racetrack by accessing the desired functions for racetrack layout (see chapter Racetracks edit modes).

5. When you are satisfied with the results click on **Save As**.
6. In the dialog box **Save Racetrack** select the folder in which you want to save the racetrack. Then enter a name for the racetrack.
7. Click on **Save**.
8. Now you can create course segmentation by accessing the desired functions for segmentations (see Determine Segmentation).
9. Press **Close** when finished

Chapter "Racetrack Dialog" describes the "Racetrack" dialog box and chapter "Racetracks edit mode" specifies in detail the methods to create or modify a racetrack.

Deleting a Racetrack

Open **Darab Explorer – Racetrack**.

Select the racetrack you want to delete.

Click on the **Delete** button.

Confirm your deletion by answering "Yes".

Important:

As a safety measure Darab Explorer prohibits the deletion of racetracks unless you have specifically enabled this function in your configuration.

Renaming a Racetrack

Open **Darab Explorer - Racetrack**

Select the racetrack you want to rename.

Click on the **Rename** button.

Change the name of the racetrack and confirm the new name by pressing **Enter** on the keyboard

Editing a Racetrack

Open **Darab Explorer - Racetrack**

Select the racetrack you want to edit.

Click on the **Modify…** button.

The dialog box **Racetrack** will appear (see chapter Racetrack Dialog).

Now you can make the desired changes to the racetrack (see chapter Racetracks edit modes).

When you are finished with your changes click on **Close**.

The Darab Explorer won't open a racetrack that was saved in the former file format (extension .lap).

Selecting a Racetrack Segmentation

Select the racetrack that contains the segmentations you want to select

Select the desired segmentation (or with pressed **Shift/Ctrl key** to select several segmentations)

Click on the "Select" button.

If you select several segmentations by pressing the **Ctrl** or **Shift key** the selection order is important for some analytic functions of WinDarab. The last selected segmentation is used by windows, supporting only a single segmentation (e.g. the Oscilloscope). The other selected segmentations are shown in the "Racetrack Window"

Racetrack Dialog

Whenever you create or modify a racetrack the dialog box "Racetrack" will be displayed. A racetrack dialog supports 2D or 3D view modes and several edit modes.
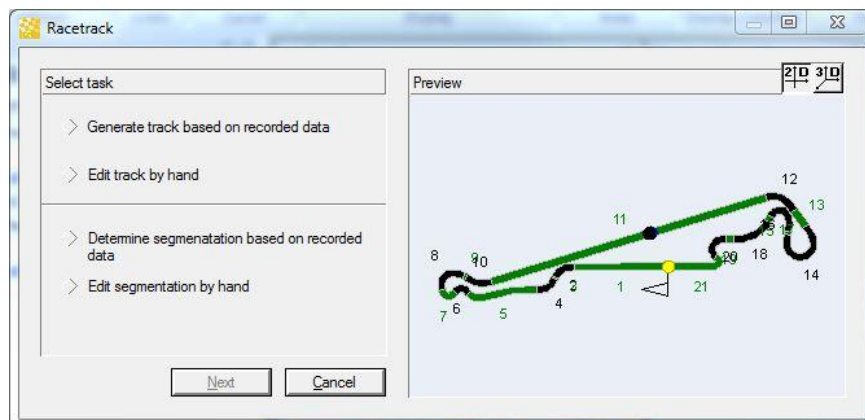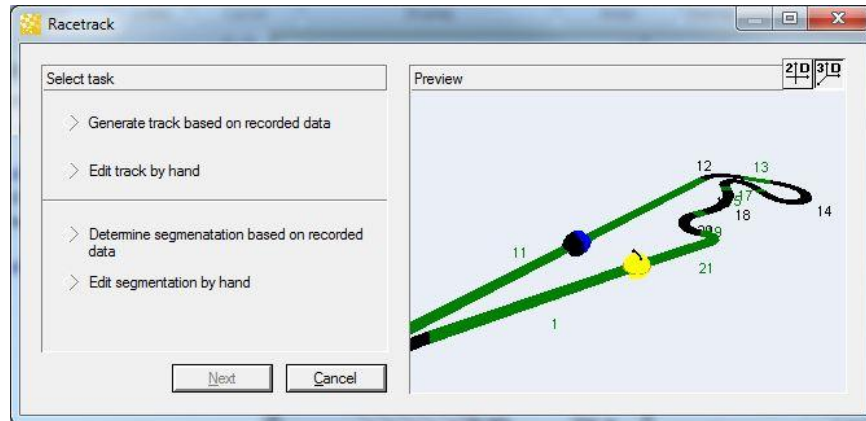
Racetracks edit modes

To generate a racetrack the following edit modes are provided (see above):

- **Generate track based on recorded data**.
  In this mode you can calculate the course using measured data like speed and transversal acceleration, GPS track boundaries and GPS laps with fixed track width.

- **Edit track by hand** (Draw racetrack).
  Here you can draw a racetrack using lines.

The following edit modes are provided for modification of an existing racetrack:

- **Determine segmentation based on recorded data**.
  This mode is useful to determine racetrack segmentation based on measured data.

- **Edit segmentation by hand**.
  Use this mode if you want to define manually the segments on the racetrack using the lap position.

The edit modes are available from the **Racetrack Dialog**.

Generate track based on recorded data

Use the **Generate track based on recorded data edit mode** to calculate the course of a racetrack from logged data.

To open the **Generate track based on recorded data** dialog box:

Open **Darab Explorer – Racetrack**

Click on the **New…** or **Modify…** button

Select **Generate track based on recorded data** option and click on the **Next** button.
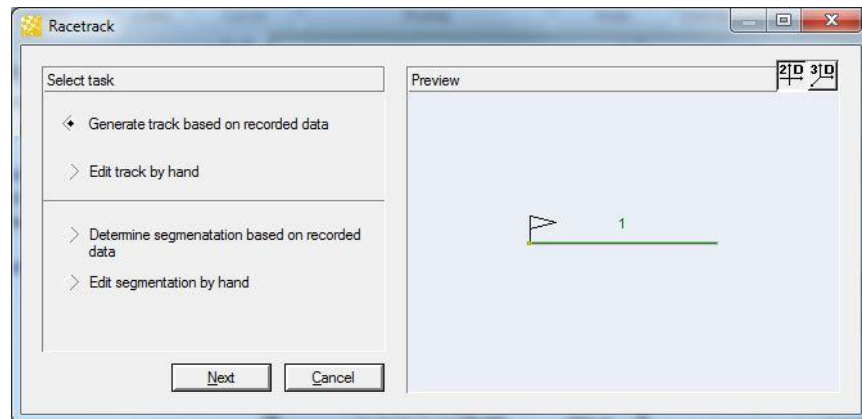
The dialog box offers three methods to generate the racetrack:
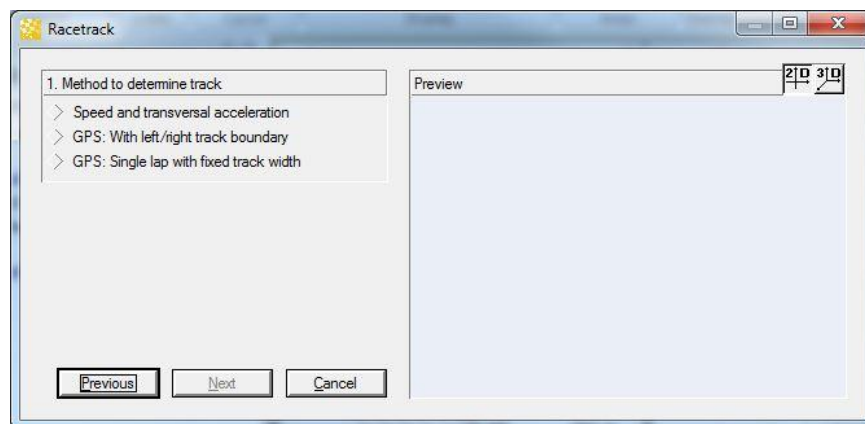
Speed and transversal acceleration

This method transforms logged speed and transversal acceleration signals to metric coordinates and shows the real racetrack.
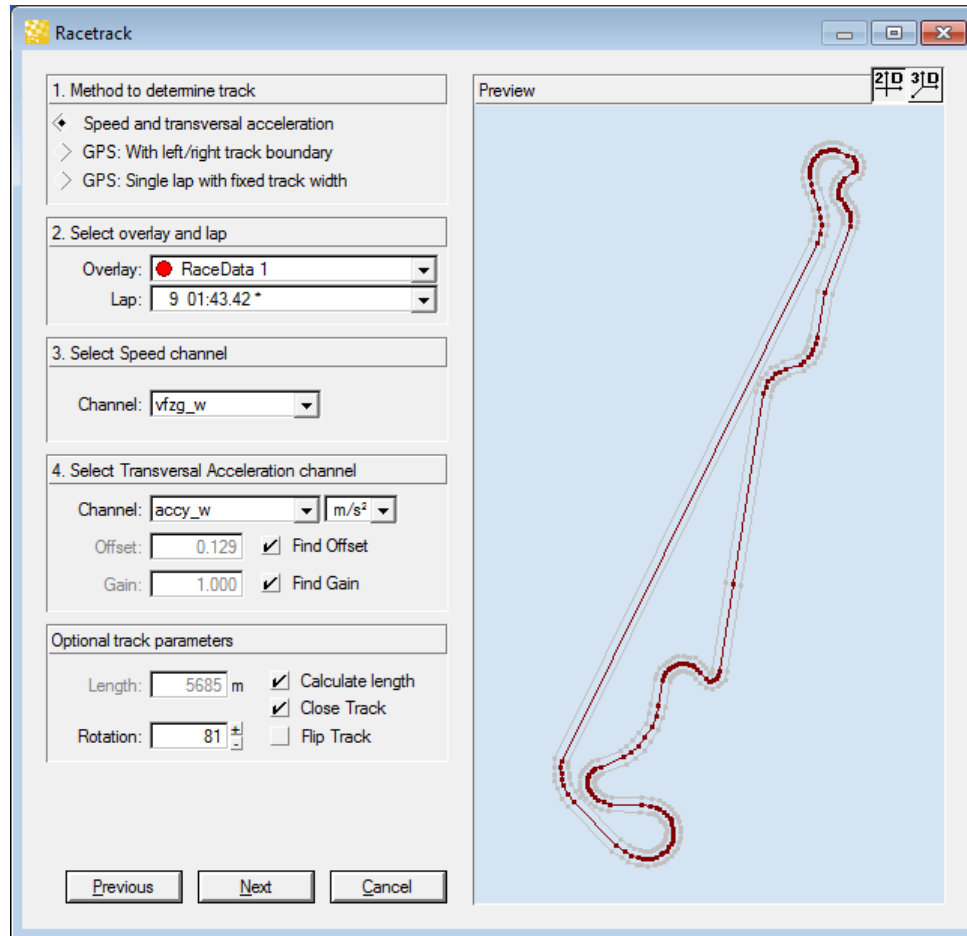
This dialog box is used to set the following parameters:

1. Method to determine track

If you choose "Speed and transversal acceleration", the corresponding option on the dialog box is ticked. To generate the racetrack you should set the parameter as follows.

2. Select overlay and lap

Overlay   In this field you select the overlay you want to use for the calculation.

Lap   In this field you select the lap that will provide the data for the calculation. Signal quality from the transversal acceleration sensors as well as the occurrences of vehicle drift and wheel slip/spin during the selected lap will greatly affect the results.

3. Select Speed channel

Select the channel used for recording speed from the list.

4. Select Transversal Acceleration channel

Channel   In this field you select the channel used to record transversal acceleration. The unit of the transversal acceleration channel can be overridden.

Offset   Enter the offset correction for the transversal acceleration in this

field. Ideally, the value entered should be zero although this is not a typically case (the course is uneven, sensors are not precisely installed, etc.).

Gain   Enter the gain for the transversal acceleration in this field. Ideally, the value entered here should be one.

When you activate the "Find Gain" option, WinDarab will automatically set gain value to 1.0.

5.   Optional track parameters

Length   Enter the precise length of the racetrack. If you activate the "Calculate Length" button WinDarab will automatically calculate the course length.

Rotation   Here you can enter a rotational angle for the course. The course will be rotated according to the angle you specify.

Flip Track   Click on this button to flip the track. Its tip always points in the direction of travel. The racetrack is flipped by reversing the traversal acceleration channel.

Close Track   Choose this option to close to a circular track. The track will consequently either be stretched out or shortened so that start and finish coincide.
De-selecting this option allows you to assess the quality of your logged data: The closer the start to the finish lies, the more accurately your data reflects the circular course.

GPS: With left/right track boundary

The method transforms logged GPS coordinates to metric coordinates and shows the real racetrack.

To generate the racetrack you need logger data file(s) containing the GPS coordinates (i.e. the GPS channels used for recording longitude, latitude and height (see point 5)) of the left -, right track boundary and optional the racing line. The logger data files can be different or identical.
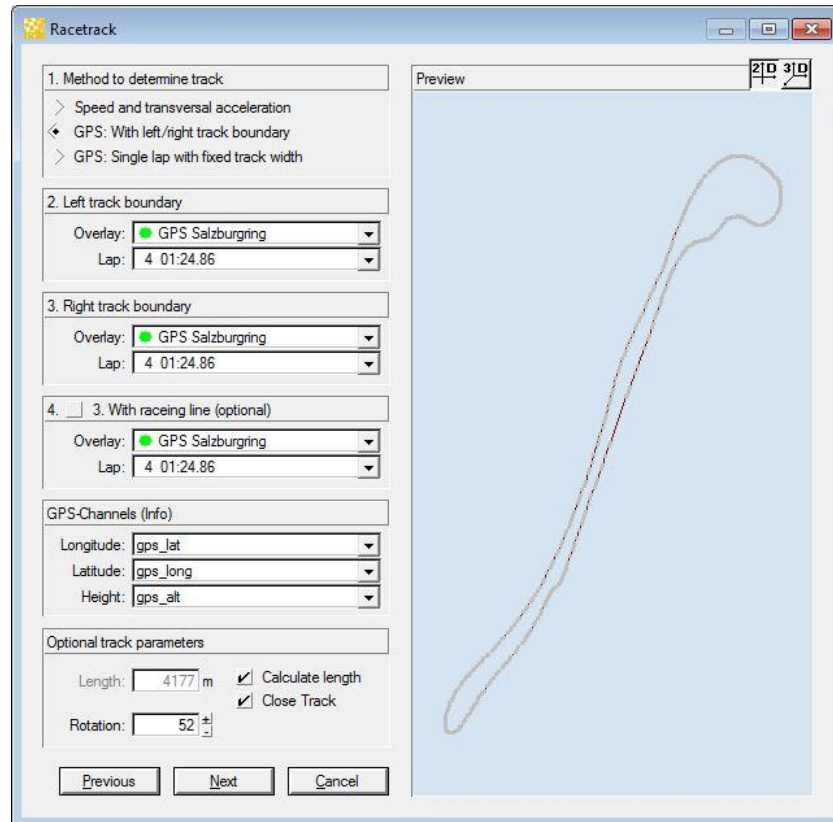
This dialog box is used to set the following parameters:

1. Method to determine track

If you choose "GPS: With left/right track boundary", the corresponding option on the dialog box is ticked. To generate the racetrack you should set the parameter as follows.

2. Left track boundary

Overlay   In this field you select overlay you want to use for calculation of the left boundary. The overlay should contain the GPS channels recording the GPS coordinates (see point 5) of the left boundary.

Lap   In this field you select the lap that will provide the data for the calculation.

3. Right track boundary

Overlay   In this field you select the overlay you want to use for calculation of the right boundary. The overlay should contain the GPS channels recording the GPS coordinates (see point 5) of the right boundary.

Lap   In this field you select the lap that will provide the data for the calculation.

The size of a car shape could become "very large" (larger than the screen) if there are invalid GPS coordinates recorded in the file.

Racetracks

4. With racing line (optional)

Overlay　　　　In this field you select the overlay you want to use for calculation of the racing line. The overlay should contain the GPS channels recording the GPS coordinates (see point 5) of the racing line.

Lap　　　　　In this field you select the lap that will provide the data for the calculation.

5. GPS Channels (Info)

Longitude　　　Select the channel used for recording the longitude of the GPS-coordinates.

Latitude　　　　Select the channel used for recording the latitude of the GPS-coordinates.

Height　　　　Select the channel used for recording the height of the GPS-coordinates.

6. Optional track parameters (see Speed and transversal acceleration)

If special channels for "GPS Longitude", "GPS Latitude" and or "GPS Height" are defined the corresponding text field will contain a special channels list and the user can select the channel to be used (see Special Channel).

GPS: Single lap with fixed track width

The method transforms logged GPS coordinates to metric coordinates and shows the real racetrack.

To generate the racetrack you need a logger data file containing the GPS coordinates (i.e. the GPS channels used for recording longitude, latitude and height (see point 5)) of the driving line. Further you need to provide the width of the driving line.
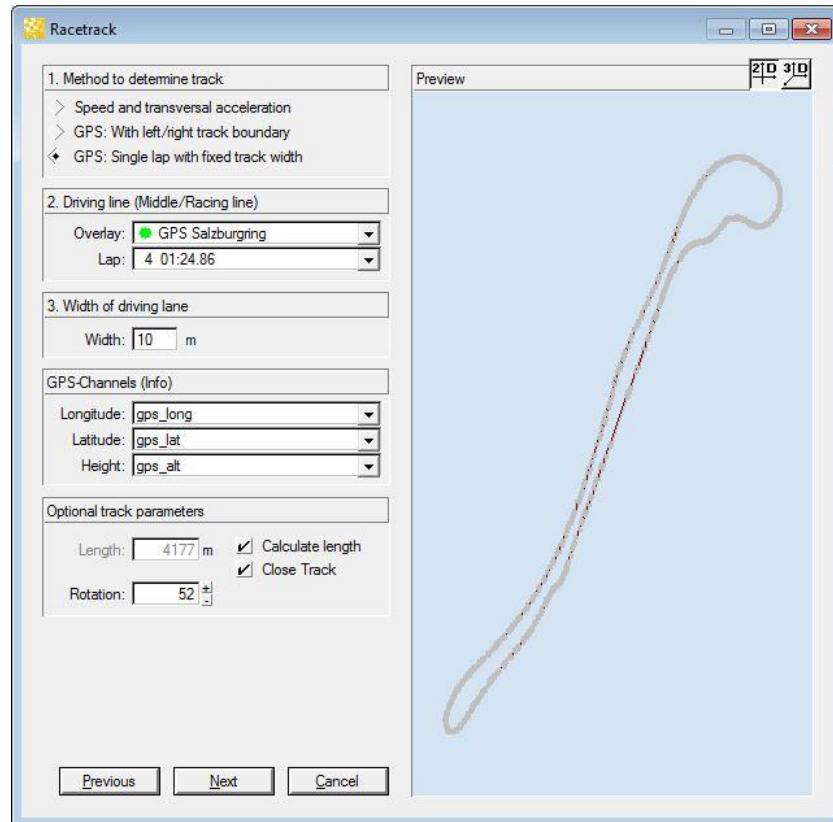
This dialog box is used to set the following parameters:

1. Method to determine track

If you choose "GPS: Single lap with fixed track width" the corresponding option on the dialog box is ticked. To generate the racetrack you should set the parameter as follows.

2. Driving line

Overlay   In this field you select the overlay you want to use for calculation of the driving line. The overlay should contain the GPS channels recording the GPS coordinates (see point 5) of the driving line.

Lap   In this field you select the lap, that will provide the data for the calculation.

3. Width of driving line

Select the width of driving line.

4. GPS Channels (see GPS: With left/right track boundary)

5. Optional track parameters (see Speed and transversal acceleration)

Edit track by hand

The "Edit track by hand" mode allows you to draw or alter the layout of a racetrack manually.

To open the "Edit track by hand" dialog box:

Open **Darab Explorer – Racetrack**

Click on the **New...** or **Modify...** button

Select **Edit track by hand** option and click on the **Next** button.
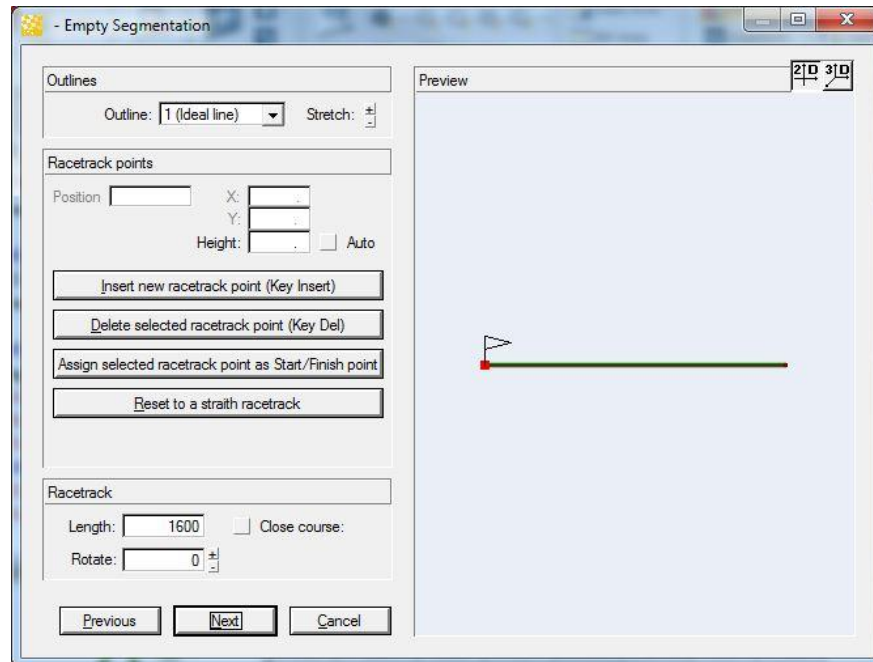
The layout of a racetrack is displayed right from the options control as a series of dots which are connected by lines. In order to change a course layout you can move, delete, or add individual dots.

"Edit track by hand" control gives you the information for the racetrack and possibility to change it.

Right from edit outlines control is the racetrack's view as dots:
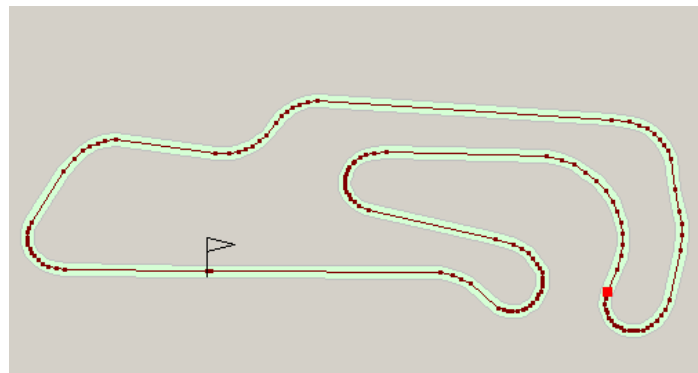
There is a quick menu available on every point of the right view. The content of the menu depends on clicked element. If a point is hit then the menu offers to delete the point, if a line is hit then the menu offers to insert a point and anywhere else it offers to save or cancel changes.
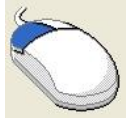
Selecting a dot

Select a **dot** by clicking on it with the left mouse button.
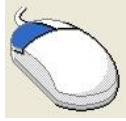
Moving a dot

Use the left mouse button (drag & drop) to ***move*** a **dot** to a new position.
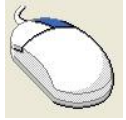
Adding a dot

Clicking onto the racetrack with your right mouse button will ***add*** a new **dot** at the current mouse position. If the location you have selected is in the immediate vicinity of the course the new dot will be inserted into the course layout at this point. Otherwise, the new dot will be placed following the selected dot. The course will now lead from the selected dot to the dot that has just been added. You can also accomplish this by clicking on the "Insert Point" button.
Holding down your right mouse button enables you to move the new dot to another position in one step.

Choosing the "Insert new racetrack point (Key Insert)" button on the screen adds a new point next to a selected point on the racetrack.

Deleting a dot

Press **Delete** on your keyboard

Choose the "Delete selected racetrack point" button on screen

Right mouse click on the dot and confirm by clicking on the **Remove point** button.

Properties of a dot

Every dot has coordinates (height also if available) and position on the racetrack. You can change all of them manually except the position. The position of the point dependents on the racetrack's length and if the track is closed or not. If the height is unknown, but neighbour points have height you can set the height to be auto and then the dot's height will be approximated from nearest neighbours with known height.

"Assign selected racetrack point Start/Finish point" Button

This button is used to define a specific point as the start/finish position. This function defines the initial point of a course layout, coinciding with the "Start" and in the case of a closed circular course the "Finish" line.

Setting the Course Length

Enter the course length in the "Length" field.

WinDarab uses this value (among other things) to standardize the length of laps during data analysis.

Rotating the Course

If necessary, use this field to rotate the entire course according to a specified angle.

"Close Course" Option

Activate this option if your course is circular. Choosing this button determines that the first and final point on the course must occupy the same position.

Determine Segmentation

In the "Determine segmentation based on the recorded data" mode you are able to automatically divide the course into segments.

To open the "Determine segmentation based on the recorded data" dialog box:

1. Open **Darab Explorer – Racetrack**.
2. Click on the **New...** or **Modify...** button.
3. Select "Determine segmentation based on the recorded data" option and click on the **Next** button.

There are two calculation methods to divide the course into segments:

- automatic generation of straightaways and curves using transversal acceleration

- free definable segmentation based on a set of rules you choose.

Calculation Method "Transversal Acceleration"

This method is a simplification of the underlying set of rules.

To carry out the actual calculation WinDarab generates a suitable set of rules
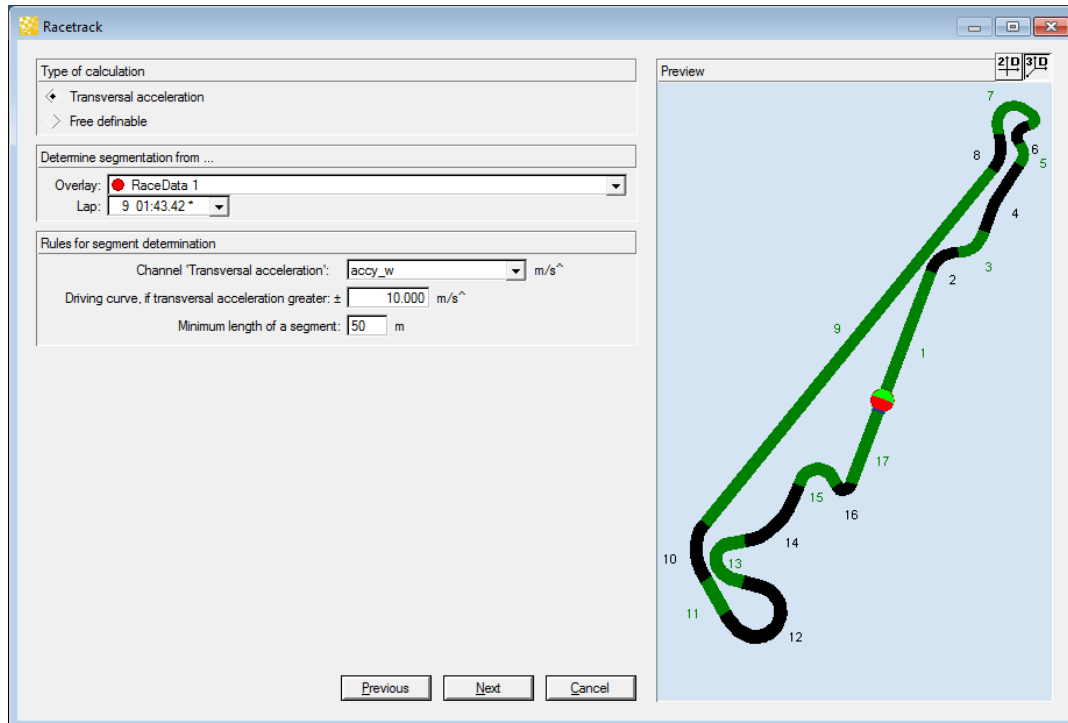
In the "File/Lap" field choose the file and lap for the course calculation.

Activate the calculation method "Transversal Acceleration".

Enter a minimum segment length.
A segment will be created only if the transversal acceleration value over this distance exceeds the set limit (threshold value).

Select the channel in which the transversal acceleration was recorded in the "Transversal Acceleration Channel"

Enter the value that must be exceeded at the beginning of a curve.

Note:
The curve ends if the g-force is 30% below the set threshold value.

Usually this calculation is carried out automatically. You can view the results immediately after you have changed a setting. This is not the case when you switch from the method that allows you to define segments ("User-Defined" button) to the automatically calculated method ("Transversal Acceleration" button). In that case you must enable the calculation manually in "Calculate Preview" or change an additional setting.

If you have used this calculation method and then select the "User-Defined" option you will be able to view and edit the rules that were generated

Calculation Method "Free definable"

This calculation method provides you with a very flexible system for determining course segmentation.

It is based on a set of rules. Each rule occupies a separate line in the table.
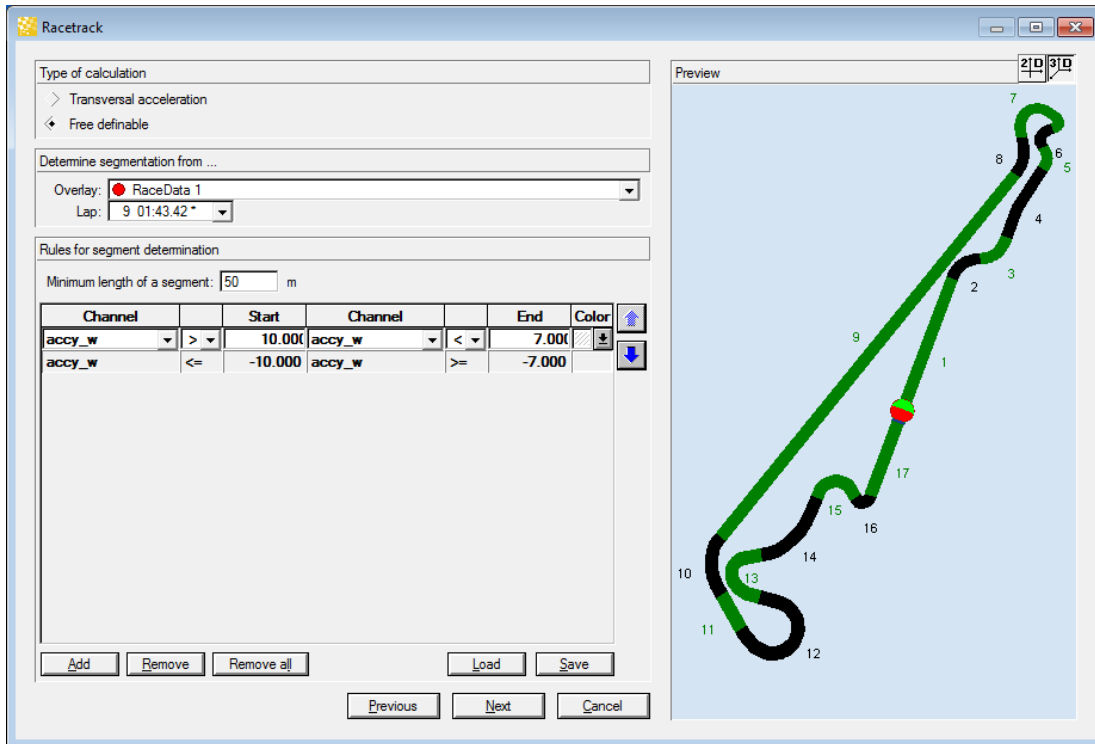
Each rule has a start and end condition as well as a color in which the segment will be drawn.

The entry fields are as follows:

Calculation Method	Activate "Free definable".
Overlay / Lap	In this filed you select the overlay and lap that will be used for calculating the course segmentation.
Minimum Segment Length	Defines the minimum length of a segment. This prevents the creation of segments caused by signal noise.
"Colors" Column	Select a specific color that will be applied to a segment if it fulfills the rule in this column. You need not to select a color (shaded button), in this case the colors you indicated in "Options/Settings/ Colors" for segments with even/odd segment numbers will be used.
Color of all other segments	This allows you to determine a color for those course segments in which no rules are fulfilled. This line appears only if you have chosen a specific color for at least one of the rules. In other words it does not appear when all rules use the default colors.
"Add" Button	Click on this button if you want to add a new rule to the set of rules.
"Remove" Button	Use this button to delete the selected rule from the set of rules.

"Remove all" Button	Delete all rules.
"Save" Button	Click on this button, to save the set of rules. This is very useful for gaining quick access to complicated rule sets.

> **Note:**
>
> The racetrack or course segmentation will not be saved. WinDarab will only save the settings made in this page.

"Load" Button	This button enables you to load a previously saved set of rules.

How WinDarab Calculates a Segmentation

1. WinDarab uses the following steps to calculate segments:
2. WinDarab drives the lap one more time.
3. While driving WinDarab checks the starting conditions for each point on the course — beginning with the first rule. WinDarab stops checking, when a starting condition applies. The vehicle now occupies this particular rule (i.e. segment). If WinDarab does not find an applicable rule it repeats its check at the next point.
4. Every subsequent point is then checked for starting conditions, beginning with the first rule. However, the check will be terminated once a starting condition is fulfilled or when WinDarab reaches the segment previously occupied by the vehicle.
5. If WinDarab finds a new rule, it terminates the preceding rule and generates a corresponding segment. The vehicle now meets the conditions of the current rule. Checking proceeds as in step 3.
6. WinDarab will then check the current rule's end condition. If this condition is not met (i.e. if the vehicle remains in this segment), then WinDarab continues checking the next course point, as in step 3. If the end condition is met, WinDarab generates the corresponding segment and checks the next rule's starting condition. If it finds a rule that has been fulfilled, WinDarab begins checking the next rule, if not it continues checking the current rule.

> **Important:**
>
> - Remember that the end condition — in a manner of speaking — represents the opposite of the starting condition.
> - You should always use a threshold value for starting and end conditions.
> - Rule sequence is important!

> **Tip:**
>
> Although a rule often uses the same channel for the first and final conditions, it is possible to vary conditions: you can use brake pressure for approaching a turn, transversal acceleration for driving curves, and throttle settings for accelerating out of curves.

Edit Segmentation

In addition to the automatic calculation of segmentation you can choose creating course segmentation manually. You can also adapt a previously calculated course for instance to remove annoying or unnecessary segments. You need to start "Edit Segmentation by hand" mode to do so.

To open the **Edit Segmentation by hand** dialog box:

Open **Darab Explorer – Racetrack**

Click on the **New...** or **Modify...** button

Select "Edit Segmentation by hand" option and click on the **Next** button

Right next to the edit segmentation control is the racetrack's view
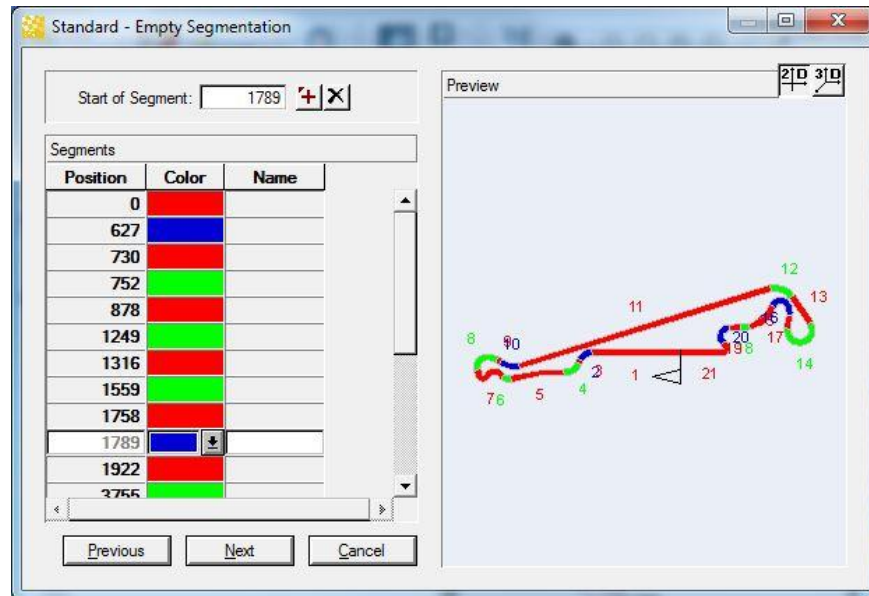
In addition to position and color this table also gives you the possibility to set a name for a segment which later will be used to identify the segment on the racetrack. If there is no name for a segment its index will be used for displaying.

 If you drag the mouse over the racetrack a small window with the current position and pointer to the racetrack shows up. So you can find a better place for adding a new segment

 There is a context menu on the racetrack. To **open** the **menu**, click with the right mouse button on a segment. The quick menu affords the following options:

1. *Change* the **color** of the selected segment.
2. *Insert* a **new segment** in the current position.
3. *Delete* the **segment** (to do this, you must exactly be on the start of the segment)
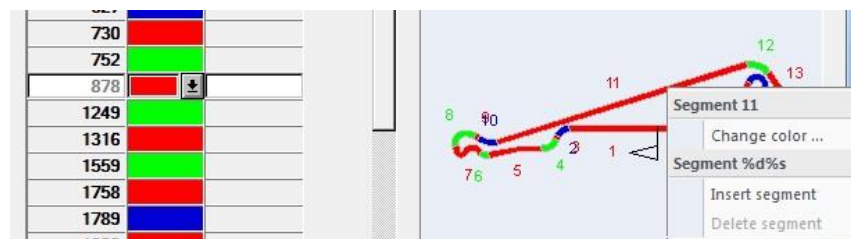
Adding a new Segment Marker

1. In the entry field, indicate a position (in meters) or use your mouse to click on the desired area in the preview. (there is a pointer with a small window showing the current position)
2. Click on button right from the segment.

A new segment marker will be added in the selected location and the segmentation is redrawn accordingly.

To add a new segment you also can use the quick menu:

1. Select the position of the new segment with the mouse.
2. Right mouse click to open the quick menu.
3. Select "Insert segment" menu point.

Deleting a Segment Marker

1. Select the position of the segment you want to delete in the table or click on the position directly on the racetrack in the preview window.
2. Click on button right from the segment.

The appropriate segment marker will be deleted from the selected location and the segmentation redraws accordingly.

The quick menu also provides the user the possibility to delete a segment marker:

1. Position the mouse pointer exactly on the start of a segment.
2. Open the quick menu by right mouse click.
3. Select "Delete segment" menu point.

Changing Segment Color

1. Choose the segment from the table.
2. Select the desired color from the list.

The segment color will be changed and the segmentation is redrawn accordingly.

The color of the segment can also be changed within the quick menu.
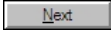
Changing Segment Name

1. Choose the segment from the table.
2. Type the desired name.

The segment name will be changed and the segmentation redraws accordingly.

If the segments name is empty then the segment index will be used to identify it on the racetrack. By default the segments don't have names.

Create segmentation

Clicking on [Next] button will show you a dialog in which you should write the name of the new segment.
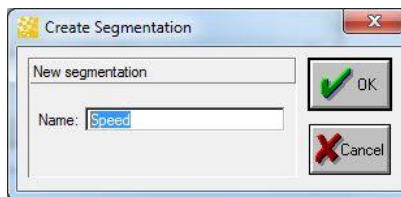
Remove segmentation

Clicking on [X] button right from segmentation deletes current segmentation. Another segmentation will be selected and displayed if available else a new empty segmentation will be added and will be shown.

Rename segmentation

You can change the name of currently edited segmentation by typing it in the edit box of the segmentations popup box.

ASCII Export

A racetrack can be export to an ASCII file.

The ASCII file contains the distance (dist) from beginning to each racetrack point and the coordinates (x, y) of each racetrack point.

 The appendix contains a format description for the ASCII file that will be created.

The file can be used to create the outline of a racetrack in other application using x/y – plot connected lines (e.g. Microsoft Excel).

There are two methods to export a racetrack to an ASCII file.

From **Racetrack** Window:

Open **Racetrack** Window as follows.

Racetrack Window   Choose **Windows/ Dockable windows/ Racetrack**.

Open context menu in the **Racetrack** Window.

Click on the **Racetrack** Window to *open* the **context menu**.
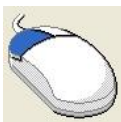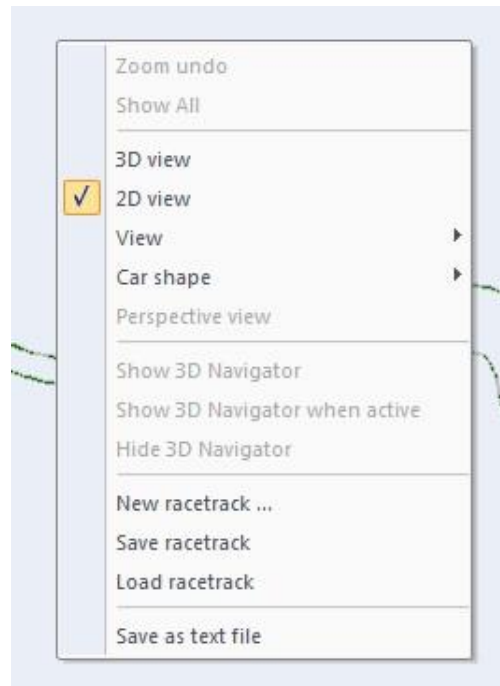
Choose **"Save as text file"** in the context menu.

The dialog **Save as** is displayed:
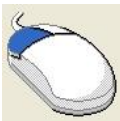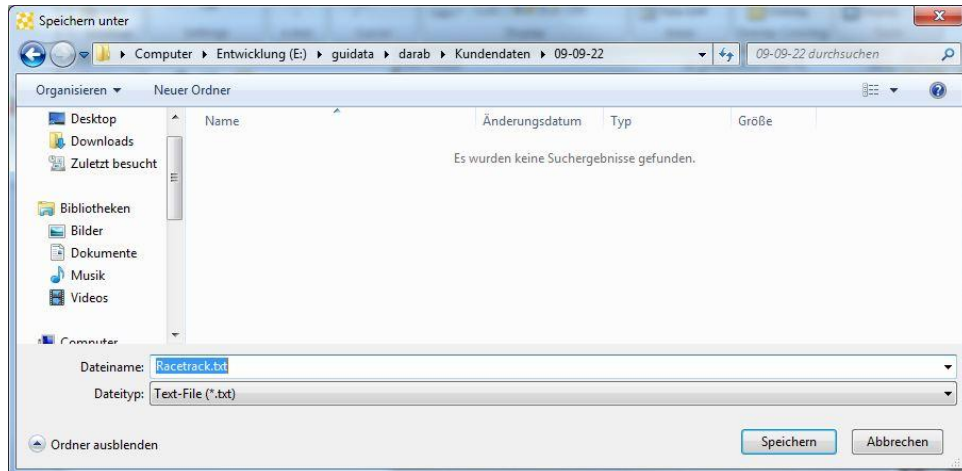
1. Select the drive and directory to *save* the **ASCII file**

2. Enter the name of the file on which the data will be written.

3. Click on the ok to confirm.

4. The racetrack will now be exported to the file you specified earlier.

From **Darab** ribbon:

Choose **WinDarab-Button/ Racetracks** in Darab ribbon

WinDarab-Button/

Racetracks

Recent Documents

Open measurement file

Close measurement file

Import/Export

Rennstrecken ...

Print

Tools

Exit
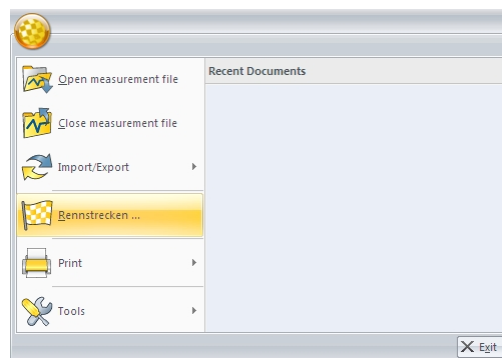
Darab Explorer – Racetrack is opened.
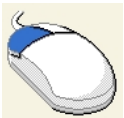
1. *Select* the **racetrack** to be exported

2. Click on the "Export" – button to *open* the **"Save racetrack as text file" – dialog** box
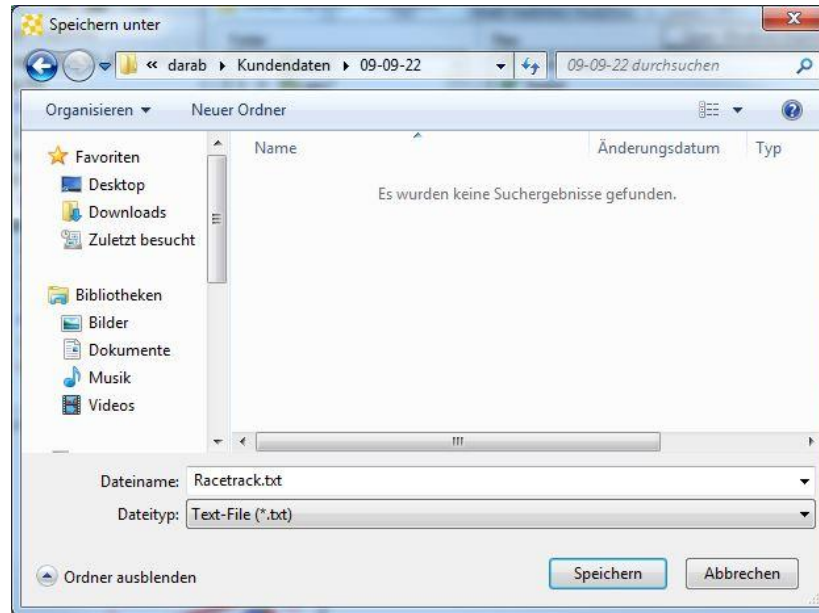
3. Select the drive and directory to *save* the **ASCII file**.

4. Enter the name of the file to which the data will be written.

5. Click on the **Save**– button.

Logged Data Files

After data recording has been completed (usually after each outing) data will be read to WinDarab and converted to the Logged Data File format.

Logged Data Files contain all data and information pertaining to a particular set of recorded data, such as signal values for individual channels, lap and status reports, and so on.

Logged Data Files are also called **Measurement Files**.

You may open more than one file and more than one copy of a file at one time. Once opened, files can be compared. In order to distinguish files clearly from each other, every file receives a specific color and index code, both of them are always displayed in conjunction with the file name.

Logged Data Files are managed in the File Explorer (Open, Delete, Import Files from other programs).

The files are closed with a separate button in the File Explorer Windows called "Close File".
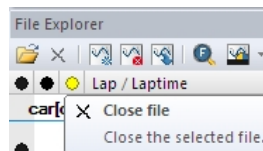
Each Logged Data File contains a default comment field in which you can enter any text comment you like. You can, however, also define additional comment fields which WinDarab provides with every Logged Data File. A distinction is made between text, numeric, and date fields.
Comment fields may also be used in mathematical channels to include settings that are specific to a particular outing (i.e. outing-specific constants).

Opening a Logged Data File

Before you can access the data and channels contained in WinDarab, you must first open a Logged Data File in the File Explorer.

Chapter "File Explorer" describes how to open the **File Explore**.

Follow these steps to open a Logged Data File:

1. Open **File Explorer**.
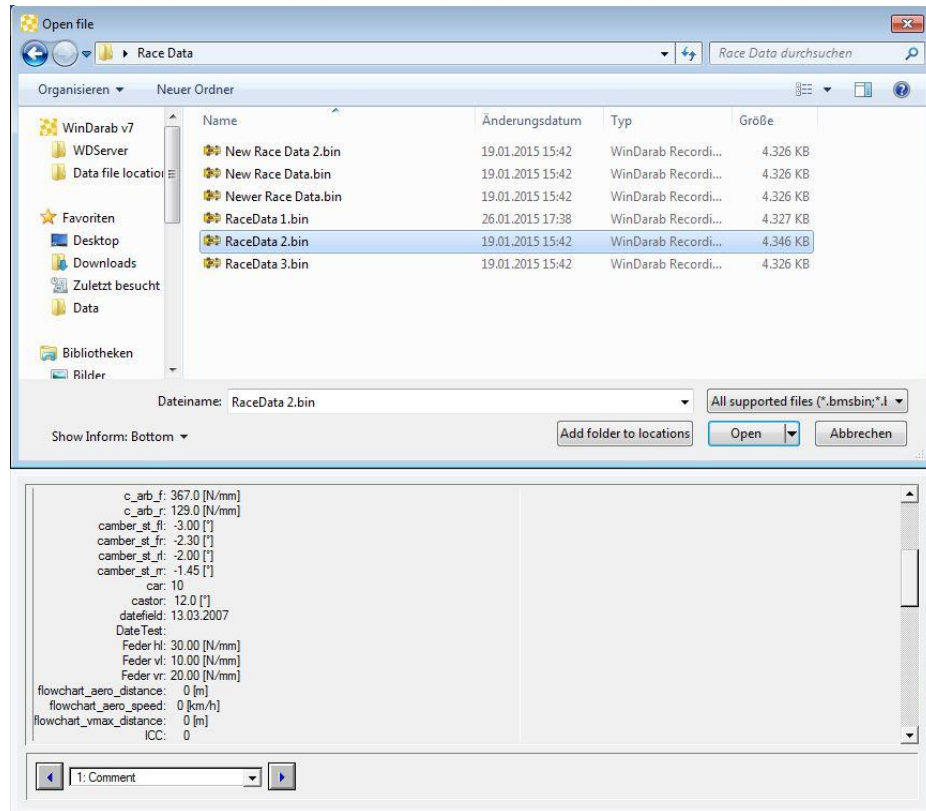
2. Select the Logged Data File(s) you want to open.

3. Click on the **Open** button.

4. The Logged Data File(s) you have chosen will now be opened.

5. If the Oscilloscope you are currently using is empty, you can now load a configuration. If the window already contains a file, the same channels as those already displayed in the Oscilloscope will now be added to the window for the file(s) you have opened

More than one file may be open at one time in WinDarab.

Oscilloscopes are also called **Measured Data Window**.

Note:

Each opened file receives a color and numerical index code. In the File Explorer Window display the color code precedes the index number and the file name follows. In this way, files are clearly distinguishable from each other. A colored dot serves as an additional identifier.

You can determine the allocation sequence for colors in **"Tools/ Program"** in the menu.

Closing a Logged Data File

There are two methods to export a racetrack to closing a Logged Data File.

From **File Explorer**:
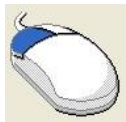
Select in the **File Explorer Window** the **Logged Data File** you want to *close*.
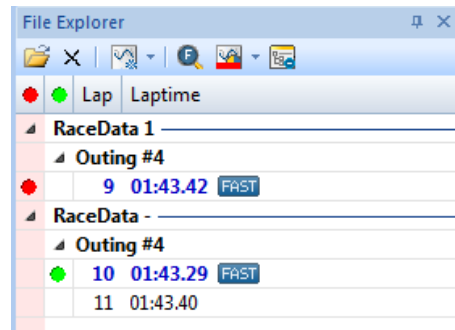
Choose **Close File** in the **File Explorer Window** Toolbar.
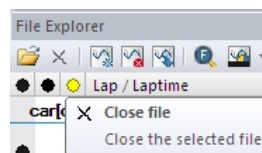
The file will now be closed and removed from the file list.

If desired, you can close several files, one after another.

From **Darab** ribbon:

Choose **WinDarab-Button/ Close measurement file** in Darab ribbon

WinDarab-Button/
Close measurement file
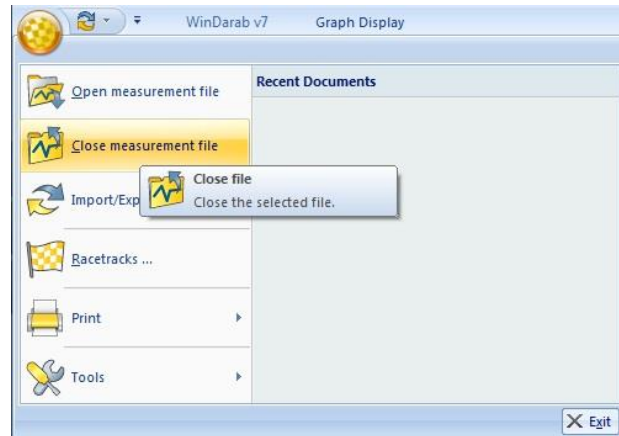
Tip:

Closing a particular Logged Data File removes all active channels from WinDarab windows. Similarly, all windows that are exclusively used to display data contained in the just closed Logged Data File will also be closed.

Creating a Logged Data File

WinDarab provides a number of possibilities for creating Logged Data Files.

Darab Explorer supports the transfer and conversion of Logged Data Files from other sources. You can:

- Read and convert Logged Data File in DOS Darab format (see chapter Historical Cars)
- Create a Logged Data File from an ASCII file.

Create a Logged Data File from an ASCII file

Follow the steps to create a Logged Data File from an ASCII file.

1. Select with **WinDarab-Button/ Import/Export/ Import from text file** the folder in which you want to save the imported Logged Data File.
2. The dialog box "Convert ASCII file" appears.
3. Select the ASCII file and the "Channel Definition" file.
4. Confirm your selection by clicking on "OK".
5. The file is being imported, now. A progress bar monitors the process.
6. After the file has been copied successfully, the dialog box "Save Logged Data File" appears.
7. The process is complete once the file has been saved.

Note:

Both file structure and format for ASCII and channel definition files are described in the Appendix.

Reading Logged Data

Various systems are available to record Bosch Motronic® vehicle data. WinDarab supports the following:

- Data Logger C55/C60/DDU8
- CardMemory C40/C5 (Windows 95 only with OmniDrive or SoftDrive; supported directly in Windows 98/Me/2000/XP/7/8)
- Others
  - MP 1.8
  - DDU8 Longterm Memory
  - CANLogger C45
  - 4 MB Reader / CanCard (PlugnPlay version) (see chapter Historical Cars)

Using the data logger import dialog

The data logger import dialog is a tool to import data from various sources such as burst, device, flashcards or USB-sticks.

WinDarab-Button/ Import/Export/ Import from data logging device

To transfer data from the hardware you must call up the menu item **"WinDarab-Button/ Import/Export/ Import from data logging device"**. The data logger import dialog will appear.

It is possible to minimize the import-dialog. It will then only accessible through the tray icon in your taskbar. However, the process can still be observed as the balloon-notification pop up whenever download is starting, a source has connected or a file was merged.

The dialog contains of three tabs to control the settings and the status of the importing.

Settings-Tab

The settings tab helps you customizing the import-procedure.
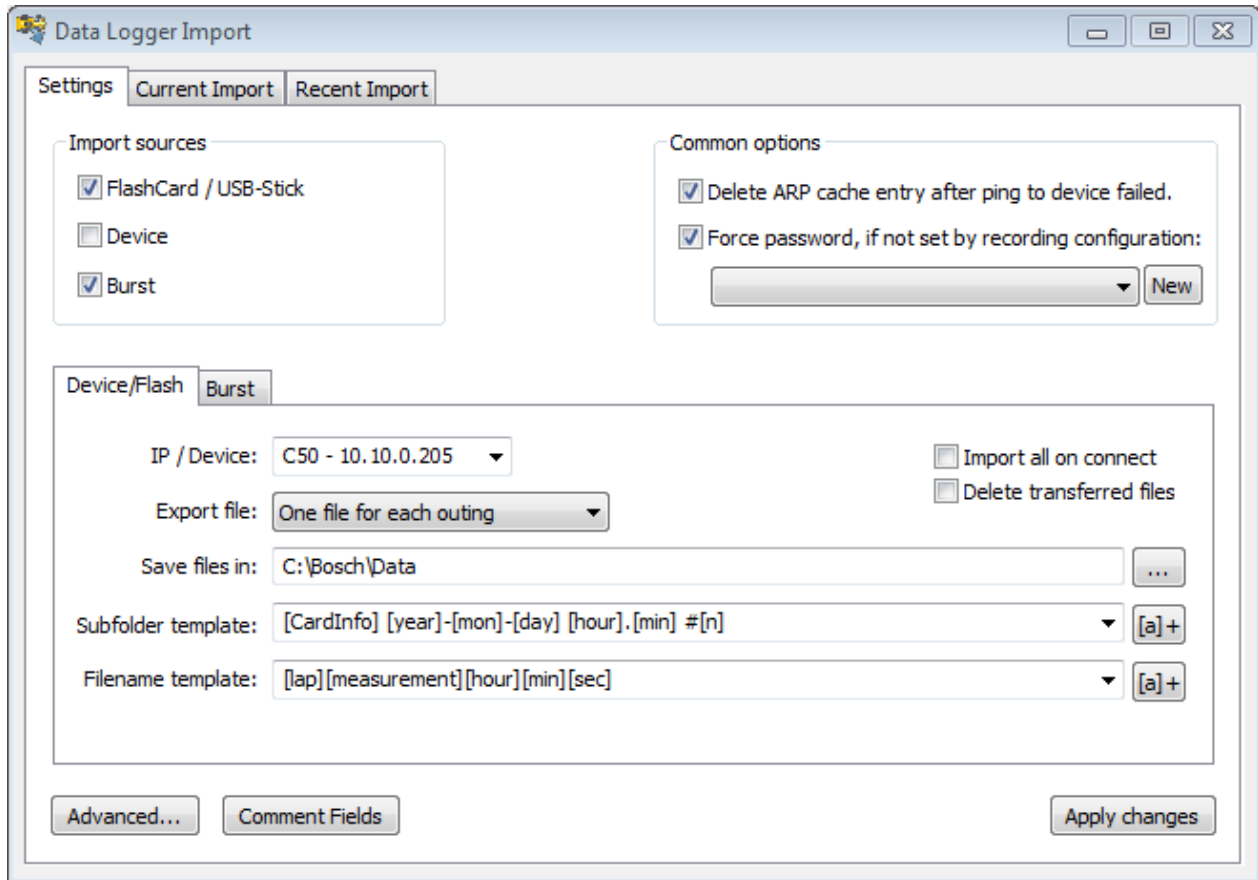
Import sources Select the sources that you want to monitor. Available are **FlashCard/USB-Stick**, **Device** and **Burst**.

If the **FlashCard/USB-Stick** checkbox is checked, an inserted **flashcard** or **USB-stick** will be recognized and shown in the current import-tab.

Device and **Burst** enables the monitoring of the IP-Addresses, that are chosen in the burst or device-settings visible in the lower half of the dialog.

Note: Device and Burst can't be selected at the same time. If you enable one of them while the other one is enabled, the other one will be disabled.

Common options Here, the password for the resulting files can be set. You can select a previously set password or create a new password with a click on the **"New"**-Button. If the **"Force password…"** option is disabled, no password is set when it is not explicitly set by the reading configuration.

The **"Delete ARP cache…"** option deletes the ARP-entry for the host-IP if a ping failed. Use this option if you can't reach the host even if you know it is reachable.
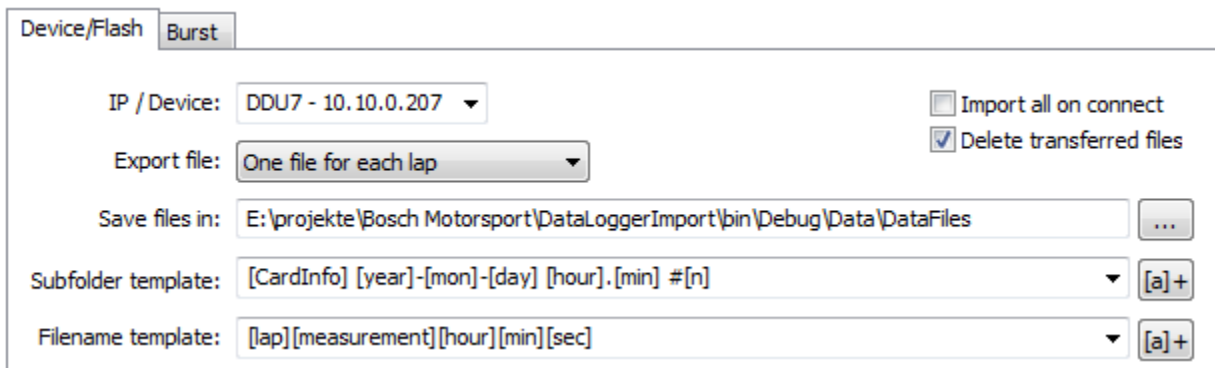
Note: The settings that you change need to be committed before they are applied to the import. Do this by clicking the **"Apply changes"**-button.

Source Settings

The individual settings for the sources are on the bottom half of the dialog.

Note: The individual settings are only shown when the corresponding import sources are enabled. For the Device/Flash settings to be shown, at least one of these sources has to be enabled. Accordingly, burst has to be enabled to show its settings.

IP / Device	In the IP/Device-field, you can chose from a set of pre-defined IP-addresses or insert your own IP-address. This has no effect for Flashcards or USB-Sticks as they have to be inserted into the PC where WinDarab is running.
Export file	The export file-selection defines which fragments should be combined to one single file. **"One file"** merges every imported fragment into one big file. The option **"as stored on the device"** doesn't merge any fragments and results in one file for each fragment. The two other options merge every fragment of the same lap or the same outing into one output-File.
Save files in	This field specifies the folder, where the resulting files are stored in.
Subfolder template/ Filename template	The subfolder- and filename-template specify the filename that is given to the resulting file(s). This field also contains a dropdown menu with the 5 most recently used templates for the subfolder- or the filename-template. There are a lot of placeholders for the template which can be inserted manually or chosen by the template-button right to the combo-box.

The following wildcards may be used:

[year]	Year*
[mon]	Month*
[day]	Day*
[hour]	Hour* (24 hour format)
[min]	Minute*
[sec]	Second*

[cardinfo]	CardInfo entry (configurable through RaceCon Recording name)
[lap]	Number of first lap
[outing]	Number of outing (of the first lap)
[fragment]	Number of file fragment
[n]	Consecutive numbering (allowed in file names only) With the exception of this wildcard all wildcards will be included in the file name. The system then searches a folder for files with the same name. The new file will receive the numerical value +1 of the file with the highest number in use so far.
[fastlap]	Fastest lap (allowed in file names only)
[fastouting]	Outing containing the fastest lap (allowed in file names only)
[fastlaptime	Laptime of the fastest lap (allowed in file names only)
[RecDateYear]	Year of the recording
[RecDateMon]	Month of the recording
[RecDateDay]	Day of the recording
[RecStart]	Time the recording was started
[RecEnd]	Time the recording ended

* of data reading

Note: If you use a finer template than the merge-option, it will be replaced by '-1'. So if for example all fragments of an outing are merged and you are using the [lap] template, it will be replaced by '-1', even if there is only one lap in the outing.

Note: You can also specify a format for the following templates: [outing], [lap], [fragment], [fastlap] and [fastouting].
Add a number between the template name and the closing bracket to specify the shown digits.
For example: if [fastouting] results in 12, [fastouting03] results in 012.

In the top right corner there are two more checkboxes.

Import all on connect	This option has to be checked in order to import all new files directly - once a source is found. For example, the flashcard-data is imported right after the card has been inserted into the computer. If this option is not enabled, you have to manually import the files in the current import tab. Additionally you can chose exactly the fragments you want to import.
Delete transferred files	With this option enabled, the source files on the device are deleted after they have been successfully imported.

Burst settings

The burst-settings-page is almost the same as the Flash/Device settings page. The difference is that there are some missing settings.

You can't chose the export file or select "**import all on connect**" or "**delete transferred files**" in the burst settings. If you import burst data, you always create one file per fragment, the import is always started right after the connection and the files are always deleted after a successful transfer.

Advanced Settings

Further settings can be found by clicking on the **"Advanced…"**-button.
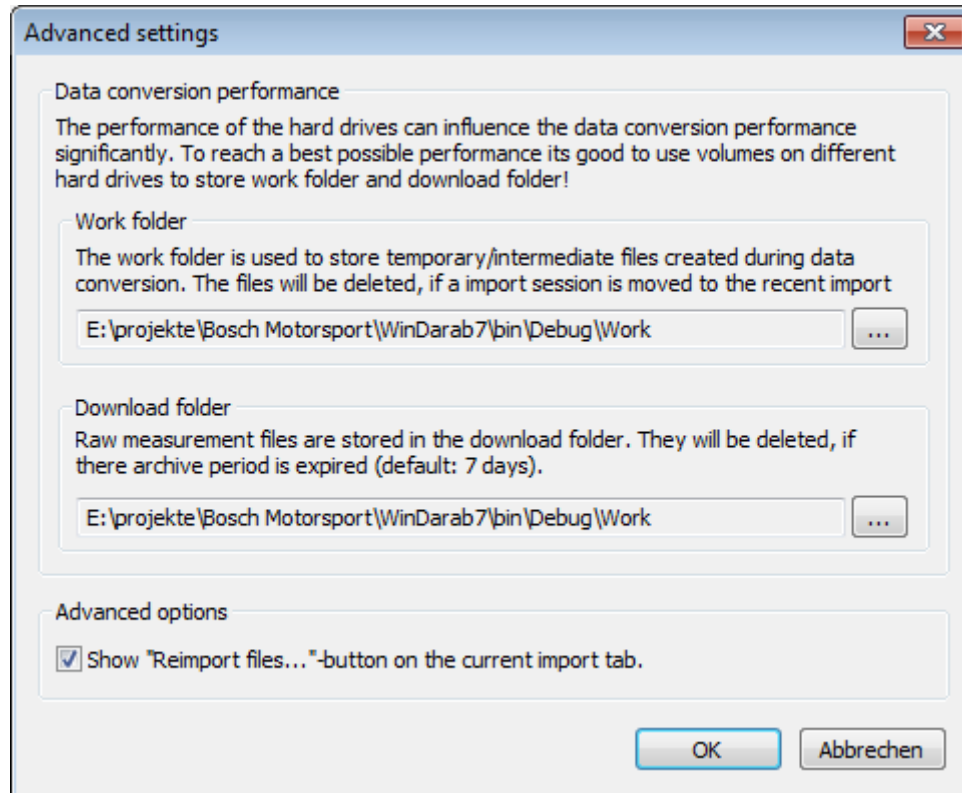
In the advanced settings, you can configure paths for your downloads and your work-path. For more information about the import-process see Import Process.

You can also change the visibility of the **"Reimport files…"-button**. For more information about the **"Reimport files…"-button** see Current import-Tab.

Comment Fields

You can view and edit the default values of the comment fields by clicking on the **"Comment Fields"** – button.
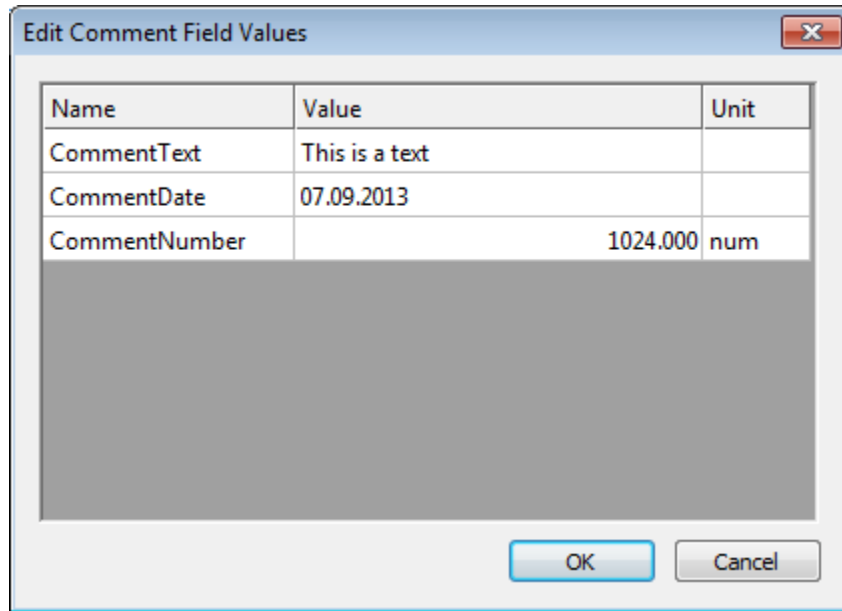
In the Comment Fields dialog you see all user defined Comment fields with their name, the unit and the default value. You can edit the default value in this dialog by selecting the value cell of the comment field.

Current import-Tab

The current import-tab shows you the import-state of your currently active data-source.
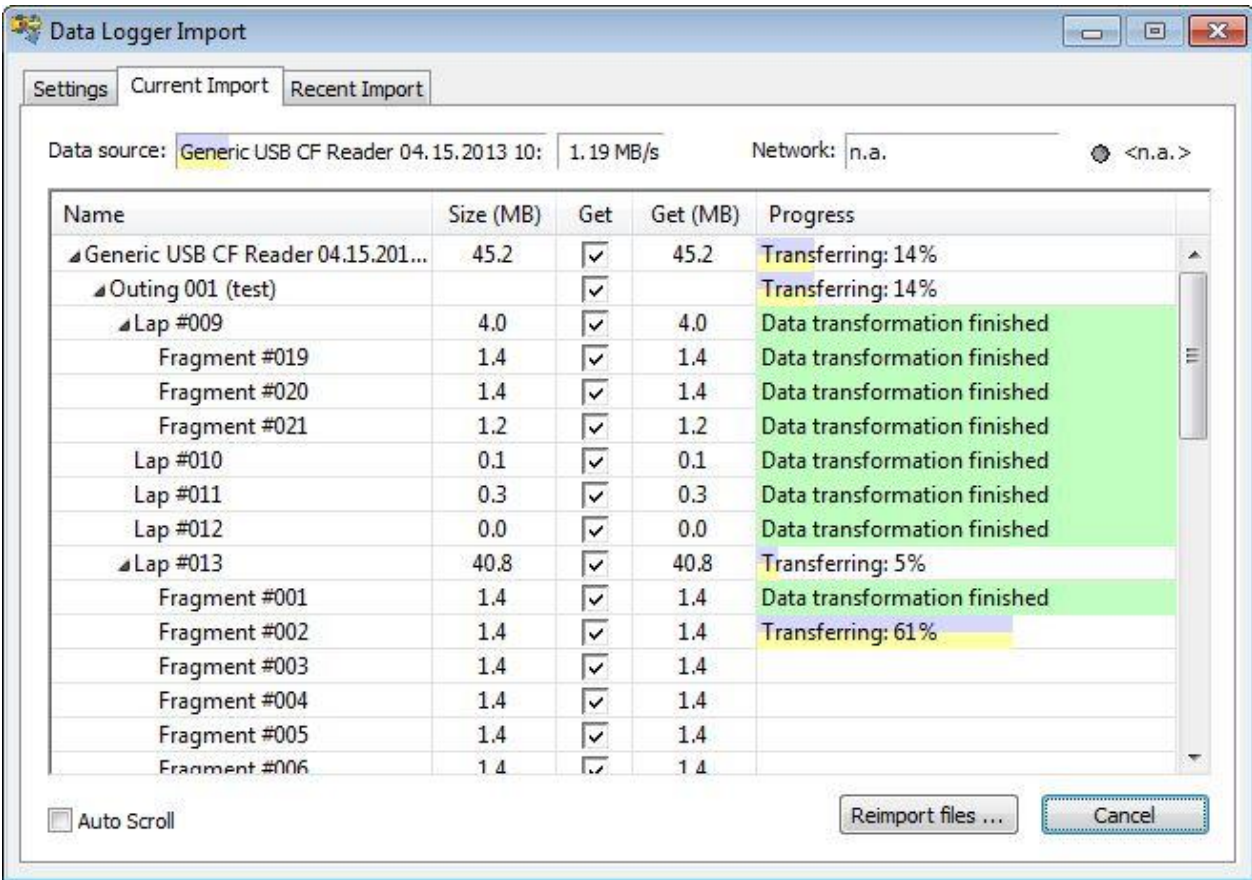
The top row changes according to the fact weather there is currently an active import or not.

Without an import-run, the **source-field** shows information about possible actions you could do like inserting a flashcard or status-information like searching for a burst-device.

If an import is active or a source is found, the **"Data source-field"** shows the connected data-source. Also it indicates the status of the import with colored bars.

The field right to the data source field shows the **download-speed**. This is only shown when you are in the download-phase of an import.

The last field shows the **network** you are connected to. In addition you see the **latency** to this host.

The main part of this page shows the import status.

At the beginning of the import run, it is possible to select or deselect any fragment you want or don't want to import in the **"Get"**-column. This requires the **"Import all on connect"**-option to be disabled in the settings-tab.

Note: Initially the check marks are set for the files that have never been imported from this source. So if a fragment is imported but not deleted once, it will not be checked initially (and thus it won't be imported when the **"import on connect"** option is enabled.

With a click on Import, the import-process is started. Now you see the status of each fragment, lap, outing and so on as the import-process progresses.

First the data is downloaded from the device and converted into a temporary file. After the conversion, the files are merged together as selected in the settings-tab. If the import was successful, the items are marked with a green background.

You can cancel the importing by clicking on **"Cancel"** while the importer is running.

The dialog automatically selects the current active fragment. You can disable this feature by using the scrollbar or disable the auto-scroll-option at the bottom of the page.

When the import is finished, the Progress-column shows the success of the import-process.
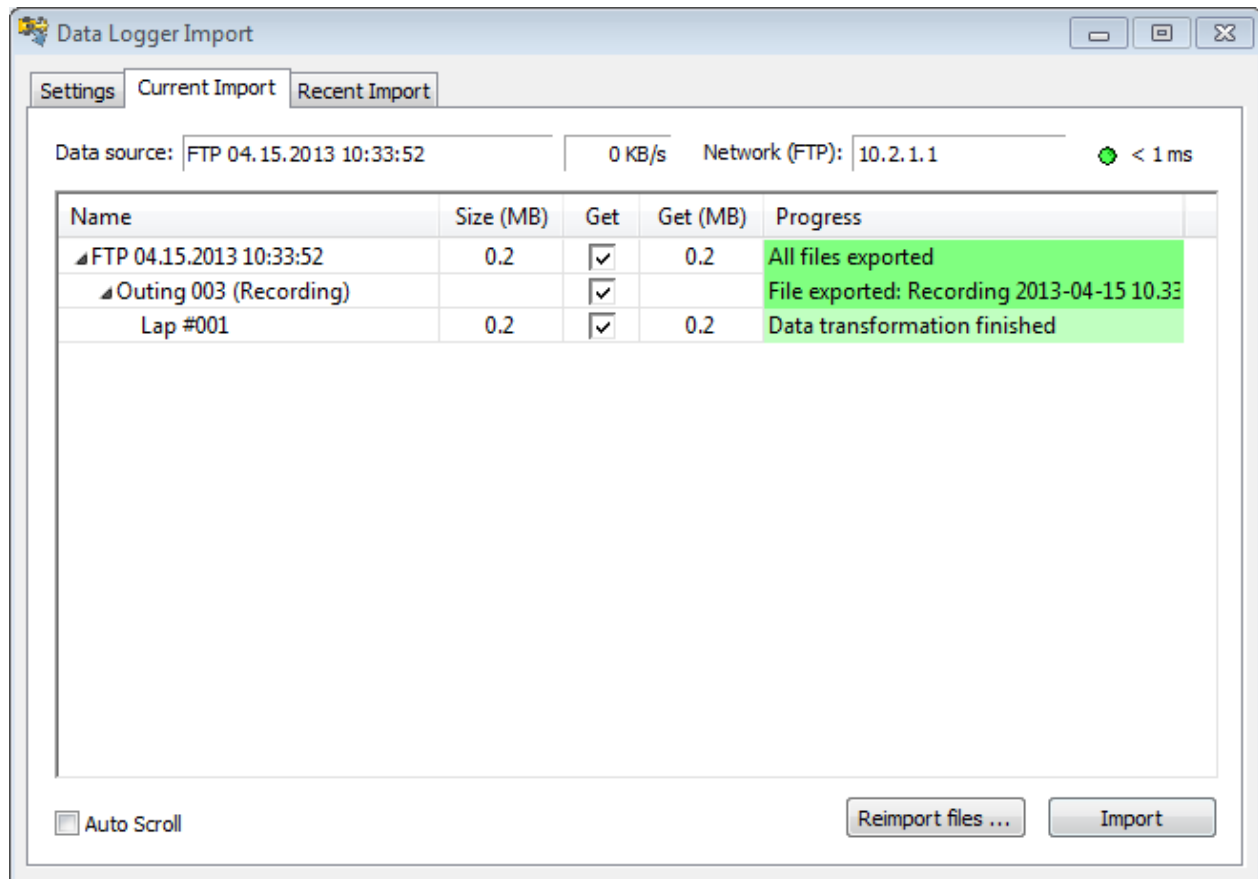
Reimport files….
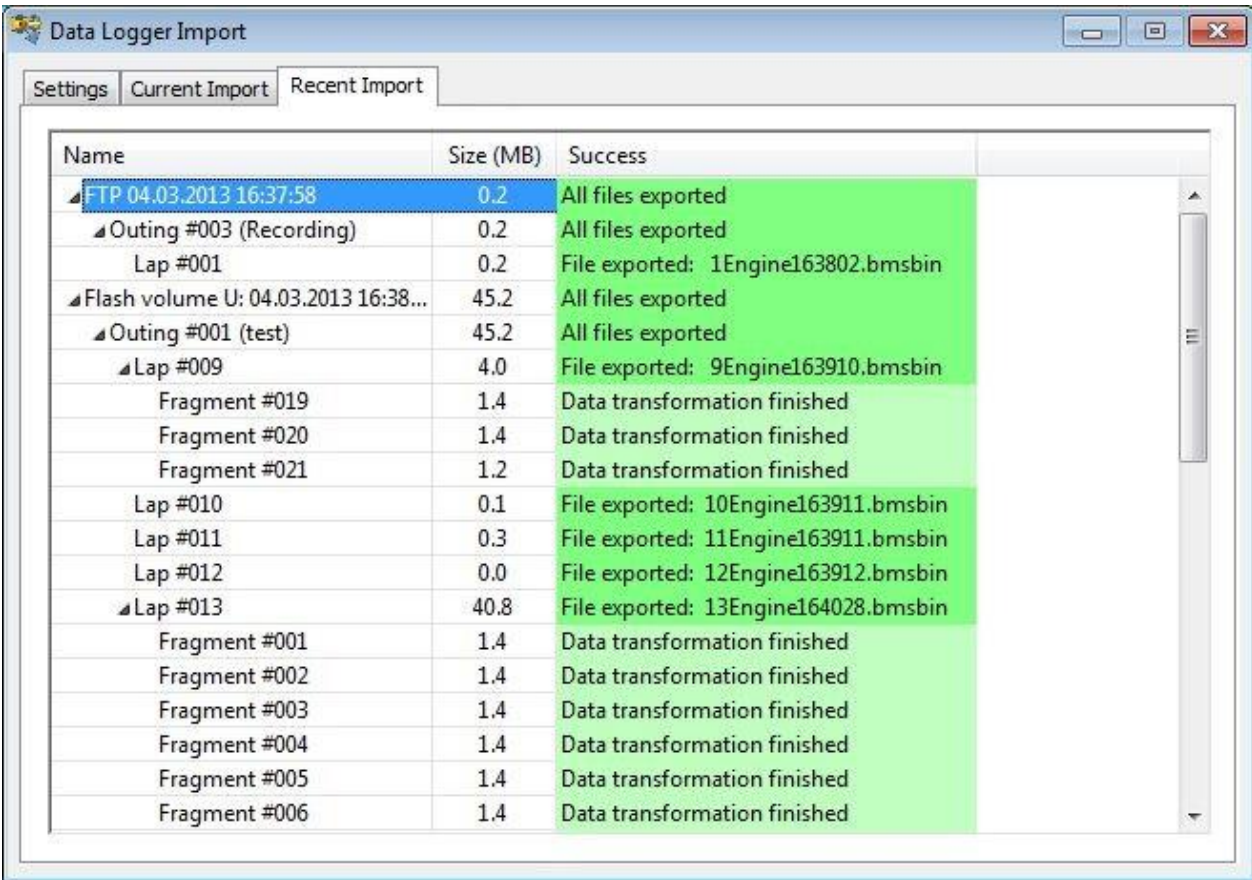
You can import already downloaded raw measurement files by clicking on **"Reimport files…"** and selecting the files from the work-folder. This is especially useful when you have to import raw data again but don't have access to the source device.

Note: If there is no **"Reimport files…"-button** on the Current import-Tab you have to make it visible by selecting the option in the Advanced settings.

Recent import-Tab

The recent import-tab shows the recent imports and their success.

In this tab you find all previous import runs and their success-status. As you can see in the screenshot above, all import-runs have their own root-entry that can be expanded like in the current import tab.

An import run is automatically added when the source is removed and the import is completely finished.

The status of every imported fragment is displayed in the success-column. Here you see which fragments were imported correctly, and which were not imported, which files have been created and which downloads have failed.

Import Process
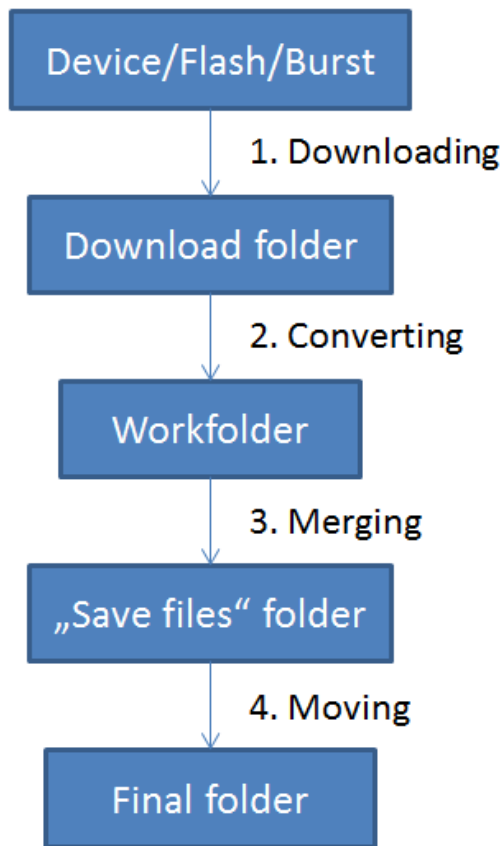
The Importing of a file runs in four steps that can run partial parallel.

1. The file is downloaded from the device. In this step, the data is copied from the device to your local hard drive. You can specify this folder under **"Settings -> Advanced... -> Download folder"**

2. The downloaded file is then converted into a temporary file. This step doesn't need the file to be downloaded completely. As soon as there is data downloaded to your hard drive it is converted. As the download folder, the work folder can also be specified in advanced settings.

3. When all parts for the final merging (according to the **"export file"**-option in the settings-page) are converted, they are merged into a single file that can be used in WinDarab. While the merging is in progress, a temporary file is written into the **"save files"**-folder specified in the device settings.

4. When the file is merged completely, it is moved to its final folder (**"save files"**-folder combined with the specified subfolder) and renamed according to the template as defined in the device-settings.

After the files have been merged successfully, the temporary files are deleted.

Tips for setting the paths

As you can see, you have many options to control the importing by setting the paths. And there is also much performance increase that can be gained by setting the paths wisely.

In almost every step, files are read and written to your hard drive(s). So if you want a fast download and have more than one hard drive installed into your pc, it is advised to select your download folder to be on another physical hard drive than your other folders. This way the download can use the full transfer rate of the hard drive.

Also have an eye on the file-sizes. The files on your device are usually compressed.

As the data-import is optimized mainly towards speed, the imported files are uncompressed and can be significantly larger than the original data. Additionally, there are the downloaded, converted and merged files saved on your hard drives during the import process, so make sure you have enough free space to support it.

Importing Longterm Data

You can import the longterm data from a device by using the longterm importer.

WinDarab- Button/ Import/Export/ Import from data logging device

To transfer data from the hardware you must call up the menu item **"WinDarab-Button/ Import/Export/ Import from data logging device"**. The data logger import dialog will appear.

As the Current Import- and the Recent Import-pages stay the same, the Settings-page differs from the usual import dialog.

Settings-Tab
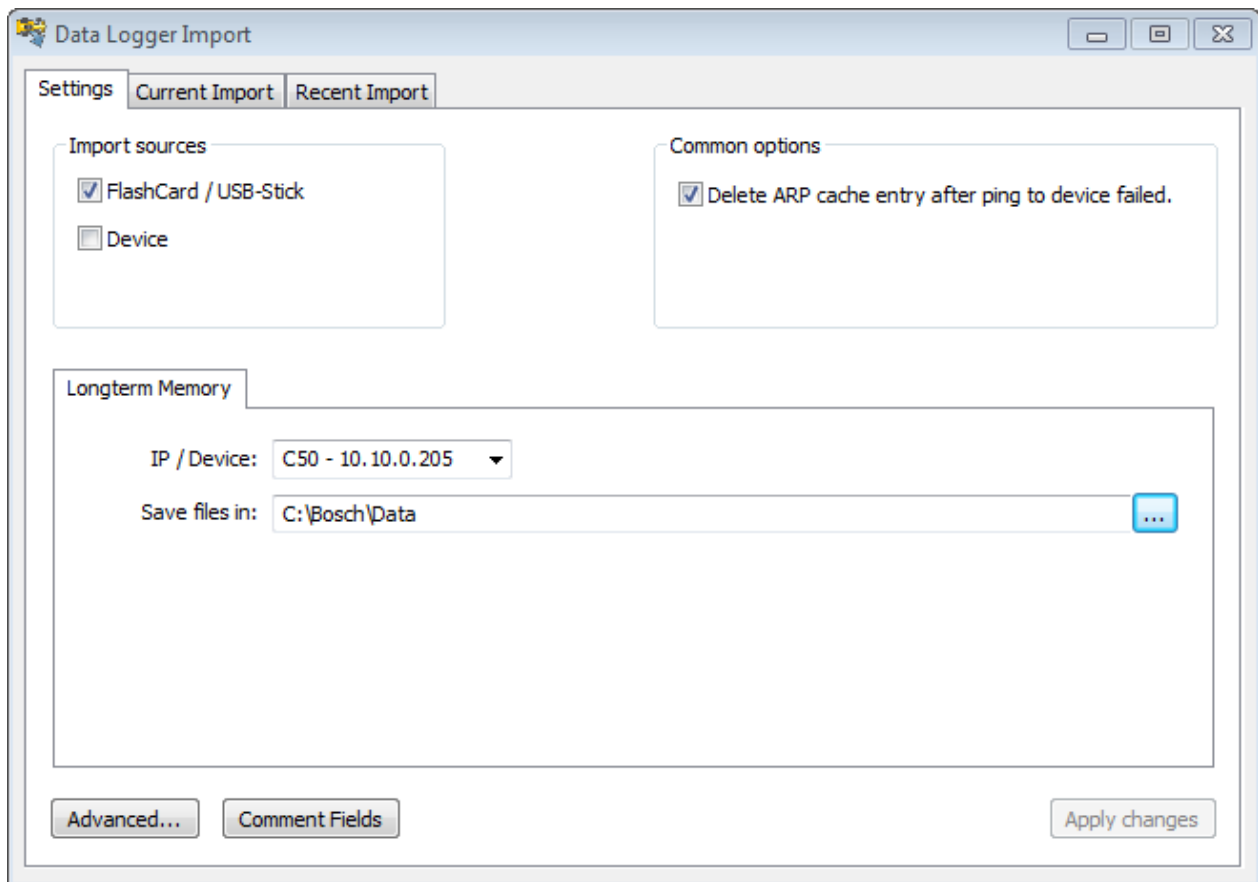
The import sources allow you to choose between FlashCard/USB-Stick and Device. The burst-transmission is not allowed in the longterm data import.

Also there is only one settings-tab for all import source.

There you can set the host to connect to for the device import source and the destination-folder for the files. The other parameters that you could manage in the usual import dialog are predefined and can't be changed.

The files that are imported from a longterm device are not merged. You get one Darab-file for every file on the device.

Additionally, the files are imported on connection and they are not deleted after the transfer.

The filename is also predefined as **"[CardInfo] [outing]-[lap]-[fragment] #[n]"**.

File Password Protection

Change File Password

You can change the password of your files after the import by selecting **WinDarab-button/ Tools/ Change file password/signature**.
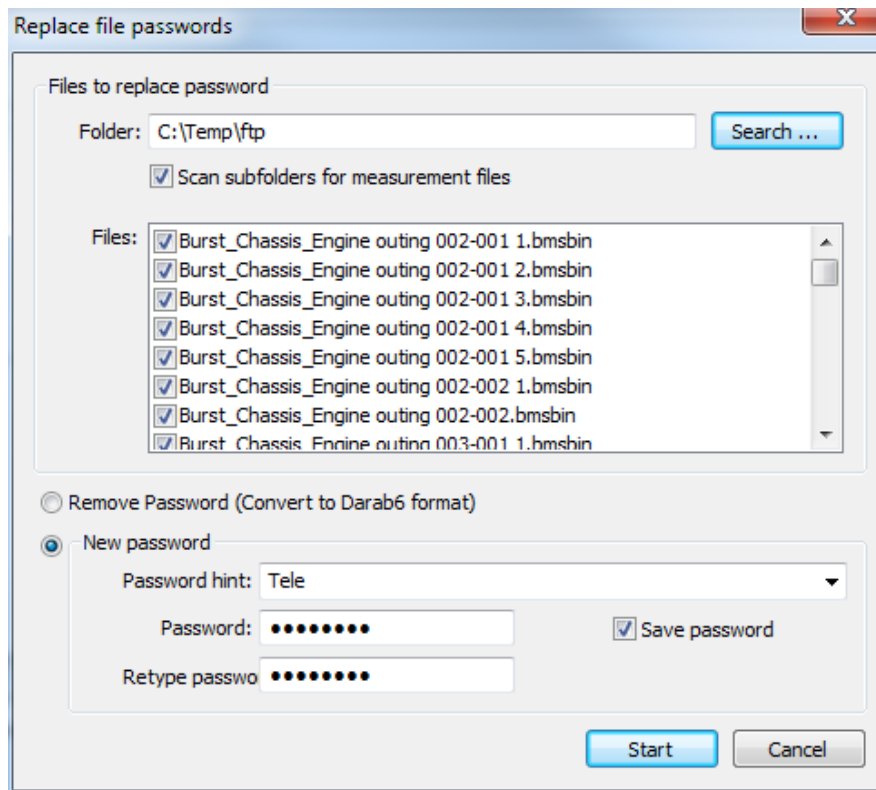
Chose the files that you want to protect with a new password. You can search for files in a given directory by clicking on "Search ..." and selecting the directory you want to search.

Check "Scan subfolders for measurement files" if you want to include the directorys subfolders into the search.

Setting a new Password	Select the option **"New password"**.
	Check all files that you want to protect with a new password. Insert a password hint and the new password into the fields and check **"Save password"** if you want the password to be saved. Alternatively you can select a predefined password hint.
	Now press **"Start"** to protect the selected files with the new password.

Removing the Password

Select the option **"Remove Password (Convert to Darab6 format)"**.
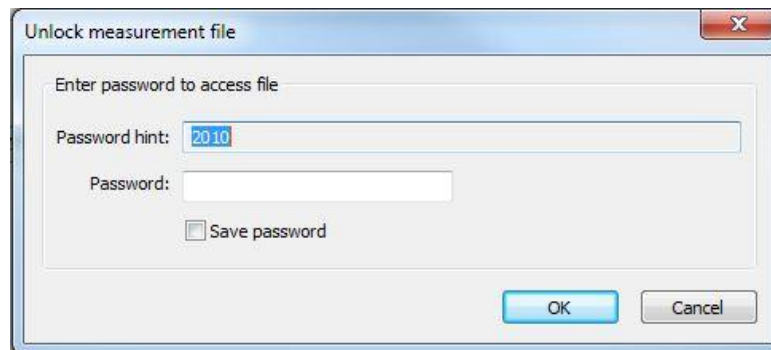
Check all files whose passwords you want to remove and press **"Start"** to remove their password.

Note:

Removing the password will reactivate the project protection of the ECU. There is no way to create a unprotected WinDarab v6 compatible file !

Open Password-protected Files
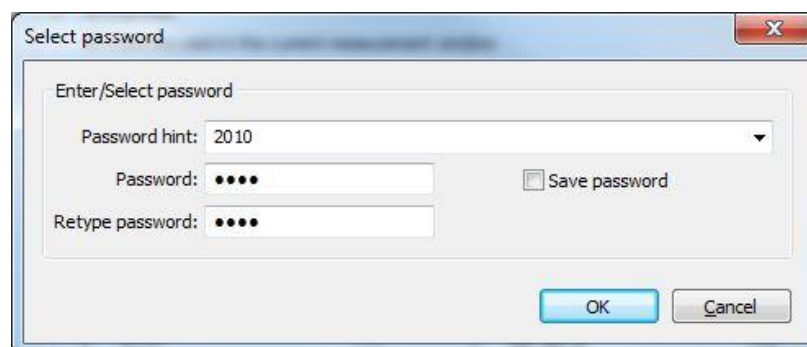
If you open a password-protected file you will get the following prompt:

Type in the password for the file associated with the hint and press **OK**. If you want the password to be saved for you, check the "**Save password**" box.

Creating a Password-protected File

If you want to protect an imported or extracted file with a password, you have to check the "**Set password**" box at the save file dialog. After you press save, a prompt is shown where you can set the password for the file.

Insert a password hint and the new password into the fields and check "**Save password**" if you want the password to be saved. Alternatively you can select a predefined password hint.

Press **OK** to complete the task.

Data Protection and Downward Compatibility

If WinDarab format is written a password can be set. The Password can also be set in RaceCon. By reading logged data into WinDarab the password is prompted if the option "Force Password" is set (see chapter Reading Logged Data)

In WinDarab v7 you must first enter the password before you can open a protected logged data file. WinDarab v6 can't open password protected logged data files.

With WinDarab v7 you can also open data written in WinDarab v6 if the license corresponds to the PCode.

Using Wildcards in Sub-Folders and File Names

You may use wildcards to allocate sub-folder and file names automatically. When you save files the variables will be replaced with information provided by the file.

The following wildcards may be used:

[year]	Year*
[mon]	Month*
[day]	Day*
[hour]	Hour* (24 hour format)
[hr12]	Hour* (12 hour clock)
[ampm]	Part of day* (12 hour format)
[min]	Minute*
[sec]	Second*
[car]	Car Identifier provided by status block (for CardMemory Measurement (C4x Logger))
[cardinfo]	CardInfo entry (configurable through RaceCon Recording name)
[fastlap]	Number of fastest lap
[fastlaptime]	Time of fastest lap
[lap]	Number of first lap
[outing]	Number of outing (of the first lap)

[fragment]   Number of file fragment

[n]   Consecutive numbering
                     (allowed in file names only)
                     With the exception of this wildcard all wildcards will be included in the file
                     name. The system then searches a folder for files with the same name. The new
                     file will receive the numerical value +1 of the file with the highest number in use
                     thus far.

* of data reading

Format directives can be appended to numeric wildcards.
E.g. [lap3] will format the lap number as right aligned text with three characters. [lap03]
will format the lap number with leading zeros instead of spaces.

The same restrictions that apply to the allocation of sub-folder and file names in Windows
apply here. As a result the use of the following characters is not permitted:

Colon ':'

Slash '/'

Backslash '\'

If WinDarab is unable to create a folder or file during saving a message is generated. Please
check the file name for disallowed characters and make the necessary corrections.

Not all wildcards are supports by the different data readers.

Desktop

While working with WinDarab a typical component constellation might look like this:

A Desktop consists of the opened dockable window and the worksheets you are currently processed. Each worksheet contains the worksheet controls whereby you analyze and visualize logged data.

WinDarab offers the possibility to save and retrieve a particular Desktop.

If loading a Desktop you can choose whether to view the files in a new Desktop or whether to restore a previous situation.

As opposed to simply loading a Desktop, restoring a previous Desktop constellation involves the closing of all currently opened files and the opening of those files that were in use as the Desktop was saved. In addition to the configured settings (e.g. channels) the windows will also restore the displayed excerpts and reproduce the situation exactly as it was before.

Every time you exit WinDarab the program saves the current Desktop and retrieves it the next time you

start the program.

> **Tip:**
> To suppress the automatic retrieval of the previous Desktop hold down the shift key when you start WinDarab

> **Note:**
> During work WinDarab saves the current Desktop periodically.

Loading a Desktop

Follow these steps to load a Desktop:

1. Select "**Start/ Desktop/ Load**" from the menu.

2. The "Load desktop" Assistant will appear.
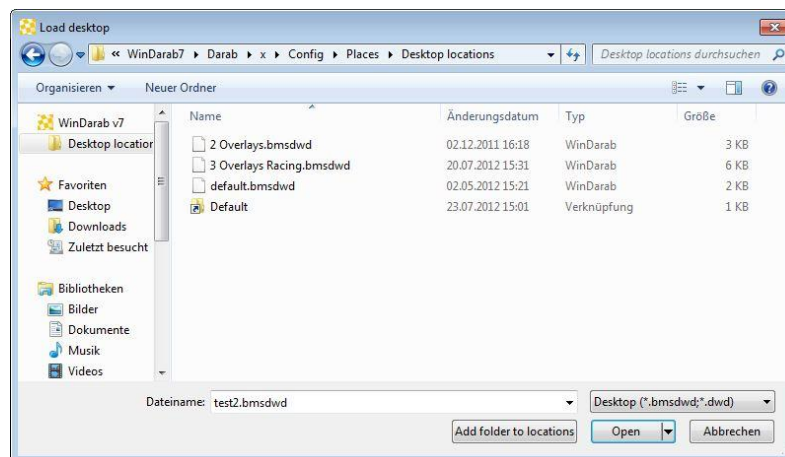
3. Now you have two options to open the selected desktop. Click on "**Open**" to open the desktop with the currently opened files
    You can also choose "**restore desktop with files**" from the dropdown menu to replace the currently opened files with the files stored in the desktop-file.

> You can also use the **mru-function** to open a desktop. Click on the **dropdown button next to the load button** and select the desktop you want to use. Press the **ctrl-key** while doing so to loaded the desktop with the files stored in the desktop-configuration.

Saving a Desktop

Follow these steps to save a Desktop:

1. Select "**Start/ Desktop/ Save**" from the menu.
2. Save Desktop will appear.
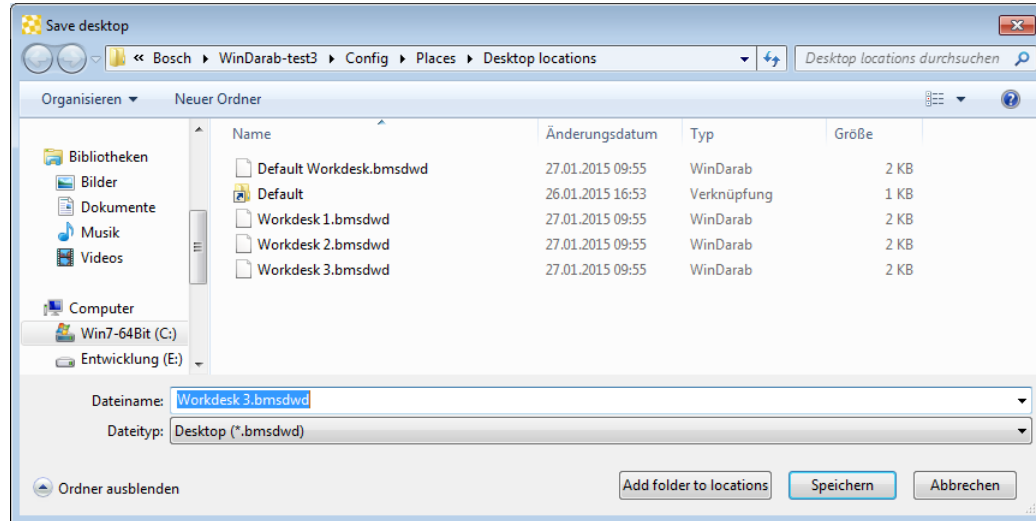
3. Select the folder in which you want to save.
4. Enter a file name for the Desktop and click on **Save**.

Oscilloscope

The Oscilloscope is the most important window for the analysis of logged data. This window displays the signal curves of channels recorded one or more Logged Data Files.

To open a new Oscilloscope, select "**Windows/ Worksheet controls/ Graph display** " and drag it onto a worksheet.

You can also *right-click on an emty part of a worksheet* and select "**Graph display**" to open an **Oscilloscope**.

Oscilloscopes are called Measured Data Window.

Important:
WinDarab allows you to open more than one Oscilloscope at the same time one of them is always the "active Oscilloscope". For example, even if you have just activated an Analysis Window WinDarab nonetheless considers the most recently used Oscilloscope as "active Oscilloscope".

An Oscilloscope displays the following:

- one or more measuring areas
- channel values for all displayed channels
- used files

Tip:

To change the width of the Channel Values display area move your mouse across the area's left-hand boundary until the mouse pointer changes. You can now drag the boundary to the right or left.

Areas

An Oscilloscope window can be divided into areas to organize your channels.

Creating an Area

There are the following methods to create an area.

From **Darab ribbon**:

Choose **Start/ Areas/ Insert** from the menu.

Start/ Areas/ Insert   An Oscilloscope always contains at least one area.

From **Oscilloscope** context menu:

An area can be created using the Oscilloscope local menu also.

Click with the left and right mouse (or middle) button on the Oscilloscope to *open* the **context menu** and choose **Area/ Create Area**
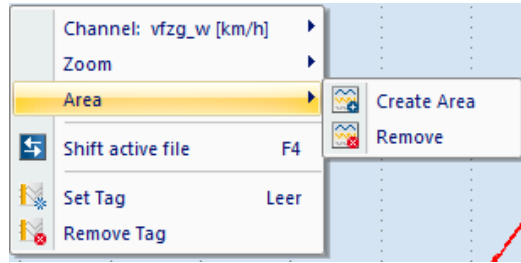
> Additional measuring areas are always added at the bottom of the window above the bit area. Adding a new area will reduce the size of already existing areas accordingly.

Changing the Size of an Area
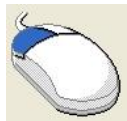
You can **change the size** of an **area** with your mouse.

Click on the upper or lower boundary of an area and then move it up or down.

> **Tip:**
> You can create a new measuring area by using the same technique. Simply click on the upper edge of the topmost measuring area (or the lower edge of the measuring area at the very bottom) and move it.

Deleting an Area

Use your mouse to place the crosshairs anywhere within the area you want to delete.

You can remove the area from the **Darab ribbon** or from **Logger Data Window** context menu.

Choose "Start/ Areas/ Remove" from Darab ribbon.
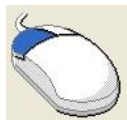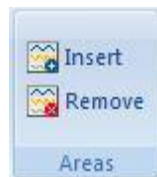
Choose "**Areas/Remove**" in **Logger Data Window** context menu.
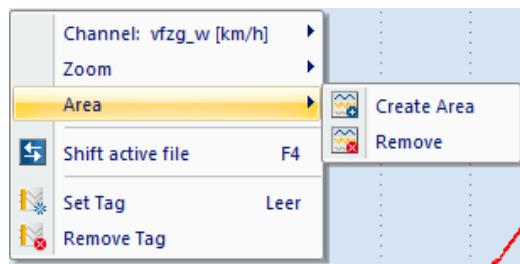
Modes

You can choose the axis mode of an area between 3 given axes-setups.
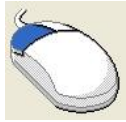
Choose "**Start/ Axis Mode**" from Darab ribbon.
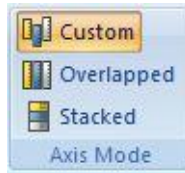
Custom   This is the default option. In this mode, you can customize the area without any restrictions.

Stacked   With this option selected, every channel gets its own axis on the first left slot.

Note:

In stacked mode, it is not possible to change the axis. To change the order of the channels, move them in the channel-area.

Overlapped   If this mode is selected, every channel uses the full height of the area and there is only one axis allowed per slot.

Note:

The channel-configuration is saved when the mode is changed, so you have the same configuration when you switch back to a previous mode.

However, if you remove a channel and insert it again, the saved data is gone.

Channels

Channels represent signals that have either been recorded in a vehicle or have been generated from mathematical calculations.

Channels are displayed in areas inside an Oscilloscope. The available range of channel display options will be described below.

Selecting and Adding Channels

There are the following methods to select and add a channel to the Oscilloscope. From

- **Channel Window**
- **Darab Ribbon**
- Add context menu

From **Channel Window**:
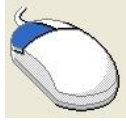
To **view** a **channel** in the Oscilloscope select the channel from **Channel Window** and drag & drop it to the Oscilloscope.

From **Darab Ribbon**:

Start/ Settings/ Edit To access channel selection choose "Start/ Settings/ Edit" from the menu. The dialog box that subsequently appears enables you to configure display options for the entire Oscilloscope.

To **open** the **Select Channel** dialog box you can even press **F5** key.

Select Channel dialog box:
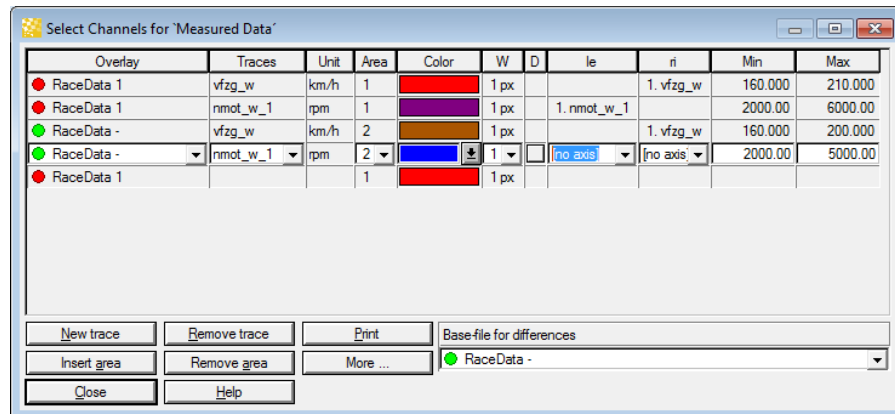
This dialog box contains a list of all currently selected channels. An empty row for the addition of another channel is always provided at the bottom of the list.

Below this list you will find a series of buttons for a number of helpful functions.

Adding a Channel Follow these steps to add an additional channel to an Oscilloscope:

1. Place the cursor in the empty row at the end of the list.
   Click on the "New trace" button to do this very quickly.
2. In the "Overlay" column select the overlay containing the channel you want to display.
3. Select the channel from the adjacent column. From this point on, the channel will be displayed in the Oscilloscope.
4. Now choose the area in which you want to display the channel.
5. Next choose the color in which you want to render the channel.

6. Column "W" allows you to determine the width of the signal display. You are able to choose a value from 1 to 5 pixels.
7. Column "D" allows you to specify whether absolute or differential channel values are to be used. (The difference will be calculated by comparison to a reference channel contained in "Base File for Differences").
8. The "le" or "ri" columns allow you to assign the selected channel to an axis-slot, create new y-axes or remove the channel from an axis.
9. In the remaining two columns you are able to set minimum and maximum channel values.

Tip:

If you want to add the same channel to all the files contained in an Oscilloscope select the channel from the "Channels Window" and drag & drop it to the Oscilloscope.

If the user is adding a channel and the channel is empty (contains no data) a message is displayed.

Notes:

Selecting a Channel from the List

To find a particular channel quickly, simply type the channel name in the entry field. A list pops up automatically. As you continue typing the channel name that matches the letter sequence you have typed so far will be displayed.

You can even use the cursor keys to scroll through the list. Pressing Enter on the keyboard confirms an items selection.

Differences

If you have selected the function for calculating differences between channels the fields for minimum and maximum values will be changed to:

MinimumNew = -(Maximum-Minimum) / 2

MaximumNew = (Maximum-Minimum) / 2

If you de-select this function, the range of values you originally entered will be restored.

Y-axes

An area can only display one y-axis, either on the right- or left-hand side. If you have

selected the right (or left) axis location for a number of channels in the same area WinDarab adjusts the measurement scale on the y-axis so that these channels will have the same minimum as well as the same maximum values. A subsequent change of a channels minimum value (for instance), will then produce a simultaneous change in the minimum value for all other channels that have the same name or are have been assigned to the same y-axis.

New Trace   Use this button to move the cursor to the last, empty row in the table.

Remove Trace   Using this button removes a channel in the selected row from the channel list as well as the Oscilloscope.

Tip:
You can quickly and efficiently remove one or several channels with the same name by selecting the channel, open the context menu with a right mouse click and choosing the "Remove"- menu point. You can select multiple channels by clicking a channel with the **Ctrl** key (add channel to selection) or **Shift** key (select a range of channels) pressed.

Print   This Button can be used to Print out Data from the Oscilloscope. When u click on it, the Print Layout will open.
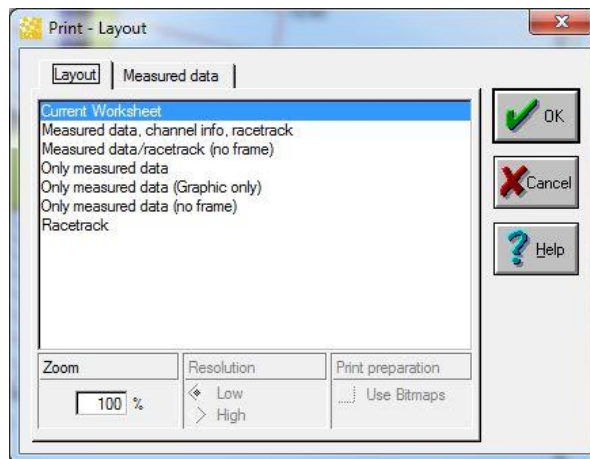
Open   Use this button to open additional files.

Insert Area   Clicking on the "Insert Area" button enables you to add a new area to the Oscilloscope. The selected channel will immediately be transferred to the new area.

More ...   Clicking on this button displays a menu with additional functions.

Remove traces from Area
This button removes all traces contained in the area selected in the current row.

Remove All traces

This button removes all traces from a configuration. Areas will not be deleted.

Close all areas
Removes all channels and areas from a configuration.

Example:

In "**Channel Settings**" the "Rev" channel has the default color "very light blue". You have selected a "red" file as well as the "Rev" channel. Accordingly, WinDarab adjusts the channel color to "very light red". In the next row you choose the "Speed" channel. Because no color has been specified in "**Channel Settings**", this channel as well is displayed in "very light red". If you now switch to a "green" file for "Speed", the channel will be rendered in "very light green."

Base File for Differences

This field displays the file used as a referent during the calculation of differences. A difference is calculated as follows:

Difference = <Value File> - <Value Base File>

In order to specify a different Base File for calculating, you must select a new Base File from the list.

Adding Channels

If you do not need to specify any particular details when you are adding a channel, you may also use the "**Add channels**" function.

Press **Insert** on the Oscilloscope to *open* the "**Add ...**" **menu**. Choose "**Channels**" to *open* the "**Add channels**" dialog box.
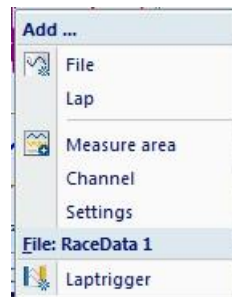
This function enables you to add one or more channels to all the files contained in an Oscilloscope.

The Add Channels dialog box appears:
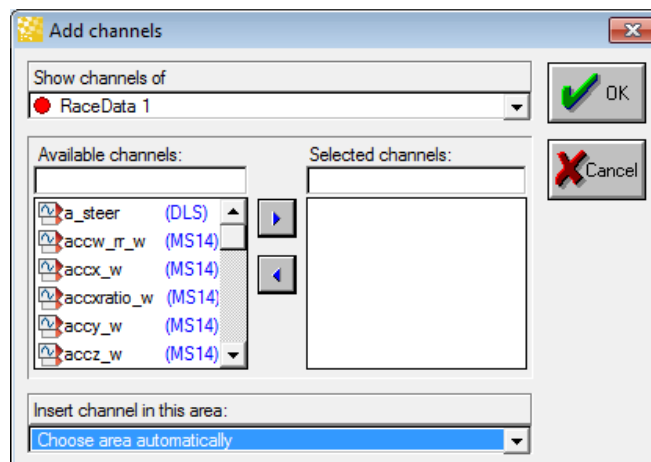
If you want to add one or more channels from each file to an Oscilloscope, use these steps:

1. Press **Insert** to open the context menu "**Add...**".
2. Select menu point "**Channel**" to open the dialog box.
3. If necessary, select the file containing the available channel names in the "Show channels of" field.
4. From the list displayed under "Available Channels" choose one or more channels. Select them by clicking on the button with the blue triangle pointed to the right.
5. Repeat step 3 until the list in "Selected Channels" contains all the desired channels.
6. Click on "OK".
7. The selected channels for each file displayed in the Oscilloscope will be added to the active area.

Tip:
You can select the channels you have chosen from the list displayed under "Available Channels" by pressing **Enter** on your keyboard. Pressing **Enter** once more (without having changed your selection) will confirm the selection.

Note:

- Channels will be added to the area to which the crosshairs are pointing.
- The color of the added channels will match the file color.

You can add channels also with dragging a channel from the "Channels Windows" or from another window that supports drag & drop (for example another Oscilloscope or another area of the same window).

Depending on the area where you drop the channel different things can happen. If you drop it on:

- the column with the channel values or inside of an area then the channel will be added in this area
- the left or right border of an area the channel will be added into the area and a y-axis for this channel will be also added on the left or right. You will see a different cursor on the left or right side of the areas during dragging.
- the top or bottom of an area a new area will be added that contains the channel. You will see a different cursor on the top and bottom sides of the areas during dragging.

If you start dragging from another area (from the same or another Oscilloscope) then depending on the **Ctrl** key the selected channel will be moved or copied. If the **Ctrl** key is pressed the channel will be copied else it will be moved. A small "+" sign appears on the right bottom of the cursor if the channel would be copied.

Removing Channels

To *remove* a **channel** from an Oscilloscope follow these steps:

1. Select the channel in the channel list of the Oscilloscope
2. Press the keys **Delete** to open the **Remove … context menu**.
3. Select from the menu whether to remove only the selected channel or all opened channels with the same name.
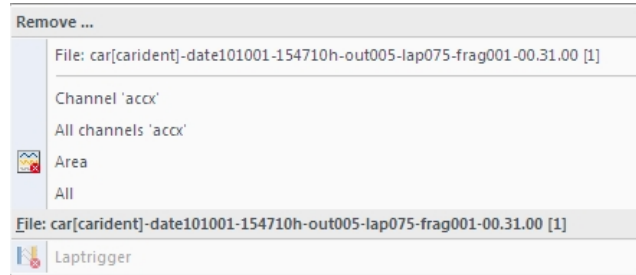
You can also remove a channel by simple dragging it to the "Channels Window" (the trash icon will appear over the "Channels Window" to indicate that the channel will be removed).

To *delete* **all channels** form Oscilloscope press **Delete – a**

Note:
You can also remove channels by using the "Add Channels" dialog box. Open the dialog box. Select the channels in the "Selected Channels" column you want to remove from the Oscilloscope and click on the button with the blue triangle pointed to the left.

Additional Functions

Additional functions available from **channel context menu**:

You can activate additional functions for each selected channel:

- Show Info
- Hiding a Channel
- Flashing Channel
- Change Color
- Change y-area
- Set Zero Point
- Delete Zero
- Use as diff-channel

To *select (or de-select)* one of these **additional functions** use your right mouse button to click on the desired channel in the Oscilloscope. A local menu will appear, listing all functions available for the selected channel. For changing the color a drag & drop can also be used.
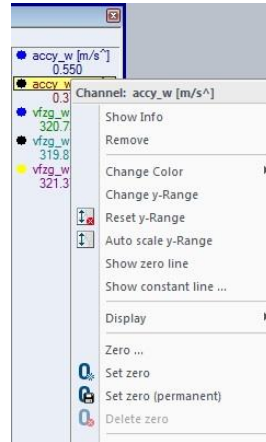
Show Info
Choosing this function opens a small window that displays a number of values for the selected channel.

If you have not previously set a marker the values for the displayed data segment will be calculated. If you did set a marker, values for the area delimited by the marker and the current cursor position will be calculated.

- Minimum
- Maximum
- Average

The following values will be calculated only if a marker has been set:

- Gradient
- Area
  Area delimited by the signal curve and the zero line of a channel
- Area 2
  Area delimited by the signal curve and the horizontal line running through the smaller y-value of the marker and cursor position

Change Color
Selecting this entry in the channel local menu displays the color palette from which you can choose a different channel color.

 You can also change the color of a channel by dragging a color from the colors Window and drop it on the channels name in value column.

Change y-Range
This function enables you to define the y-range for the selected channel. The function is also available in the channel local menu.
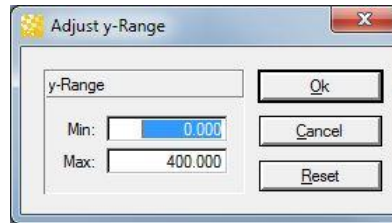
Enter the new maximum and minimum values.

Click on "OK".

To restore the previous values click on the "Reset" button.

 Press y-key to *open* the "**Adjust y-Range**" dialog box.

The y minimum and maximum values can also be inserted into the fields „**Max**" and "**Min**" under the menu "**Channel/ y - Scale**".

 If you want to simply shift the visible range of a selection keeping the range the same size, use **Ctrl + up/down**.

Restore y Range	To return the y-Range to its previous settings, choose this function from cannel context menu or "**Channel/ y-Scale/ Reset**" from menu list.
Auto scale y-Range	To scale y-Range automatically, choose this function from cannel local menu or "**Channel/ y-Scale/ Adjust**" from menu.
Show zero line	The zero line is shown on the Logger Data window if this option is selected.
Show constant lines	To show or hide constant lines, choose this function from the cannel context menu.

1. Select (Check) the constant lines you want to display in the graph window.
2. Insert the values for the selected constant lines. (Zero line doesn't need a value)
3. Click on "OK".

Set Zero Point   WinDarab allows you to set the zero point for a channel

The zero point is only valid while a file remains opened. If you close the file the new zero point will not be saved. Setting a **permanent** zero point saves them into a Logged Data File and retrieves them when this file is re-opened.

If you want to set a zero point, check the appropriate menu item to specify what type of zero you want to set.

Replacing a permanent zero point with a temporary zero point causes the permanent zero point to be deleted.

Zero …   Selecting this function opens a small dialog box in which you are able to enter the absolute value that is to equal the 0 value.
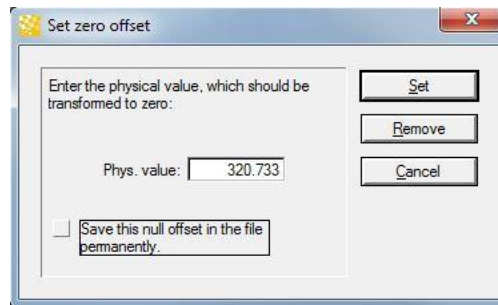
You can open the "**Set zero offset**" dialog box from Darab menu list **Channel/ Set zero/ Dialog** …

To *open* the "**Set zero offset**" dialog box you also can press the **z**-key.

Set Zero Point/   Use the function to set your zero point. The cursor used to mark the channels y-value
Set Zero Point   which serves as the new zero point.
(permanently)
The function is selected either from the channel local menu

- "**Set Zero Point**" and
- "**Set Permanent Zero Point**" or from the menu
- "**Channel/ Set zero/ Set zero**" and
- "**Channel/ Set zero/ Set zero and save to file**"

Delete Zero   Select this menu item from the channel local menu if you want to delete the previously
Point   set zero point for a particular channel.

Use as diff-   Selecting this function displays the selected channel(s) as diff channel. The displayed
channel   values are the calculated difference between the selected channel and a reference
                     channel contained in "Base File for Differences".

Additional functions available from **Darab menu tab Channel**.

You can activate additional functions for each selected channel:

- Filter a Channel

- Showing/Hiding a Channel
- Blinking Channel Display
- Thickness

Filter a Channel   WinDarab enables you to define a series of mathematical functions (i.e. filters) in order to reduce interference and channel noise (see chapter Mathematical Channels and Filters).

To activate a channel filter select the desired filter from the menu bar "**Channel/ Filter**". Selecting the filter again will deactivate the filtering.

> **Important:**
> The menu always displays the ten most recently used filters. If more than ten filters have been defined in WinDarab, an additional menu item containing a list of all defined filters will be displayed.

> When a channel is filtered the filters name is appended to the channel name. This new, filtered channel will be displayed on all channel selection lists while it is used in WinDarab.
> As a result the filtered channel can be used in all other program functions such as the Analysis Window, for example.

Showing/Hiding a Channel   You can temporarily suppress the display of a channel without actually removing it from the current configuration.

To suppress a channel select "Channel/ Display/ Hide" in the menu. Selecting this option again to show the channel.

> You can select or de-select this function also by clicking on the color dot beside the channel name with your left mouse button.

Blinking Channel Display   To emphasize a channel you can choose a blinking display. To do this, select "**Channel/ Display/ Blink**" in the menu. Selecting this option once again will de-activate the flashing display.

> You can also activate (or de-activate) this function by clicking on the color dot beside the channel name with your right mouse button.

> **Important:**
> A blinking display means that a channel is alternately displayed in its own and the background color. This may cause problems with the display of other channels.

Thickness   This feature allows setting the thickness of a channel curve in the Oscilloscope. The thickness of a channel can be specified in "**Channel/ Display/ Thickness**".

1. Select the channel in the Oscilloscope
2. Select a thickness from 1 to 5 in "**Channel/ Display/ Thickness**" menu.

> If not specified the channel curve gets the standard thickness of 1.

Axes

Axes can be assigned to multiple channels to display their value-range in the Measured Data Window.

Adding a new Axis

There are many ways to add an Axis to a Measured Data Window.

Drag and drop
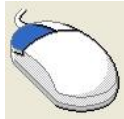
Drag one more channels to the left or right border of the Area until the New-Axis-Icon appears. Release the Mousebutton to clip the new Axis to the left/right border of the Area.

If there is more than one channel selected, all channels are bound to the same axis.

There is more information in the section "Customizing Axes"

> **Note:**
> If you drag Channels from a different Area, these Channels will be moved to the new Area.

Channel Context-menu

Rightclick on one or more selected channels to open the contextmenu. In the display-submenu you can choose to bind the selected channels to an existing axis or to create a new one.

> **Note:**
> If the selected channels are from different areas, the option to bind the channels to an existing axis is not given. If you choose to create a new axis, a new axis will be created in every area with a selected channel.

Ribbon
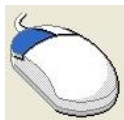
In the Ribbon under "**Channel/ y-Scale/ Y-Axis**", you can add or remove the axes of the selected channels.

> **Note:**
> If the ribbon-button is marked as checked, all selected channels lose their axis. If it isn't checked, all selected channels are bound to the same new axis.
>
> If the selected channels are from different areas, a new axis will be created in every area with a selected channel.

Customizing Axes

Axes can be customized regarding their design, size and values

Size

Axes, usually use the entire height of the area. But you can resize them for example to have two individual axes in the same axis slot.

Note:
Resizing the axes only works in Custom mode. You can find more about Areamodes in the Section "**Areas/Modes**"
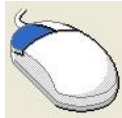
Drag and drop   Click on the top border of an axis and drag it to resize the axis.

Note:
You can't overlap axes. If you want to maximize an axis height, doubleclick on the top or the bottoms border.
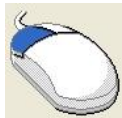
You can divide an existing axis into two equal parts by dropping a new axis on the top or bottom third of the axis. The new axis will take exactly the half of the targeted axis which will be resized to the other half.
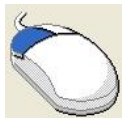
Dropping a new axis on the left or right side of an existing axis, the new axis is created right next to the targeted axis with the same size and position.
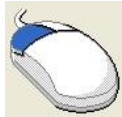
Dropping a new Axis onto free space over or under an existing axis, the new axis will fill out the maximum space without overlapping with another axis.

Every drop-zone will be displayed by a blue region, to let you know which drop-zone is currently selected.

Value

Axes show the channels value-range by default. In the contextmenu you can configure it otherwise.
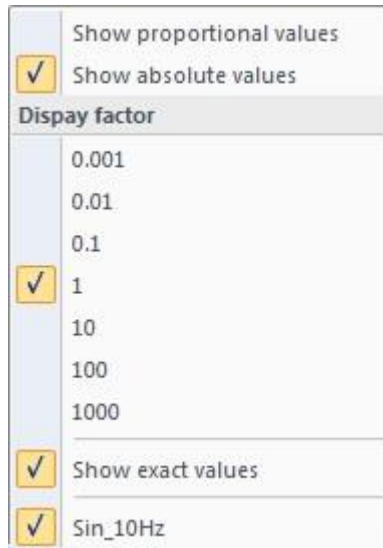
Proportional values

This option changes the value-range to axis-percentage. The values go from 0 to 100%.

Note:
When the axis is in proportional value mode, the channels on one axis are not linked anymore. You can scale one channel, without the others being scaled as well.

Absolute values

This option is the default-setting. The axis has a minimum and maximum value to which all the attached channels are scaled.

Display factor

The Display factor can help you to show exceptional big/small values in a better readable way. If the factor is different to 1, it is shown with the axis unit.

Note:
The Display factor can only be modified in absolute value mode.
Also, the Display factor can not be modified in Darab-v6-mode.

Show exact values

If this option is not checked, the displayed values are rounded to the next natural number.

Every drop-zone will be displayed by a blue region, to let you know which drop-zone is currently selected.

Configurations

A configuration/setting contains the particular composition of an Oscilloscope. You can save a window configuration and retrieve it any time.

A configuration contains the following information:

- Number and size of areas
- Number of files used (not the file names of opened files)
- Active File
- Base File for calculating differences
- Channels and their colors as well as Min/Max values
- Y-axes
- Active x-axis

Saving a Configuration

Start/ Settings/ Save

Follow these steps to save the current configuration for an Oscilloscope:

1. Activate the Oscilloscope whose configuration you want to save.
2. Select "**Start/ Settings/ Save**" from the menu.
3. The File Explorer dialog box "**Save settings**" appears.
4. Select the folder in which you want to save the configuration.
5. Enter a name under which you want to save the configuration.
6. Complete the optional fields.
7. Click on "**OK**".

Press **F7** to *open* the **File Explorer**

The following entry fields are available in the "Save Settings" dialog box:
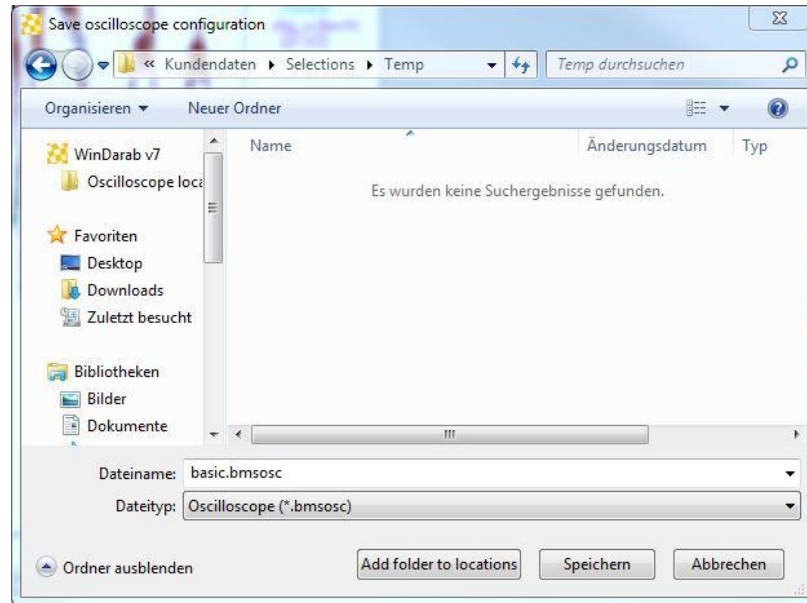
Folder   In this field select the folder in which you want to save the configuration.

Files   This area provides information about all configurations and sub-folders contained in the current folder.

Name   In this field you enter the name under which you want to save the configuration.

Note:

WinDarab uses File Explorer to manage configurations. For information about how to work with File Explorer, please refer to the "File Explorer" chapter in this manual.

Add Settings

The "**Add Settings**" function loads an additional configuration for all displayed files.

If an Oscilloscope does not yet contain a file, choosing this menu item performs the "**Load Settings**" function (see below).

Follow these steps to load an additional configuration into an Oscilloscope:
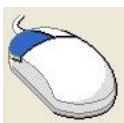
Click with the left mouse button to *activate* the **Oscilloscope** to which you want to load the new configuration.

Press **Insert** to *open* the "**Add ...**" context menu and select "**Settings**".
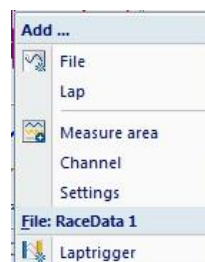
The dialog box "**Select oscilloscope configuration**" will appear.
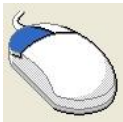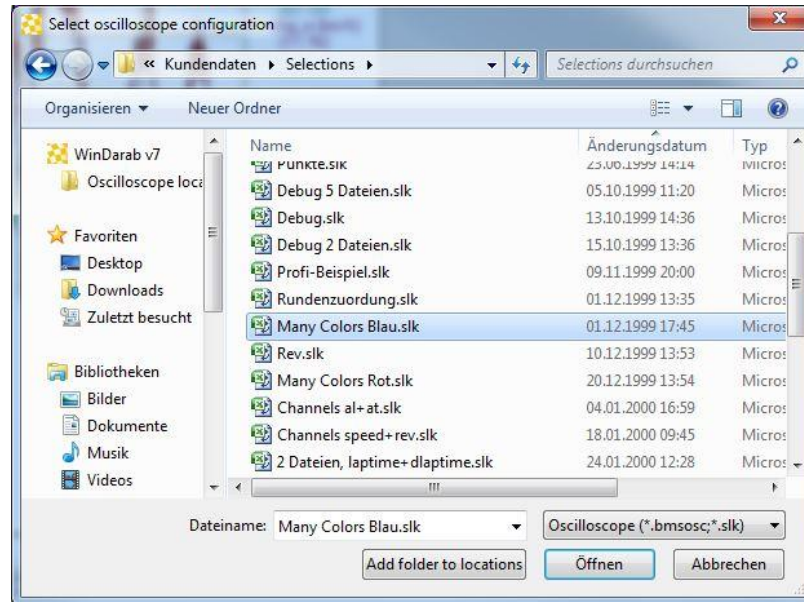
Select the **configuration** you want to *add* to the Oscilloscope and click on "Open".

The configuration will now be loaded into the Oscilloscope.

Load Settings

The "**Load Settings**" function removes all channels from the Oscilloscope and loads a new configuration for the **selected file only**.

Follow these steps to load a new configuration to an Oscilloscope:
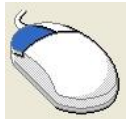
Click with the left mouse button to *activate* the **Oscilloscope** to which you want to load the new configuration.
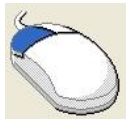
Select "**Start/ Settings/ Load**" from the menu.

The dialog box "**Select oscilloscope configuration**" will appear.
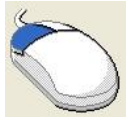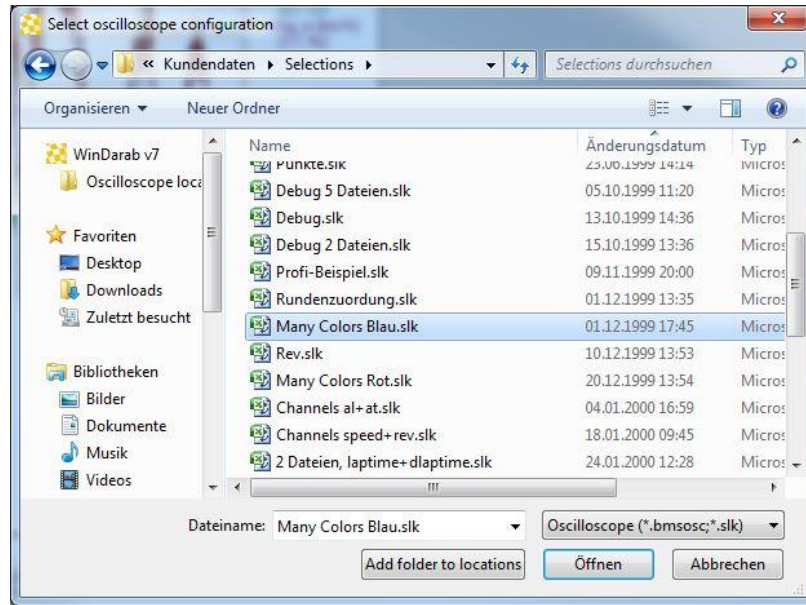
Select the **configuration** you want to *load* and click on "Open".

The configuration will now be loaded into the Oscilloscope.

If a file has no Logged Data File assigned the channels in that file will not be loaded.

If you load a new configuration but use the same set of Logged Data Files the current logged data segment will be retained.

If you have not yet opened any Logged Data Files when you begin loading a new configuration, File Explorer will open.

Replace Settings

The "**Replace Settings**" function loads a different configuration for all displayed files.

Follow these steps to replace a configuration
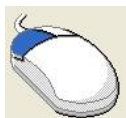
Click with the left mouse button to *activate* the **Oscilloscope** to which you want to load the new configuration.

Press **R** to *open* the "**Replace ...**" local menu and select "**Settings**"
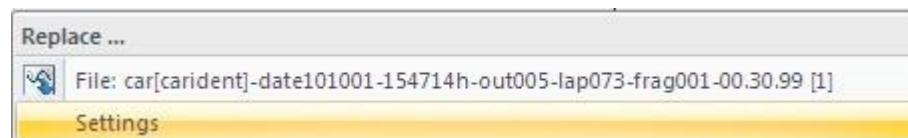
The dialog box "**Select oscilloscope configuration**" will appear. Select the configuration you want to load.

Click on "Open".

The configuration will now be loaded into the Oscilloscope.

If a file has no Logged Data File assigned, the channels in that file will not be loaded.

If you load a new configuration but use the same Logged Data Files the current logged data segment will be retained.

Select oscilloscope configuration Dialog Box

The "Select oscilloscope configuration" Dialog Box allows you to browse through drives, folders, locations and select the configuration file.
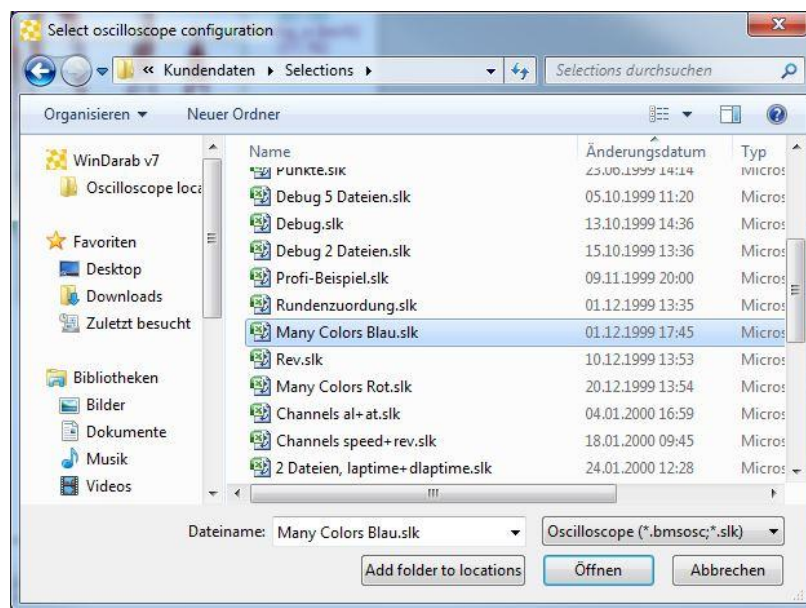
Note:
WinDarab uses File Explorer to manage configurations. For information about how to work with Darab Explorer, please refer to the "File Explorer" section in this manual.

If the user loads an Oscilloscope configuration containing a channel, which is not provided by a file, WinDarab automatically adds a "dummy" channel to the file.

The same happens, if a mathematical channel is deleted by the user or because of changing the base folder where mathematical channels are loaded from.

If the user later adds a mathematical channel with the same name or changes the base folder to a different folder, containing a mathematical channel definition for the "dummy" channel name, WinDarab automatically replaces the "dummy" channel by the mathematical channel.

Working with Oscilloscopes

Crosshairs

Within an Oscilloscope the crosshairs mark the current position to which all displayed values refer.
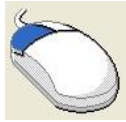
Use the left mouse button to *place or move* the **crosshairs**.

Follows Signal

When the "**Start/ Cursor/ Shape/ Follows Signal**" option has been selected, the crosshairs vertical line follows the value of the selected channel.
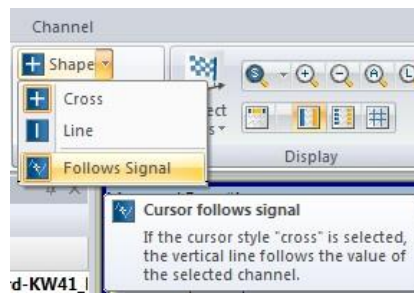
Note:

If the „Cursor Follows Signal" option is activated and the value of the currently selected signal is outside of the visible range, the horizontal line of the cursor will not be drawn any more. The horizontal line is not visible in the area above/below of the area the signal resides in.

If the „Follows Signal" option is activated and the user moves the cursor from an empty area to another area, the horizontal line of the cursor will not be removed from the old position.

Tag

A tag can be set any place in an Oscilloscope. Only one tag can be set within the Oscilloscope windows.

Setting a tag is an optional function used to delimit a logged data segment for analysis.

Set Tag

There are the following methods to set a tag:

Choose **Start/ Tools/Tag/ Marker/ Set** in **Darab ribbon**

Click with the left and right mouse (or middle) button on the Oscilloscope to *open* the **context menu** and choose "**Set Tag**".

	Channel: accx\MS5.0 [g] ▸
	Zoom ▸
	Area ▸
	Shift active file
	Set Tag
	Remove Tag
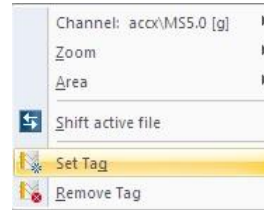
Press **Space** to *set* a **tag**.

Setting of tags also display the differences of channels

Note:

All Oscilloscope windows have the same tag. Therefore, a tag set in one Oscilloscope is also set in every other Oscilloscope.

Remove Tag

There are the following methods to remove a tag:

Choose **Start/ Tools/ Tag/ Marker/ Remove** in **Darab ribbon**

Tag/Marker ▾
Set
Remove
Tools

Click with the left and right mouse (or middle) button on the Oscilloscope to *open* the **context menu** and choose "**Remove Tag**".

Channel: accx\MS5.0 [g] ▸
Zoom ▸
Area ▸
Shift active file
Set Tag
Remove Tag
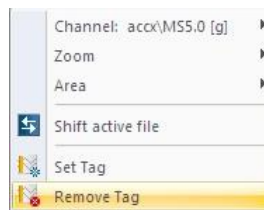
Press **Space** twice to *remove* a **tag**.

You can also delete a tag by setting a tag in the same location in which you have already set one.

Marked range (Selection)

If you select a range of data this is called marked range or selection. Only some functions of the windows can use the marked range.
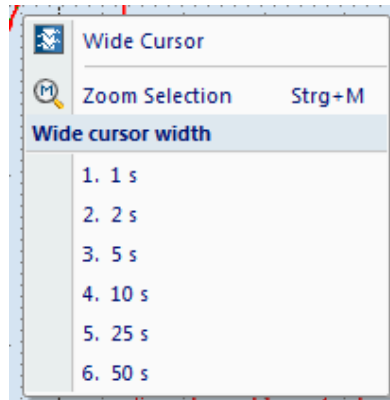
You can mark range with the mouse, with the keys or from the quick menu (which is opened with the key **W**).

By default the marked range will disappear after you click somewhere in the window. But there is also a possibility to fix the marked range to a certain range around the current cursor. In this case the marked range is moved with the cursor.

Wide Cursor

You can choose the marked range to follow the cursor by the marked range quick menu or by selecting "**Start/ Cursor/ Wide Cursor**" the menu.
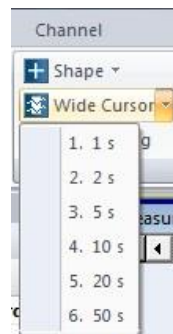
Mouse Functions

You are able to perform every important function with your mouse. What types of functions are available depends on the area you have clicked on.

Clicking on a display item with the right mouse button will typically *open* a **local menu**.

Note:
An exception occurs in the signal display area where you must use the center mouse button to open the local menu.

Important:
If your mouse only has two buttons, you can still call up the functions of the third (center) mouse button by holding down the right and left mouse buttons. Press the buttons one after the other - **not simultaneously**.

You can access the following functions with your mouse in an Oscilloscope:
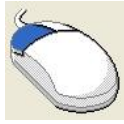
Left-click/Hold

Sets crosshairs in the selected location.

Using the scroll wheel on your mouse will shift the selected file in relation to all other files.

Clicking onto the line separating two measuring areas allows you to change their size.

If **Shift key** is pressed the range between start and end will be marked.

Right-click/Hold

When you begin marking a range

holding down the **Shift key** marks a rectangular range within the window.

moving the mouse laterally marks range along the x-axis.

moving the mouse vertically marks a range along the y-axis of the selected channel.

Releasing the mouse button allows you to zoom in on the marked range. Using this function on the y-axis will zoom in on the marked channel only.
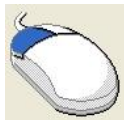
Double-click left

Zooms 2x

Double-click right

Undo zoom 2x

Click center mouse button

Displays local menu with a selection of menu commands.

Mouse wheel

Move crosshair in the corresponding direction. The movement is faster when you increase the distance between the mouse and the crosshair.

Keyboard Functions

The following (important) keys are defined within Oscilloscope:

Left/ Right/ Top/   Move crosshairs in the corresponding direction

Bottom

Ctrl+Left	Scroll logged data display to the left by one-half the display width
Ctrl+Right	Scroll logged data display to the right by one-half the display width
Ctrl+Up	Shift the range of the selected channel(s) to the top.
Ctrl+Down	Shift the range of the selected channel(s) to the bottom.
+	Zoom 2x
	If **Shift key** is pressed then marked range around the cursor will increase.
-	Undo zoom 2x
	If **Shift key** is pressed then marked range around the cursor will decrease.
A	Zoom All
L	Zoom Lap
M	Zoom the marked range
W	Opens a menu which contains functions connected with the marked range. For example you can select the size of the marked range around the cursor position.
S	Zoom Segment
T	Zoom Tag
Ctrl+Z	Undo Zoom
Space Bar	Set Tag (pressing twice: delete tag)
B	Turn on/off flashing channel display
C	Change color of the selected channel
H	Display/ Suppress display of the selected channel
I	Activate "Show Info" for the selected channel
Y	Restore default values for the y-range of the selected channel
Ctrl -Y	Set the y-range for the selected channel
Z	Set zero for the selected channel
Ctrl+L	Select Laps
F4	Activates / de-activates "**Shift File**" mode
F5	Select Channels
F6	Load Settings
F7	Save Settings
F9	Load desktop
Insert	Opens local "Add ..." menu
Delete	Opens local "Remove ..." menu

Zooming and Comparing

There are a variety of options to enlarge or reduce the size of data segments when you want to evaluate the recorded data.

The following zoom functions can be called up from the "Start/ Display" menu:

🔍	"+"	Zoom 2x	Doubles the zoom factor.
🔍	"-"	Zoom out 2x	Reduces the zoom factor by one half.
🔍	M	Zoom Marked Range	The currently marked range will be enlarged.
🔍	L	Zoom Lap	The lap in the crosshairs will be enlarged.
🔍	S	Zoom Segment	The course segment in the crosshairs will be enlarged.
🔍	T	Zoom Tag	The range delimited by the tag position and crosshairs will be enlarged.
🔍	A	Zoom All	Displays all data in the Oscilloscope.
Ctrl+Z		Zoom Undo	Undoes the previous zoom functions.

Adding a File

Additional Files can be added to an Oscilloscope with the local menu "**Add ... / File**". The file will be first added to the **File Explorer Window**.

Follow these steps to add an additional file to an Oscilloscope:

1. Select the file you want to use as a template for the addition.
2. Press **Insert** to open the "**Add ...**" context menu or
3. Click on "**File**" in the menu.
4. Select the files you want to add to the Oscilloscope in File Explorer.
5. Click on "**Open**".
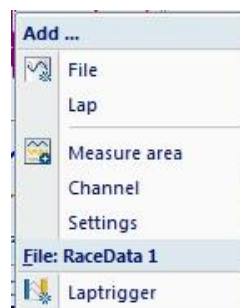
The file is added and displayed in the File Explorer Window. To add the file lap to the Oscilloscope an

overlay has to be assigned to it.

There are the following methods to assign an overlay to a file lap.

Create an overlay

1. Create an overlay in the File Explorer Window (see File Explorer Window)
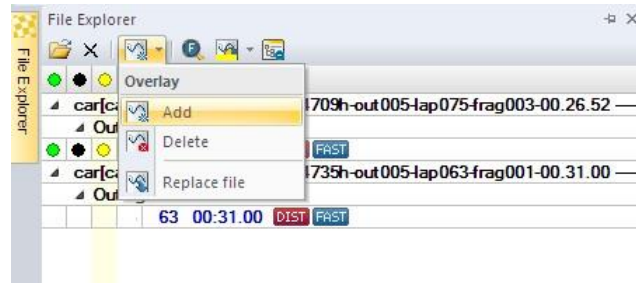
or
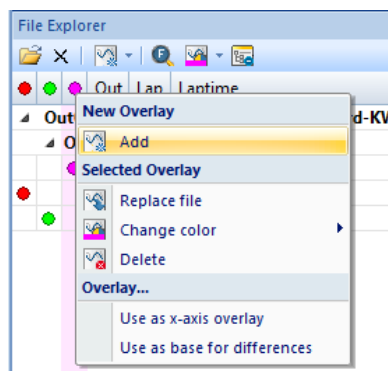
2. Click on the the overlay-column (the new added column ⬤) next to the file lap you want to add to the Oscilloscope

Adding a lap

1. Press **Insert** to open the "**Add ...**" context menu of the oscilloscope window
2. Click on "**Add .../ Lap**" in the menu.
3. Click on "**OK**".
4. A new overlay is added to the lap currently active in the oscilloscope (see Adding lap)
5. Drag & drop the overlay to the file lap you want to add to the Oscilloscope

Replace file

There are three methods:

1. Drag & drop an existing overlay ⬤ to the desired file lap in the **File Explorer Window**

or

2. Open the context menu of an overlay ⬤ in the **File Explorer Window** and choose **Selected Overlay/ Replace file**
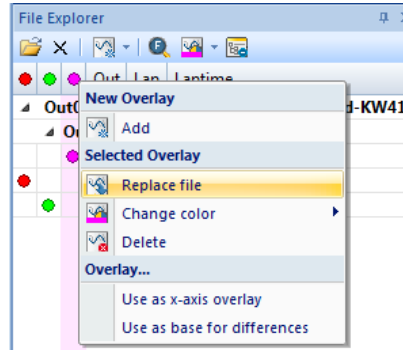
or

3. Click with the right mouse button on the file bar of the Oscilloscope to open the "**File**" context menu an choose **File:/ Replace**
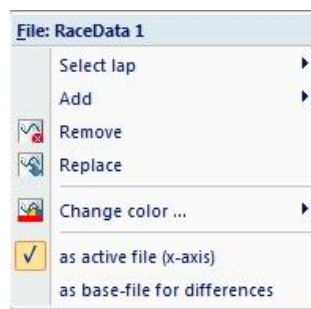
1. The **File Explorer** is opened
2. Choose the file and click on "Open"
3. The overlay is assigned to the file lap

A file can also be added to the File Explorer as follows.

1. Click with the right mouse button on the file bar of the Oscilloscope to open the "File" context menu
2. Click on "**File .../ Add / File**" in the menu.
3. Select the files you want to add to the Oscilloscope in Darab-Explorer.
4. Click on "Open".
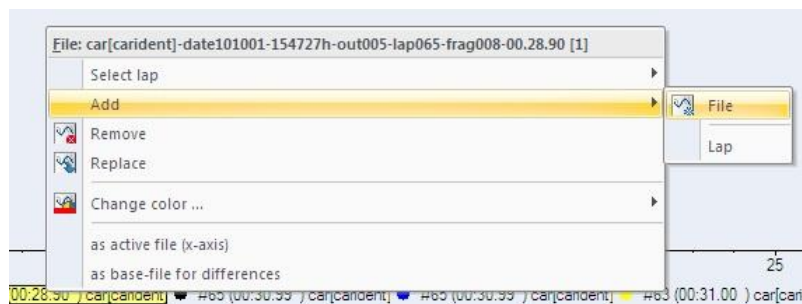
Adding a Lap

The function "**Add .../ Lap**" an overlay to the currently active lap. Adding an overlay to the same file allows you to make lap comparisons. Each overlay has its own color to enable you to differentiate one copy from another.

Follow these steps to add an additional lap (or copy of a file):

Press **Insert** to open the "**Add ...**" local menu

Click on "**Add .../ Lap**" in the menu.

Click on "OK".
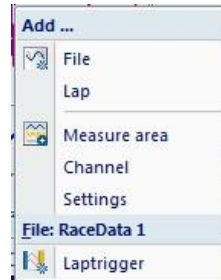
A lap can also be inserted by using the "**File**" local menu on the file bar of the Oscilloscope.
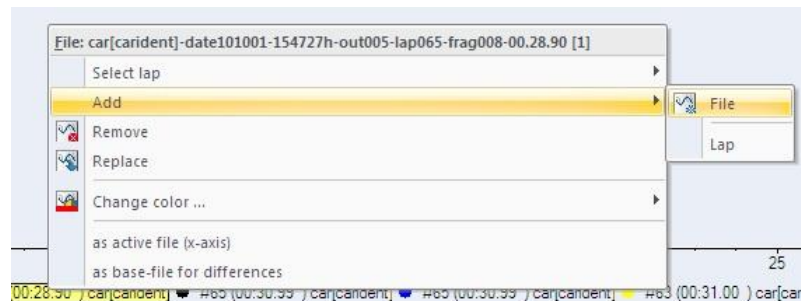
Change Color

You can change the file color at any time from local menu (Click right mouse button) on the files name. This menu is only for this file. You can see the file name and its color as a title of the menu.

You can also change the color of a file by dragging a color from the "Colors Window" and drop it on the file's name in channel values column.

Deleting a File

There are the following methods to remove a file:

Select the **file** you want to *remove* in the file bar of the Oscilloscope.

Press **Delete** to open the "**Remove ...**" local menu.

Select "**File**" to remove the file
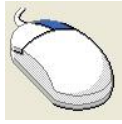
1. Select the **file** you want to *delete* from the file bar of the Oscilloscope.
2. Click on the file tab with the right mouse button to *open* the "**File**" menu and then click on "**Remove …**".
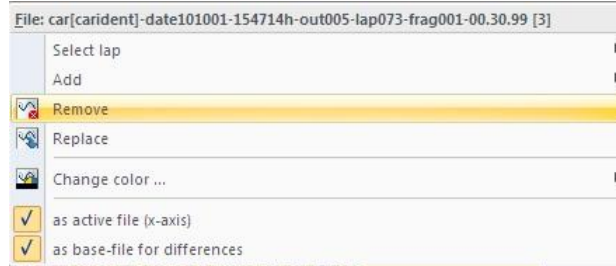
Tip:

Deleting a file from an Oscilloscope does not automatically close this file. The menu item "**Tools/ Settings/ Program/ Miscellaneous**" is used to determine whether Win Darab closes files automatically when they are no longer needed, or not. If you did not select this option you have to close the file manually using the "**Close File**" function in the File Explorer Window or "**WinDarab-Button/ Close measurement file**" in menu.

Replacing a File

To replace a file within an Oscilloscope with another file follow these steps:

1. Select the file you want to replace in the file bar of the Oscilloscopes.
2. Press **R** to open the "**Replace …**" context menu and select "**File**"

or

Click with the right mouse button on the file tab to open the "**File**" local menu and choose **"Replace"**
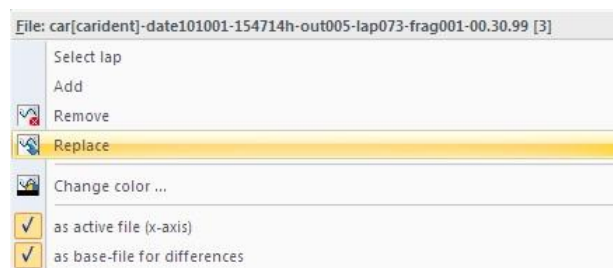
3. In File Explorer open the file with which you want to replace the selected file in the Oscilloscope.
4. Click on "Open".

Note:
The file you have selected will only be replaced in the active Oscilloscope. In all other windows the selected file remains open.

Tip:

You can also replace a file by dragging the new one from the files/channels/colors toolbar or from another window (for example another Oscilloscope) into its file name area. If the new file is from another window then the file can be copied (default) or moved (**Shift**). A small "+" sign appears on the right bottom of the cursor if the file will be copied.

Replacing file settings

To replace file settings follow these steps:

1. Select the file whose settings you want to replace in the file bar of the Oscilloscope.
2. Press **R** to open the "**Replace ...**" context menu and select "**Settings**".
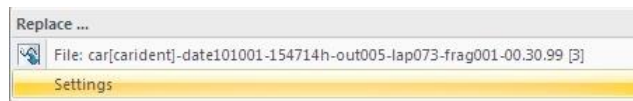
3. In Darab Explorer open the settings that will replace the file settings.
4. Click on "**Open**".

File colors and all channel colors will remain unchanged.

Selecting Laps

Several functions are available to select a particular lap in the current display. Select **Start/ Display/ Select Laps**
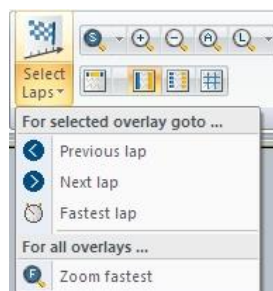
Select Laps	The "**Start/ Select Laps**" function allows you to conveniently select particular laps from the
Ctrl+L	loaded files for immediate comparison.
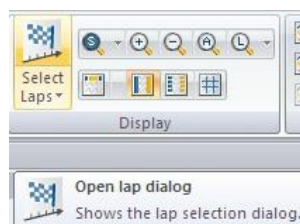
Previous	This selection displays the same course segment from the preceding lap for the selected file.

lap	The display of all other data remains unchanged.
P Key	
Next lap **N Key**	This selection displays the same course segment from the next lap for the selected file. The display of all other data remains unchanged.
Fastest **F Key**	This selection displays the same course segment from the fastest lap for the selected file. The display of all other data remains unchanged.
Zoom **fastest** **Ctrl +F**	This selection displays the same course segment from the fastest lap in all displayed files. If several laps within the same file are displayed, then the fastest and the second fastest lap will be shown.

The "**File**" context menu on the file bar of the Oscilloscope contains also the "**Select lap/ Previous, Next and Fastest**" entries.
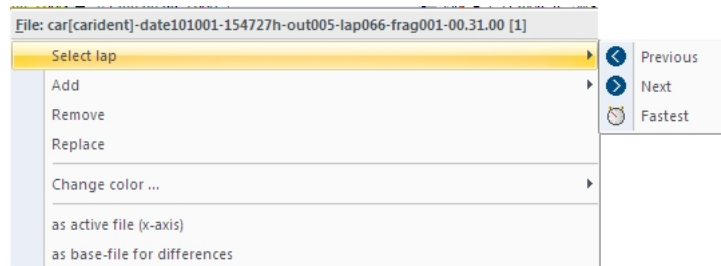
To *open* the "**File**" context menu click with the right mouse button on the file bar of the Oscilloscopes.

Tip:
Use the "Select Fastest Laps" button if you want to find the fastest laps in the files you have selected.
If you have opened a file more than once (in order to make lap comparisons within a single file) the "Select Fastest Laps" function will find the next slower lap for each additional instance the file has been opened.

Lining Up Axes

While comparing two files, it is very unlikely that the particular sections you want to compare will be aligned over top of one another. You will have to move the files so that the sections of interest line up.

WinDarab automatically makes this adjustment when comparing laps. You can find more information about this function in the section called "Selecting Laps".

If you want to compare particular areas, you have to line up the corresponding files yourself.

WinDarab provides the "File Shifting" function that enable you to accomplish this.

File **Shifting**	In the "**File Shifting**" mode some keyboard and mouse functions are different. You activate and de-activate this function by selecting "**Start/ Cursor/File Shifting**" from the menu.

Once you have activated this mode the following keyboard and mouse functions will be altered:

Left / Right   The selected file will be moved exactly one pixel to the right or to the left.

Ctrl+Left   The selected file will be moved exactly five pixels to the right or to the left.
Ctrl+Right

File movement will correspond to your mouse movements.

'<' or '>'   The selected file will be moved exactly one pixel to the right or to the left.

Ctrl + '<' or   The selected file will be moved exactly five pixels to the right or left.
ctrl + '>'

Ctrl + left mouse button
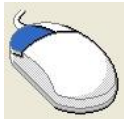
File movement will correspond to your mouse movements.

left mouse button and Scroll Wheel   Holding down the left mouse button enables you to move a file using the scroll wheel. It is possible to control the scrolling speed: the greater the distance separating the mouse pointer from the crosshairs the faster a file will be scrolled.

The "**File Shifting**" option can be activate from the Logger Data Window context menu. Click the left and right mouse button on the window to open the local menu and select "**Shift active file**"
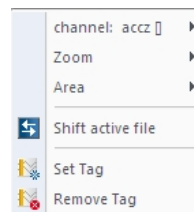
To *open* the **context menu** click with the right mouse button on the Oscilloscope.

Manual Lap Trigger

In cases where lap trigger signals could not be received by the vehicle due to a malfunction, you are able to add them manually in WinDarab.

When you add a lap trigger manually a lap is divided in two sections and lap times are adjusted accordingly. Because lap numbering remains unaffected by the addition you will find a duplicate lap number in the lap display. The added lap will be differentiated by the letter "m" beside the lap time.

Only lap triggers that have been set manually can be removed. Lap triggers recorded by the hardware can neither be changed nor removed.

Note:
Manually added lap triggers will be permanently saved in the Logged Data File.

Insert Lap Trigger

Follow these steps to add a manual lap trigger:

1. Open an Oscilloscope and add some channels.
2. Place the crosshairs where you want to add a lap trigger.
3. Select "**Start/ Tools/ Laptrigger/ Insert**" from the menu.

A **lap trigger** can be *inserted* by choosing "Laptrigger" in the "**Add ...**" context menu of the Oscilloscope.

Press **Insert key** to open the "**Add ...**" context menu.

Deleting a Lap Trigger

Follow these steps to delete a manual lap trigger:

1. Open an Oscilloscope and add some channels
2. Place the crosshairs in the lap you generated by adding the manual lap trigger. A lap that has been manually generated is differentiated by the letter "m" beside the lap time.

3. Select "**Start/ Tools/ Laptrigger/ Delete**" from the menu.

A **lap trigger** can be *deleted* by choosing "Laptrigger" in the "**Remove ...**" context menu of the Oscilloscope.

Press **Delete key** to open the "**Remove ...**" context menu.

Laptrigger Manager

The Laptrigger-dialog helps you manage your Laptriggers for a specific File. It lists all your Laptriggers for this file with the distance- and time-position as well as the length of the resulting lap. You can delete your manual Laptriggers or add new Laptriggers to a position of your choice.

Lap	Lap time	Lap dista...	Time	Distance	
4-9	01.43.42	5686	0.000	0	OUT IN
4-10	00.15.92	1000	103.424	5686	OUT IN DIST FAST
4-10	01.27.37	4684	119.341	6686	OUT IN DIST MAN
4-11	01.43.40	5677	206.710	11370	OUT IN

Start/ Tools/ Laptrigger/ Laptriggers ... The Laptrigger Manager can be accessed by clicking on "**Start/ Tools/ Laptrigger/ Laptriggers ...**". The Laptrigger Manager will display current selected File.

Defining an In-/Outlap

A lap can be *marked/un-marked as an in-/outlap* by clicking on the **In** or **Out-Symbols** of the chosen lap

Ignore as Fastest Lap
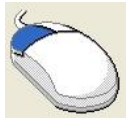
A lap can be *ignored from the fastest lap calculation* by clicking on the **fastest lap Symbol** of the chosen lap. The next fastest lap will be declared as fastest lap. The ignored lap can be reconsidered as fastest lap by clicking on the **Ignored as fastest lap icon.**

Deleting a Laptrigger
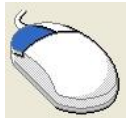
A **lap trigger** can be *deleted* by selecting a manual Laptrigger from the List and then pressing the **Delete**-Button.

Note:

You can only delete manual Laptriggers. The manual Laptriggers are marked by a gray "Man"-Symbol.

Split Lap

You can Insert a Laptrigger by splitting an existing Lap into two or more parts.

First, you have to select the Lap you want to split. Then press the Split-Button. A new window appears.
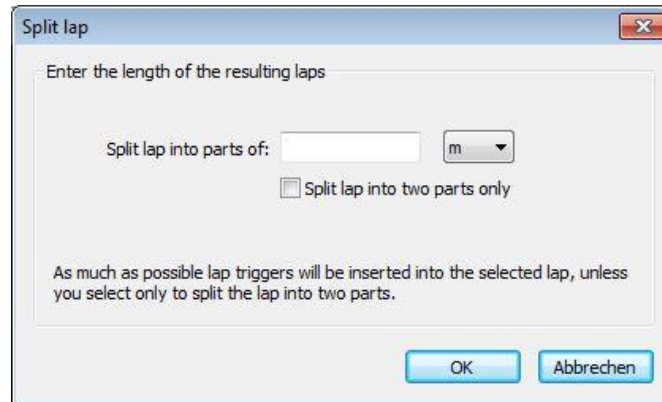
Put the length of the new laps in this window and press the OK-Button.

You can enter the length as a distance or time-value. When you press OK, the lap will be split into a maximum number of parts of the chosen length.

As an additional feature, you can split the lap just once by selecting the "Split lap into two parts only" - checkbox.

With this option selected, you create just one new laptrigger at the chosen distance from the start of the selected lap.

Note:
You can input a distance- or a time- position. While you insert the distance- position as a number, there are more ways to insert a time- position.

Like the distance, you can enter a number to specify the position in seconds.

In addition the formats hh:mm:ss and mm:ss are supported.

Generate Laptrigger

You can generate lapgriggers with the laptrigger generator built into WinDarab. It allows you to generate Laptriggers by characteristics of a given channel.

The laptrigger generator can be opened by clicking "Generate…" in the laptrigger manager.
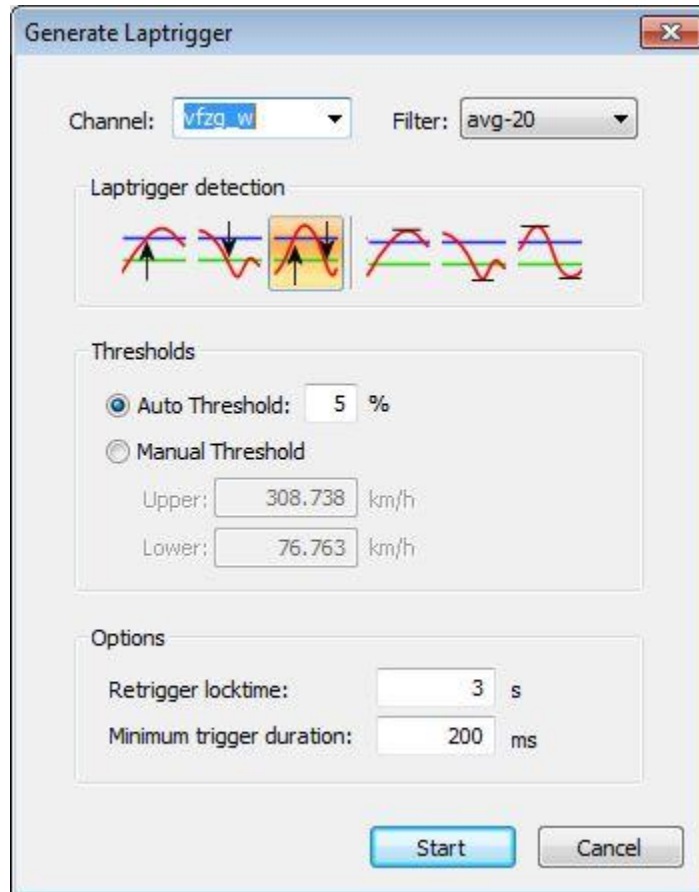
Channel	Insert the base-channel for calculating the laptriggers. This channel can be a mathchannel too.
Filter	The combobox includes a few pre-defined filters, which can be applied to the Channel to smooth the values.
Laptrigger detection	Select the detection-rules. You can choose between crossing the upper threshold, crossing the lower threshold or crossing both either threshold (first three options). The last three options are basically the same with the difference, that not the crossing of the threshold marks the laptrigger-position but the following extremum.

Important:
The laptriggers are generated alternating. After crossing the high threshold the lower threshold has to be crossed in order to produce another high threshold laptrigger.

If you just want a single value as threshold, set the same value for upper and lower threshold.

Thresholds	The thresholds are defined by absolute values for the upper or the lower threshold (Manual Threshold) or by a percentage-value (Auto Threshold). The upper threshold will be the maximum value minus the percentage of the difference between the maximum and the minimum value.

The lower threshold will be the minimum value plus the percentage of the value range.

Retrigger locktime	This option defines the time after a laptrigger in which no other laptrigger can appear.
Minimum trigger duration	The Minimum trigger duration is much like the Retrigger locktime, but instead of blocking the next laptrigger if it's within the time range, the laptrigger is not set if the next laptrigger is in the time-range of the minimum trigger duration.

Display Options

X-Axes

Win Darab provides a number of different x-axes for your display. At the moment you can choose from the following x-axes:

Use the menu item "**Start/ x-Axis**" to set the axis for the current Oscilloscope.

Star/ x-Axis/ Distance	• distance
Start/x-Axis/ Time	• time
Start/ x-Axis/ Samples	• samples
Start/ x-Axis/ Lap based scale	The "**x-Axis/ Lap based scale**" option allows you to change the units of measure along the x-axis. Activating this option displays the relative values with respect to the beginning of a lap. If it is de-activated, absolute values with respect to the start of data recording will be shown.

 Important:
Please remember that displaying the difference between "lapdist" channels (or "laptime" channels) will not produce meaningful results if you have chosen "distance" (or "time") for your x-axis.

Note:
The "Samples" x-axis is primarily used for troubleshooting and running diagnostics.

Lap Trigger

Start/ Display/ Lap trigger

You activate or de-activate the function that allows you to show lap triggers in the Oscilloscope in the "**Start/ Display/ Lap Trigger**" menu item.
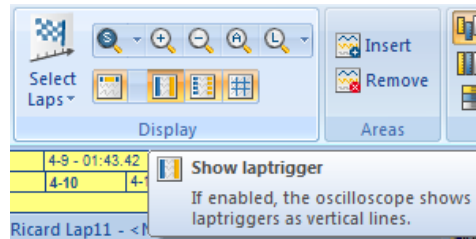
Racetrack Segments

You can display the lap segmentation near the top edge of the Oscilloscope.

Start/ Display/
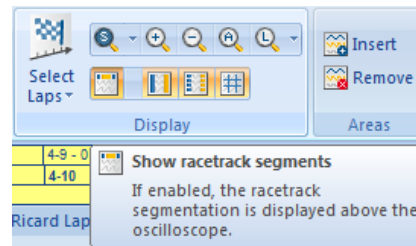
Racetrack

segments

Use the menu item "**Start/ Display/ Lap segments**" to activate or de-activate the display.

Segment borders

Start/ Display/ Segment boarders

The menu item "**Start/ Display/ Segment borders**" enables you to activate or de-activate the display of individual segment markers.
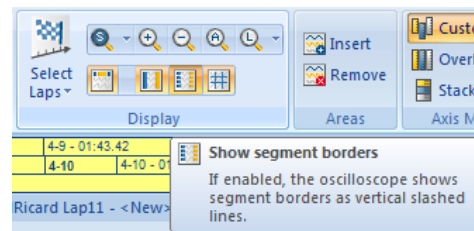
Grid

Start/ Display/ Grid

To display gridlines within an Oscilloscope use the "**Start/ Display/ Grid**" the menu item.
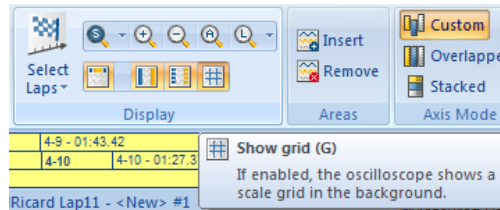
You can *show* the **grid** by pressing **G** – key also.

Selecting this function draws a line through each main calibration point along the x- and y-axes. If a range has two y-axes a grid based on the left y-axis will be displayed.

Replay offline files

In WinDarab you can replay recorded data in real time. When playing such a file then only the data left from the cursor is supposed to be available for the tools and functions you are using to watch the data (e.g. Racetrack Window, Analysis windows, Telemetry windows and others).

The "**Replay**" option in menu "**Start/ Tools**" allows you to select one from both possible modes for simulation. Click on the option to activate or de-activate the corresponding simulation mode.
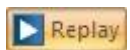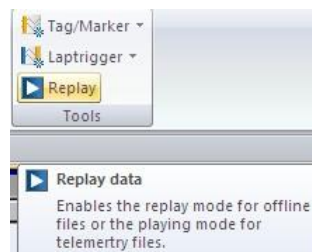
Follow (offline data). In this mode the logged data cursor will move to the next data of currently selected offline file and channel. The speed of movement is synchronized with the files time channel.
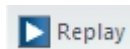
Pause (offline data). This is the default mode. The cursor of the Oscilloscope will not follow it.

Oscilloscope will be switched to pause mode automatically if you scroll the window.

Additional Windows

The Racetrack Window

In the Racetrack Window WinDarab displays the current racetrack/course segmentation.

While you are working the Racetrack Window always remains in the foreground. You can position the window wherever you like on the screen.

You can choose between a 2D and 3D view of the racetrack.

You can switch to 3D view at any time from the context menu. The **context menu** will be *opened* by clicking with the right mouse on the Racetrack Window.
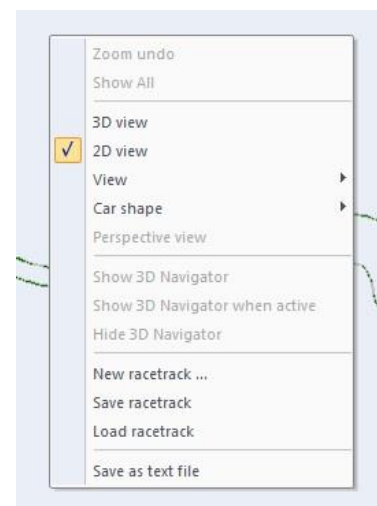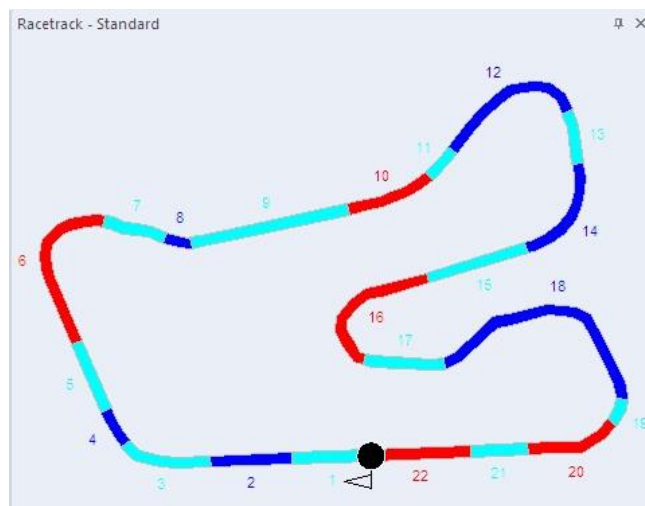
If the **Racetrack Window** is in **3D view** the **context menu** is accessible through clicking the right and left (or middle) mouse button on the window.
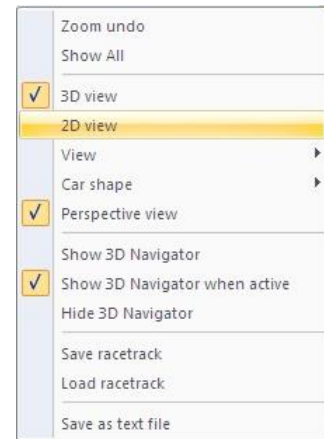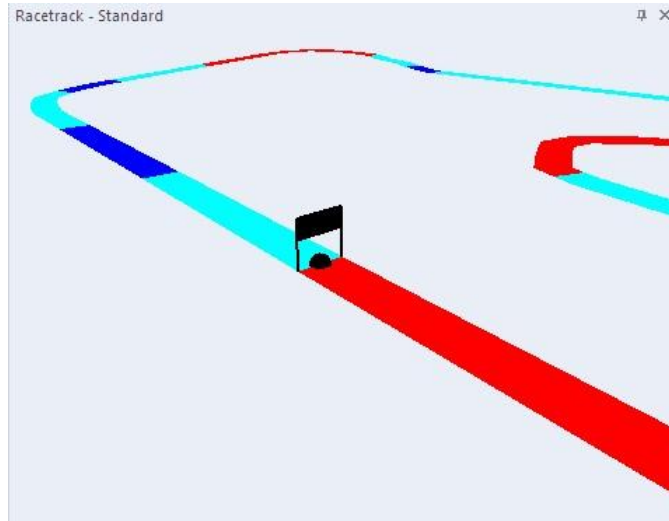
The 3D view has 2 sub views:

3D overview

3D overview is the default view when you switch to 3D. You can switch to this view from the local menu.

Context Menu/ View/ Segmentations
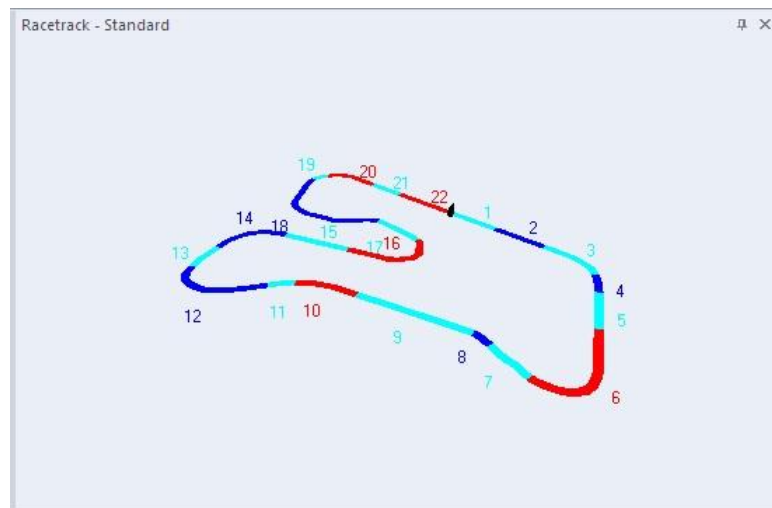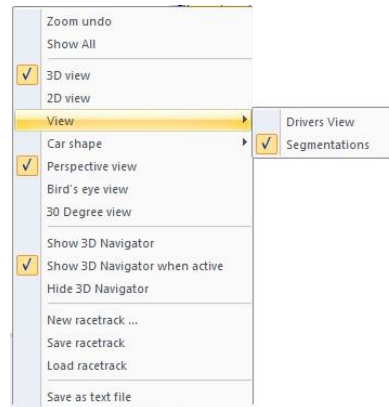
3D drivers view

Local Menu/View /Drivers View

You can switch to 3D drivers view only if you are in 3D view. You can use the context menu.

Racetrack - Standard
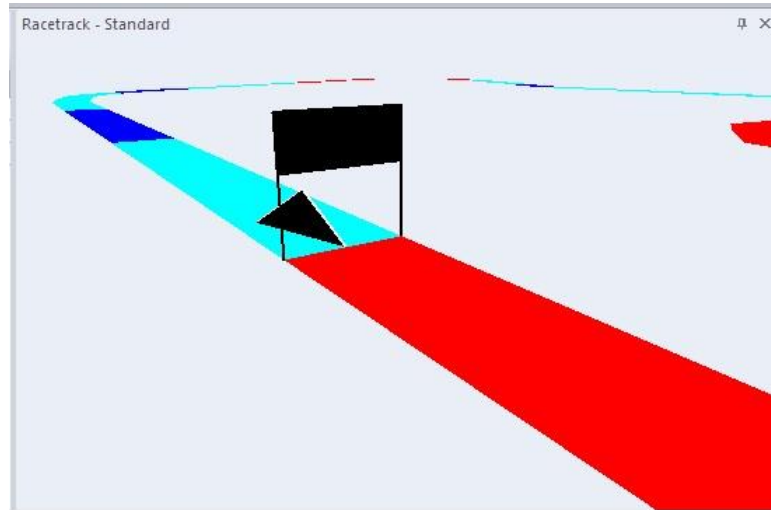
In the both 3D sub views you can zoom in, zoom out and rotate for better point of view.

The Vehicle Position

Vehicle positions on the track are indicated by colored triangles (or spheres). Their tips indicate the current position of each vehicle. Moving the crosshairs in the Oscilloscope repositions the vehicles along the track. In this way you are always able to view the position a vehicle occupied at a particular point in time.

Important:
Each vehicle position corresponds to the vehicle position of the current lap of the current Oscilloscope. The cursor in the Oscilloscope gives the position of the vehicle for active file. All other vehicles in the window may be shifted by some distance and so the lap position must not be the same.

Because of this the racetrack window shows different lap positions for the each vehicle on the course reflect the position marked with the crosshairs. Because the active file determines the x-axis of the Oscilloscope the position of the crosshairs corresponds to the position occupied by the vehicle of the active file. All other vehicle positions on the course correspond to the position they occupy in relation to their own x-axes (which are not displayed).

Windows/ Dockable windows/ Racetrack

Note:
If the Racetrack Window has been closed, for some reason, you can open it again by choosing "**Windows/ Dockable windows/ Racetrack**" from the menu.

Clicking on any point on the racetrack moves the crosshairs in the Oscilloscope to that point on the course.

This action moves also the vehicle in the active file to the same position.

Note:
The vehicle in the active file always remains in the active lap. For example, if a vehicle is located in front of the Start/Finish line and you click on a position behind the Start/Finish line the vehicle will return to the Start position in that same lap.

Note:
The size of vehicle cannot exceed a certain size. This could happen if the length of the racetrack is very small (< 2000m).

Multi Segmentations

If you have selected several segmentations within the racetrack explorer you can see them in the Racetrack Window (in 2D view and 3D overview, but not in 3D drivers view). In this view you can compare several segmentations.

The first segmentation is shown as most inner segmentation and the segmentation are going in direction outside, so the last (it is the default and it is marked with * in Racetrack Windows title) segmentation is the most outer (for the order of segmentations see Selecting Racetracks Segmentation). Only for the first and the last segmentation the segments will be shown.

If in the current Oscilloscope are two files opened then the Racetrack Window will show the different cars in different segmentations. (First car on first segmentation and so on…).
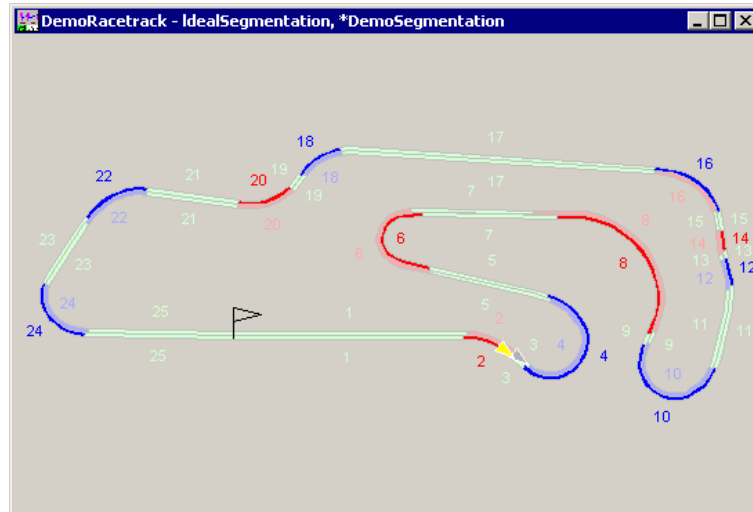
You can switch to 3D view and zoom in parts of the racetrack to see the difference closely. Moving over a segment from any of the segmentations a small yellow window (quick segment info) will appear. There the segmentations name and the segments name (if the name is not available then the index) are listed.
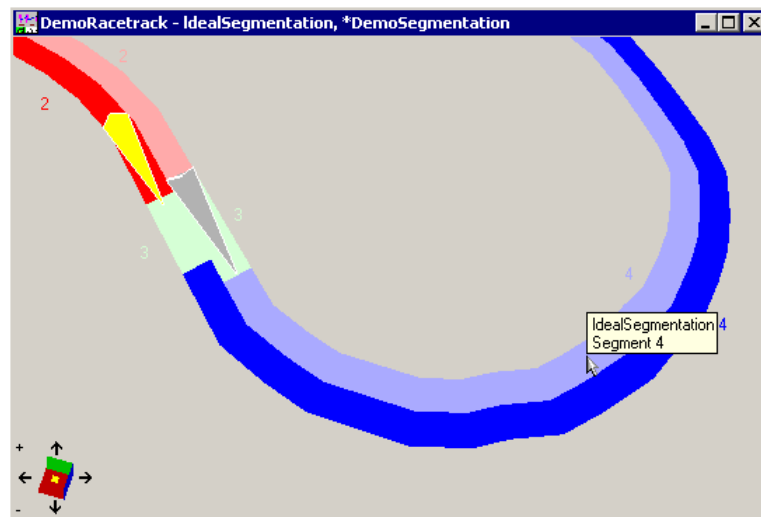
If you switch to some of the edit modes or to 3D drivers view only the default segmentation will stay (last selected one during the multi selection in Racetrack Explorer).

3D Zoom & 3D Drag functions

"+"   Zoom in. You can use "+" on 3D navigator and also double click with left mouse button or scroll in with the mouse scroll wheel.

"-"   Zoom out. You can use "-"on 3D navigator and also double click with right mouse button or scroll out with the mouse scroll wheel.

Context menu/   Show All. The scene will be moved to the default position.

Show All

Context menu/ Zoom undo   Zoom undo. The previous zoom state will be returned.

If you drag the scene with the left mouse button the scene will be rotated. If you drag the scene with the right mouse button then the scene will be translated.

Zooming a Segment

Double-clicking on a course segment causes the current Oscilloscope display to zoom in on the selected segment. At the same time the crosshairs (and hence the vehicle in the active file) will move to the position you have double-clicked.

Context Menu

In the Racetrack Window a local menu is available. In 2D view the menu is opened by clicking with the right mouse button on the Racetrack Window, in 3D view by clicking with the middle mouse button or by clicking with the left and right mouse button simultaneously. There you will find various functions: changing the view, save, load a racetrack, changing the car shape, e.g. If you are in edit mode the local menu gives you possibilities to save or cancel changes and other function depending on the hit of current mouse position.

Context menu/ Load Racetrack   You can select a segmentation/segmentations from Racetrack Window with "Context menu/Load Racetrack" the other windows will be informed for the new selected segmentation (for example the Oscilloscopes will show the new selected segmentation).

In the menu you will find a submenu for the car shape. You can select the size and the shape of the car.

The Video Window

The video window allows you to synchronize a video with the data. The video position will then be always aligned with the data position and lets you compare the data to the video.

WinDarab does not provide video codecs!
In order to use the videos, the corresponding codecs have to be installed on the PC by the user. For example, the free LAV Filters codec pack provides video codecs for the most used formats.

Open the Video Window by *right clicking* in an empty worksheet space and *selecting "Video"* in the appearing **Darab context menu**.
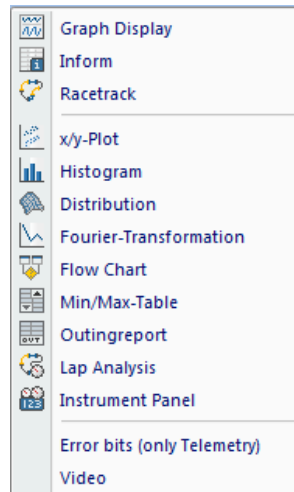
The Video Window allows you to synchronize a video with the displayed data.

Settings Dialog

Open a video by pressing F5 or by selecting "Settings …" in the video windows context menu.
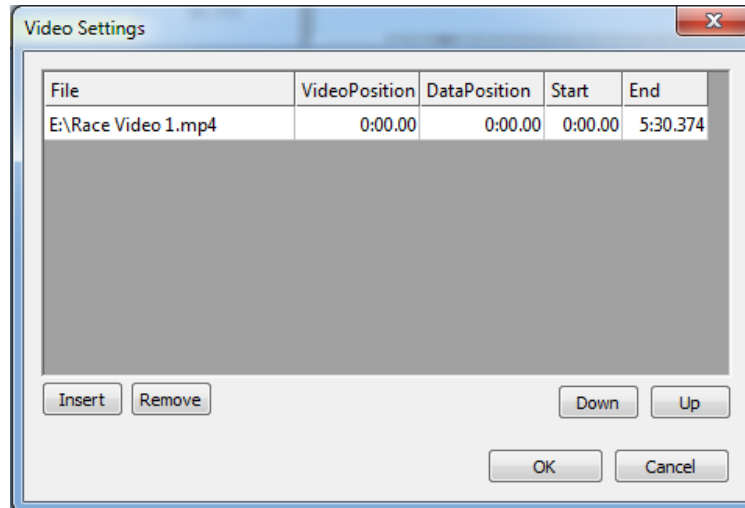
The Settings Window shows the used videos and their synchronization parameters.

File	The file name of the selected video
Video-/DataPosition	The synchronization points in the video and the data. These are the connected time values.
Start/End	The start and End-Time of the video.

You can use more than one video, by using the **Insert** button again. The used videos can be rearranged by the **Up** and **Down** buttons or deleted from the configuration by selecting a video and clicking the **Remove** button.

Apply the settings with the **OK** button.

Navigation

The Navigation in the video player is done in the **navigation bar** below the video.

The bar changes according to the locked state. This state can be changed by clicking the **Lock/Unlock** button on the right side.
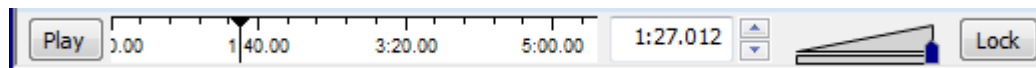
In the **locked state**, the navigation bar can't be used. Instead the video is **controlled by the cursor position in the oscilloscope**. The play button and the time field are disabled and the navigation bar is hidden by default. It can be **temporarily revealed** by moving the mouse to the bottom of the video window.

In the unlocked state, the video is controlled by the navigation bar. You can change the position of the video by either clicking into the time bar, dragging the cursor, changing the time in the time field or clicking on the up/down arrow. These arrows jump **one frame forward** or **backward**.

The Play button allows you to play or stop the video. The sound volume can be changed by the audio slider.

When the video and the data are aligned, press Lock to save the synchronization point. You can now control the position by the oscilloscope.

You can zoom into the navigation bar by *right clicking* on the **navigation bar** and *selecting* the **new navigation range**.
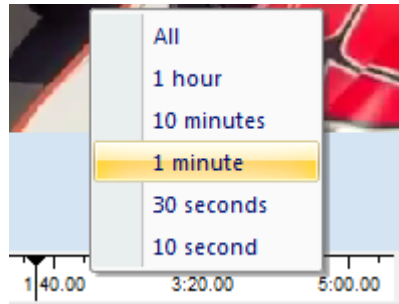
The Inform Window

Drag "**Windows/ Worksheet controls/ Inform**" from the menu into your workspace to open the Inform Window.
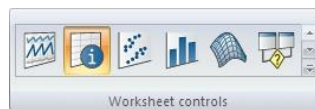
The Inform window can also be opened by selecting "Inform" in Darab context menu.
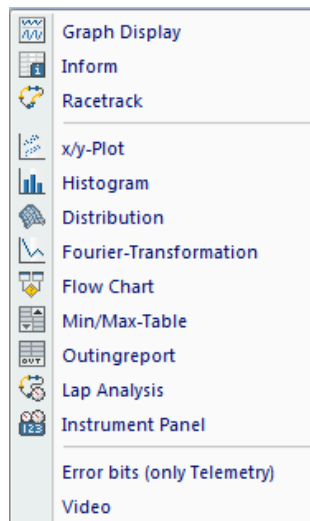
Data relating to the status blocks of the recording equipment is displayed in the Inform Window. It usually includes a number of Inform Pages with information grouped into different categories.

The number, layout and content of these pages depend on the controller you are using and how it is configured. These pages will therefore be generated specifically for each user.
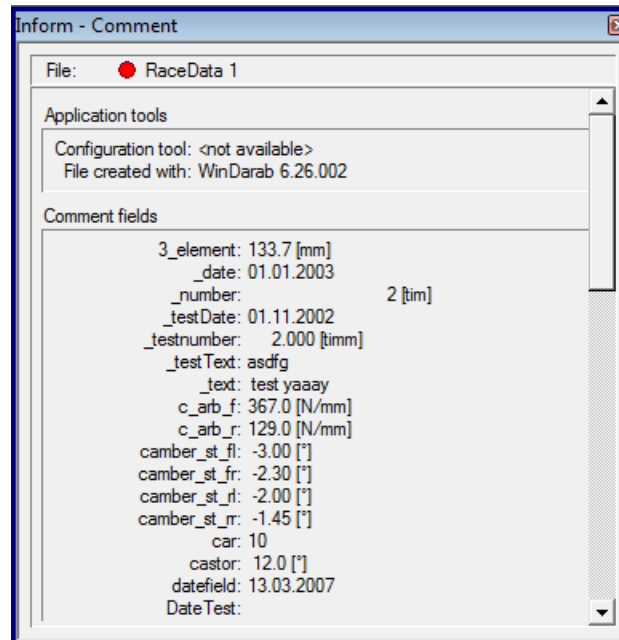
Note:

Likewise, File Explorers Preview Function displays Inform Pages when a Logged Data File has been selected.

Note:

.ini and .dat Files are provided with the ECU software.

Open

To open the Inform Window select "**Windows/ Worksheet controls/ Inform**" from the menu (see above).

Note:

If more than one Logged Data File is opened choose which file should be displayed.

Start/ Inform page/ Show page
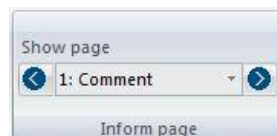
If you would like to view another Inform Page choose the desired page from the list in the Ribbon bar. Alternatively, you can browse backward or forward through the Inform Pages using the buttons to the left- or right-hand side of the list respectively.

Start/ Edit/ Comments

Clicking on this button displays a dialog box for entering comments related to the selected file. You will also be able to enter information into any previously defined comment fields.
All comments will be saved in the Logged Data File and displayed on the Inform Page called "Comments".

Comment fields tab
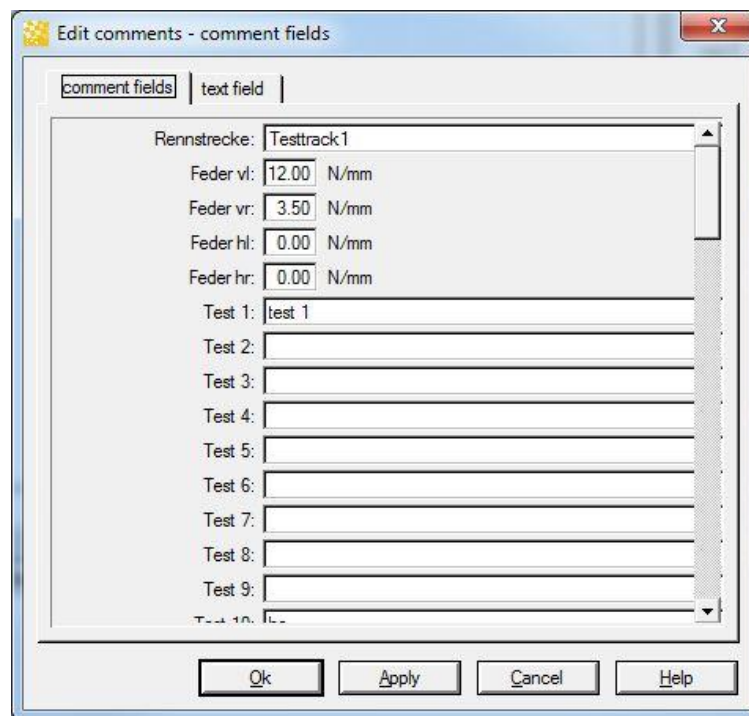
In this tab, you can assign values to the in the Comments tab in Tools/Settings/Program defined Comment-Fields.

Fields that are removed from the Comment-Field-List in the Comments tab remain in the File. The Comments in the Comment-Field-List are always shown first.

Text field tab

In every File, you have a texfield that you can edit in this tab.

Start/ Overlay
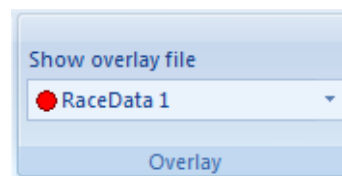
Clicking on an Item in the list displays the chosen file in the Inform Window.

Special Channels

Internal Channels

In addition to defining recorded channels WinDarab also defines its own, internal channels. Apart from some exceptions, internal channels are treated like recorded channels.

The following table lists all internal channels and their descriptions:

Channel	Description
laptrig	1 when vehicle is passing lap trigger, otherwise 0
Mux	Row values identifier (Reader/CanCard) Block number (CardMemory)
samples	Record Number/ Samples Channel has a special plotting routine, plotting every record as a single dot.
xdist	Distance axis Logged distance travelled will be standardised to accord with the course length-- if the difference between the actually logged lap length and specified lap length is less than 5 %. If the channel has not been recorded directly, it will be calculated using values from rawdist and laptrig.
xtime	Time axis
lapdist	Distance travelled on the course since last lap trigger.
laptime	Time logged since last lap trigger.
rawdist	Logged distance
dbg_time	Block time stamp (CardMemory with active debug switch only)
dbg_dist	Block distance stamp (CardMemory with active debug switch only)

Note:
Some channels may not be available depending on the hardware you use and its configuration.

Special channels settings

WinDarab provides the user the possibility to define a list of special channels for the following categories:

Category	Description
Distance	Contains cannels or calculation methods WinDarab should use to generate distance information. Predefined calculation methods are "Calculate using GPS", "Calculate using speed" and "Default distance".
Speed	List of channels, which contain speed information. If a file contains one of the channels in the list, it will be used in other calculation (e.g. calculation of the distance information). Predefined channels are "vfz_w", "vfzg" and "speed".
GPS Horizontal (meter)	List of channels, which contain horizontal GPS-coordinates containing data, which is transformed into relative meters.
GPS Vertical (meter)	List of channels, which contain vertical GPS-coordinates containing data, which is transformed into relative meters.
GPS Height (meter)	List of channels containing GPS-height in meters.
GPS Longitude (Earth)	List of channels, which contain the longitude of the GPS-coordinates. The channel "gps_long" is predefined.
GPS Latitude (Earth)	List of channels, which contain the latitude of the GPS-coordinates. The channel "gps_lat" is predefined.
GPS Height (Earth)	List of channels, which contain the height of the GPS-coordinates. The channel "gps_alt" is predefined.

The user can selects special channels for each category. WinDarab uses these lists to find channels containing needed information. If a file contains one of the channels listed it will be used for further calculations. If a file contains more than one of the channels in the list, the user selects must select one.

Defining a special channels list

To define special channels for the categories listed above or to modify a special channels list, follow these steps:

1. Use "**Tools/ Settings/ Program**" to open the "Options" dialog box.
2. Select the "**Special channels**" tab.
3. Select the category from the "**Special channels for**" – list.
4. The "**Special channels**" tab contains two channel lists - list of all channels in the measurement file and list of special cannels for the selected category. To add a channel to special channels list, select the channel from the list of all channels and click on the

    "add" –button.
   To delete a channel from the special channel list, select the channel from the special channel list and click on the

    "delete"-button.
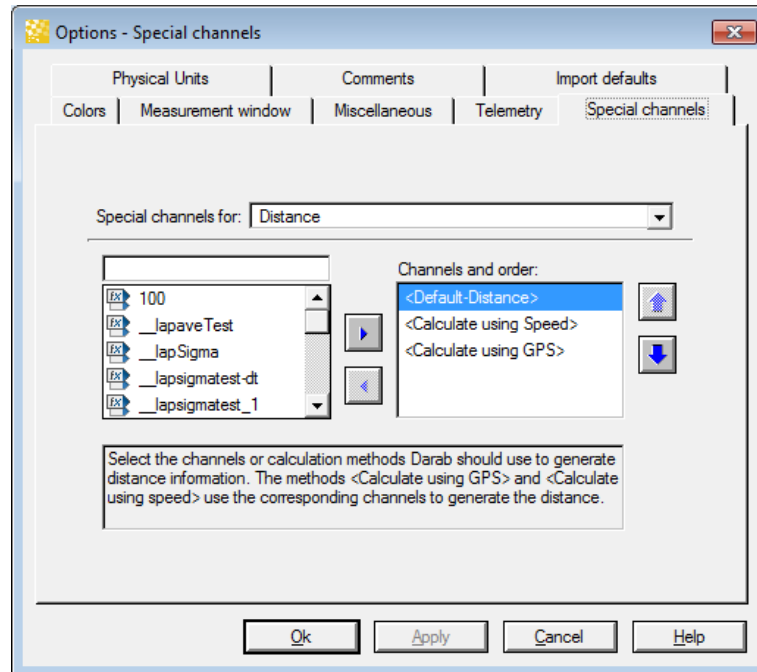
The channels and the calculation methods in the special channel list have a priority. The position of a channel in the list determines its priority. The higher the position of a channel in the list, the higher the priority. The user can change the priority of a channel or calculation method as follow:

1. Select the desired channel in the list

2. Place the channel to the desired position by using the "up" and "down" – buttons.

Note:

The reserved entries in a special channel list cannot be removed.

For distance generation information is valid - the calculation methods always have a higher priority than the channels selected by the user.

If the channel provided for distance information was not sampled with each sample block, the interpolation of the channel for each sample position was not "good" enough, if the channels quantization didn't offer enough precision for the interpolated values.

The channels provided for distance generation shall be monotonically strictly increased. Except the first two samples of the channel. In this very special case the first sample will be extrapolated based on the values of the second and third sample. WinDarab carries out additional checks to ensure this and shows a message, if a distance channel cannot be used.

If the distance channel isn't strictly increasing, WinDarab interpolates the duplicate values and saves the results to the file. This is done once when the channel is accepted as distance channel.

If distance information is calculated by GPS (the calculation method "Calculate using GPS" is selected) and WinDarab runs with a unit converter for the distance channel, the lap normalization is not working in the case the user changes the current racetrack. Only in this case the data is not normalized (but as measured!).

Mathematical Channels (and Filters)

In WinDarab it is possible to define filters or new channels with the help of mathematical functions.

Win Darab provides functions and filters that will help you to solve a great number of mathematical and analytical problems.

In a mathematical function it is possible to use comment fields — in addition to recorded channels — to include conditions that are specific to a particular outing (outing-specific constants). Comment fields are included in the channel selection list and marked with "C".

Mathematical Channels

A mathematical channel calculates values based on other channels in a file. Which channels will be used is determined by the mathematical function.

If a mathematical channel has been defined, it will be listed in the channel display area in a Logged Data File. Much like internal channels, mathematical channels can be used (in nearly all instances) as if they were logged data.

If new a mathematical channel is added, but another mathematical channel with the same name already exists, the existing mathematical channel is replaced by the new one. In case of a manual creation of the new math channel (Extras/Functions/New) a message box appears, informing the user about the duplicated math channel name.

Mathematical Filters

Unlike a mathematical channel a filter does not require the definition of a new channel. Instead, it only requires the definition of a mathematical function, which can be used in conjunction with any channel you wish.

In order to define the values within a function those are to be replaced by values of a filtered channels, add the channel "Default" as a wildcard.

The section "Additional Channel Functions" describes how to assign a filter to a channel.

A mathematical filter does not need to filter a channel necessarily. A filter can also be used to calculate derivations and integrals.

Working with Functions

Mathematical functions can be saved in a folder in the File Explorer.

In addition to the selected folder and its contents, the lower area of this window shows a summary of the most important information pertaining to a mathematical function.
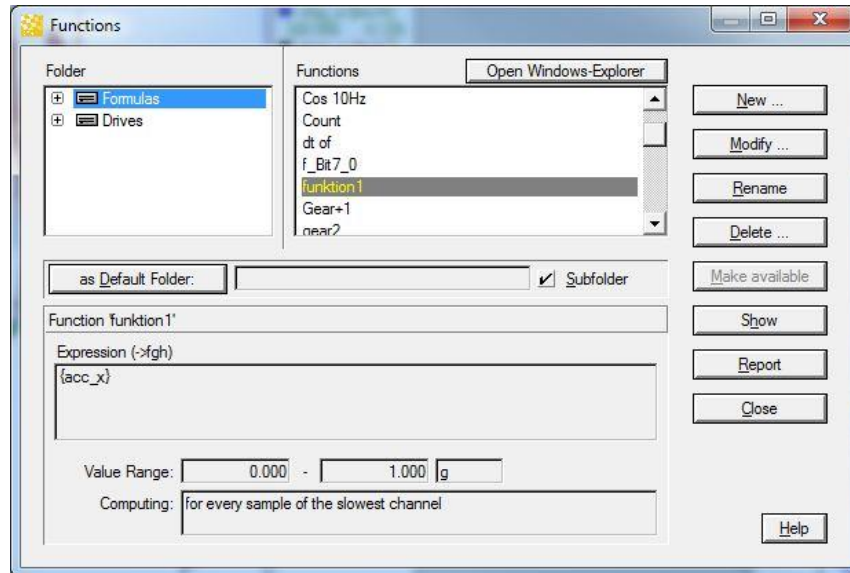
"As Default Folder" Button
If you start the program WinDarab makes all functions saved to a specified folder available. To specify this folder select it from the directory and click on the "As Default Folder" button.

"Sub-Folder" Option
Activating this button makes all formulae contained in the default folder as well as those contained in the sub-folders available.

"Make Available" Button
If you would like to make a mathematical function available you will have to do this explicitly.
This button is not active when the mathematical function has already been made available.

Report
This button enables you to create a report that contains all the settings for a mathematical function. If you press this button, a dialog box for choosing a report template appears.

WinDarab provides one template for each language it supports.

If two functions with the same name exist WinDarab will only use the first one.

Defining a Function

To define a new mathematical function or to modify an existing one, follow these steps:

3. Use "**Tools/ Mathematics/ Functions**" to open the "Functions" dialog box.
4. Select the mathematical function you want to edit and click on "Modify" or "New/Function" if you want to create a new function.
5. The dialog box "Edit Function" appears. It contains three tabs in which you can make all entries pertaining to the mathematical formula.
6. Once you have finished making your entries or modifications, click on "Save" or "Save as".

When a mathematical function is modified, WinDarab immediately makes a new calculation if this function is displayed or in use in a window.

You can also create a new function by right-clicking into the Channels window and select Math function/ New. If you right-click on a function in the Channels window, you can chose to edit this function.

Math formulas and channels with identical name are not allowed.

If a user tries to save a math formula but at least one opened file contains a data channel with the same name, WinDarab asks the user to select a different name for the math formula.
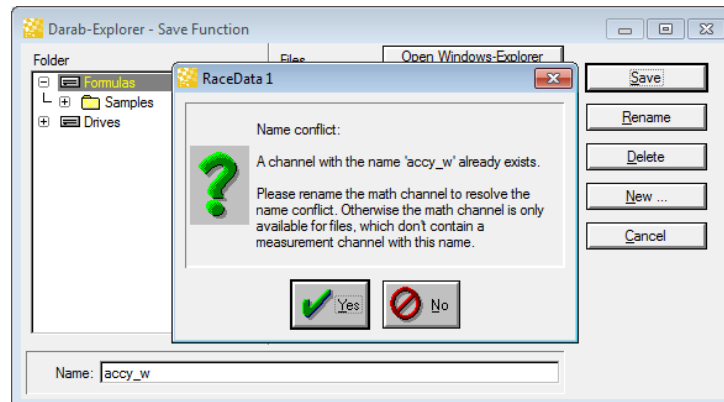
WinDarab shows a message, if a file contains data channels with the same name as already defined math formulas. The data channels are not available until the user changes the name of the math formula and reopens the file.

See also:

- "Expression" Tab
- "Format" Tab
- "Calculation" Tab

Expression Tab

Enter the mathematical expression you want to calculate in the "Expression" tab. In addition, it is possible to enter a brief description that will be displayed in channel selection lists.

In the "Check expression with…" field you are able to select a file containing a list of channel names. This enables you to access the choice of available channels by using the "Channel…" button. WinDarab will then test the syntax of the expression.
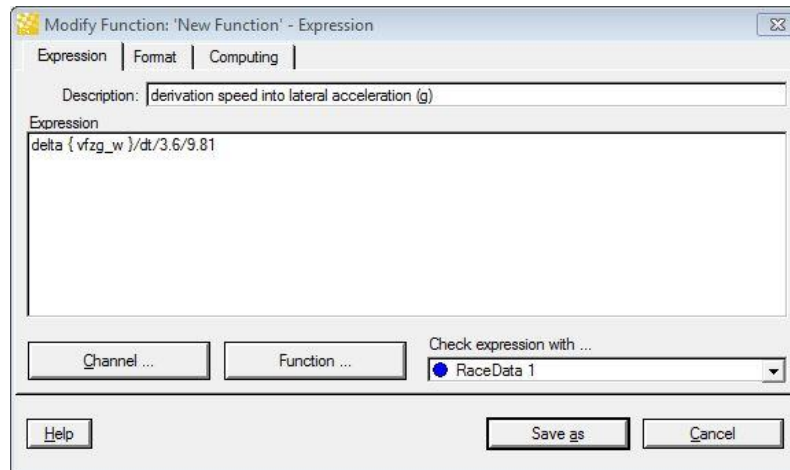
Selecting a Channel

Channel names should always be enclosed in braces. Alternatively, you can insert a channel in an expression by using the "**Channel…**" button:

1. Position the cursor at the point in the expression where you want to insert the channel.
2. Click on "Channel…".
3. Select the desired channel from the list and then click on "Selection".

The "Default" channel is not a channel belonging to a particular file but rather a wildcard used in mathematical filters.

In a mathematical function it is possible to use comment fields — in addition to recorded channels — to include conditions that are specific to a particular outing (outing-specific constants). Comment fields are included in the channel selection list and marked with "C".

A formula is not limited to a single line. If a formula is too long or complex for one line, you can split it by using multiple lines.

Selecting a Function

Similar steps are involved in inserting a function or an operator into the formula:

1. Position the cursor at the point in the expression where you want to insert a function or an operator.
2. Click on "Function …"
3. Select the desired operator from the list and then click on "Select".
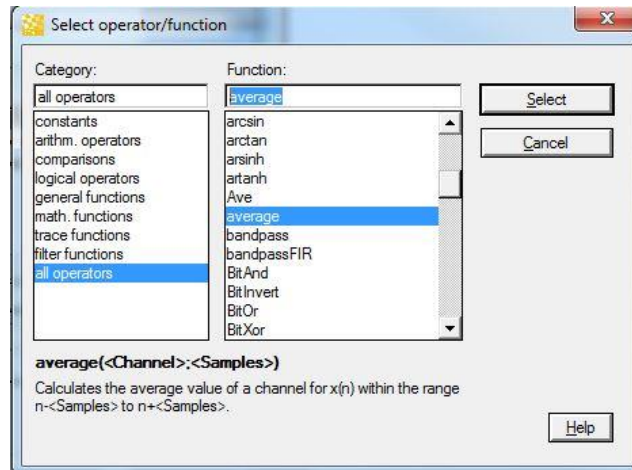
All functions and operators are organised in groups. You are able to limit the elements contained in the list of functions by pre-selecting a group of functions.

If you select a function from the list, a brief description will be shown in the lower area of the window.

For further information see Appendix / Mathematical Expressions.

Format Tab

All the settings pertaining to the display of a mathematical channel are made in the "Format" tab.
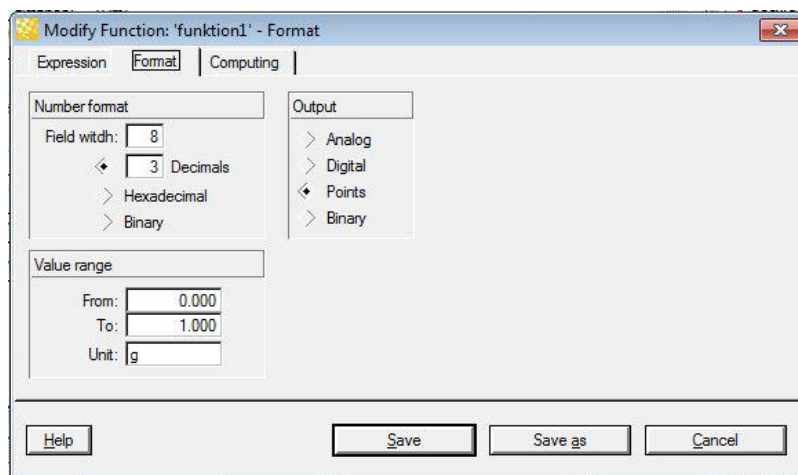
Numerical Format

Make the settings for the numeric display of the channel values. You are able to choose either a "decimal", "hexadecimal" or "binary" format. If you choose the decimal format, you are able to specify the width of the entry field as well as the number of decimal places.

Range of Values

Use this area to set the range of values (from - to) and the unit of measure to be used in the default channel display.

Computing Tab

This tab is used to determine the sampling rate for calculating a mathematical function.

Depending on the configuration for recording data, data is sampled at varying intervals. As a result, the number of samples per unit of time may vary.

Depending on the meaning of the variables in a mathematical calculation, it may be necessary to vary the sampling rate. This setting determines the quality and speed of processing.
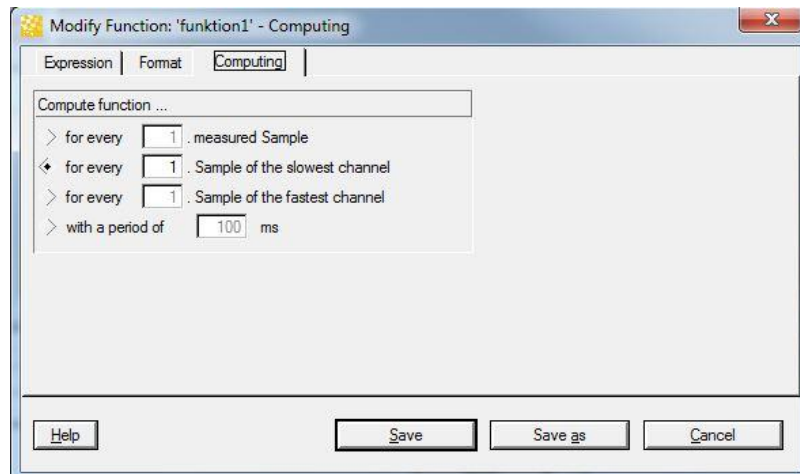
Compute function …for every <x>th measured Sample
This setting calculates the function independent of the samples that were recorded and which belong to channels included in the formula. Calculations are instead carried out for each <x>th sample recorded in the file.

Compute function …for every <x>th sample of the slowest channel recorded
Before WinDarab begins the calculation, it searches for the channel with the lowest sampling rate. The calculation is then only carried out for every <x>th sample record in this channel.

Compute function …for every <x>th sample of the fastest channel recorded
Before WinDarab begins the calculation, it searches for the channel with the highest sampling rate. The calculation is then carried out for every <x>th sample record in this channel.

> If WinDarab searches for the fastest channel it ignores standard channels such as xtime or xdist because they are included in every block and thus would always be the fastest channels. If you would indeed like to make a calculation for these samples, you must select the first option (i.e. "for each <x>th sample recorded").

Compute function with a period of <x> ms
WinDarab calculates the function at a specified minimum time interval.

> WinDarab is able to perform a calculation only if sample positions are valid. As a result, WinDarab always searches for the sample separated from the previous sample by at least the specified value. It is therefore possible that the intervals vary – but they are always at least as big as the specified value.

Computing Example:

Assume there are two channels (v_wheel_rb and v_wheel_rf) with different sampling rates (100ms and 130ms) and you want to create a math-channel to calculate the average of these channels.

You can create a math-channel with the expression "({v_wheel_rb} + {v_wheel_rf})/2".

The Image below shows the two channels in the upper area and the math-channel with the computing-configurations "every sample from the fastest channel", "every sample from the slowest channel", "every 150ms" (from top to bottom in the bottom area).
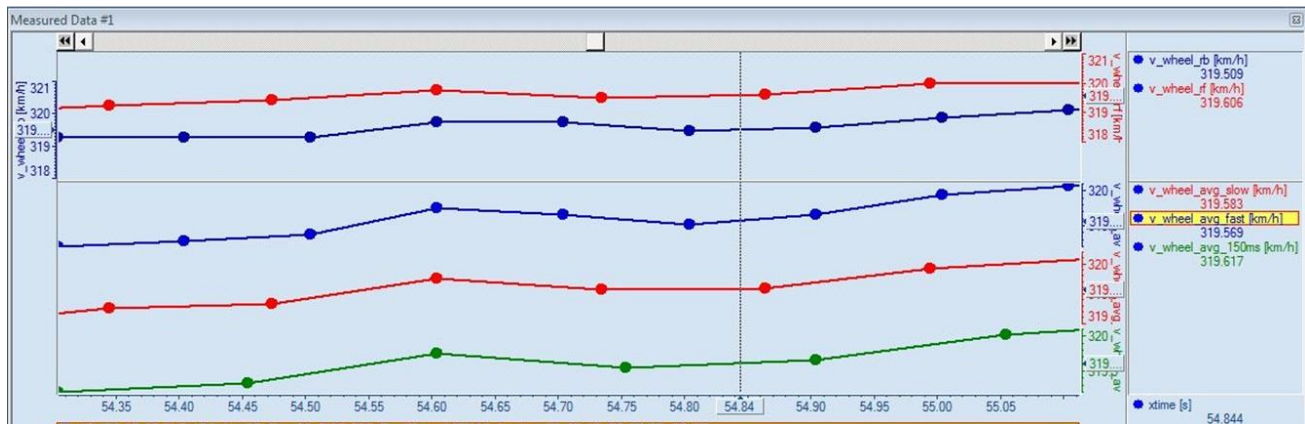
As you can see, the blue math-channel has the same sample-points as the blue channel (fastest).

The red math-channel has the same sample-points as the red channel (slowest).

The green math-channel has its own sample-points that are not in any way linked to the included channels sample-points.

The sample-points that are not on the channels-points are interpolated by the surrounding sample-points of the channel. As an example, the blue math-channel uses the exact values from the blue channel and interpolates (where the sample-points are not at the same point by a coincident) the red channel.

Filter Functions

For this section, mathematical foundations of filters are required.

WinDarab provides a lot of mathematical filters, such as:

- bandpass
- bandpassFIR
- filter
- lowpass
- lowpassFIR
- highpass
- highpassFIR
- stopband

They can all be found in the function window in the category "filter functions".
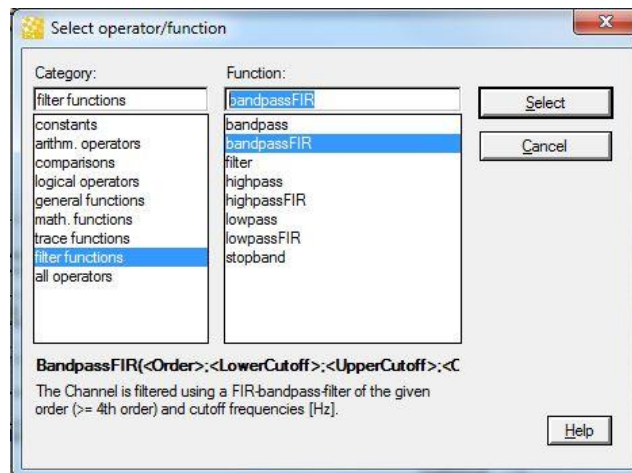
Bandpass, Lowpass, Highpass, and Stopband Functions

These Functions all have an editor, where you can insert the parameters for the function.
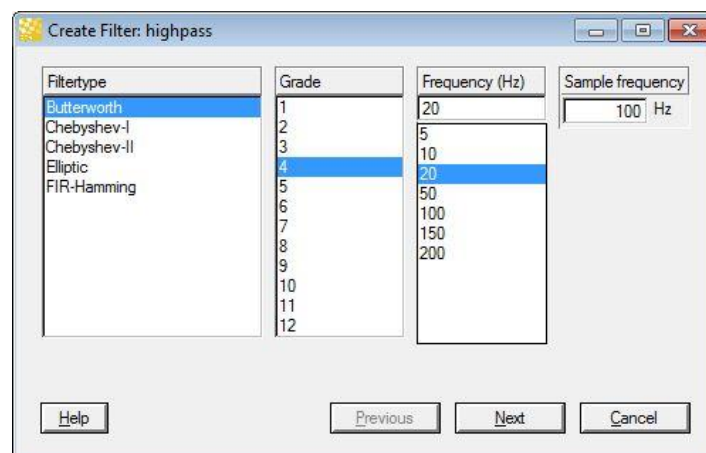
In the filtertype-list are up to six different filtertypes, that you can chose for your filter. Each of the filters have individual strengths and weaknesses regarding roll-off rate or phase response

The other lists supply the parameters for the order, the frequency and the sample frequency.

Keep in mind, that the sample frequency has to be at least twice the frequency.

When all parameters are configured, you can go to the expression editor by clicking on Next.
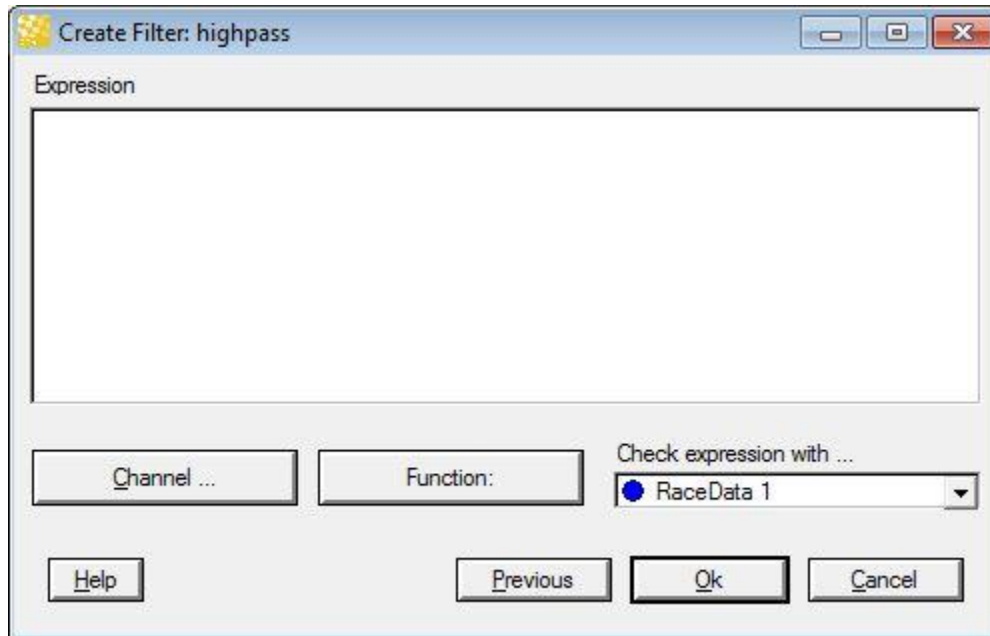
In this dialog, you have to enter the expression you want to filter. The expression can be as complex as you require.

It is also possible to define another filter within this expression by calling up the Assistant again.

Filter Function

The Assistants first dialog box is used to specify the expression you want to filter.

The second dialog box allows you to enter the coeffizients for the filter function.
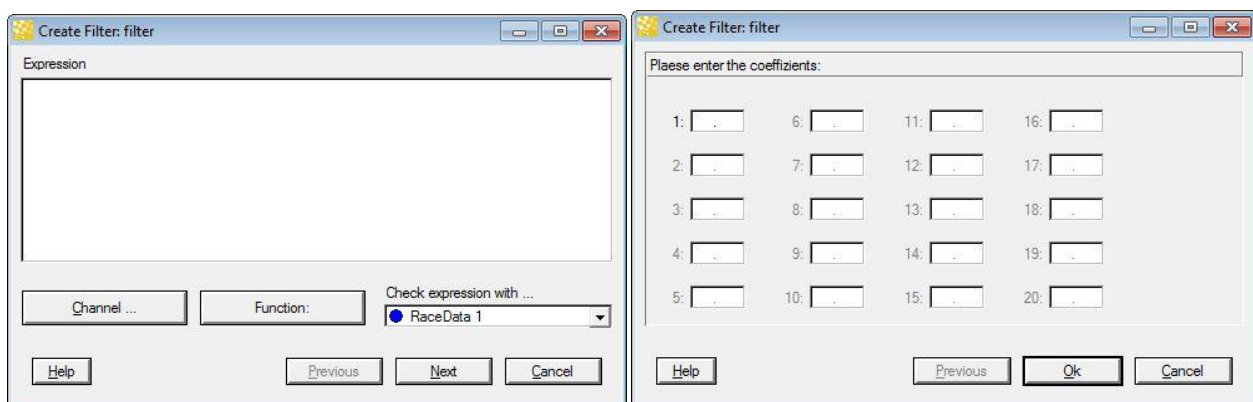
The Assistant supports up to 20 sampling points. If your filter requires more sampling points, you have to add these manually to the expression created by the assistant.

BandpassFIR, LowpassFIR and HighpassFIR Functions

These functions don't require an assistant. They all require as parameters the order of the filter, the channel to filter and at least one cut-off frequency (bandpass requires two).

All other parameters like the frequency are calculated.

Lookup Tables

WinDarab allows you to define tables to map input values to a specified output-value.

Defining a Lookup Table

To define a new lookup table or to modify an existing one, follow these steps:

1. Use "**Tools/ Mathematics/ Functions**" to open the "Functions" dialog box.
2. Select the lookup table you want to edit and click on "Modify" or "New/Lookup Table" if you want to create a new lookup table.
3. The lookup table dialog box appears.
4. Once you have finished making your entries or modifications, click on "Save" or "Save as" to save the lookup table.

Lookup Table dialog
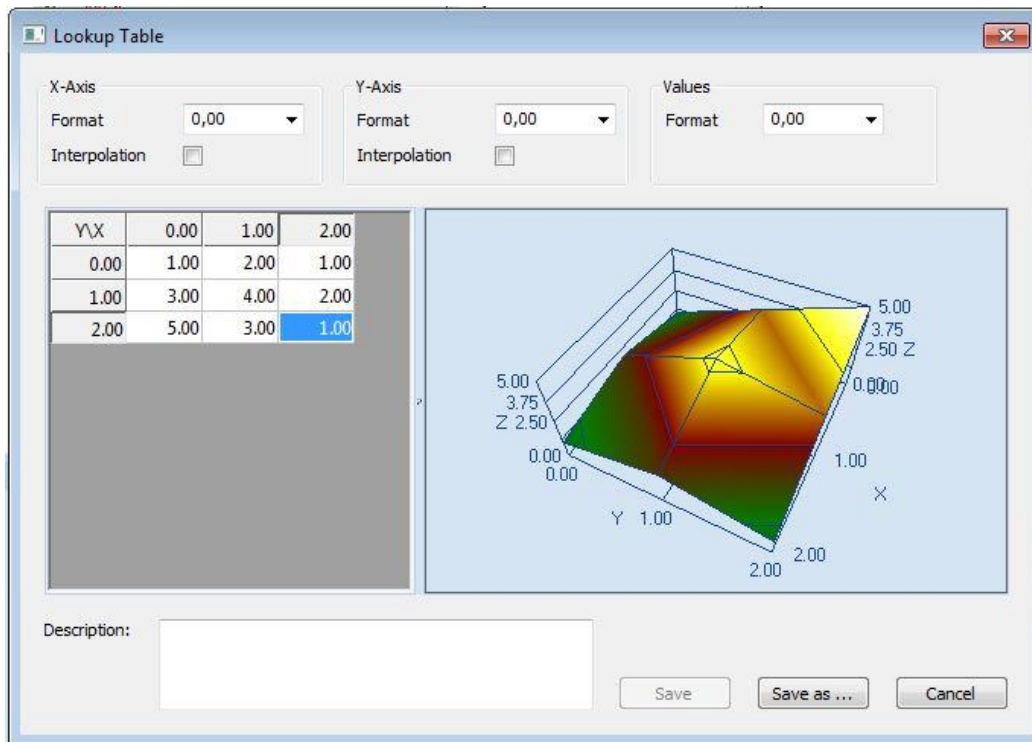
On the top of the dialog, there is the configuration-area. It allows you to set the formats of the axes and values as well as setting the interpolation for the axes.

The interpolation property controls the behavior of the table when facing input parameters that are between two axis values. If the interpolation is turned off, the input parameter is handled as the nearest axis value. With interpolation turned on, the result value is calculated by interpolating over the surrounding values.

Example:

Lets take the Table in the Picture above as an example and use the input parameters X = 0.9 and Y = 0.9;

- If there was no interpolation, the return value would be 4.00, because the input parameters are handled as X = 1.0 and Y = 1.0.
- If only X were Interpolated, the return value would be 3.9, because the y parameter is handled as 1.0 and the return value is interpolated between the values 3.0 and 4.0.
- If only Y were Interpolated, the return value would be 3.8, because the x parameter is handled as 1.0 and the return value is interpolated between the values 2.0 and 4.0.
- If both axes were interpolated the return value would be 3.7. There is a pane built out of three surrounding points to calculate the interpolated value.

Note:

There are 2 possible triangles – that include the target point - which can be chosen for the interpolation pane.

On the bottom of the dialog, the description of the lookup table can be placed.

On the left is the most important part of the dialog. There you can edit the values and axes of your lookup table. You can add new rows/columns by rightclicking on an existing row/column and press add Insert before/Insert behind. Removing a row/column works the same just by clicking remove in the appearing context menu.

> You can add as many rows/columns as you like. Note, that after removing a row/column there has to be at least one row and two columns to work.

You can change the values of the table and the axes freely as long as they are valid decimal numbers.

> The axis-values have to be strictly increasing. While editing the values, wrong numbers will be marked red and have to be changed before the table can be saved.

On the right side is a 3D-model of your table. It can be rotated (left mouse button drag), moved (right mouse button drag) and scaled (mouse wheel).

> If there is only one row in the table, the 3D-model shows a 2D-model like in the FFT analysis window

> The 3D-Model always shows the values as if both axes were interpolated.
>
> Note that if the axis is not interpolated, values between two axis values are interpreted as the nearest axis value.

Using the lookup table

Lookup tables can be accessed in any math function like math channels or math filters by using the "Lookup" function:

```
Lookup(<LookupTableName>; <X-Param> [; <Y-Param>])
```

> Because the function only takes the name of the lookup table to be used, WinDarab has to search for the lookup table in the formula folders.
> WinDarab looks for the lookup table in the following sequence:
>
> - In the same folder as the math function is saved in.
>
> - In the parent folders of the math function.
>
> - Any lookup table matching the name.

> The parameter count has to match the dimension of the lookup table.

Lookup table example

You have a measurement sensor attached to a ADC pin of your ECU (named ADCPin). But you want to display the physical values instead of the voltage values in WinDarab.

For this you create a lookup table "ADCPin_Characteristic" containing the characteristic curve of your sensor. The output values will be interpolated corresponding to the input value and its neighbor breakpoints on the x-axis.
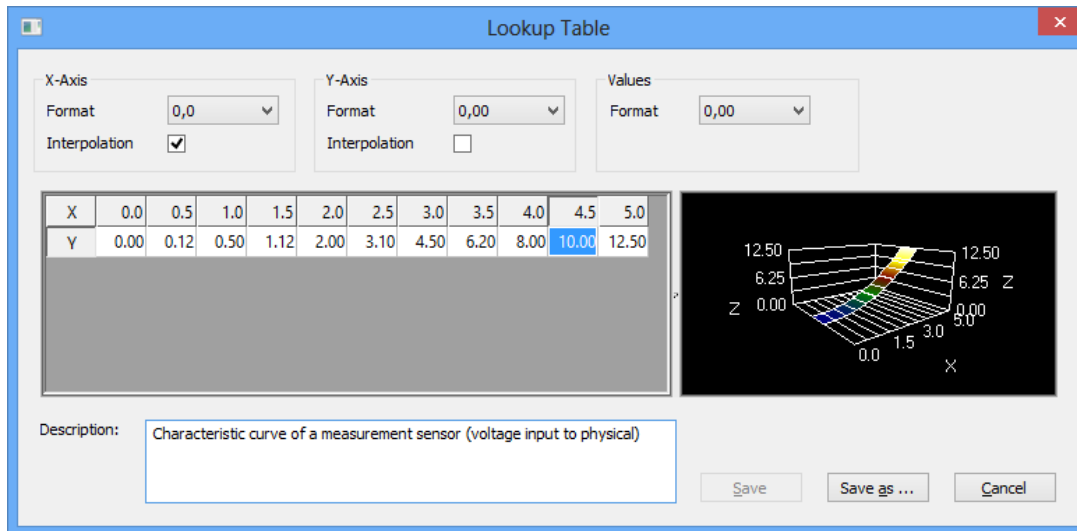
In addition you create a math channel "ADCPin_Physical", which converts the voltage input value to the corresponding physical value. We only need a single input argument, because the lookup table contains a single row only:
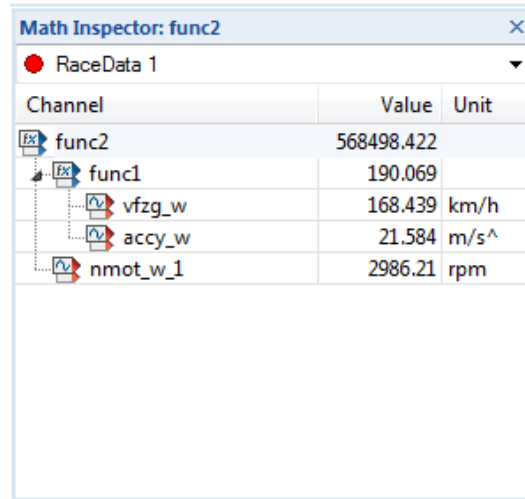
```
ADCPin_Physical =  Lookup ({ADCPin_Characteristic}; {ADCPin})
```

The results of the math channel would be something like this:

Input value (ADCPin)	Output value (ADXPin_Physical)
1,0	0,5
1,1	0,62
2,75	3,8
4,9	12

Math-Inspector

The Math-Inspector is a tool to show you the components of your function with their corresponding values at the current position.

Note:

The Math-Inspector shows when an error occurs. This could be a recursion, an invalid channelname or a used mathchannel with an error.

In these cases the valid values will be displayed anyway. Only the errors will be shown by a warning sign and an error message.

The same channel/ constant/ function will be shown only once as a child of a function.

Open the Math-Inspector by right-clicking on a math-channel in the channels-pane and selecting **Math Inspector** from the appearing menu.
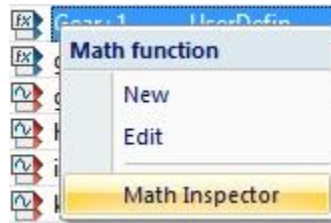
Overlay   You can assign a different overlay to the math inspector by selecting it in the combobox on top of the window.

Conditions

Conditions are used in conjunction with analysis functions in order to exclude certain data from consideration during data analysis.

A given data point is ignored unless the condition applies — i.e. if its evaluation results in "true". A condition is "true" if the formula for the given data results in a value that is not equal to 0.

Defining a Condition

Follow these steps to define a condition:

1. Use "**Tools/ Mathematics/ Conditions**" to open the "Conditions" dialog box.
2. If desired and available you can choose a condition from the "Name" list. You are able to modify this condition or else use it as a template.
3. Enter the condition in the "Formula" field. The formula can be as complex as required and may occupy more than one line.
4. Click on the "Save" button.
5. A small dialog box appears. Enter the name for the condition here.
6. Click on "OK". If the name you have entered is already in use you will be asked to confirm your entry.

Click on "Cancel" to close the "Conditions" dialog box.
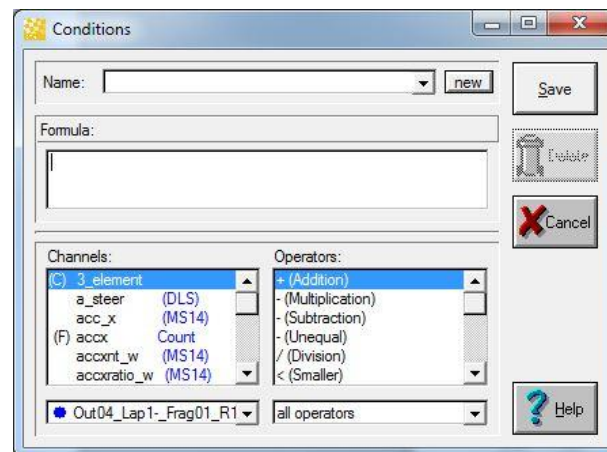
To facilitate the entry of formulae the dialog box contains lists with available channels and operators.

Channels
This area displays all the channels available for the selected file. You select a channel by double-clicking on it. The channel name will then be inserted in the formula field at the current cursor position.
Below the list you will find a field from which you can choose the file the channels should be selected from.

Operators
WinDarab provides numerous operators and mathematical formulae.
Select an operator from the list with double-click. It will then be inserted into the formula at the current cursor position.
Please consult the Appendix for more detailed information about operators and mathematical formulae.
Operators are grouped according to type. The list displays all operators belonging to the selected operator group.
Below the list of operators you can select the operator group.

Deleting a Condition

Follow these steps to delete a condition:

Use "**Tools/ Conditions**" to open the "Conditions" dialog box.

Select the name of the condition you want to delete from the "Name" list.

Click on the "Delete" button.

Confirm your deletion by clicking on "OK".

Temporary Conditions

Conditions don't have to be predefined to be used by WinDarab.

Instead of defining and selecting a condition in the analysis windows setup-dialog, you can simply type in the condition you want to apply.
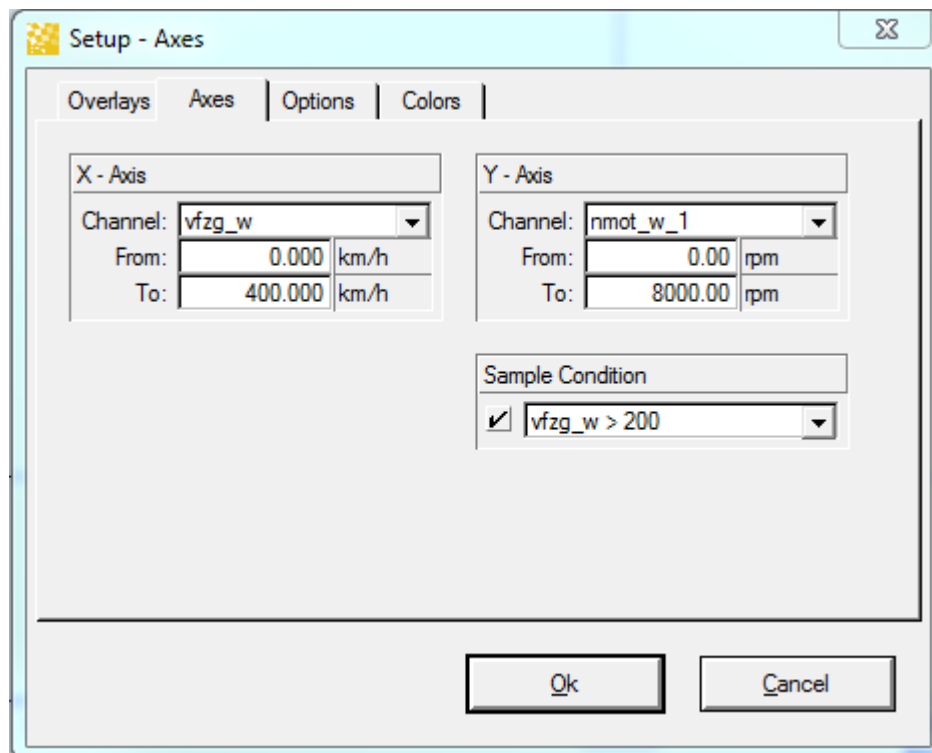
You can type in any expression that can be parsed as a condition.

Visualizing the Condition
Like a normal channel or a math channel, a condition can also be added to the oscilloscope. The values are 1 for "condition is true" or 0 for "condition is false".

Analysis

WinDarab provides a number of functions and algorithms which enable you to present and analyse data.

The following functions are accessible in the "**Windows/ Worksheet controls**" menu in **Darab Ribbon**:

- (x, y)-Plot
- Distribution
- Histogram
- FFT
- Flowchart
- Min/Max Table
- Outing Report
- Lap Analysis
- Instrument panel

Selecting one of these options opens a window which displays the results of the calculation that was performed. There is a separate dialog box for the settings.

The functions are also available from the WinDarab context menu. The WinDarab local menu can be opened by clicking with the right mouse button on an empty area.
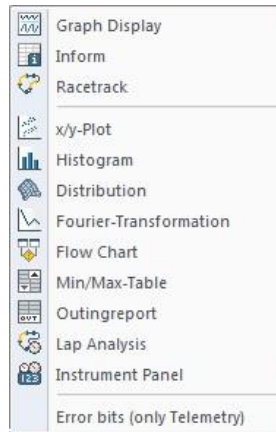
Some of these analysis functions are only available if the active Oscilloscope contains at least one channel.

If an analysis function is opened, the recently active Oscilloscope will provide data.

Some Analysis Windows only reflect a situation at a particular moment in time. Modifications (re-definition of functions, conditions, differences, or file alignments) will be reflected only when a new calculation has been performed. Re-opening and re-confirming the "Settings" dialog box accomplishes this.

You can move the Analysis Windows on your workspace by dragging the caption-bar or change the size by dragging the borders. Double-clicking the borders increases the size until it hits another Analysis Window. If windows are docked together you can move all borders at once by pressing shift while dragging the border.

Pressing **Shift + F4** will close the active window.

Settings in Analysis Windows

Nearly all Analysis Windows allow you to save the settings you have made for future use. Let us denote with [Analysis Window] the name of a analysis window you are interested in (for example x-y Plot, Distribution, Histogram, FFT, Flowchart, Min/Max Table/ Outing report or Lap analysis) then:

Load Settings

You can access pre-defined settings for analysis window by selecting "[Analysis Window] and select Start/ Settings/ Load" from the menu. It is available in the context menu of the Analysis Window too.

1. Select "**Start/ Settings/ Load**" from the menu.
2. Darab Explorer will appear.
3. Select the desired setting from the "Files" list.
4. Select the desired file with settings and confirm your selection by choosing "Open".
5. Your saved settings will be restored.

 Press **F6** to *open* the "**Load configuration**" File Explorer

 You can also load a configuration by chosing a recently used configuration from the MRU (**M**ost **R**ecently **U**sed) dropdown list right to the Load-Button.

Save Settings

You can save your current settings by selecting the **Analysis Window** and select "**Start/ Settings/ Save**" from the menu. It is available in the context menu too.

1. Select "**Start/ Settings/ Save**" from the menu.
2. File Explorer appears.
3. Select the file in which you want to save the settings or enter a name.
4. Confirm your selection by choosing "Save".

 Press **F7** to *open* the "**Save configuration**" File Explorer

---

Analysis

Common Overlays Settings

The most Analysis Windows are using common overlay settings. Therefore their setup dialog boxes contain the same tab called Overlays Tab.

This tab is used to select Overlays and their sample range for evaluation.
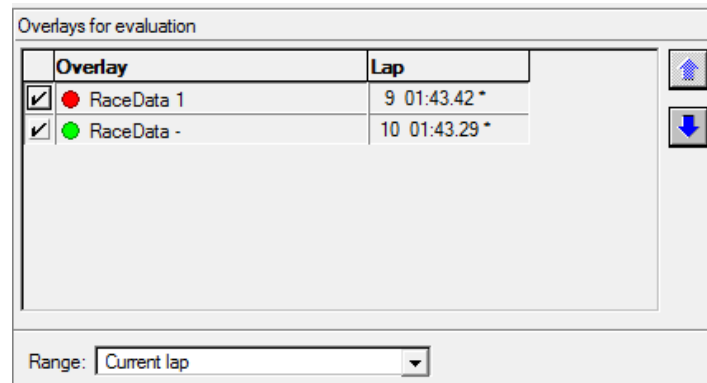
| | Table with Overlays | The table contains all overlays in the File Explorer Windows. The rows can contain any lap from the overlay. You can change the order of the overlays with the Up / Down arrows. |

ASCII Export

Some Analysis Windows allow you to export their results to an ASCII file.

You will find more detailed information about this topic under the ASCII Extract heading.

The Appendix contains a format description for the ASCII file that will be created.

(x, y)-Plot

In a (x, y) graph coordinates are calculated for a pair of values (x, y) in two channels (x- and y-axis).

This type of function allows you to determine very quickly which (x, y) pairs occur frequently as these tend to show up as a dense, cloud-like accumulation of dots.

A (x, y)-Plot window can display multiple (x, y)-Plots that are bound to different axes. This way it is very easy to compare the results of different (x, y) graphs.

In addition each (x, y)-Plot can be used to separate the clouds in components and then to find the regression line for every component. Also for every component the correlation of dots inside is found. You can also add your own (user) lines for comparison. In the next figure you can see 4 components and 4 regression lines which correspond to the gears of the car. There is also one user line indicated by the two movable handles (displayed as dots on the line).
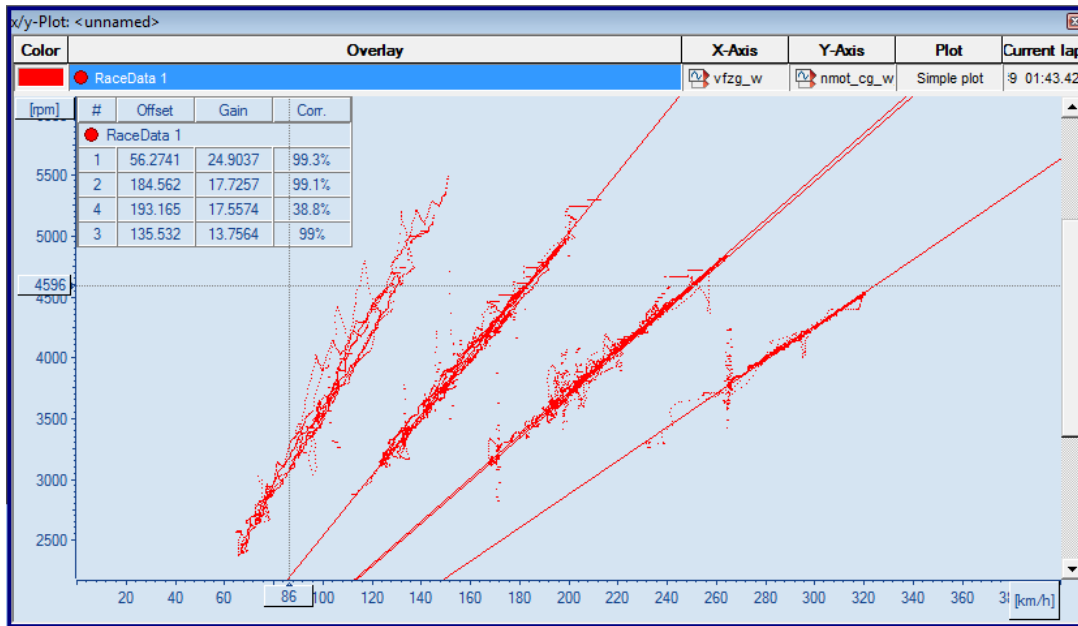
Open x/y plot

Drag "**Start/ Worksheet controls/ x/y-Plot**" into your workspace to open the (x, y)-Plot Window.
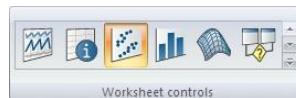
Middle Mouse Click (Left Button + Right Button)
Clicking your middle mouse button (or left + right mouse buttons) *displays* a **local menu** with options.

Calculations
Calculations are started after changes in the settings are made, or after "Update View" from the menu was clicked.

Because calculations can take much time especially if the range is above the whole file, they are made inside of another thread and you will see a progress bar in this case. Instead of waiting for them you can work in some other windows.

For online files calculations are started automatically after changing the range (new samples have been received) and the view will be updated too. The cross will be moved to the last drawn dot.

(x, y)-Plot Settings

A dialog box with settings appears automatically when you open the (x, y)-Plot Window for the first time. Closing the window will save the settings. The next time the window is opened this dialog will not appear again. Instead the saved settings will be loaded.

If you want to modify these settings or carry out a new calculation you can re-open the dialog box using the

---

menu item "**Start/ Settings/ Edit**" or by clicking "**Settings...**" in the context menu.

You can also press the **F5** key to *open* the "**Load Settings**" dialog box.

How to save and load settings see: Settings in Analysis Windows.

Global options Tab
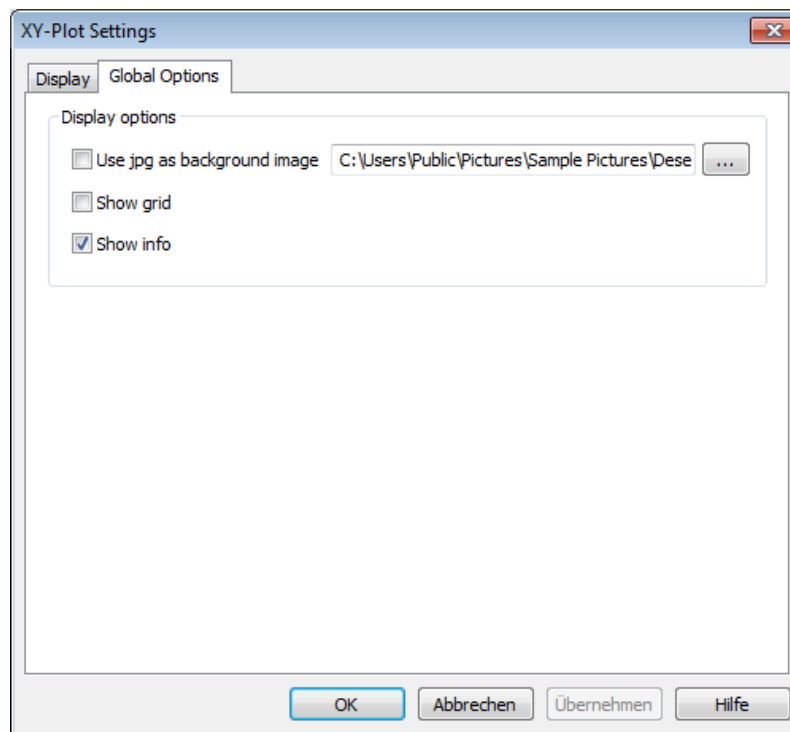
In the global options tab are the options that are regarding all (x, y)-Plots or the window itself.

Use jpg as background image	If desired, you can activate this option and select a jpg-file that you want to have displayed in the Background. Zooming in on the (x, y)-Plot also zooms in on the Picture.
Show grid	This option shows a grid in the background.
Show info	Shows the Offset and Gain of all the lines (regression lines and user lines) and the correlation of the regression lines.

Display Tab

This tab is used to show all the existing plots, to manage them as well as setting the data source and the

Condition, that is applies to the data from the data source.
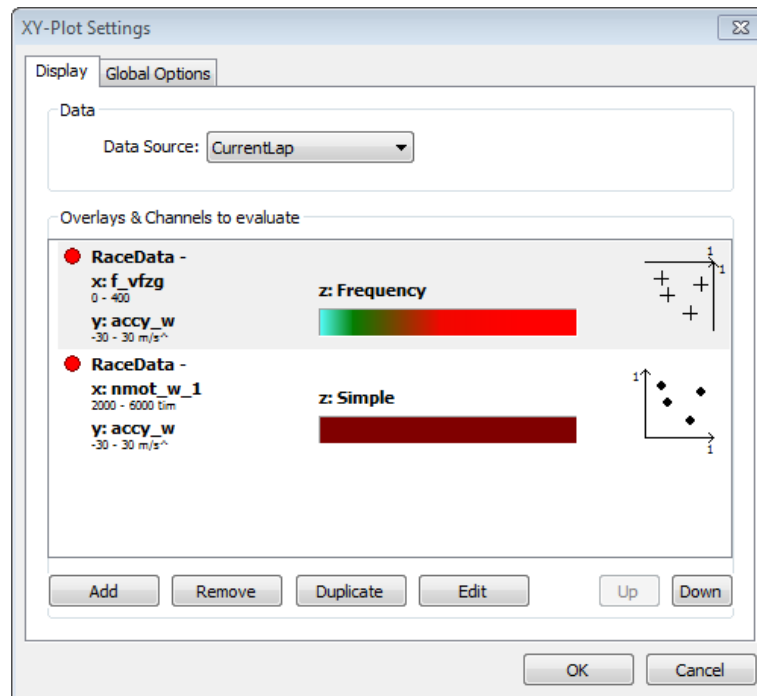
Data Source   Chose the data that is analyzed by the (x, y)-Plot. The combo box offers a wide range of selectable options.

**Overlays &   In this list, you see all the (x, y)-Plots that are currently defined. Every item shows the
Channels to   essential parameters selected for this particularly plot.
evaluate**
It shows the selected overlay, the channels for the x- and y-axis, the plot mode, the color/colorscale for this plot, the marker style and the selected axes.

You can manage these configurations with the buttons under tis list.

They allow you to rearrange the plots as well as removing, adding, editing and duplicating them.

Note:

If two or more plots have the same x- or y-axis the topmost plot defines the shown range for this axis.

Plot Edit Dialog

This dialog is shown when you mark an existing (x, y)-Plot and press Modify or by adding a new (x, y)-Plot to the list (Add-Button), or duplicating an existing one.
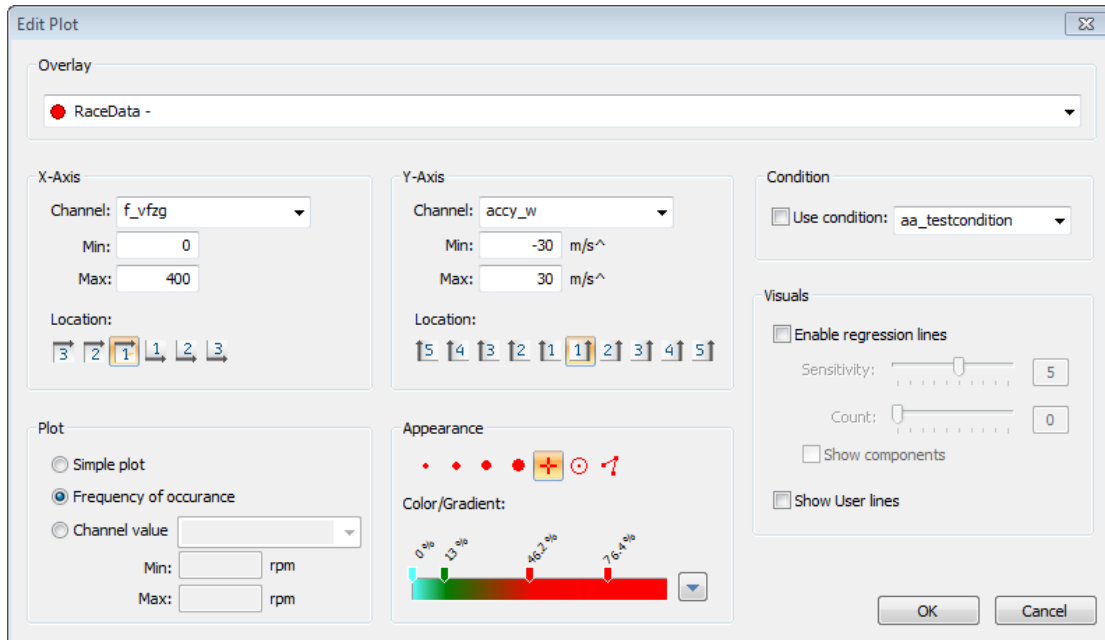
The difference between adding a new plot or duplicating an existing plot is that the fields are filled with default values when adding or with the values from the source plot when duplicating a plot.
The plot itself is only created after pressing OK.

Overlay	Choose the overlay from which the data is analyzed.
x-Axis	In this section, select the channel, the range and the axis-position for the x-axis.
y-Axis	In this section, select the channel, the range and the axis-position for the y-axis.
Plot	You can choose between 3 color modes.

	Simple plot	The simple plot is the basic mode. Every plot-sample has the color specified for this plot.
	Frequency	This mode colors the plot-samples according to the number of other plot-samples around them. This enables you to highlight the more frequented areas of the plot.
	Channel	With this option, you have to choose a channel and its range. The plot-sample are then colored according to the value of the selected channel they correlate with.

Appearance	Chose the appearance of the plot-samples within the plot-window. First select the form of each sample. You can choose between a various size of dots, a cross or circle. The last option connects the dots according to the order on the timeline.
	In addition to that, you need to select a color for the samples. Using the simple plot, just select a color that is used for all the samples.
	In frequency- or channel-mode, you have to select multiple colors for various values.
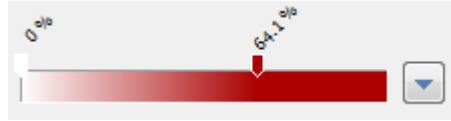
You can move the sliders by dragging them to another position. Remove sliders by pulling them away from the scale, and create new sliders by clicking inside the gradient.

You can also use predefined color scale settings by using the button on the right side of the color scale.

Use condition You can select a predefined condition or write a new one directly into the field. The data is filtered by the condition before it is being analyzed by the (x, y)-Plot.

Visuals In this section you can configure the regression lines, components and user lines.

Select the sensitivity and the maximal component-count to regulate the outcome.
The sensitivity controls how far two samples can be away without losing the connection. The higher the sensitivity, the more separate components are found, but also the jitter is reduced.

The maximal component-count limits the components and therefor the regression lines. The x best components are calculated.

The components can be optional displayed to see with which samples the regression lines are calculated.

The last option is to make the user lines visible or hide them.

For more information see: Components and regression lines

Components and regression lines

Components are calculated based on the (x, y)-Plot data to find and visualize connected data samples.

To analyze the data, the (x, y)-Plot is divided into a grid. First, the field containing the most samples is set as a starting field of a component. Starting from this field, the surrounding fields are analyzed. If a surrounding field contains more samples than a given threshold, the field is added to the component. Now the surrounding fields of the enlarged component are analyzed and maybe added. This step is repeated until there is no surrounding field with a high enough sample count to add to the component.

Now the next field with the highest sample count outside a previously found component is chosen to start a new component. A new component is being built from this. These steps are repeated until the last component is found.

The higher the sensitivity is set, the finer the grid will be. Also the sample count threshold rises with the sensitivity.

The count sets the maximum number of components. So if the maximum number of components is reached, no new component will be started.

After the components have been found, the regression lines are calculated for each component.

Working with the (x, y)-Plot Window

Basic

In the (x, y)-Plot window, the chosen (x, y)-Plots are shown as overlapping layers. Clicking on the window will show you a cross and the values of the selected dot can be seen in the surrounding axes.

You can zoom in an area of the window and see more details.

Zoom functions:

	Press right mouse button and *drag down or up*. The **marked area** will be all x-values and only the y-values between the y-start drag and y-end drag values.
	Press right mouse button and *drag left or right*. The **marked area** will be all y-values and only the x-values between the x-start drag and x-end drag values.
	Press **Shift-key** and then *drag a box* with pressed right mouse button. The **marked area** will be x-values and y-values inside of the dragged box.

Start/ Display/ Zoom in Context Menu/ Zoom in

   ⊕

Double Click left

Zoom in. The picture will be *zoomed in* by a factor 2 so the cross dot will stay unmoved.

Start/ Display/ Zoom out Context Menu/ Zoom out

   ⊖

Double Click right

Zoom out. The picture will be *zoomed out* by a factor 2 so the cross dot will stay unmoved.

Start/Display/ **Show All** **Context Menu/** **Show All** **A – Key**	All dots will be shown.

Additional elements

There are components, regression lines, user lines and a InfoBox with the information about the files and lines. They are additional to the base picture. Additional elements can be hidden and shown from the menu or the toolbar.

Start/ Display/ **ShowInfo** **Context Menu/** **ShowInfo** **F11**	The InfoBox contains information about files and user lines. Information for every file consists from files ranges, correlation of the range and file regression lines. Information about file lines can be shown or hidden by using +/- in front of the file name. Every line has offset, gain and correlation.
Start/ Display/ **ShowScale** **S**	Displays or hides the colorscale for the selected (x,y) plot in the (x,y) window. This is only available in frequency or channel mode.
Start/ Background/ **Use Image**	Select and use an image for the background of your (x,y) plot.
Start/ Visibility/ **Transparency** **H, 0-9**	Shows/Hides the selected (x,y) plot **(H)**. You can adjust the transparency of the (x,y) plot by clicking on the drop down button and change the slider position or by using hotkeys **(0-9)** to set the transparency **(0% - 90%)**.
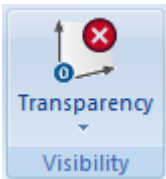
Start/ Display/ Components Context Menu/ Components **Ctrl+Space**	Components are continuous regions with approximately the same dense of dots.
Start/ Regression lines /Regression lines Context Menu/ Regression Line **Space**	Regression lines are calculated and they depend on calculated components. For every regression line there is a correlation.
Start/User Lines /User Lines Context Menu/ User Lines **Space**	User lines are free placeable lines that are inserted into a (x,y) plot by the user.
Start/ User Lines/ Insert Context Menu/ Insert line	Adding a new user line is done by pressing Insert-key or "Insert line" from the menu.
	Lines If the mouse cursor is near a line then the equation of the line will be shown in a small window. The information about the lines can be seen also in the InfoBox. You can select a line by clicking near to the line or by clicking in the InfoBox on some row with line information. If the line belongs to the component the component will be selected too.
Start/ User Lines/	You can delete the selected line and component by pressing Del-key or "Remove"
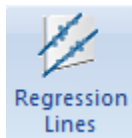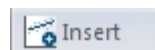
Remove (User line)
Context Menu/
Remove line

Remove

Del

from the menu.
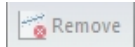
The inserted line is going through the (0, 0) and the cross point. A user line can be dragged using two small rectangles on the line.

Axes

- The axes of (x, y)-Plot have their own context menus. Using it you can:
- change the position of channel names or units
- hide or show the channel names or units
- hide or show the current values
- change the style of the numbers in axis: all numbers have the same style

---

Distribution

A Distributions Table displays frequency distributions of pairs of values (x, y) in (‰) for two channels (x- and y-axis) or else the average value for a third channel.

Distribution: vfzg_w vs nmot_w_1

	Overlay	Whole file
● RaceData 1		0.00 m - #9 5684.84
● RaceData -		0.00 m - #11 5684.7

vfzg_w [km/h]

nmot_ [rpm]	65.000	80.000	95.000	110.000	125.000	140.000	155.000	170.000	185.000	200.000	
1350.00											
1700.00											
2050.00		8									
2400.00		25	0		1			0			
2750.00		6	18	1	8	17		8	8		(
3100.00			14	23		47	12	1	12	21	0	0	2
3450.00			0	25	4	0	58	6	0	26	49	2	(
3800.00				6	18	0	2	48	10	1	10	51	1
4150.00				1	11	7	0	2	35	3		0	4
4500.00				0	1	8	0		1	26	1	
4850.00						3	4			0	1	
5200.00						1				0		

In addition the Distribution window can show the results as 3D surface where z is the cell value from the Distributions Table. The surface can be rotated, zoomed in and out.
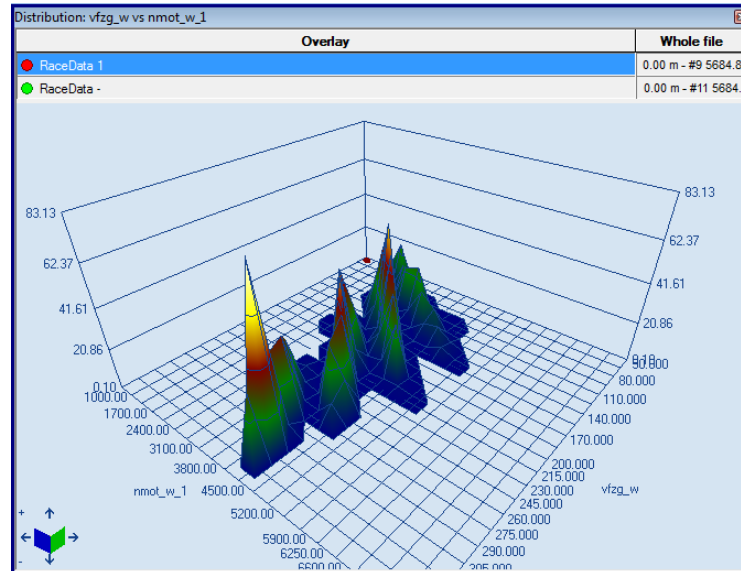
The surface can be divided into slices with user defined planes.
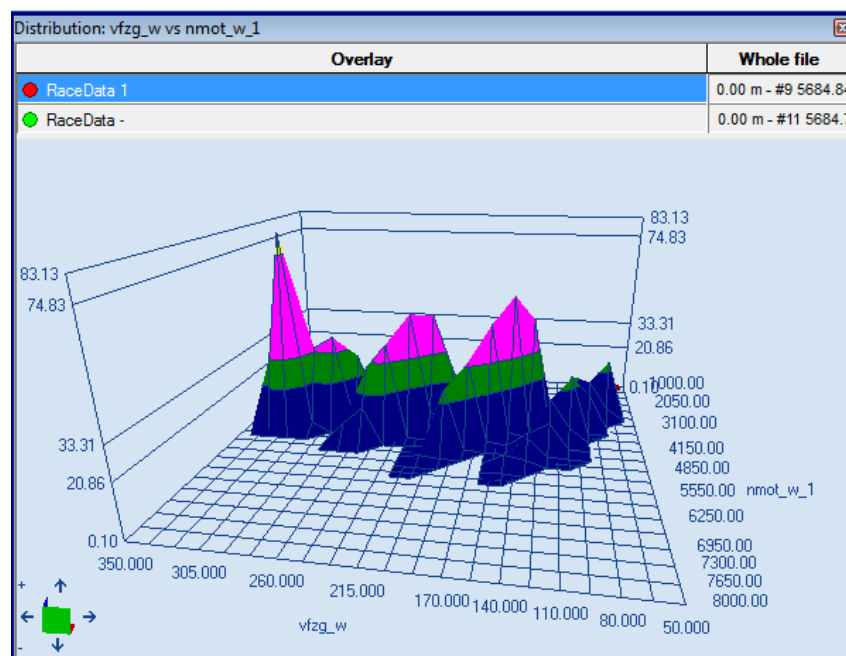
Open a Distribution Window

Use "**Start/ Worksheet Distribution**" to open a Distribution Window.
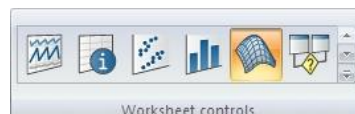
Middle Mouse Click (Left Button + Right Button)
Clicking your middle mouse button (or left + right mouse buttons) *displays* a **local menu** with options.

Calculations
Calculations are started after changes in settings or after "Local Menu/ Update View" from the menu.

Because the calculations can take much time especially if the whole file should be calculated, they are made inside of a working thread. You will see a progress bar then. Instead of waiting you can work with other windows.

For online files, calculations are started automatically after changing the range (new samples has been received) and the view will be updated too.

Distribution Settings

A dialog box with settings appears automatically when you open the Distribution Window for the first time. Closing the window will save its settings. The next time the window is opened the dialog will not appear again, instead the saved settings will be loaded.

If you want to modify settings or carry out a new calculation, you can re-open the dialog box using the menu item "**Start/ Settings/ Edit**" or by using the context menu with pressing middle mouse button or left + right mouse buttons.

You can press the **F5** key to *open* the "**Load Settings**" dialog box.

How to save and load settings see: Settings in Analysis Windows.

Overlays Tab
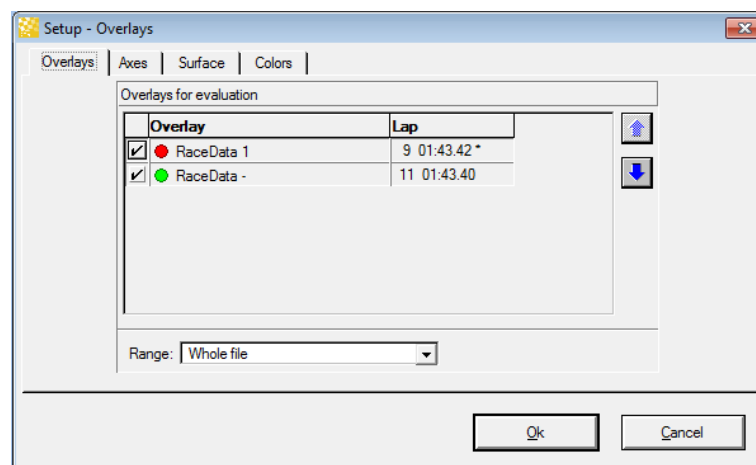
Axes Tab
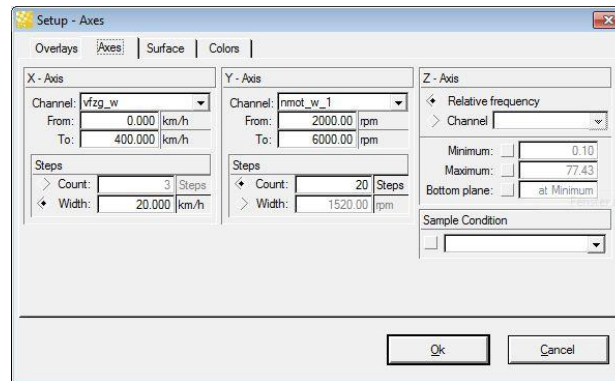
This tab is used to select channels for evaluation and their range for visualization and set/unset data filter. There are z axis options and relative frequency parameters.

Steps	Steps defines the division of the axis range.
	There are two possibilities to specify the division:
	Count: Specify the number of steps into which you want to sub-divide the value range of the channel.
	Width: Specify the width of the individual bars. The stepcount is calculated according to the width.

Note:

In case the range can't be divided without a remainder. The range is extended to a range that can be divided by the width without a remainder

Z-axis	For the Z-axis you can chose between two alternatives
	Channel:
	For every (x,y) – pair, the z-value of the distribution is the average of the selected channels values fitting the (x,y) – pair.
	Relative Frequency:
	For every of values (x,y), the z-value of the distribution is the portion of values (in ‰) fitting the (x,y) - pair in relation to all values in the selected range (see [Files Tab](#)).
Boundaries	If check box **Auto Max** is checked then the maximal **z** value will be equal to the maximal value from Distribution Table else the value is written into the edit box right to it. It will be taken as maximal **z** value.
	If check box **Auto Min** is checked then the minimal **z** value will be equal to the minimal value from Distribution Table else the value is written into the edit box right to it. It will be taken as minimal **z** value.
Sample Condition	If desired, you can activate this option and select a condition from the list. Only the data samples that fulfill the selected condition will be used to calculate the distribution.

Note:

Changing relative frequency or channel will change calculated Min and Max values so it is good to check the Surface Tab-Slices again to see if the limits are the expected ones.

Surface Tab

This tab is used to define the slices and the appearance of the surface.
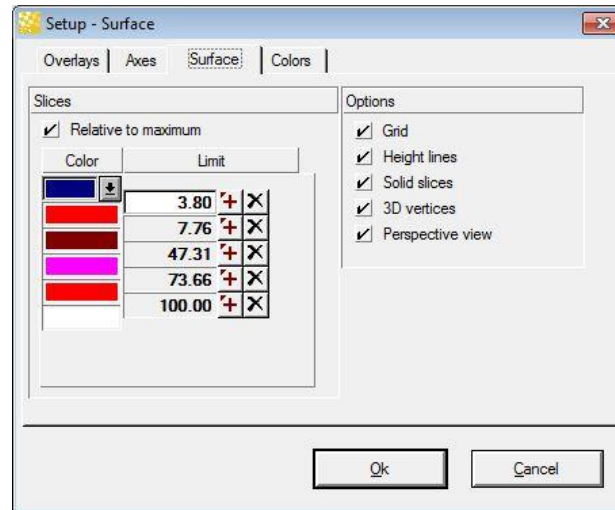
Slices   A slice is a row with color and limit and it is the part of a surface between planes z = PrevLimit and z = Limit. PrevLimit is the Limit of previous row or minimum. If **relative to maximum** is checked then limits are displayed in percent and the last row has the limit 100 and cannot be changed. 100% corresponds to the defined Z-axis maximum and 0% corresponds to the Z-axis minimum. If **relative to maximum** is not checked then the limits are supposed to be absolute values.

You can insert a new slice with the [+] button. A new row will appear above the pressed button with limit between the two neighbor limits.

You can delete a row with making it current and then pressing [X] button.

Options   You can change the appearance of the surface (you can do this by using the menu or tool bar too):

- **3D-Grid**: the surface is made from 3D quadrangles and you can see their borders switching this option on.
- **Height lines**: switching this option on you will see the intersection lines between the surface and planes of slices.
- **Solid slices**: switching this option on you will see every slice in only one color.
- **3D-vertices**: switching this option on you will see the dots that correspond to the Distribution Table cells.
- **Perspective**: switches to perspective 3d view.

Notes:

Do not forget to check limits after changes in [Axes Tab](#) **relative frequency** and **relative to maximum** is not checked.

Colors Tab

This tab is used to choose the colors of axes-grid, background, grid-points, height lines, selected point, surface-grid and text.
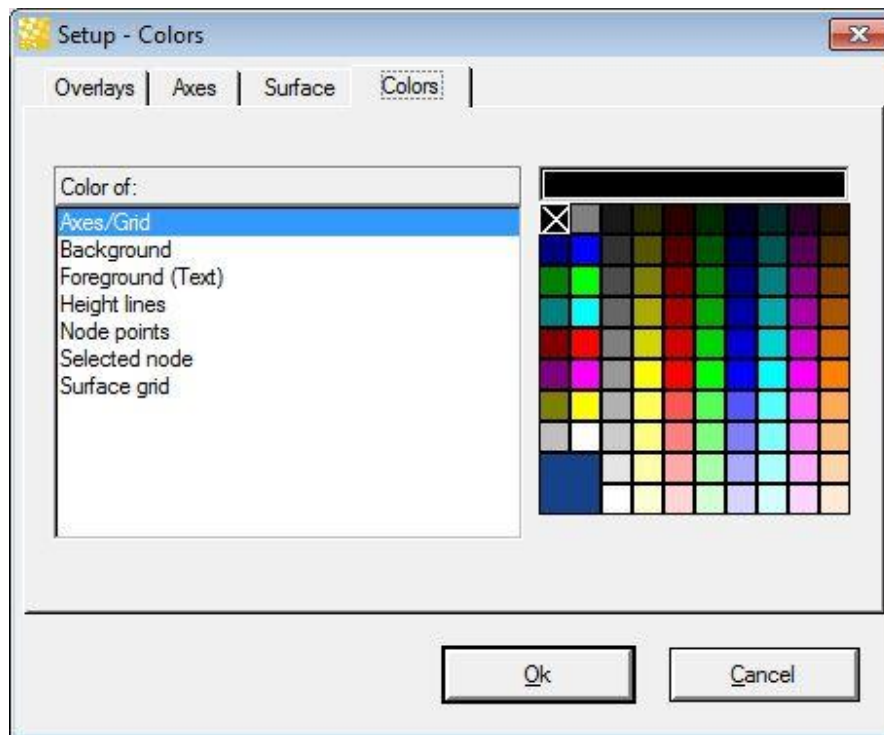
Working with the Distribution Window

The results are shown in Distribution Table in 2D view or as a surface in 3D view where Z is the cell value from the Distributions Table.

Start/ Display/ 2D View

Ctrl+2

2D view

In Distribution Table in 2D view every row has at the right side the sum of all cells in the according row and every column has at bottom the sum of all cells in the according column.

Start/ Display/ 3D View

Ctrl+3

3D view

In 3D view you can *rotate* the **3D surface** using the keys: Left, Up, Right, Down

Moving the mouse wheel with/without SHIFT key will rotate the 3D surface horizontally/vertically. You can rotate the 3D surface with just dragging the

mouse cursor over the window in different directions.

Start/ Display/ 3D Navigator

3D view contains 3D navigator. You can use it for rotation too by pressing on arrows. Using the menu 3D navigator can be shown, hidden or shown only when the window is active.

By pressing on the button instead of selecting an item from the list, you change the perspective to a 30 degree perspective.

Start/ Display/ Birds view

Pressing this buttons changes the perspective of the 3D-View to a top-down perspective.

When changing to the Birds view, the perspective view is disabled automatically.

Zoom functions

Use menu, toolbar, "+" on 3D navigator or double click with left mouse button to zoom in.

Ctrl+Up

Use menu, toolbar, "-" on 3D navigator or double click with right mouse button to zoom out.

Ctrl+Down

Distribution/ Show All

Surface will be moved to its default position by Show All.

A

Additional functions

Context Menu/ Update View

At any time you can recreate 3D surface using Update View.

Start/ Display/3D vertices

You can see the table cells as 3D-vertices.

Ctrl+V

Start/ Display/ Solid slices
Surface can be shown in slices (different solid colors between the height lines).

Ctrl+S

Start/ Display/ Grid
The 3D-grid on the surface can be shown or hidden.

Ctrl + G

Start/ Display/ Grid
Height lines are the intersection between the user-defined planes and the surface and you can show or hide them.

Ctrl + G

Start/ Display/ Perspective
If you desire you can switch to perspective 3D view.

Ctrl + P

Drag & Drop

You can change settings of the Distribution by drag & drop an object from the channel toolbar. You can drag a channel over the x or y axis (in both 2D and 3D axes) and if you drop it then the corresponded channel will be replaced.

Copy to clipboard

Select **Context Menu/ Copy** to copy the Distribution to clipboard. Plain text and rich text formats are supported.

Press **Ctrl + C** to copy the Distribution to clipboard. Plain text and rich text formats are supported.

Histogram

A histogram shows the relative frequency distributions of channel values in percentages. Because histograms can use a channel from a number of different files, they enable you to make direct

comparisons.
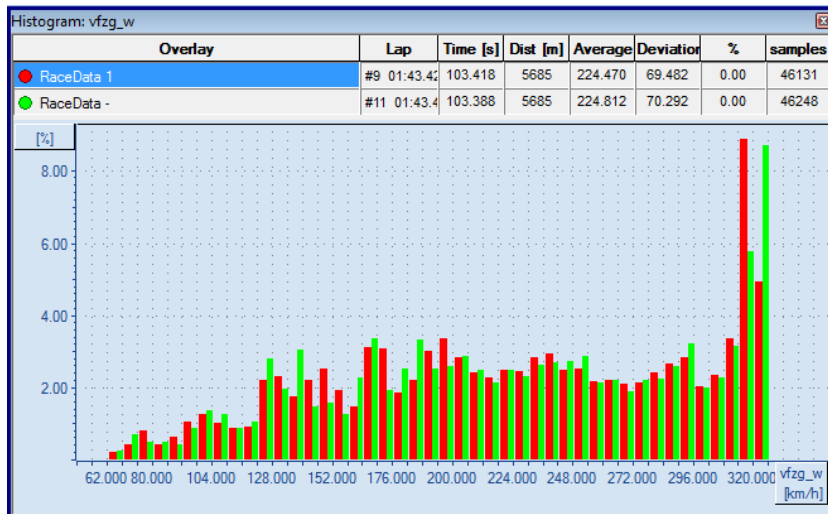
In addition the Histogram window can show the results as lines.
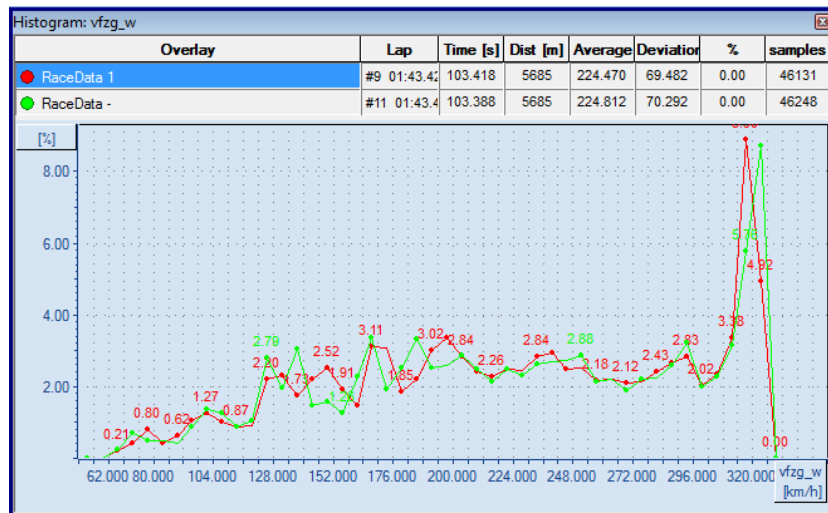
Histogram window can also show the results as 3D bars. The 3D Bars can be rotated, zoomed in and zoomed out. 2D bars and 3D bars can be selected and you can see the percentage of the selected bars.
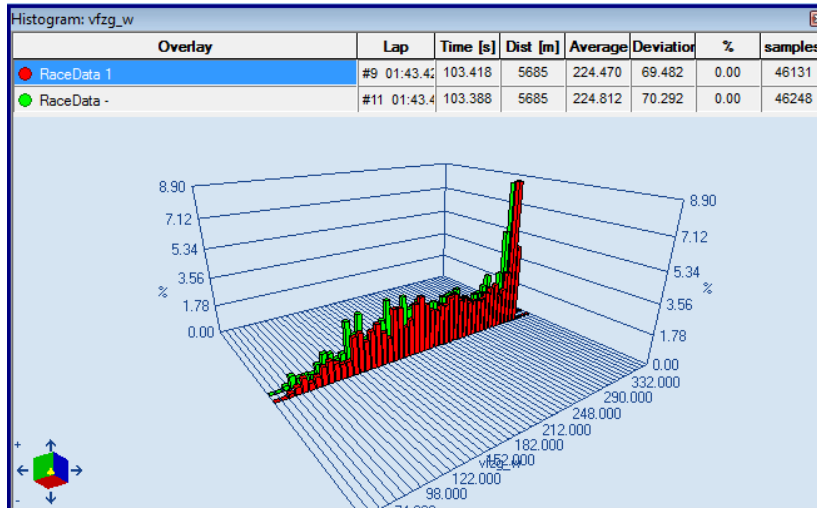
Open a Histogram Window

To open a Histogram Window select "**Start/ Windows/ Worksheet/ Histogram**" and drag it into your workspace.
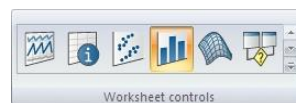
Middle Mouse Click (Left Button + Right Button)

Clicking your middle mouse button (or left + right mouse button) displays a local menu with options.

Calculations

Calculations are started after changes in settings were made.

> Because the calculations can take much time especially if the range is above whole file, they are made inside of a working thread and you will see a progress bar in this case. Instead of waiting you can work with some other window.

> For online files, calculations are started automatically after change in the range (new samples have been received) and the view will be updated.

Histogram Settings

A dialog box with settings appears automatically when you open the Histogram Window for the first time. Closing the window will save the settings and the next time the window is opened the dialog will not appear instead the saved settings will be loaded.

If you want to modify settings or carry out a new calculation, you can re-open the dialog box using the menu item "**Start/ Settings/ Edit**" or by using the context menu by pressing middle mouse button or left + right mouse buttons.

You can press the **F5** key to *open* the "**Load Settings**" dialog box.

How to save and load settings see: Settings in Analysis.

Overlays Tab

This Tab is used to Select the visible overlays as well as the range in which the histogram is calculated.
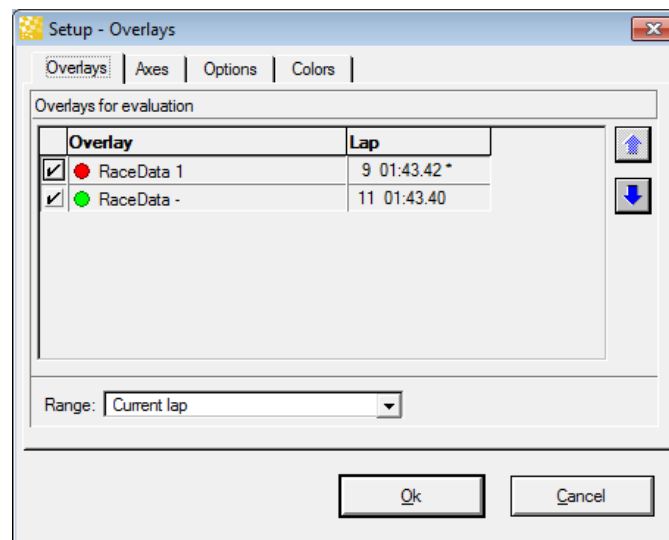
Axes Tab

This tab is used to select channel/channels for evaluation and set / unset data file
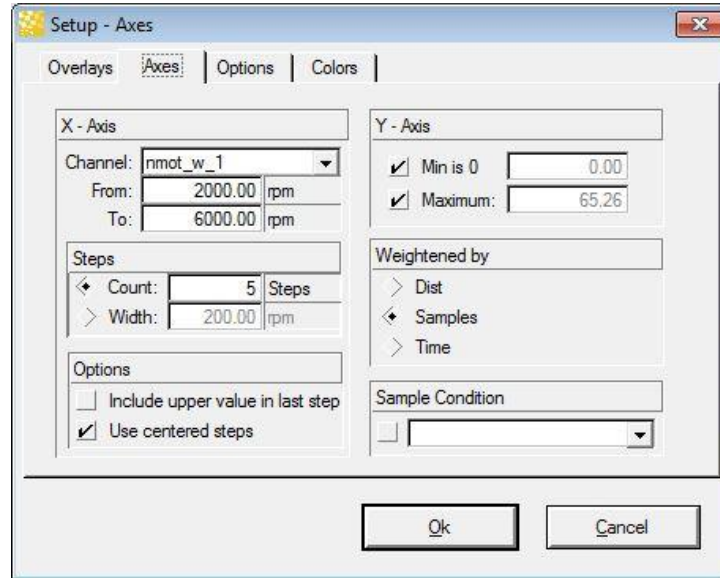
Channel/Channels Specify the channel you want to evaluate. You can select several channels which has similar meaning. If you open the list box of the popup control of a channel and hold **shift** or **ctrl key** then you can select several channels.
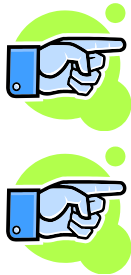
If you select several channels then they should have approximately the same range (in histogram this is x-range).

If the selected channels have the names with the same prefix then the histogram will show the common prefix of channels as a name for the x-axis.

Steps Steps defines the division of the histogram range.

There are two possibilities to specify the division:

Count: Specify the number of steps into which you want to sub-divide the value range of the channel.

Width: Specify the width of the individual bars. The Stepcount is calculated according to the width.

Note:

In case the range can't be divided without a remainder. The range is extended to a range that can be divided by the width without a remainder

Include upper value in last step The histogram usually calculates the steps with
lower boundary <= values < upper boundary

This option includes the upper boundary in the last step of the histogram.

Use centered steps Use this option when you don't want specific boundaries, but specific values to analyze.

Use this option for example for a gear analysis or if you want to analyze the revolution speed around defined values.

Note:

If you select centered steps, the value-range is extended automatically so that the minimum x value is the center of the first step. Also the last step is extended by a half step (if you are using steps) or until the range can be divided by the width without a remainder (if you are using width).

Weighted by   Choose how the samples should be weighted.

- **Samples**: values count once
- **Time**: weighting according to the time intervals between samples
- **Distance**: weighting according to the distance covered between samples

Sample Condition   If desired, you can activate this option and select a condition from the list. Only the data samples that fulfill the selected condition will be used to calculate the histogram.

Axis Y   Specify the visible range for the y axis. The default value for minimum y is 0. The default (automatically calculated) value for maximum y is the maximum value of a bar.

Option Tab

This tab is used to select or not the perspective geometry. This option can be changed from the menu and tool bar.

Colors Tab

This tab is used to choose the colors of axes-grid, background and text.
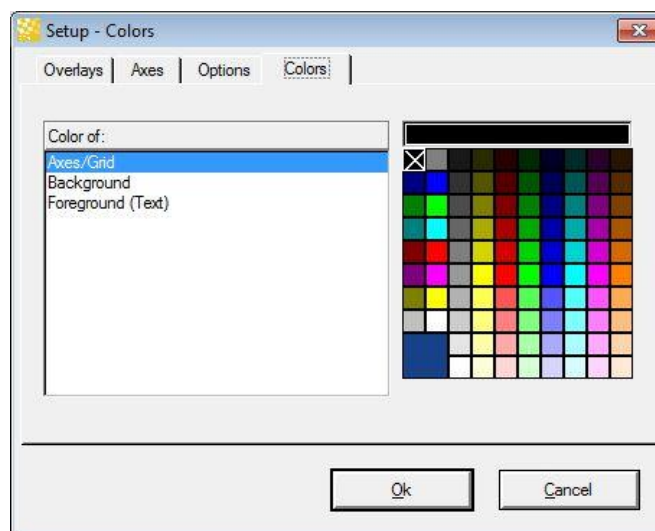
Working with the Histogram Window

Results can be shown in different views: 2d view as bars, 2d view as lines and 3d view. It is possible to zoom inside of all views and it is possible to select bars or lines.

2D view as bars

Start/ Display/2D view as bars

Ctrl+2

The feature of 2D view as bars is the possibility of multi-selection of bars and showing their sum for different files (% column in table above the bars).

How to select/unselect bars see Selecting/Unselecting Bars.

You can zoom in to see more details from the results. See also: 2D Zooming.

2D view as lines

Start/ Display/2D view as lines

Ctrl+L

Clicking on 2D view as lines will show you a cross in the same x-position as the mouse position, but the y-position will depend on the value in this x-position.

You can zoom in to see more details from the results. See also: 2D Zooming.

In 2D lines view you have one selected line. In the beginning it is the line of the first file. You can select a new current line by clicking near it.

If you drag with the mouse on the screen (or just click) then a cross appear which follows the mouse with its vertical line and follows the current line y-value with its horizontal line.

If you move the mouse near a line then a small yellow window with the value of the line in this point appears.

3D view

Start/ Display/3D view

Ctrl+3

In 3D view you can rotate the 3D bars using the keys: Left, Up, Right, Down.

Moving the mouse wheel with/without SHIFT key will rotate the 3D bars horizontally/vertically. You can rotate the 3D bars with just dragging the mouse cursor over the window in different directions.

Start/ Display/ 3D Navigator

3D view contains 3D navigator. You can use it for rotation too by pressing on arrows. Using the menu 3D navigator can be shown, hidden or shown only when the window is active.

By pressing on the button instead of selecting an item from the list, you

change the perspective to a 30 degree perspective.

Start/ Display/ Birds view

Pressing this buttons changes the perspective of the 3D-View to a top-down perspective.

When changing to the Birds view, the perspective view is disabled automatically.

Start/ Display/ Perspective

If you desire you can switch to perspective 3D view.

Ctrl + P

You can also zoom the view with Zoom functions.

2D Zooming

You can zoom an area of the window to see more details.

- Press right mouse button and drag down or up. The marked area will be all x-values and only the y-values between the y-start drag and y-end drag values.
- Press right mouse button and drag left or right. The marked area will be all y-values and only the x-values between the x-start drag and x-end drag values.
- Press SHIFT-key and then drag a box with pressed right mouse button. The marked area will be x-values and y-values inside of the dragged box.

You can also zoom the view with Zoom functions.

Zoom functions

Ctrl + Up

Zoom in. In 3D view you can use also "+" on 3D navigator. In both 2D and 3D views you can double click with left mouse button.

Ctrl + Down

Zoom out. In 3D view you can use also "-" on 3D navigator. In both 2D and 3D views you can double click with right mouse button.

Histogram/ Show All

Show All. In 3d view the scene will be moved to the default position. In 2D view the whole bars will be shown.

A

Multi-channel mode

The main difference between one-channel mode and multichannel mode is the colors of the bars. In one-channel mode the bar has the color of its file. To distinguish the different channels from one file you can change the color of bars from one channel.

You can change the color directly by opening the context-menu over a bar for 2d bars and 3d bars mode. For the line mode you can change the color of a line by opening the context-menu near a line. You also can change a color from a color legend in right up corner of the 2d views (as 2d bars and 2d lines).

Histogram: accx_waccy_w							
Overlay	**Lap**	**Time [s]**	**Dist [m]**	**Average**	**Deviation**	**%**	**samples**
accx_w - RaceData 1	#9 01:43.42	103.418	5685	1.255	5.101	0.00	44066
accy_w - RaceData 1	#9 01:43.42	103.418	5685	0.995	9.400	0.00	41041
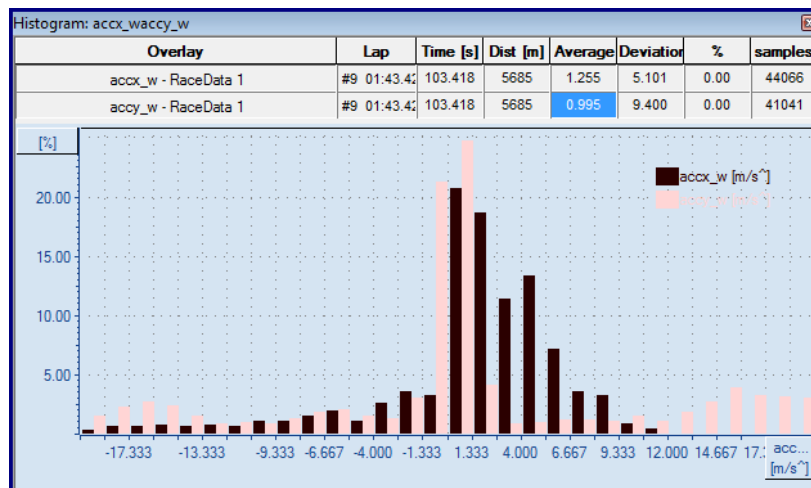
Selecting/Unselecting Bars

Press on a bar and you will select it. If you already have selected bars then pressing on a bar will unselect others. Hold CTRL-key if you do not want to unselect others selected bars when selecting new one. If you want to select a range:

1. select a bar
2. press and hold **Shift- key** and press on another bar
3. bars between will be selected

The last selected bar is remembered and if you hold SHIFT-key and press on another bar then you will select the range between them.

Click on empty space to unselect all bars.

The selected bars are not cleared if you switch between views.

Drag & Drop

You can change settings of the Distribution by dragging and dropping an object from the file/channel/color toolbar. You can drag a channel over the x or y axis (in both 2D and 3D axes) and if you drop it then the corresponded channel will be replaced. If you drag a file you can drop it on the Distribution then the file will be added to these files for evaluations (just like added it manually from the Files Tab), but if you drop it over a row in the files table then the file in the row will be replaced with the dragged one.

FFT

Use this function if you want to approximate a channel's signal curve using a Fourier Series. The resulting coefficient values will be displayed in a diagram. This diagram can show the results as lines, 2D-bars and 3D-bars.

The line-view shows all selected channels as lines in the diagram.
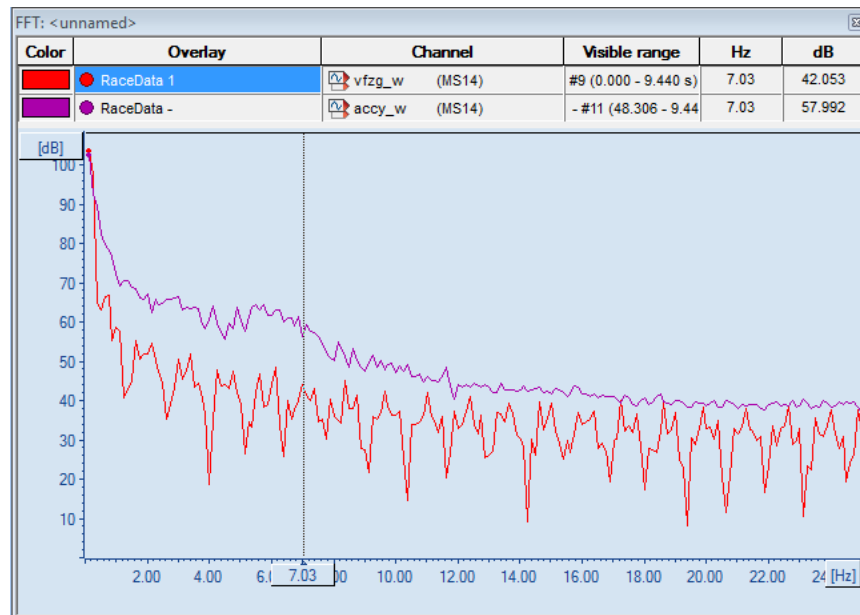
The 2D-bar-view shows the channels values for every step. For every step, the bars of the selected channels are shown next to each other.
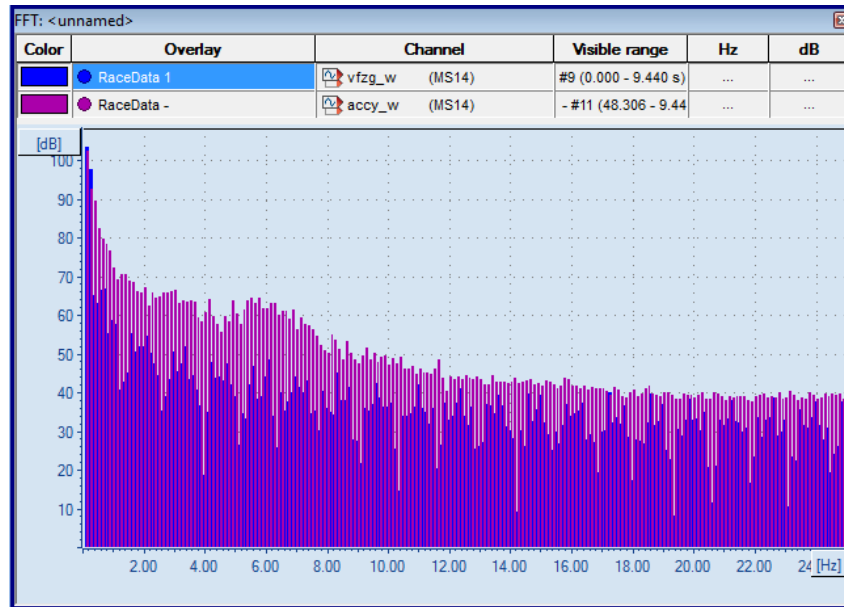
The 3D-bar-view shows all selected channels as different layers in the 3D-bar-diagram.
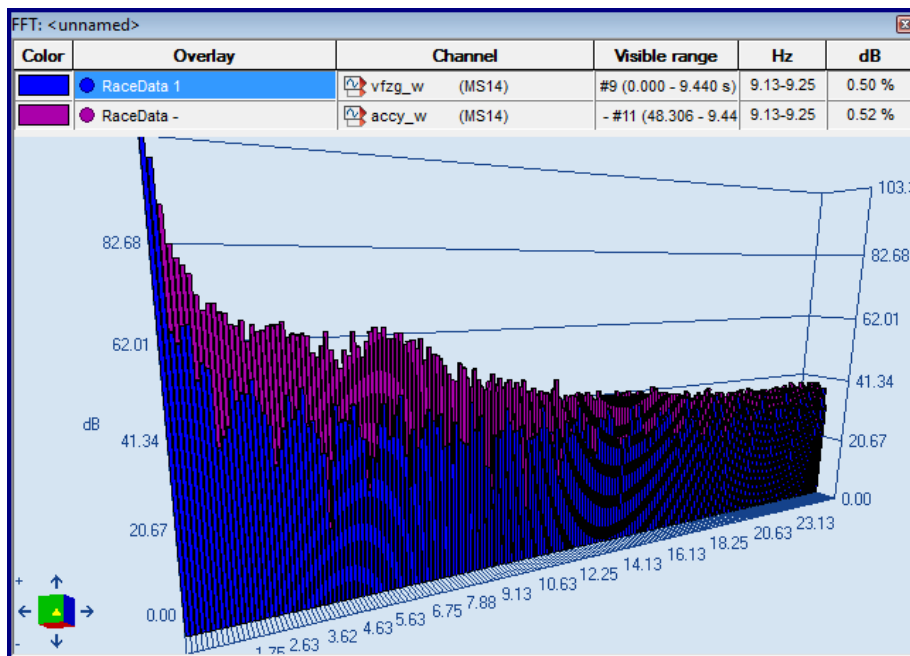
The 3D-view can be rotated, zoomed in and out.

Open the "Fast Fourier Transform" Window

Drag the menu item "**Start/ Windows/ Worksheet controls/ Fourier transformation**" into your workspace to open the "Fast Fourier Transform" Window.

Middle Mouse Click (Left Button + Right Button)

Clicking your middle mouse button (or left + right mouse buttons) displays a local menu with options.

For online files, calculations are started automatically after change in range (new samples have been received) and the view will be updated.

FFT Settings

A dialog box with settings appears automatically when you open the FFT Window for the first time. Closing the window will save its settings and the next time the window is opened the dialog will not appear instead the saved settings will be loaded.

If you want to modify settings or carry out a new calculation, you can re-open the dialog box using the menu item "**Start/ Settings/ Edit**" or by using the context menu with pressing middle mouse button or left + right mouse buttons.

You can press the **F5** key to *open* the "**Load Settings**" dialog box.

How to save and load settings see: Settings in Analysis.

Input Tab

This Tab is used to select the input variables for the Fast Fourier Transformation.
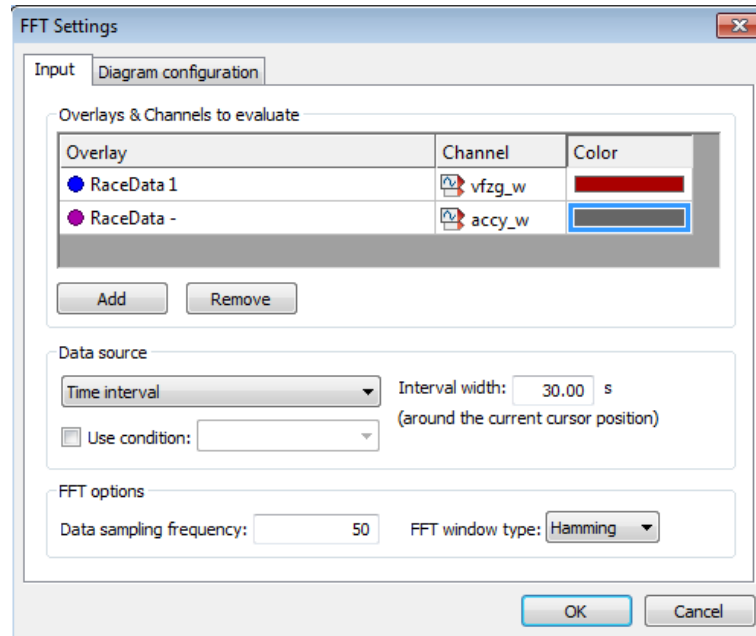
Overlays & Channels to evaluate

Use the "Add" and "Remove" Button to manage the channels that are calculated. In the table, you can alter the overlay, the channel and the color in which the channel is displayed.

Data source

You can configure the fft input parameters in this section. Chose an input range between a time interval, the current lap, the whole file, a segment, and the marked, tagged and visible range. You can also add a condition to filter the values used in the fft.

FFT options

In this section, you can select the data sampling frequency and the window type.

Data sampling frequency: Frequency which is used for calculation of FFT.

FFT window type: Select the approximation technique you want the Fourier Transform to use:

- Hamming Window
- Hanning Window
- Rectangular Window
- Blackman Window

Note:

The sampling frequency should be at least twice the maximum frequency to provide reasonable results.

Diagram configuration Tab

This tab is used to set different parameters for the appearance of the diagram.
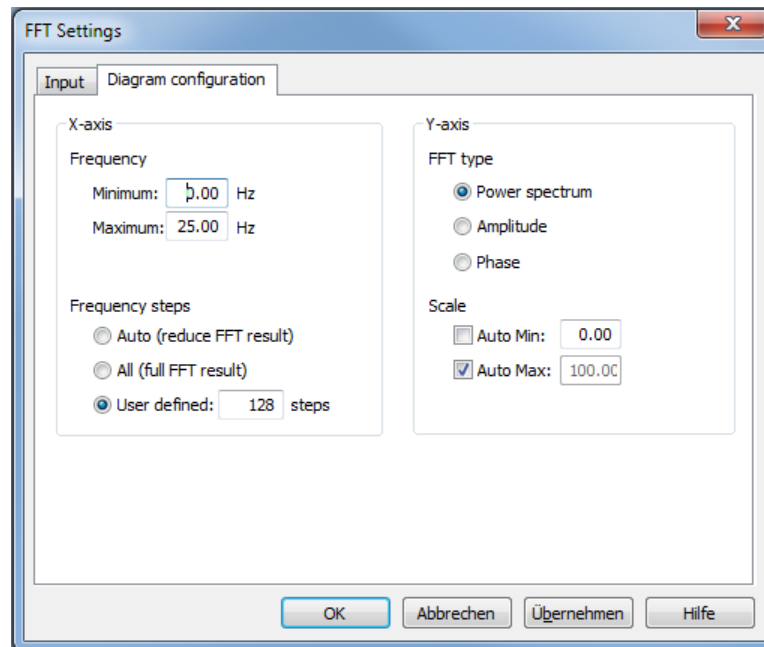
X-axis   **Frequency**: Chose the shown frequency range.

Frequency steps: Select a number of steps shown in the FFT window or let it be calculated

- Auto (reduce FFT result): The step-count is calculated by Darab
- All (full FFT result): Use this option show all calculated FFT results
- User defined: Chose the number of steps yourself

Data Window: Select the approximation technique you want the Fourier Transform to use:

- Hamming Window
- Hanning Window
- Rectangular Window
- Blackman Window

Note:

The maximum frequency should be at most be half the sampling frequency to provide reasonable results.

Y-axis   **FFT Type**: Select the values / coefficients you want to display:

- Power Spectrum
- Phase
- Amplitude

Scale: Set the desired **Min** and **Max** values or set them to **Auto Max/Min**. With that option set, the **Min** or **Max** will be calculated.

Working with the FFT Window

Start/ Display / 2D view as bars

Ctrl + B

2D view as bars

The feature of 2D view as bars has the possibility of multi-selection of bars and showing their sum for different files (% column in table above the bars).

Press on a bar and you will select it. If you already have selected bars then pressing on the bar will unselect others. Hold CTRL-key if you do not want to unselect others selected bars when selecting new one. If you want to select a range:

1. select a bar
2. press and hold **Shift- key** and press on another bar
3. bars between will be selected

Click on empty space to unselect all bars.

You can zoom in this window to see more details from the results (see FFT 2D Zooming).

If you have more than one channel shown in the window, the bars are alternated.

Start/ Display/2D view as lines

Ctrl+2

2D view as lines

Clicking on 2D view as lines will show you a cross in the same x-position as the mouse position, but the y-position will depend on the value in this x-position.

You can zoom this window to see more details from the results (see FFT 2D Zooming).

If you drag with the mouse on the screen (or just click) then a cross appear which follows the mouse with its vertical line and follows the line y-value with its horizontal line.

If you move the mouse near a line then a small yellow window with the value of the line at this point will appear.

Start/ Display/3D view

Ctrl+3

3D view as bars

In 3D view you can rotate the bars using the keys: Left, Up, Right, Down.

Moving the mouse wheel with/without **Shift- key** will rotate the 3D surface horizontally/vertically. You can rotate the 3D surface with just dragging the mouse cursor over the window in different directions.

Start/ Display/ 3D Navigator	3D view contains 3D navigator. You can use it for rotation too by pressing on arrows. Using the menu 3D navigator can be shown, hidden or shown only when the window is active.

By pressing on the button instead of selecting an item from the list, you change the perspective to a 30 degree perspective. |
| **Start/ Display/ Birds view** | Pressing this buttons changes the perspective of the 3D-View to a top-down perspective. |
| | When changing to the Birds view, the perspective view is disabled automatically. |
| **Start/ Display / Perspective**

Ctrl + P | If you desire you can switch to perspective 3D view. |

FFT 2D Zooming

You can zoom an area of the window and see more details. Zoom functions:

- Press right mouse button and drag down or up. The marked area will be all x-values and only the y-values between the y-start drag and y-end drag values.
- Press right mouse button and drag left or right. The marked area will be all y-values and only the x-values between the x-start drag and x-end drag values.
- Press **Shift- key** and then drag a box with pressed right mouse button. The marked area will be x-values and y-values inside of the dragged box.
- Double click with the right mouse button to zoom out.
- Double click with the left mouse button to zoom in.

FFT 3D Zooming

Ctrl + Up | Zoom in. You can use also "+" on 3D navigator. You can also double click with left mouse button. |
|

Ctrl + Down | Zoom out. You can use also "-" on 3D navigator. You can also double click with right mouse button. |

Flowchart

A Flowchart is a powerful tool with which you can filter and analyse data. For example you can make left curves as red segments and right curves as blue segments on a racetrack like below.
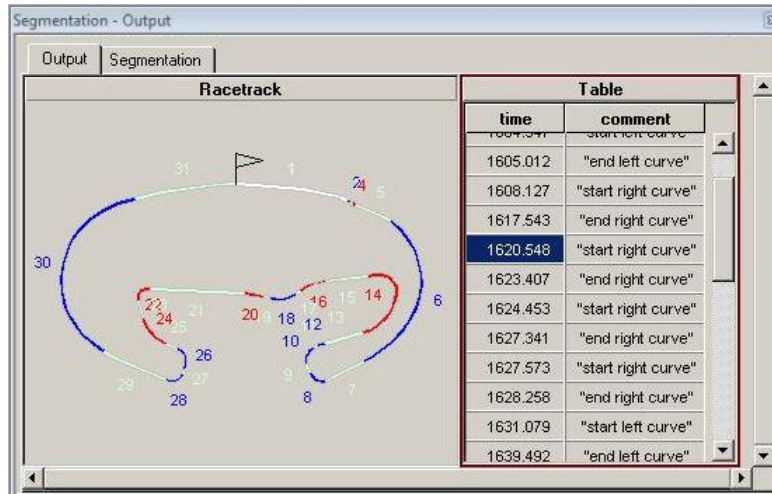
In flowcharts you have flow-objects like compare, formula, range, print value in table or value, insert/delete segment in a racetrack and others. The flow-objects have parameters: constants, user defined variables or channels. In the picture below the color is a user defined variable which holds the current segment color, f_ay1 is a channel.
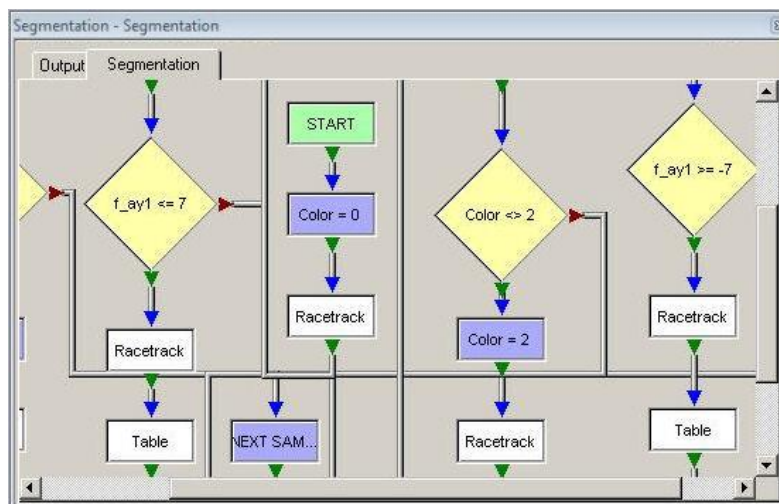
Open the Flowchart Window

To open a Flowchart Window select "**Windows/ Worksheet controls/ Flowchart**" and drag it into your Workspace.

Flowchart Settings

A dialog box with settings appears automatically when you open the Flowchart Window for the first time. Closing the window will save the settings. The next time the window is opened the dialog will not appear instead the saved settings will be loaded.

If you want to modify settings or carry out a new calculation, you can re-open the dialog box using the menu item "**Start/ Settings/ Edit**" or by using the context menu with pressing middle mouse button or left + right mouse buttons.

You can press the **F5** key to *open* the "**Load Settings**" dialog box.

Flowchart has flow-objects and most of them have parameters and Setup dialogs. If there is an active flow-object (it has a red frame) then calling "Setting" will open its Setup dialog. Click on an empty space on the flow page and then call "Settings".

How to save and load settings see: Settings in Analysis.

Setup Dialog Box

This dialog box is similar to Common Overlays Settings, but because only one overlay is used instead of a table with overlays there is a list box to choose only one overlay. There is also a additional edit box to change the name of current page of the flow register.
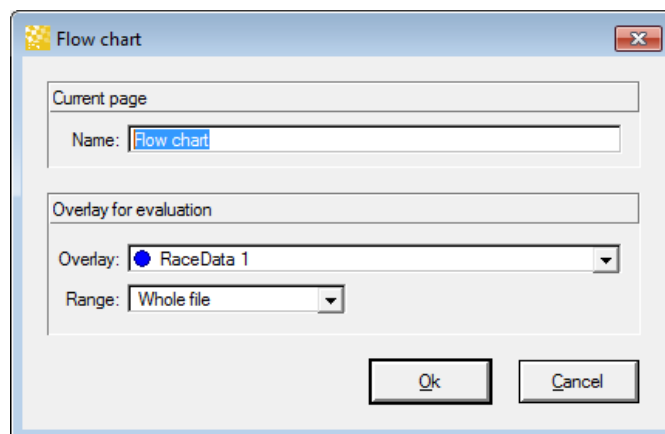
Flowchart Pages

There is one output page where the results of calculation are shown and one or more design pages. Design page can represent a flowchart or a part of a flowchart called container. During editing you can:

Start/ Pages/ New flow
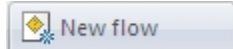
- add a new Design Page.

Start/ Pages/ Delete flow
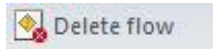
- delete current Design Page.

Start/ Pages/ Close container
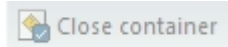
- If the Design Page represent a container then you cannot delete it, but you can close it. Closing means the objects from Design Page get invisible.

Elements of Output Page

This page contains analyzed or filtered results, calculated during flows run. The result can be a simple value, multiple values organized as a table or segments in a racetrack.

Output Table

Insert/Table

During calculations you can print different values in cells of the Output table.

Context Menu/ Settings F5

If the object is active (with red frame), you can open and modify its settings. You can open the "Output Table" dialog box using the context menu by pressing right mouse button.
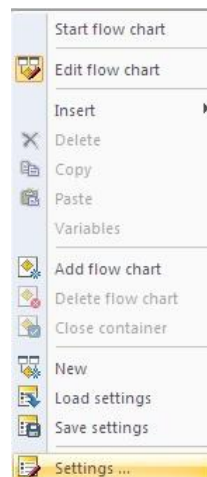
A column in the Output Table is a row in the Column Definition Table in the setup.
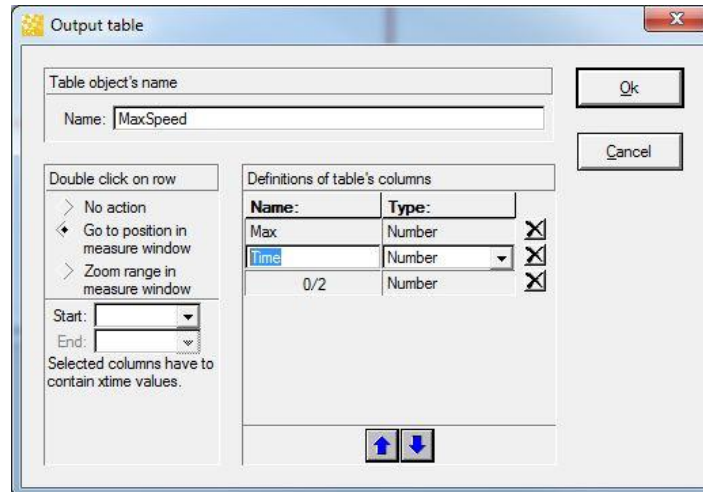
A row in the Column Definition Table contains the Name and the Type of the column. If the column is a number then you also can choose the length and the decimals of the number. You can change the order of columns with Up / Down arrows.

A new row is added if you make the last empty row current and enter a name. A new empty row will be added after insertion.

If you want to delete a row then make the row current and then press the delete button.

You can choose two columns from defined ones for zooming a range in bounded Oscilloscope on double click defined by two time values (Start and End) or you can choose to move the position of the bounded Oscilloscope to a time value (Start) on double click on a row.

For example: in the picture above if you double click on the value of time column then Oscilloscope position will be moved to this time and there speed will be equal to Max column value.

You can sort the results of the table by a click on the column of the table. By default after calculations the results are sorted by the column chosen in the field Start (In our example it is Time column). If you click on other column results will be resorted and the new sort column will be remembered and after next calculations will be used to sort the results. If there are several results with the same value for current sort column then they will be sorted by the values of column in Start field.

Into the table cannot be inserted more than 10000 rows. If there are already 10000 rows, the new rows will be ignored.

Output Value

Insert/Value

During calculations you can print a value in Output Value, and after the calculation you can analyze it.

MAX SPEED

322.833
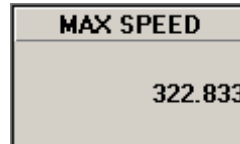
Context Menu/
Settings
F5 or **Double Click**

If, the object is active (with red frame), you can open and modify its settings. You can open the dialog box using the context menu with pressing right mouse button.

Output value

Value object's name

Name: MaxSpeed

If Double-Clicked

☐ Value is a time value

Ok   Cancel
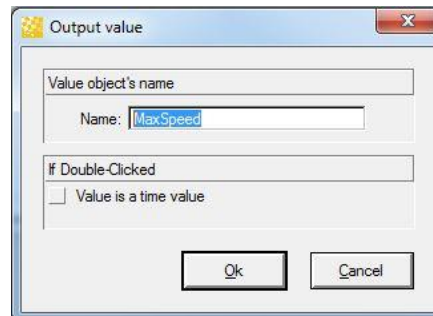
If you select the check box "Value is a time value" this means that the printed value is a time and with double clicking on it the bounded Oscilloscope will go on this time position.

Output Racetrack

Insert/Racetrack

During calculations you can insert segment, delete segments or delete all segments in an Output Racetrack.
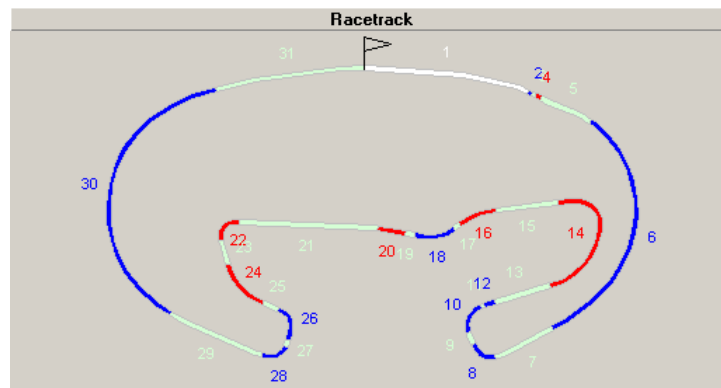
Racetrack

Local Menu/
Settings

F5

If, the object is active (with red frame), you can open and modify its settings. You can open the dialog box using the context menu with pressing right mouse button.

The setup of Output Racetrack contains only the name of the object and number of segmentations of the racetrack. See also: Racetrack Dialog.
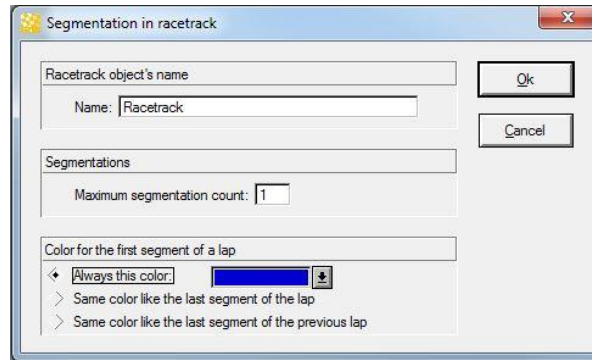
Elements of Design page

Flow objects

There are compare, condition, container, exit, formula, join, next sample, range select case, print in table, print in value and segmentation in racetrack.
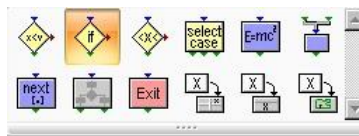
Variables

You need variables to store values, for example the flow-object formula needs variable to store the calculated result.

Variables From most setups of flow object it is possible to open the table with all user defined variables.
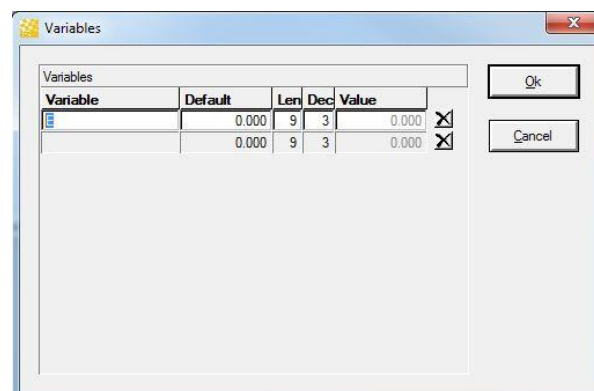
A table row contains a variable and its value.

A new row is added if you make the last empty row current and write variable name. A new empty row will be added after insertion.

Delete

If you want to delete a row then make the row current and then press the delete button.

Parameters

While working with flow-object you can use constant-numbers (1, 2, 5.6 …), constant-strings ("This is a string"), channels and variables. We will call parameter any of these objects.

Start

It is used as a start object of a flow. It has no settings. You cannot insert or delete it.

Compare

Insert/Compare

It is used to compare 2 parameters.
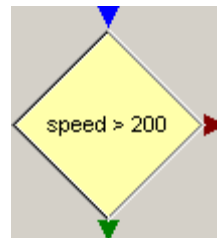
The red arrow means the comparison is false; the green one means it is true.

Tip:
If you move the mouse cursor over them a small window appears with their meaning.

Settings

F5

If the object is active (with red frame), you can open and modify its settings. You can open the dialog box using the context menu with pressing right mouse button.
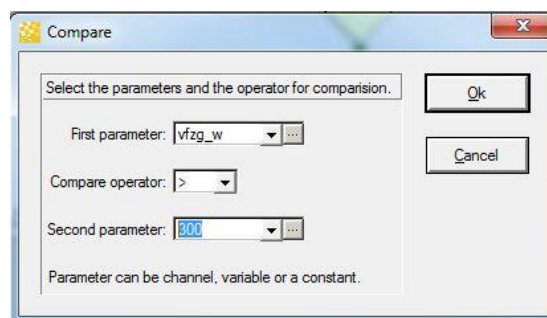
Condition

Insert/Condition

You can use any earlier created condition.
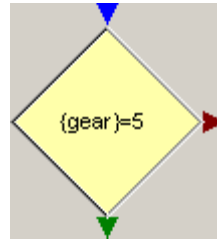
The red arrow means the comparison is false; the green one means it is true.

Tip:
If you move the mouse cursor over them a small window appears with their meaning.

Settings

F5

If the object is active (with red frame), you can open and modify its settings. You can open the dialog box using the context menu with pressing right mouse button.
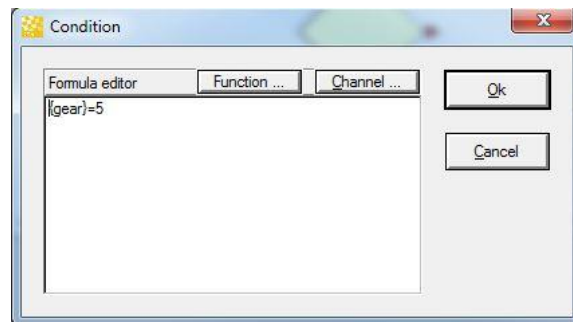
You can build the condition using a dialog with the available functions and a dialog with the available channels (buttons Function... and Channel...). See also: Defining a Condition.

Container

Insert/Container
If you want to use a block of objects several times you can create a container and then use it as a single object.

Settings

F5

If the object is active (with red frame), you can open and modify its settings. You can open the dialog box using the context menu with pressing right mouse button.

Tip:
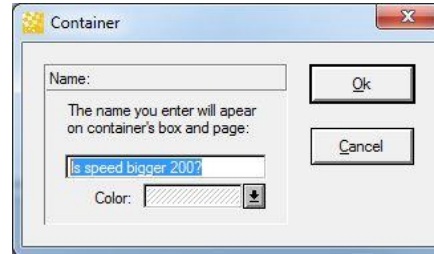
Double clicking on the container will open a new page with the content of the container. To open setup of the container press F5 or use the menu

An example of a container:
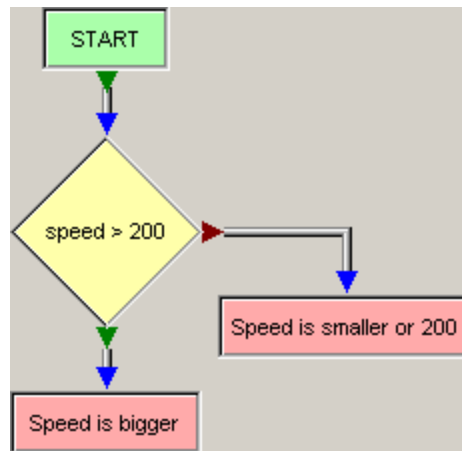
Where "Speed is bigger " and "Speed is smaller" are exits of the container.

See also: Exit.

Tip:
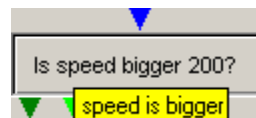
The green arrows of the container are the exits, if you move the mouse cursor over them a small window appears with the name of the exit.

Exit

Insert/Exit   It is used to exit the current container and return to its parent.

   See also: Container.
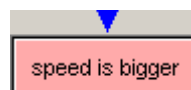
Settings   If the object is active (with red frame), you can open and modify its settings. You can open
F5   the dialog box using the context menu with pressing right mouse button.

Tip:
Double clicking on the container will open a new page with the content of the container. To open setup of the container press **F5** or use the menu

Formula

Insert/Formula

You can calculate a formula for the current sample like mathematical functions; the result is remembered into a variable.

See also: [Mathematical Channels and Filters](#) and [Container](#)

Settings

F5

If the object is active (with red frame), you can open and modify its settings. You can open the dialog box using the context menu with pressing right mouse button.
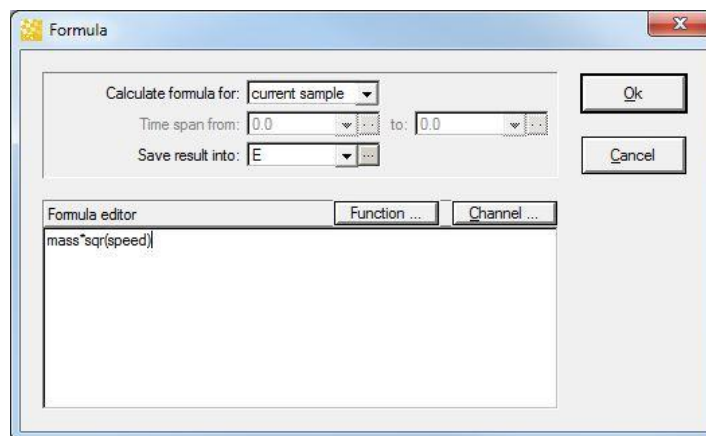
Or you can calculate the Ave, Min, Max, Sum of a formula for every sample in a range by selecting Ave, Min, Max or Sum from "Calculate formula for:". The range is between two parameters ("from", "to").

You can build the formula using a dialog with the available functions and a dialog with the available channels (buttons **Function…** and **Channel…**). See also: [Defining a Function](#).

Join

Insert/Join   Join has no settings. It is used to collect several links and from it can start several links too.

Next sample

Insert/Next sample   Next sample has no settings. It will increase the current sample.

Range

Insert/Range   Range is used to check if a parameter is between two limits. A limit can be constant or a variable.
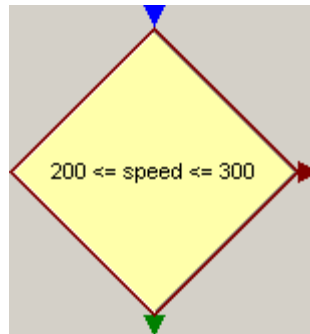
The red arrow means out of range, the green one means it is inside.

Tip:
If you move the mouse cursor over them a small info window appears.

Settings
F5

If the object is active (with red frame), you can open and modify its settings. You can open the dialog box using the context menu by pressing right mouse button.
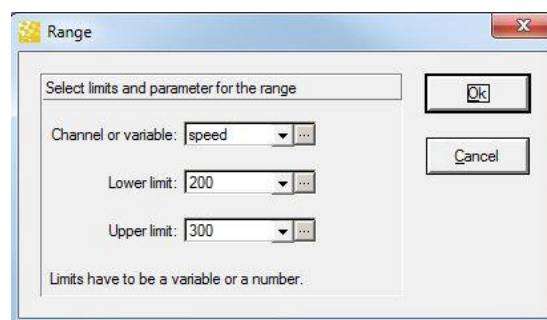
Select case

Insert/Select case

The select case allows selection among multiple comparisons, depending on the select parameter.

Settings

F5

If the object is active (with red frame), you can open and modify its settings. You can open the dialog box using the context menu with pressing right mouse button.
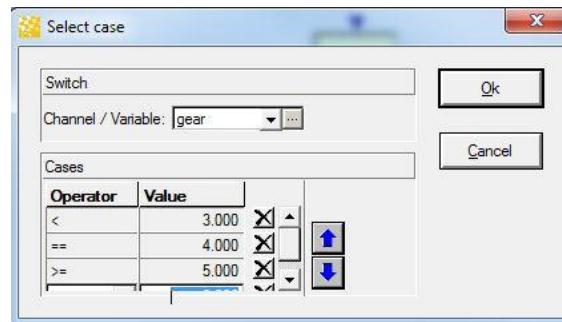
A table row contains a case. You can change the order of cases with Up / Down arrows.

Checking of cases starts from the first one and continues down until some comparison succeeds.

A new row is added if you click the last empty row and select an operator or enter a value. A new empty row will be added after insertion.

Delete

If you want to delete a row then select the desired row and press the delete button.

Print in table

Insert/Print in table

Use Print in table to define in which Output Table and which cell of the table you want to print. The Print in table has the name of its Output Table.

Settings
F5

If the object is active (with red frame), you can open and modify its settings. You can open the dialog box using the context menu with pressing right mouse button.
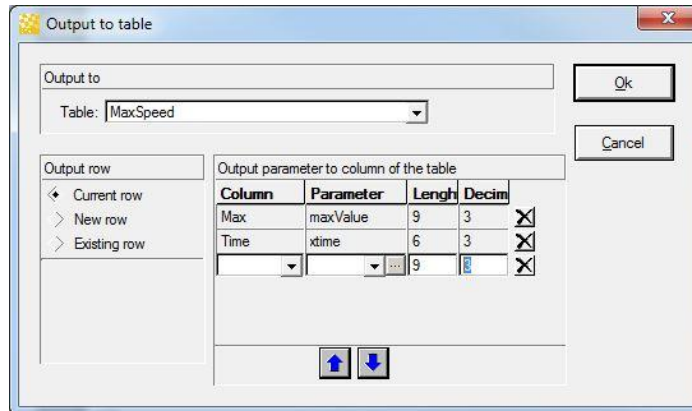
A setup-table row contains a column from Output table and the value you want to print there. You can change the order of rows with Up / Down arrows.

A new row is added if you select the last empty row and then select a column name and parameter. A new empty row will be added after insertion.

Delete

Select the desired row you want to delete and then press the delete button.

"**Output row**" is used to find the row in which to print. If "**New row**" is checked then a new row is inserted into the Output Table. If you want you can save the row in some variable for later use.

If you check "**Existing row**" the row is the value from a variable specified below. See also: Output Table.

Print in value

Insert/Print in value

Use Print in value to define in which Output Value you want to print. The Print in value has the name of its Output Value.

See also: <u>Output Value</u>.

Settings

F5

If the object is active (with red frame), you can open and modify its settings. You can open the dialog box using the context menu with pressing right mouse button.
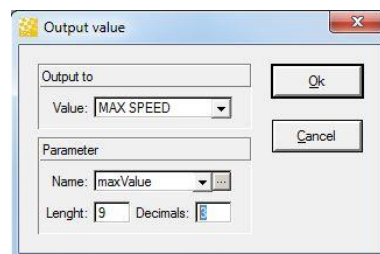
Segmentation in racetrack

Insert/Segmentation in racetrack

Use Segmentation in racetrack to define which Output Racetrack you want to modify. The Segmentation in racetrack has the name of its Output Racetrack.


Settings

F5

If the object is active (with red frame), you can open and modify its settings. You can open the dialog box using the context menu with pressing right mouse button.
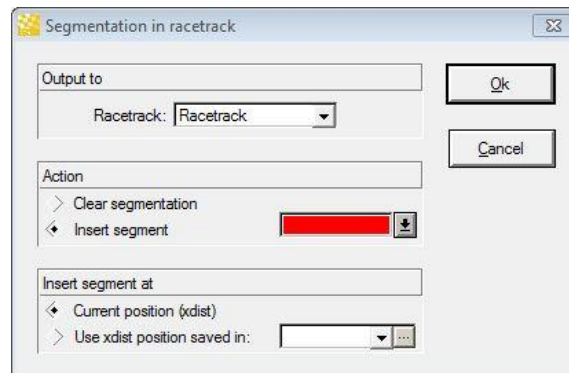

You can select action:

- clear all segments (Insert segment will not be available)
- Insert a new segment with a specified color. The inserted segment can be at
  - current lap position (current sample).
  - any value saved in a variable.

See also: Output Racetrack.

Links

The objects in design pages are connected with links.

See also: Connecting objects with links in design pages.

Working with the Flowchart Window

Bases

There are **Edit** and **Auto Run** modes for the flow window.

Edit mode

Start/ **Settings/** **Design**	In edit mode you can insert, delete, copy, paste objects or links. It is available at the toolbar window under the menu item "**Start/ Pages/New Flow**".

Run mode
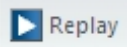
Start/ Tools/ **Replay**	"Run mode" will switch "edit mode" off and all flows will be executed; one after another using the order of pages. In this mode there are several cases in which calculation will start again automatically:

- The sample range has changed (moving in bounded measure window, online files).
- Settings have changed (after OK of setup dialog or after loading).

In "Run mode" you can move, rearrange objects and change links, but it is not allowed to make changes in the logic of the flow.

"Edit mode" will stop current execution (see also Executing the flows) and "Run mode" will be switched off. This means that you will not have any updated results in output objects until you switch on "Run mode" mode again.

Active object

Active object

Clicking on object will make it active. The active object has a red frame.

Settings **F5**	You can *open and modify* the **active object settings**. Open the dialog box using the context menu with pressing the right mouse button.

Only one object can be active at one time but you can mark several objects.

Selected objects

Selected objects

Hold **Ctrl- key** down and click on objects you want to mark. The last clicked object will be also active object. You can mark objects also by dragging a rectangle on the page with left mouse button. All objects that are inside the rectangle will be selected. The selected objects have a blue frame.

If two objects from a design page are selected then the links between them are also highlighted.

Inserting, deleting objects

Flow objects- Design page   **Insert**
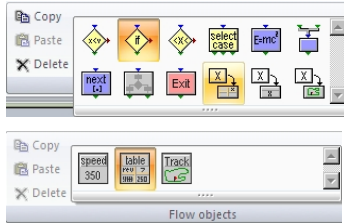
You can insert an object in Output or Design page using the menu Start/Flow objects. Drag an Item into your Flow chart to use it. The context menu is opened with the right mouse click. From the menu select the item "**Insert**" and then select an object.

The toolbars are only available in edit mode.

Delete
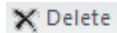
Delete marked objects

The marked objects (all with blue frame) will be deleted. The links between marked objects will be deleted too.

If you press a button with flow object on the toolbar you enter the insert mode for the object. In insert mode every click on the page will be inserting an object. Only if the click is not associated with other actions with bigger priority (move, resize, connect-click on connectors of an object).

Switch off insert object mode

Press the arrow on the toolbar to switch off the insert mode.

Moving, copying, pasting

If there are marked objects you can move them. Moving one of the marked objects will move all others too. The relative arrangement between them will stay. Only the links which connect marked object with not marked one will be changed during the movement. You can use the keys Up, Down, Right, Left to move the selected objects also.

Copy
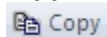
Will copy all marked objects (and their links if any) to clipboard.

Paste
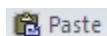
If clipboard is not empty you can paste the objects (and their links if any) from the clipboard.

You cannot paste output objects to design page and vice versa, but you can copy objects from one design page and then paste them in another design page.

Resizing objects

If only a part of the name of an object is visible you can resize the object for the best fit. First make the object active and then if the mouse is near the frame of the object, you will see the resize mouse cursor and then you can resize it.

Connectors of an object

Every object in design pages has one input and/or one or several outputs. They are used to define the direction of execution. The object "**Start**" has no input connector.

In-Connector

If the link which enters the connector is active then the object will start working.

Out Connector (True) ▼	If this output is found as the results of the calculation of an object then the link which starts from with will be activated. In compare, condition and range this connector is chosen if the result is true.
Out-Connector (False) ▶	In compare, condition and range this connector is chosen if the result is false.

Connecting objects with links in design pages

You can connect objects with links. A link is created if you drag from output connector to input connector (or vice versa) with left mouse button down. The line will follow the mouse cursor.

If the mouse cursor is over a connector it's getting lighter. You can start a link when it is in edit mode and the connector is lighter. You can finish the link if the end connector is lighter.

If the mouse cursor is over an output connector and you press right mouse button (content menu will appear) and select an object from "Insert" item of the content menu then the input connector of inserted object will be connected with the output connector.

Editing links

Clicking on a link will make it active. Then you can change the link with moving some parts (sub lines) of it left, right, top or bottom.
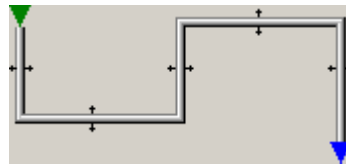
Executing the flows

State of a flow

The state of the flow is defined by current sample and working object or active link.

Working

In any time of execution there is only one working object or active link. The working object can use the current sample with some channel and/or information saved in variables and can save result in a variable or print it in output object. Depending on the result the working object will select an active link. The task of active link is to select next working object and so on.

If the working object cannot select a new active link then the current sample will be increased automatically and the last active link will be activated.

For example you have a compare object with an input link, output link if comparison is true and no link if

the comparison is false.
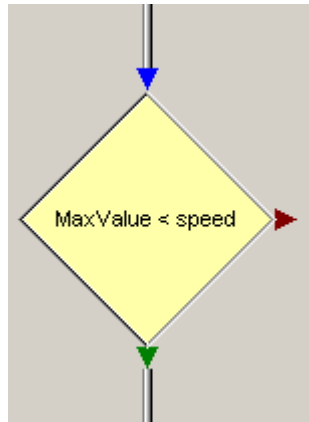
Then the input link will make the object working and suppose that the MaxValue >= speed for current sample. Then there is no link right to continue and then the current sample will be increased automatically and the input link will be activated again. The result of this is: the output link will be reached when a sample for which the speed is bigger as the MaxValue appears.

The equivalent scheme is
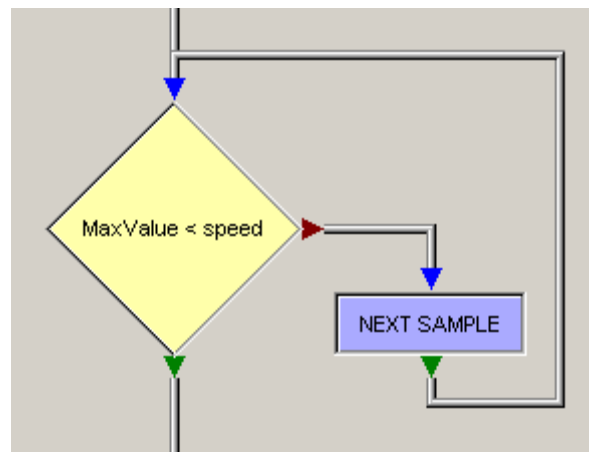
You should be careful when using no link feature. If it is used wrong then unexpected results are possible.

Iteration

The execution of a flow for a single sample is a iteration. You can go to the next iteration simply by going to the "NEXT" object. See also: Next sample.

Because the calculations can take much time especially if the range is about the whole file, they are made inside of a working thread. Pause and Stop button are enabled. Instead of waiting for them you can work with some other windows.

For online files, calculations are started automatically after a change in the range is made (new samples have been received) or the output page will be updated.

Errors

Invalid entry

Before running a flow the application prepares some elements of the flow. For example locking used channels. If there is an element which cannot be prepared from the application (for example channel name which is not in current file) then a message appears and the setup of the element will be opened and the wrong parameter will have a light red background. The calculation will not start.
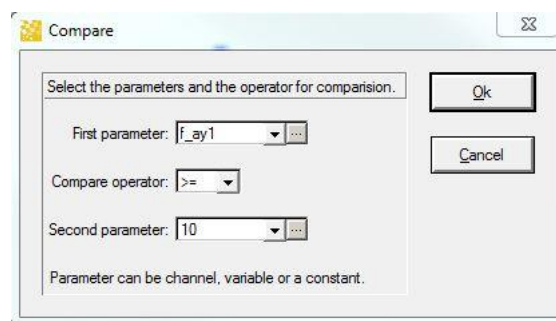
Endless loop

During execution of a flow the application checks if there is a progress of execution. Usually if the flow stays more than some time executing one sample then it is supposed that there is an endless loop and the calculation will stop. The supposed endless loop will be shown to the user.
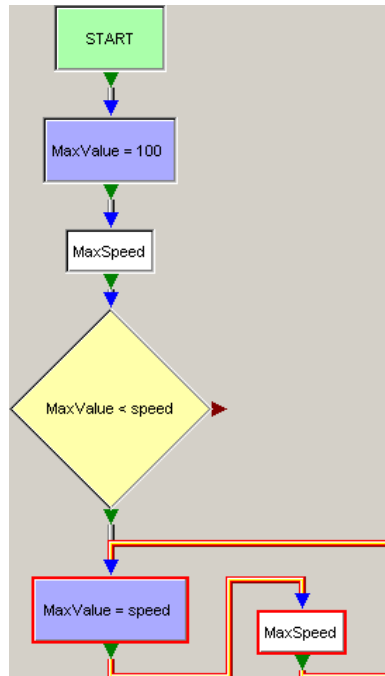
Table Min/Max

A Min/Ave/Max Table enables you to determine the minimum, average and maximum values for every lap and the whole file for selected channels. The results are shown in a table.

	Min	Ave	Max	Min	Ave	Max
	● RaceData 1					
	accy_w			vfzg_w		
Overlay values	-26.875	2.472	27.875	65.000	197.914	320.625
#9(01:43.42 *)	**-26.875**	2.472	**27.875**	**65.000**	197.914	**320.625**
	● RaceData 1					
	nmot_w_1					
Overlay values	2114.00	3836.37	5410.00			
#9(01:43.42 *)	**2114.00**	3836.37	**5410.00**			
	● RaceData -					
	accy_w			vfzg_w		
Overlay values	-26.875	2.423	27.750	63.875	197.873	321.625
#10(01:43.29 *)	-25.375	2.444	**27.750**	67.500	198.099	321.375
#11(01:43.40)	**-26.875**	2.402	27.250	**63.875**	197.647	**321.625**
	● RaceData -					
	nmot_w_1					
Overlay values	2142.00	3835.64	5352.00			
#10(01:43.29 *)	2372.00	3849.39	5090.00			
#11(01:43.40)	**2142.00**	3821.88	**5352.00**			

Open a Table Min/Max Window

To open a Table Min/Max Window select "**Windows/ Worksheet controls / Min/Max-Table**" and drag it into your workspace.

Worksheet controls
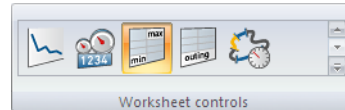
Min/Max Table Settings

A dialog box with settings appears automatically when you open the Min/Ave/Max Window for the first time. Closing the window will save its settings and the next time the window is opened the dialog will not appear again. Instead the saved settings will be loaded.

If you want to modify the settings or carry out a new calculation, you can re-open the dialog box using the menu item "**Start/ Settings/ Edit**" or by using the context menu with pressing middle mouse button or left + right mouse buttons.

Settings

You can press the **F5** key to *open* the "**Load Settings**" dialog box.

How to save and load settings see: Settings in Analysis.

Overlays Tab

This tab is used to select files for evaluation and it is similar to Common Overlays Settings. The only difference is that there is no range available to be selected because the min, ave and max are calculated for the whole overlays.
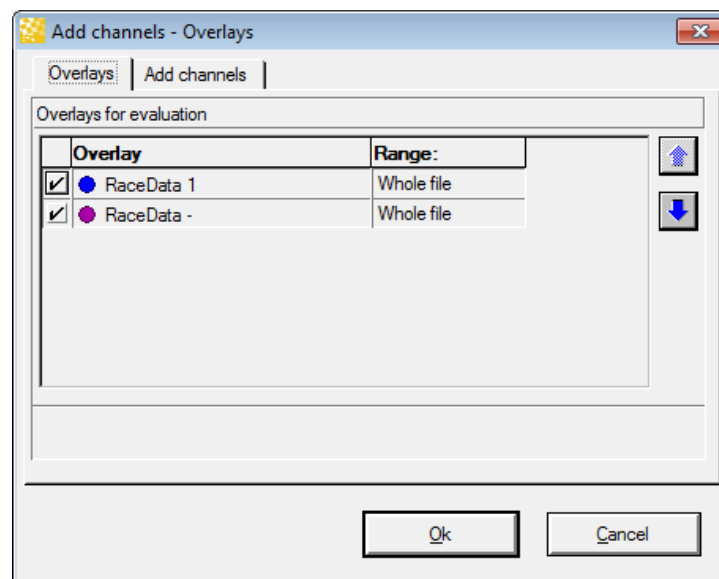
Add Channels Tab

This tab is used to select channels for evaluation.
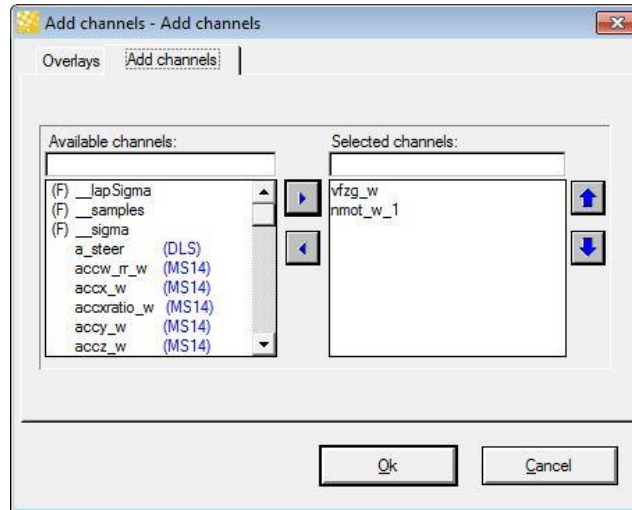
Available channels	This list box shows all channels of the selected file.
Selected channels	This list box contains channels for evaluation. Select a channel from "Available channels" and press enter or the right arrow button to copy it to selected channels.

If there is a selected channel in the list box - "Available channels" then with **Enter –key** the channel will be moved to "Selected channels". Pressing the **Enter – key** for the second time will close the setup dialog.

Left/ Right arrows	Remove or add a channel to the selected channels list box.
Up/ Down arrows	If wish you can reorder the selected channels.

Results Window

This window displays the results of all min/ave/max calculations carried out for the selected file in the Oscilloscope.

When you carry out a new calculation for a new channel, the results for the other channels remain unchanged.

Emphasized (yellow background and bold font) values are absolute values (Min/Ave/Max) for channels.

Because the calculations can take much time they are made inside of a working thread. You will see "…" in uncalculated cells and a flashing channel name for currently calculating channels. Instead of waiting for them you can work with some other windows or look over the already calculated once.

For online files, calculations are started automatically after changing the range (new lap have been received) or update the view.

The number of channels displayed horizontally depends on the size of the window and the size of the columns.

You can copy the result in the clipboard. The plain text and rich text formats are supported.

Outing Report

The Outing Report provides a quick overview of an individual outing. Values for individual laps or lap segments, from any number of different channels are brought together and displayed in an organised fashion.

In addition, an outing report enables you to quickly access points of interest in an Oscilloscope.

Open an Outing Report Window

Drag the menu item "**Windows/ Worksheet controls/ Outing report**" into your workspace to open an Outing Report Window. The window that appears will display the results that were calculated based on the most recent settings.
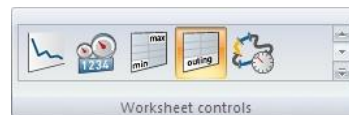
Outing Reports are typically used to for the performance of two different tasks:

- For the evaluation of files in the Oscilloscope, so that the results for all opened files will be displayed automatically. If you open further files (or if you close already opened files) WinDarab automatically updates the Outing Report.
- For the evaluation and documentation of certain files, in which case only the files that have been specified for evaluation will be added to the Outing Report.

You can save the settings you make for an Outing Report in an Outing Report File (file extension *.bmsor). Among other information this file saves the evaluation results so that - if an Outing Report is opened - new calculations will not be necessarily. This can save a lot of time

Outing Report Window

In an Outing Report evaluation results are displayed as a table.

	Evaluate	Lap	Laptime	Fuel	Max vfzg_w	Min accy_w	Ave nmot_w_1
⊟	**RaceData -**						
	Demo Comment						
		#10	01:43.29 *	2.63	321.625	-26.875	3846.74
⊞		theoretical	01:42.75				
⊞		running	01:43.10				
⊞	✓	#9	01:43.42	2.63	320.625	-26.875	3846.74
⊞	✓ FAST	#10	01:43.29 *	2.63	321.375	-25.375	3836.25
⊞	✓	#11	01:43.40	2.63	321.625	-26.875	3827.10

Segment Times Table If the evaluation of a lap segment has been activated, it is possible to display a table containing segment times. This display format provides an excellent overview of individual and fastest segment times.

Evaluate	Lap	Laptime	Fuel	1	2	3	4	5	6	7	8	9	10	
RaceData -														
Demo Comment														
	theoretical	01:42.75		7.977	3.055	2.888	5.490	2.470	4.000	9.750	2.710	21.725	3.857	2
	running	01:43.10		7.977	3.130	2.930	5.538	2.540	4.050	9.750	2.710	21.725	3.857	2
✓	#9	01:43.42	2.63	8.003	3.055	2.910	5.490	2.470	4.140	9.852	2.738	21.831	3.870	
✓ FAST	#10	01:43.29 *	2.63	7.986	3.082	2.888	5.517	2.503	4.000	9.817	2.733	21.785	3.960	
✓	#11	01:43.40	2.63	7.977	3.130	2.930	5.538	2.540	4.050	9.750	2.710	21.725	3.857	2

Working with the Results Table
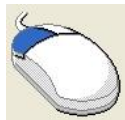
Changing the Column Width
Use the mouse to *adjust* **column width**.

Double-clicking Cells
Double-clicking on a cell prompts WinDarab to look for an Oscilloscope that displays the file. If WinDarab find one Oscilloscope the cursor will jump to the location at which the selected value occurred. If nothing is found (as is the case for lap averages) Win Darab zooms in on the entire lap.

Right Mouse Click
Clicking your right mouse button *displays* a **local menu** with the following options:

- Display Laps
- Display min/max row
- Display Comment
- Display Comment Fields
- Edit Comment
- Load Settings
- Save Settings
- Settings…
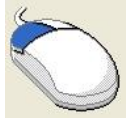
Expanding the Lap

In "Channel results"-mode click on the + or – sign in front of the lap to show or hide the laps segments and their individual results.

Expand/ Collapse all laps	Use the ribbon buttons or context menu entries to expand or collapse all laps at once.
Evaluate Lap	This column indicates if the lap is evaluated into the running or theoretical lap. You can manually include or exclude a lap by clicking on the checkbox. A blue checkbox indicates that the evaluation was manually changed.

Settings

A dialog box with settings appears automatically when you open the Outing Report Window for the first time. Closing the window will save its settings and the next time the window is opened the dialog will not appear again. Instead the saved settings will be loaded.

If you want to modify settings or carry out a new calculation, you can re-open the dialog box using the menu item "**Start/ Settings/ Edit**" or by using the context menu with pressing the middle mouse button or left + right mouse buttons.

You can even press the **F5** key to *open* the "**Load Settings**" dialog box.

Note:
In addition to the actual settings the calculated values of files displayed in the Outing Report will be saved. No new calculations will have to be made when you re-open the Outing Report at a later time.

How to save and load settings see: Settings in Analysis.

Columns Tab

This tab is used to configure columns that are displayed in the Outing Report.
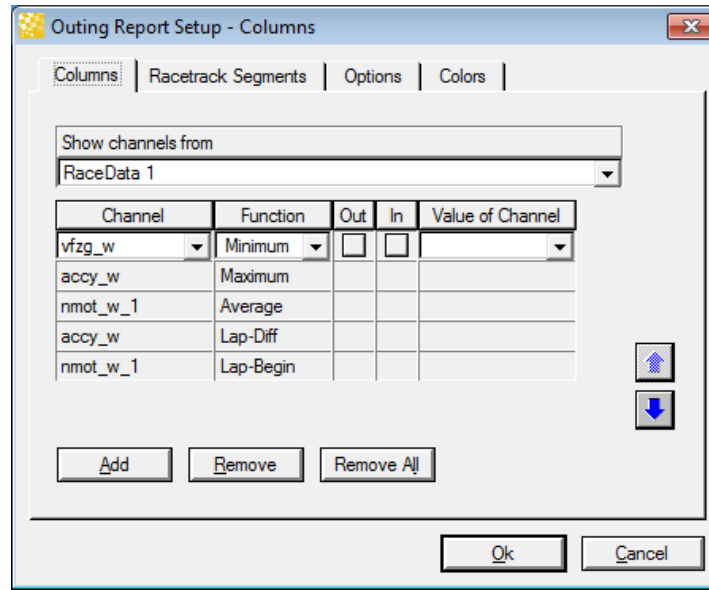
Show Channels from	Use this field to select the file that will provide the list of available channels. It is also possible to enter the channel names you want in the channel fields. However, selecting a file name saves time and eliminates the possibility of making typos when entering the channel names.
Outing Report Table	Each row in this table represents a column in an Outing Report. Use the first two fields, "**Channel**" and "**Function**" to specify which value has to be calculated for the selected channel. Use the "**Out**" and "**In**" option buttons to specify whether you want to consider the first lap ("Out") or the last lap ("In") of an outing for the calculation of the "most extreme" value (which will be displayed in the min/max row of the Outing Report). Finally, in the "**Value of Channel**" field specify the value you want to display in the Outing Report. If you do not make an entry in this field, the value for the specified channel will be displayed. But you also have the option of displaying the value for another channel, at the moment the specified condition pertains.
Add	Use this button to add a new row to the table.
Remove	Use this button to remove the current row from the table.
Remove All	All rows will be removed from the table.
Up / Down Arrows	Clicking on the arrows will move the current row up or down. This determines the column sequence in the outing report.

Segments Tab

Activate the evaluation of course segments in the "Segments" tab.

You have the option of defining segments with segment triggers for which you need hardware (i.e. segment triggers) whose signals have to be recorded in the vehicle.

Another option is the use of a racetrack with a course segmentation of your choice.
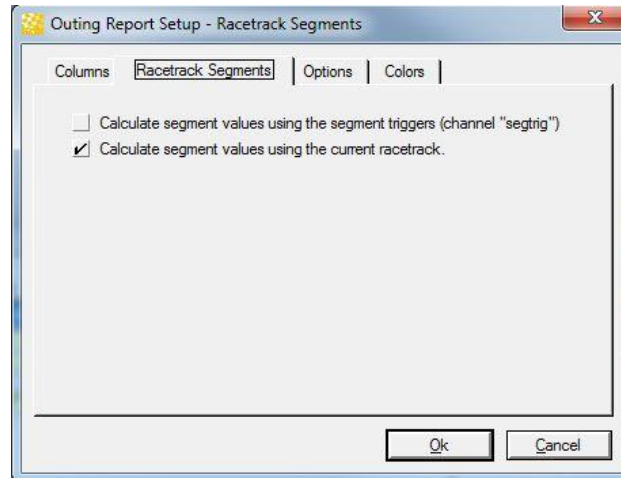
Note:

The segmentation in the Outing Report is separated from the selection in the Racetrack Window. Therefore if you switch racetracks using "File / Racetracks ..." the selected racetrack in the Outing Report will not be changed.

Options Tab

In this tab, you can select display options that determine the appearance of the Outing Report.

You can activate (or de-activated) most of these options within the Outing Report with your right mouse button.
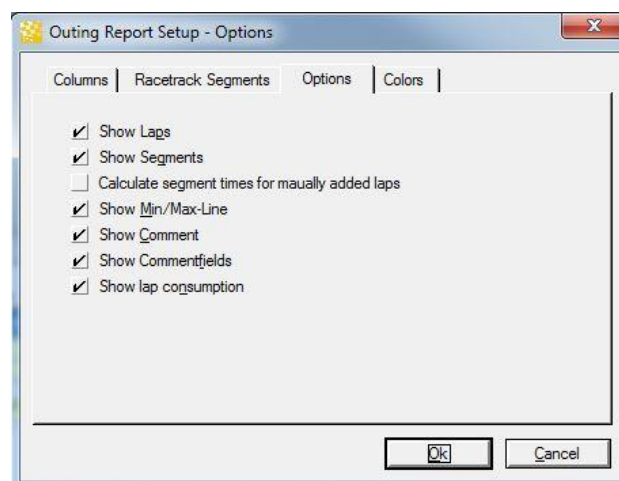
Show Laps	When you have activated this option, a row containing the respective values for each lap will be displayed in the Outing Report.
Show Segments	Activating this option enables you to turn on or off the display of individual segment results for each lap by clicking on the "+" or "-" button. If this option has not been activated, segment results will not be displayed.

Show Min/Max Row	If you have activated this option, a row with min/max values will be displayed in the Outing Report. This row displays the "most extreme" values of all laps. Depending on how you have configured a column, values for the first lap ("**Out**") or the last lap ("**In**") may also be considered.
Show Comment	Selecting this option displays the contents of the comment field in addition to the file name.
Show Comment Fields	Selecting this option displays the contents of all comment fields in addition to the file name.
Show Fuel Consumption	Activate this option, if you would like to view the fuel consumption for each lap in addition to the lap number and lap time.

You can activate (or de-activated) most of these options in the Outing Report with the right mouse button.

Colors Tab

This register enables you to make your color choices for the Outing Report.
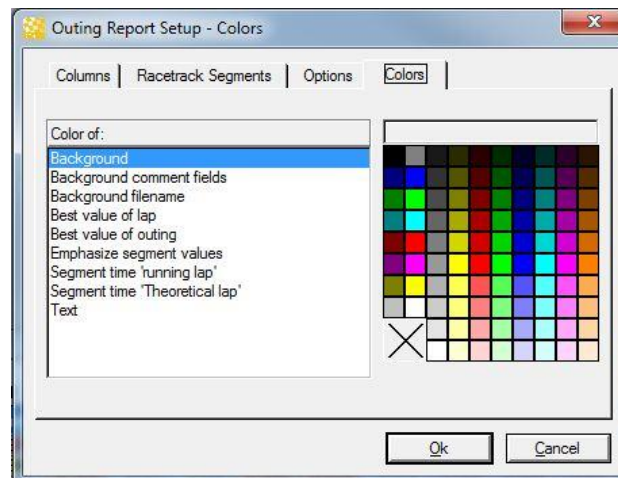
The palette is an excerpt from the "Colors" Tab in "Options/ Settings".

Start/ Overlay   In this Menu, you can chose the Overlay file, that will be shown in the Outing Report-window.

Show overlay file
⬢ Out04_Lap1-_Frag01_R10 ▾
Overlay

Lap Analysis

Lap analysis represents a comprehensive analysis tool for course segmentations, which enables you to evaluate and display logged data in a racetrack diagram.

Lap Analysis supports the calculation and display of several course segmentations in a single window, displaying individual segmentations as parallel tracks (something that cannot be done in regular course segmentation).

The evaluation for individual laps will be carried out according to the course segmentation. You can define as many evaluation rules as you like. Each evaluation rule describes which function has to be used to evaluate a channel, which segments have to be examined, and how the results should be displayed in the diagram.

To learn more about the evaluation algorithm, please refer to "Analysis Rules".

Open a Lap Analysis Window

Select "**Windows/ Worksheet controls / Lap analysis**" from the menu and drag it into your workspace to open a Lap Analysis Window. The Lap Analysis Window appears and its content indicate which calculations will be carried out, based on the most recent settings you made.

Worksheet controls
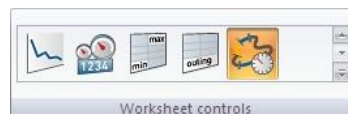
It is possible to save settings for a particular lap analysis in a Lap Analysis File (file extension *. bmsla).

Analysis Rules

An individual analysis rule contains the following information:

- The name of the channel you want to analyze
- The function you want to use to analyze the channel
- The specification of the course segment for which the channel will be analyzed
- The name of the channel whose value should be displayed if a matching course position has been identified during the analysis of the specified channel.
- A combination of style attributes that determine how the results will be displayed.

Channel Names

You are able to choose two channels per rule; the first is the channel you want to analyze. It will be

evaluated with the help of a function.

If you want to identify a particular course position (e.g. the maximum value of the channel) the found maximum value for that channel will be displayed accordingly if you **did not select** a channel in the "Value of" field. However, if you **did select** a channel in that field, the value of this specified channel will be displayed (for the identified position) and not the value of the channel that is being analyzed.

> Selecting a channel in the "Value of" field is only possible, if the function you have selected belongs to the type of function that are able to match a course position to a particular result.

Functions

You need to distinguish between three different types of functions:

1. Functions whose results cannot be matched to a particular position on the course (e.g. average or standard deviation). Since there is no matching course position for these results, they are always displayed at the center of the evaluated course segment.
    During a comparison with a reference lap, the value for the reference lap is calculated for the same course segment.
2. Functions whose results can be matched to a particular position on the course. Depending on the function, the number of results can range from none to many (e.g., Minimum, Maximum and Change). Results will be displayed in close proximity to the course position that has been identified. A line indicates the position on the course.
    During a comparison with a reference lap the channel value of the reference lap is extracted at the same location on the course.
3. The "Course Position" Function. This function does not analyze a channel. Instead, it indicates a channel value for a vehicles current position on the course. The Lap Display Window enables you to change the position of the vehicle and to insert additional values at as many points on the course as you like.
    Results will be displayed in close proximity to the course position that has been identified. A line indicates the position on the course.
    During a comparison with a reference lap the channel value of the reference lap is extracted at the same location on the course.

Course Segment

Rules are evaluated for particular course segments. In the selection list you will find "each segment", "entire lap" and the colors for individual course segments as specified in the racetrack segmentation.

If you select "entire lap" the lap will be evaluated independent of the racetrack segmentation. In all other cases, course segments are evaluated independently.
For example: the "Maximum" function does not identify the maximum channel value for segments with the same color; rather it finds the maximum channel value for each segment with the selected color.

Style Attributes

Style attributes are used to format the display of results. You will find more detailed descriptions of individual attributes in the "Style and Display" section.

The Lap Analysis Window

This window displays the results of the lap analysis in the racetrack diagram.
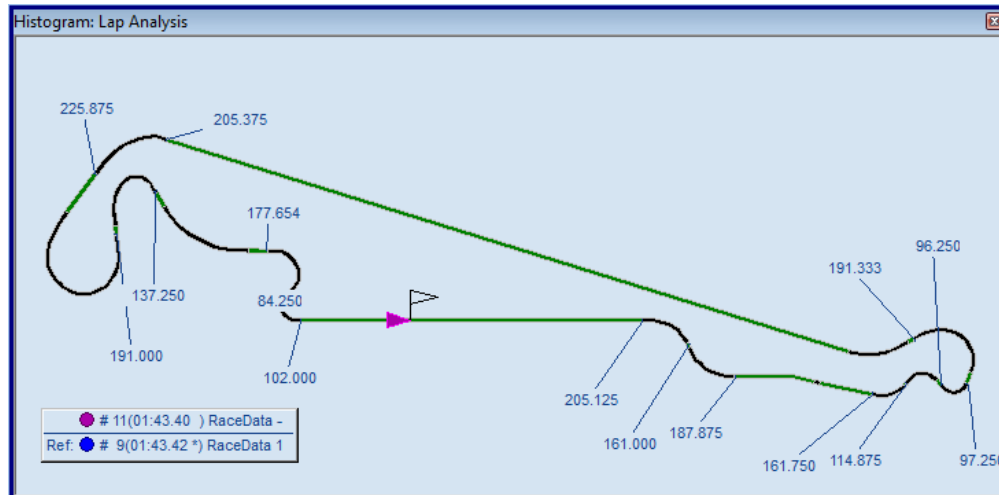
Working with the Lap Analysis Window

A Lap Analysis Window works much like a Racetrack Window. The following additional functions are also available:
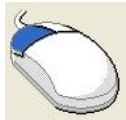
Left Mouse Click
Select a **result** by clicking on it. This will move the corresponding vehicle to the appropriate position. The selected result will be framed in the same color as the segment in which it occurred.

- **Right Mouse Click**
  Clicking on the window displays a local menu with the following options:
- Delete Result
  This function deletes the result you have selected from the diagram.
- Insert Result from "Point rules"
  This function evaluates all rules of the "Course Position" type for the vehicles current positions on the course and displays the results.
- Recalculate all
  This function deletes all displayed results from the diagram, carries out a new calculation and places the new results near the racetrack.
- Show file legend
  This function displays a list of file names and evaluated laps. You are able to adjust the size of this list and place it anywhere you like.
- Car shape
  Select the shape of the car that is displayed.
- Load Settings
- Save Settings
- Settings…

Delete Result

Context Menu/ **Delete result** **Start/ Results/** **Clear** 🏁 Clear	1. Click on the result you want to delete. It will be displayed in a color rectangle. 2. Select "**Start/ Clear**" from the menu or "**Delete result**"
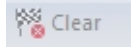
Insert Result of Position Rules

Local Menu/ **Insert Result** **from "Point"** **rules**	1. Move the vehicle (with a mouse click) to the position on the course for which you want to evaluate the "Course Position" rules. 2. Select "Insert Result from "Point" rules" from local menu. 3. WinDarab will now calculate the rules for the current vehicle position and display the results in the diagram.

If you have calculated a number of rules for the same position you will have to arrange the results in a useful way.

Recalculate all

Context Menu/ **Recalculate all** **Start/ Results/** **Recalculate** 🏁 Recalculate	Select "**Start/ Recalculate**" from the menu or "**Recalculate all**" from the context menu to delete all results from the display and to carry out a new calculation.

Show file Legend

Context Menu/ Show **file legend** **Start/ Display/** **Legend**	Select "**Start/ Display/ Legend**" from menu or "**Show file Legend**" from local menu to activate the display of file names and the evaluated laps.

Legend
Display

You can change the size and placement of this window.

Lap Analysis Settings

A dialog box with settings appears automatically if you open the Lap Analysis Window for the first time. Closing the window will save its settings. The next time the window is opened the dialog will not appear again. Instead the saved settings will be loaded.

If you want to modify settings or carry out a new calculation, you can re-open the dialog box using the menu item "**Start/ Settings/ Edit**" or by using the context menu with pressing middle mouse button or left + right mouse buttons.

You can press the **F5** key to *open* the "**Load Settings**" dialog box.

How to save and load settings see: Settings in Analysis.

Overlays Tab

Use the "Overlays" tab to specify the overlays or laps for which you want to calculate a lap analysis.
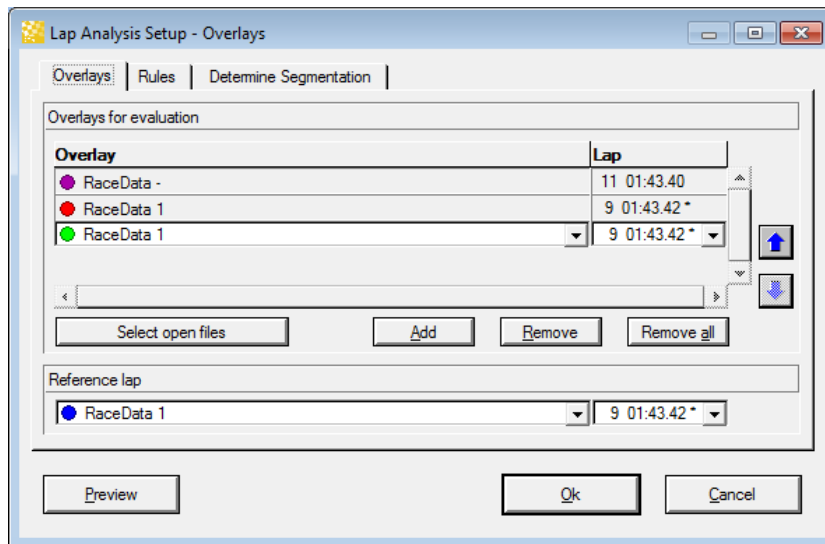
Overlays for evaluation	This list includes all files and laps for which the rules will be calculated during lap analysis.
	Use the arrow buttons on the right-hand side to change the sequence. This is important if you want to calculate the course segmentation for a single file, where WinDarab always uses the first file in the list.
"Select Open Files" Button	Use this button if you want to analyse the same files and laps that are already opened in the Oscilloscope. The appropriate entries will then appear in the list.
"Add" Button	Click on this button if you want to add another line to the table.
"Remove" Button	Click on this button if you want to remove the current row from the table. The fields will display the file and lap selection that was pre-selected for the current Oscilloscope.
"Remove All "	Click on this button to empty the table and to add a new row showing the current Base

Button	File and laps for the Oscilloscope.
Reference Lap	Click on this button to select the file and lap you want to use as the reference lap.

A reference lap will only be used if a rule uses a display mode that pertains to the reference lap. Otherwise the setting has no effect.

Rules Tab

The "Rules" tab is used to specify the analysis rules. An analysis rule stipulates which function will be used to evaluate a channel for a particular course segment as well as how the result will be displayed.
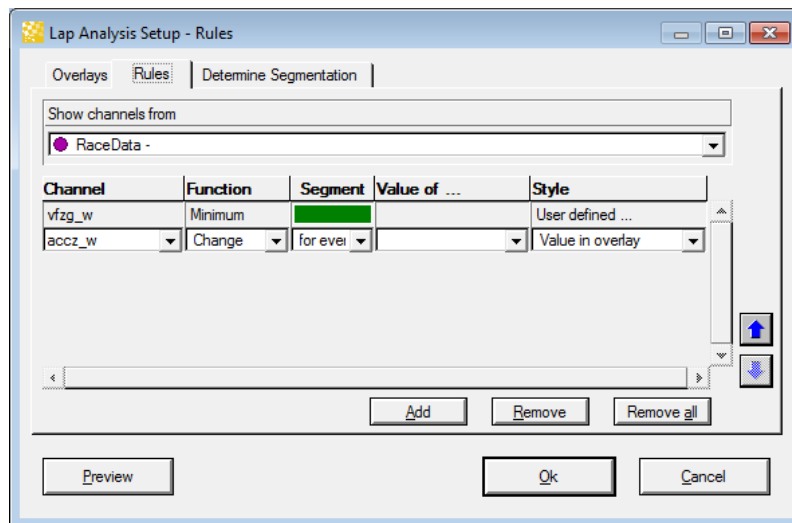
The following specifications can be made for each rule:

Channel	Name of the Channel you want to evaluate. Depending on the function you select, this channel is used to calculate a particular position or result.
Function	Choose the function that has to be used for the analysis of the selected channel. If the result for this function also contains a course position (such as Minimum or Maximum values) the rule is then able to return the result for a second channel at the same position on the course (see "Value of …").

Following functions are provided:

- Maximum
- Exit
- Change
- Standard deviation
- Average
- Difference Exit-Entry
- Dist position
- Entry
- Minimum

Segment	Select the course segments you want to analyze individually in this area. Results are calculated for each course segment. Depending on the function (e.g. "Change"), several results might be calculated for each segment. The "Entire Lap" option is a special case. When this option is selected segments are not evaluated individually but rather the lap in its entirety.
Value of …	It is possible to select a second channel here. As a consequence the returned value will reflect the result for the channel you have specified in this field and not the value for the channel to be analyzed. This option is only available if you have selected a function whose calculation yields a course position (e.g., Minimum or Maximum values)
Style	Here you can choose from a number of result display styles, including the option "user-defined …". Selecting this option will open a separate dialog box called "Style and Display" that enables you to select individual options. All other buttons and entry fields in this tab are described below:
Show Channels for	In this field, select the file you want to use. The channels for the selected file will be shown in the channel selection fields under "Channel" and "Value of".
"Add" Button	Click on this button if you want to add an empty row to the table.
"Remove" Button	Use this button to remove the selected row from the table.
"Remove All" Button	Clicking on the "Remove All" button removes all rules from the table.

Calculate Segmentation Tab

You are able to define rules for the automatic calculation of course segmentation in the "Calculate Segmentation" tab.

If you choose neither the "for the first selected file only" nor "for all selected files", then the automatic calculation of a course segmentation will be de-activated. In this case the course segmentation for the selected racetrack applies. (See Load Racetracks).
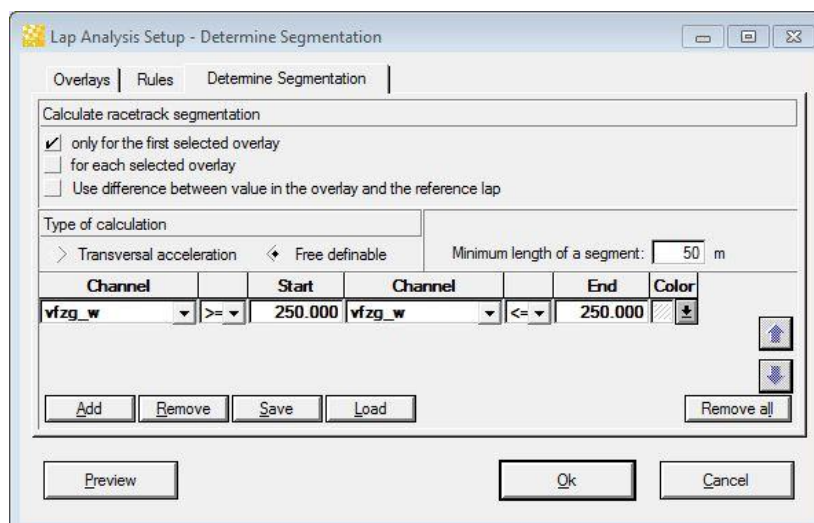
Calculate racetrack Segmentation

Use the buttons in this area to determine for which file(s) segmentation will be calculated.

For the first selected file only	If you select this option the calculation is made only for the first file listed in the "files" tab. This means that the course will be broken down into a single segmentation that applies to all files equally during the evaluation of rules that have been specified in the "rules" tab.
For all selected files	Choosing this option means that course segmentations will be calculated for all the files you have selected in the "Files" tab. This results in a course with a number of parallel tracks. Each track represents the course segmentation for a particular file. Vehicle icons move along their individual tracks.
Use the "difference between file values" and the "values of the reference lap"	When course segments are calculated, WinDarab does not use the absolute values for each channel; instead it calculates the difference between the values of the specified file and the reference lap you specified in the "Files" tab. This button corresponds to the "Difference" button in the " Calculate Segmentation" tab inside the "Racetrack and Segmentation" dialog box.

This tab corresponds in most respects to the "Calculate Segmentation" tab in the "Racetrack and Segmentation" dialog box.

Style and Display

This dialog box enables you to make decisions about the style and presentation of analysis results. It will be shown when u select "User defined ..." as Style in the Rules-Tab.
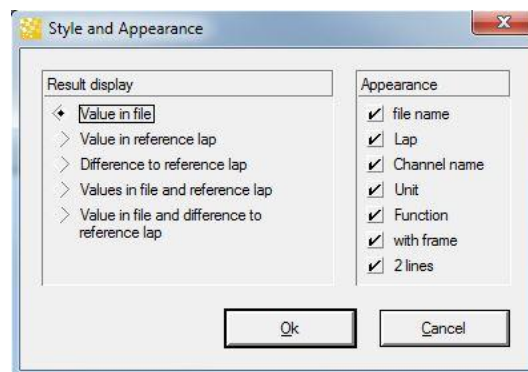
You are able to decide which results will be displayed through in the "Display Results" options list.

Value in File	If this option is selected, only the result for the evaluated file will be displayed.
Value in Reference Lap	If this option is selected, only the result for the reference lap will be displayed.

Difference   Choosing this option displays the difference between the result for the evaluated file and the result for the reference lap.

Values from File and Reference Lap   Results for the evaluated lap and the reference lap will be displayed. The value for the reference lap will be displayed in brackets.

Value in File and Difference   This option displays the result for the evaluated lap and the difference to the reference lap. The value for the difference is displayed in brackets.

The "Display" options enable you to select a number of style attributes used to modify the appearance of the results display. Instead of explaining these options in great detail, we suggest that you view the various possibilities using a sample lap.

The result for the reference lap is calculated in the following way:

- If the result for the evaluated lap has been assigned to a specific position on the course (e.g. Minimum or maximum) the result will be extracted from the reference lap at exactly this position.
- If the result for the evaluated lap has not been assigned to a specific position on the course (e.g. Average) the result for the reference lap will be calculated for the same course segment.

Telemetry

In WinDarab there are several tools for analyzing and monitoring telemetry online data (online files). In online files the end is unknown and new data can be received at any time and the new samples become available immediately. Oscilloscope can open online files just as they are normal files and you can work with them in the same way.

Re-establishing connection after network disconnect

If an active connection to the WDServer is required (e.g. after a network disconnection because of unplugging the network cable) WinDarab tries to reconnect to the WDServer by sending server search packets to the WDServer.

Instrument panel is used to show channel values using different display options and shapes. It also allows tracking the channel values and raising alert events if they exceed user defined ranges.

Open an Instrument Panel Window

Drag "**Start/ Worksheet controls / Instrument panel**" from the menu into your workspace or select "**Instrument panel**" from local menu to open an Instrument Panel Window.

To open an online file use Select File/ Telemetry in the context menu of the Instrument Panel Window. After the online file is opened you can selected it in an Oscilloscope as a normal file. If the file is online then in the menu and tool bar of the Oscilloscope become available in two modes:

Replay ON	Follow (online data). This is the default mode. In this mode the logged data cursor will move to the new received data of currently selected online file and channel.
Replay OFF	Pause (online data). The data will continue to arrive, but the cursor of the Oscilloscope will not follow it. In this mode you can examine the received data until the present.
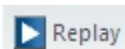
Oscilloscope will automatically switch to "Pause mode" if you scroll the window.

Instrument panel
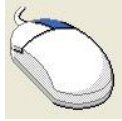
Context Menu - Right Mouse Click

Clicking your right mouse button *displays* a **local menu** with the following options:

Instrument panel / Select File/ Telemetry	You can select an online file (car) to monitor.
Instrument panel/ Add	You can add a display object in Instrument Panel Window. Available objects: Value Display, Bar Display, Round scale, Previous laptime, Current laptime.
Instrument panel/ Remove	Removes the selected display object.
Instrument panel/ Copy Ctrl + C	You can copy the selected displayed object into the clipboard.
Instrument panel/ Paste Ctrl + V	You can paste the clipboards displayed object into the window.
Instrument panel/Clone Ctrl + D	You can make another displayed object (clone) with the same properties like the selected one.
Instrument panel/ Load Settings F6	Load Settings.
Instrument panel/ Add Settings Ctrl + F6	Union current settings with settings from a file.
Instrument panel/ Properties F5	Show properties of the selected display object, if there is selected one.

If a new round scale is added to the instrumented panel by dragging a channel into the window the round scale will not render correctly until the properties window is opened/applied or closed/reopened.

If the user assigns a channel to an instrument (Value display, Bar display, Round scale, Bit Panel) and the channel is empty (contains no data) a message is displayed to notify the user about this.
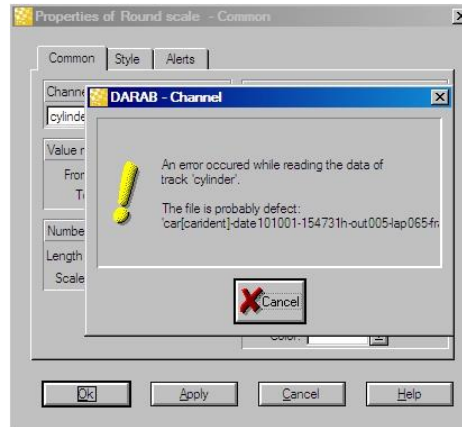
Value display

Displays the current channel value as a number. This display shows simply the channel name and its current value.
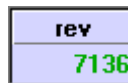
In addition you can see the value in different colors (states) depending on defined ranges for the channel in the setup common page.

Properties   If the object is active (indicated by a blue frame) you can open and modify its properties. You
F5   can open the dialog box using the context menu with pressing right mouse button.

A dialog box with 2 tabs appears:

Common Tab

This tab is used to select channel, its ranges and number format. You can divide the channel range up to 3 sub ranges upper, middle and lower. You can choose a color for any of the sub ranges. This display doesn't have a scale so the scale factor is not available. You can also change the background color of the item.
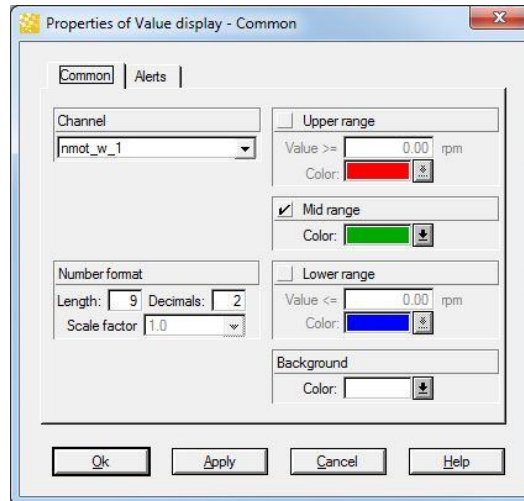
Alerts Tab

This tab is used to set parameters of the alerts and events. An alert is raised (flashing channel name in the color of the range) if the current value of a channel satisfies the condition in a range and if it is checked in this tab. If you check an alert you can specify a hysteresis for it. The alert will stop if the value reaches the range limit minus hysteresis. In this tab you can also specify alert notifications time.
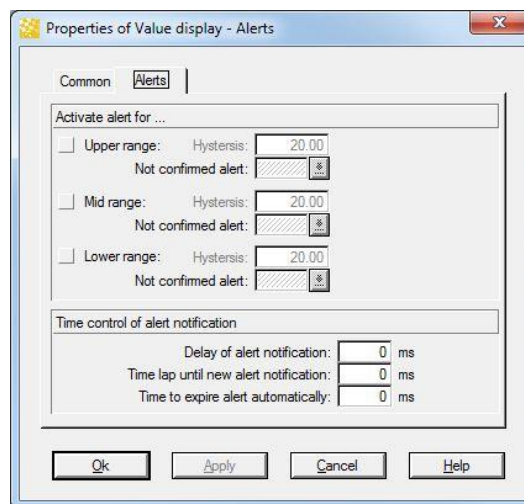
An Event is also raised and displayed in the **Event Window** (see Events).

Bar display

Displays the current channel value as a bar. You can see a current value as a number and in the same time as a bar. Additionally you can select and see whole range, upper, middle and lower sub ranges. You can also see the Min/Max of the previous lap. There are small arrows at right side which shows the min and max of the channel for the whole online session. If the arrow is red then this value is measured by the car else it is black and evaluated from Darab. In the middle of the bar there are two signs that shows the local min and max peaks.

Properties
F5
Double
Click
If the object is active (indicated by a blue frame) you can open and modify its properties. You can open the dialog box using the context menu by pressing the right mouse button.

A dialog box with three tabs appears:

Common Tab

This tab is used to select channel, its ranges and number format. You can divide the channel range up to 3 sub ranges upper, middle and lower. You can choose a color for any sub range. Scale factor is used to show or hide digits from numbers of the scale. You can also select the display value range for the channel.
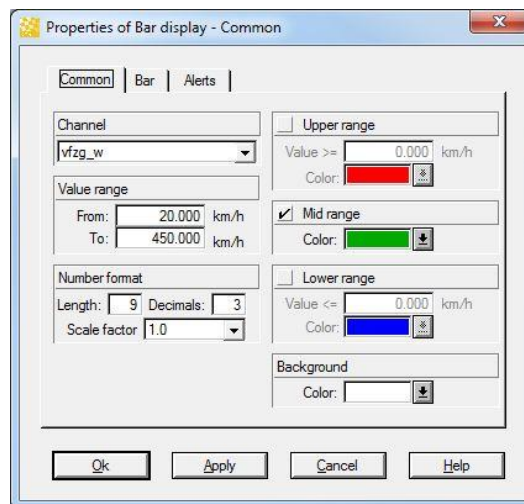
Bar Tab

This tab is used to show/hide peaks and the shape of the color. If you set a peak you can specify how long the peak should be held if a new peak does not appear in this time. After this time the peak will be cleared. Instead of a color you can select the appearance of the bar. The option stacked bar will show the bar in several colors depending on sub ranges and current value. Solid bar will fill the whole bar in the color of the current range. If the channel has negative values then the bar can be drawn always starting at the current value or starting at physical value 0.
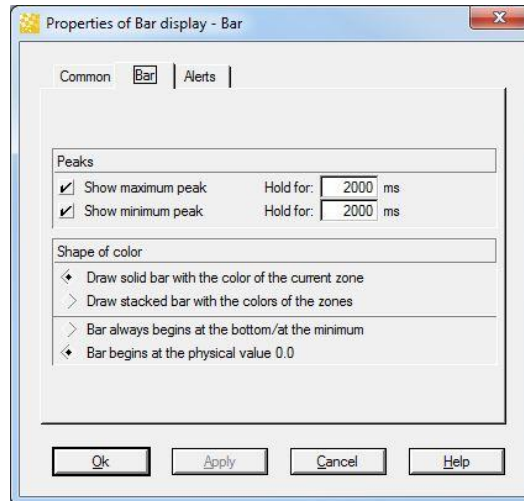
Alerts Tab

This tab is used to set parameters of the alerts. An alert is raised (flashing channel name in the color of the range) if the current value of a channel satisfies the condition in a range. If you check an alert you can specify a hysteresis for it. The alert will stop if the value reaches the range limit minus hysteresis. In this tab you can also specify alert notifications time.
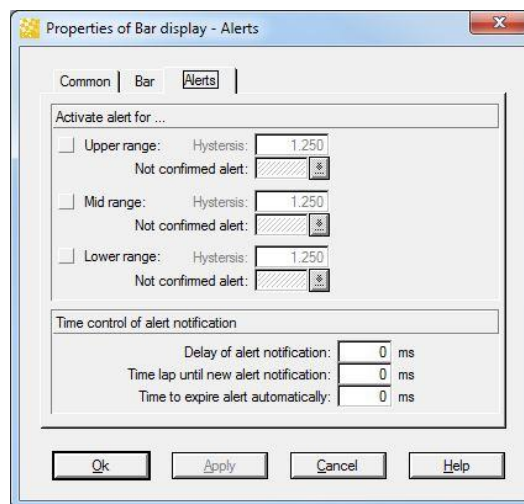
Round scale display

Displays the current channel value as a channel-meter. This display shows the data as real life devices. The display is a circle with arc scale and there is an arrow which shows the current value. You can see the current value as a number too. You can choose from eight different styles: two different radius for the circles (big and small) and four scale layouts (270 degree or 90 degree left or right sided).

Here are some examples of round scales:
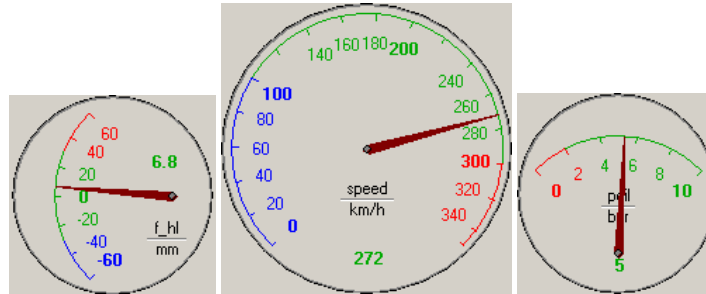
Properties
F5
Double
Click

If the object is active (indicated by a blue frame) you can open and modify its properties. You can open the dialog box using the context menu by pressing right mouse button.

A dialog box with two tabs appears:

Common Tab

This tab is used to select channels, its ranges and number format. You can divide the channel range in three sub ranges upper, middle and lower. You can choose a color for any sub range. Scale factor is used to show or hide digits from numbers of the scale. You can also select the display range for the channel and the background color for the item.
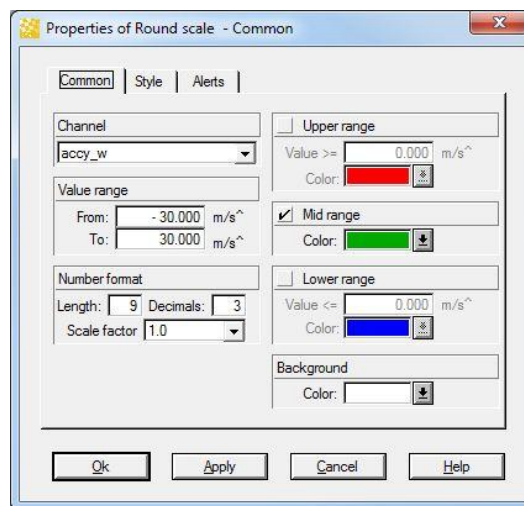
Style Tab

This tab is used to select the size and the arc of the meter. You can choose from eight different styles: two different radiuses for the circles (big and small) and four scale layouts (270 degree or 90 degree, left or right sided).
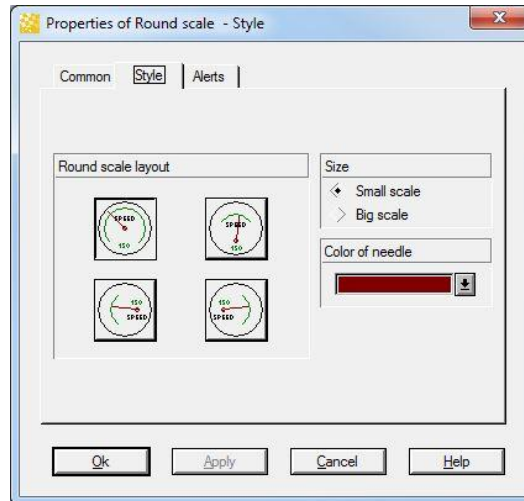
Alerts Tab

This tab is used to set parameters of the alerts. An alert is raised (flashing channel name in the color of the range) if the current value of a channel satisfies the condition in a range. If you check an alert you can specify a hysteresis for it. The alert will stop if the value reaches the range limit minus hysteresis. In this tab you can also specify alert notifications times.
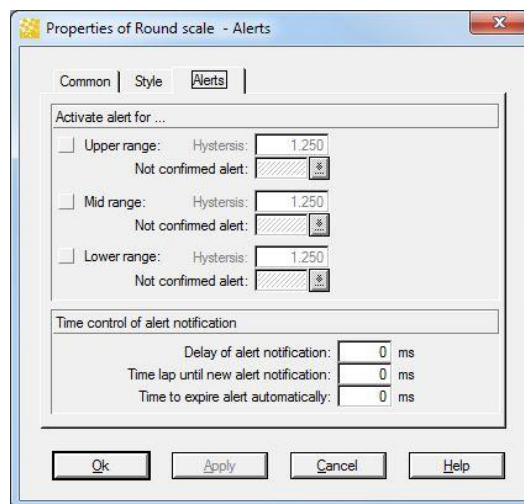
Bit panel

A bit panel allows the display of each bit of a channel. This display shows the channel name and the current value of each bit of a channel.
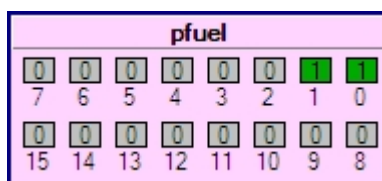
In addition each bit can be displayed in two different colors depending on the bit state ("0" or "1").

| Properties F5 Double Click	If the object is active (indicated by a blue frame) you can open and modify its properties. You can open the dialog box using the context menu with pressing right mouse button.
	A dialog box with two tabs appears:

Common Tab

This tab is used to select the channel, the bit panel layout and the colors.

- **Channel** – select the channel which bits should be displayed
- **Bit panel layout**
  - **Bit Count and Bitnumber-Format** – determine the number of bits displayed on the panel und the bit number format. The user can select hexadecimal or decimal format.
  - **Bits per row and Bit counting** – determine the number of bits per row and if bit counting is 0- or 1-based
- **Colors** – select the color of bit state ("0" and "1"), panel background, active alert and unacknowledged recent alert.
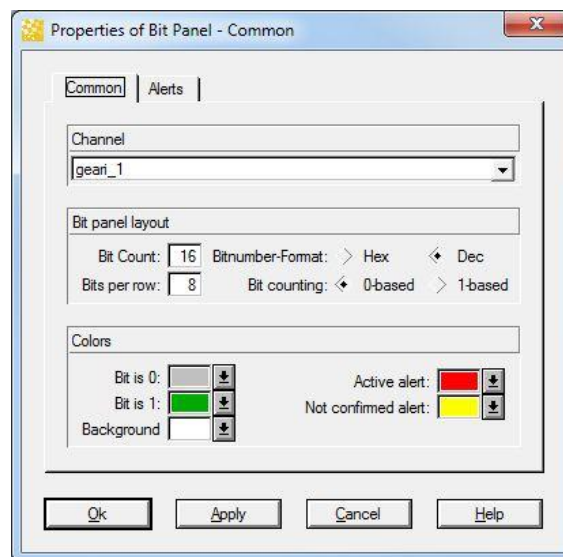
Alerts Tab

This tab is used to set the parameters of alerts. An alert is raised if the current value of a channel satisfies the condition defined in this tab.

The user can define an alert condition for each bit of the channel. By default the value of each bit is masked. Through double click the bit value by which an alert will be reported can be defined.

If an alert is active the channel name and the bits raised the alert will be flashed in the colors select by **Properties of bit panel - Common / Active alert**.

In this tab you can also specify alert notifications times.
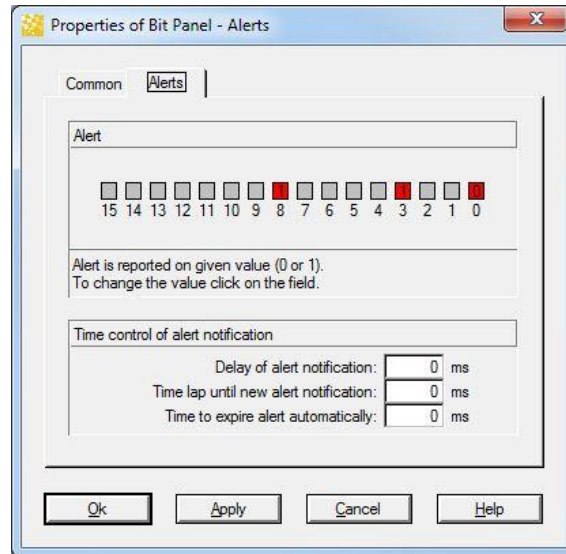
Current lap time

Displays the current lap time. The color can be customized.

Properties	Active object properties can be modified. You can open the dialog box by using the context
F5	menu.

A dialog box appears:

Common Tab

This tab is used to select the color of the current lap time.

Previous lap time

Displays the previous lap time. The color can be customized.

Properties	Active object properties can be modified. You can open the dialog box by using the context
F5	menu.

A dialog box with appears:

Common Tab

This tab is used to select the color of the previous lap time.

Settings

There is no dialog box for telemetry settings. The settings of telemetry monitoring can be adjusted by the applied displays and their properties.

Load Settings

You can access pre-defined settings by selecting "**Start/ Settings/ Load**" from the menu, by using the context menu with right mouse button or by pressing the **F6-key**.

1. Select "Start/ Settings/ Load" from the menu.
2. File Explorer will appear.
3. Select the desired settings file and confirm your selection by choosing "Open".

Save Settings

You can save your current settings, by selecting "**Start/ Settings/ Save**" or by pressing **F7-key**.

1. Select "**Start/ Settings/ Save**" from the menu.
2. File Explorer appears.
3. Select the file in which you want to save the settings or enter a name.
4. Confirm your selection by choosing "Save".

Add Settings

Instrument panel/ Add Settings You can union different settings by adding them one after another. To add settings select "**Add settings**" from local menu
Ctrl+F6

Wrong Settings

If for some reason a channel in a display object is unavailable then it will be red-crossed.
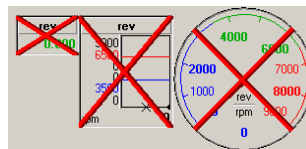
You can combine the telemetry window (for example with round scales of speed and revolution channels) with the racetrack's driver's view.

Events

This window displays all events of the current online session. Telemetry events are set from a car in alert state or from a display object of Telemetry Monitoring (Value Display, Bar Display, and Round Scale Display).

Time	Car	From	To	Duration	Channel	Min	Max
14:27:25	Out04_Lap-	76.563	77.146	0.563	vfzg_w	200.215	200.5
14:27:32	Out04_Lap-	80.266	80.826	0.561	vfzg_w	98.420	100.3
14:27:38	Out04_Lap-	85.382	88.440	3.058	vfzg_w	200.931	240.2
14:27:46	Out04_Lap-	94.117	97.987	3.869	vfzg_w	65.770	101.2
14:27:53	Out04_Lap-	101.35	111.57	10.218	vfzg_w	200.913	296.5
14:28:10	Out04_Lap-	117.97	122.55	4.585	vfzg_w	199.961	256.7
14:28:19	Out04_Lap-	126.61	127.78	1.169	vfzg_w	97.003	99.44
14:28:22	Out04_Lap-	129.46	132.77	3.307	vfzg_w	73.575	100.1
14:28:32	Out04_Lap-	139.63	164.35	24.713	vfzg_w	198.875	321.0
14:28:59	Out04_Lap-	167.28	171.09	3.802	vfzg_w	200.260	246.9
14:29:12	Out04_Lap-	179.82	180.51	0.687	vfzg_w	200.304	207.4
14:29:16	Out04_Lap-	183.44	184.44	1.002	vfzg_w	96.002	100.9
14:29:21	Out04_Lap-	188.56	191.80	3.240	vfzg_w	200.576	241.2
14:29:29	Out04_Lap-	197.36	201.22	3.868	vfzg_w	67.676	101.2
14:29:37	Out04_Lap-	204.60	214.64	10.046	vfzg_w	200.055	296.2

- **Time** - shows the system time of the error.
- **Car** – which car (online file) sent the event
- **From** - is a value from the time channel of online file when the error started
- **To** - is a value from the time channel of online file when the error ended.
- **Duration** - is the time in which the error was active.
- **Channel** – is the name of the channel which causes the event.
- **Min** - minimum of the channel between the start and end of the event.
- **Max** - maximum of the channel between the start and end of the event.

> If the event is an error bit instead of channel, min and max you will see the error bit description.

Settings

It is possible to modify the way WinDarab by a number of different settings.

To modify options select "**Tools/ Program**" from the menu. The "Options" dialog box with the following tabs will appear:

- Colors – Configure color settings for the on-screen display.
- Measurement window – Configure settings that affect the display of channel signal curves in the Oscilloscope.
- Miscellaneous – Automate certain WinDarab operations and dialog boxes.
- Comments – Define additional comment fields.
- Physical Units – Edit free conversation rules to transform the physical values of Bosch to its own units.
- Special channels (see Special channels).
- Telemetry – Define if WinDarab is activated in case of on alert.
- Import defaults – Configure the default settings for importing data.

Color Tab

In the Color Tab you are able to configure color settings for the on-screen display.
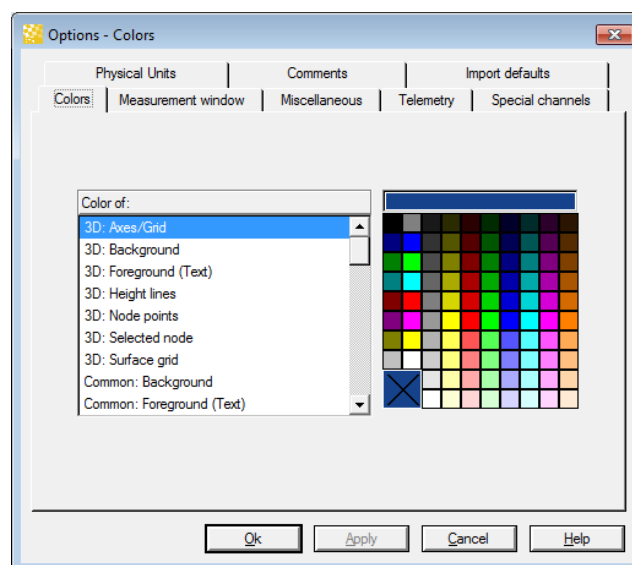
The list includes all WinDarab elements for which you can specify a particular display color.

Choose the color you would. The currently selected color is marked with an x.

The large color square on the lower left has a specific function:

- It indicates the default colors for some elements. For instance: in "General: Background" it displays the color the system assigns to the background in dialog boxes; in "Outing Report: Background" it indicates the current selection for the general setting "General: Background".
- Selecting the shaded square (= transparent) "switches off" the display for some elements.

Confirm your changes by clicking on either "Apply" or "OK", or by selecting another tab. All windows in WinDarab will be displayed with the new color settings.

Measurement windows

The Display Tab is used for settings that affect the display of channel signal curves in the Oscilloscopes.
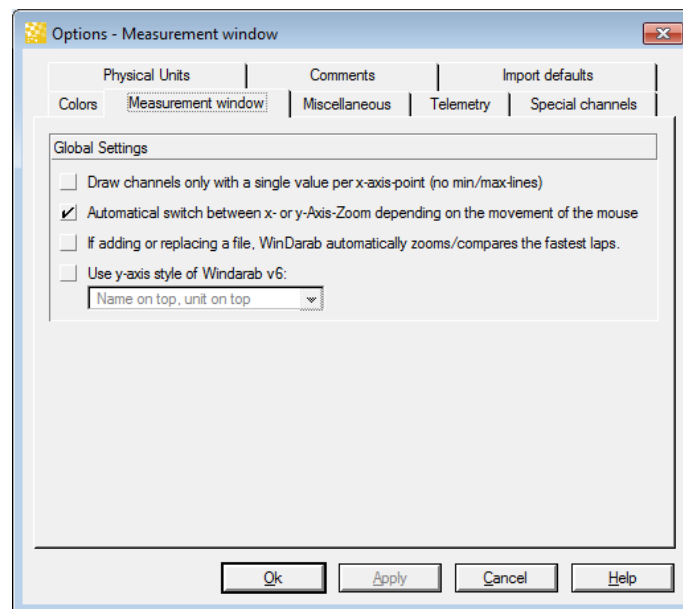
Draw Channels only with a single value per x-axis-point…

If you activate this button WinDarab produces a simplified display of the signal curve.

In a standard display WinDarab determines extreme signal values for each screen pixel along the x-axis; and if necessary it generates a vertical line in order to make this value range clearly visible. As a result the display of channels with extreme signal fluctuations (digital channels, for instance) will produce color areas that obscure other signal curves.

In the simplified display WinDarab determines, by spot sampling a single signal value for each screen pixel along the x-axis. These signal values are then used to render the signal curve.

Situations may occur where not all anomalies (e.g. outliers or digital triggers) of a signal curve can be shown in the simplified display.

Automatically switch between x-or y-Axis-Zoom…

This option allows you to switch the direction of the zoom in the measurement window while zooming. So if you start with a y-axis-zoom and move to the right, it becomes an x-axis zoom.

Without this option set it would still be an y-axis-zoom.

Changing the y-axis view

Use the Windarab v6 y-axis view
With this option, you can change the new y-axis-view with the name and unit on the side back to the v6-design with the name and unit on top/bottom of the axis.
When this option is selected, you can chose globally the position of the name and unit from the dropdown-menu.

Miscellaneous Tab

The Miscellaneous Tab allows you to automate certain WinDarab operations and dialog boxes.
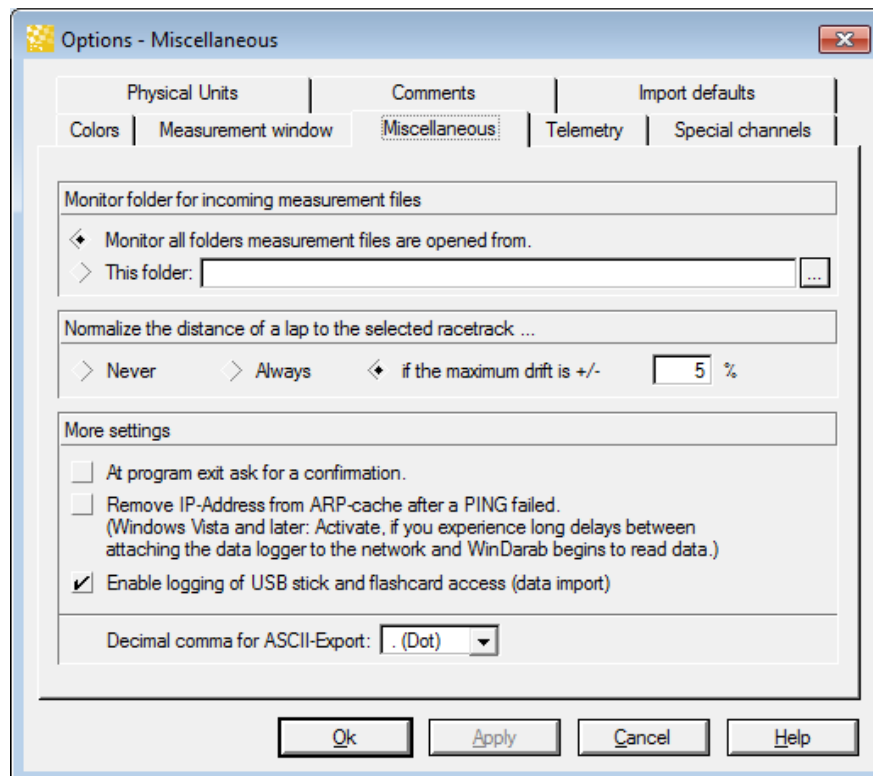
Use this tab to make the following settings:

Monitor folder

Chose the folder where Darab should look out for new darab-files or select the option to monitor every folder with opened files in it. (see File System Monitoring)

Normalization Options

Chose which laps should be normalized in Darab.

At program exit ask for a confirmation

Before you exit WinDarab, you will be asked to confirm the changes or selections you may have made.

Decimal comma for ASCII-Export

You can chose, whether your decimal separator for the ASCII-Export should be a comma or a point.

Comments Tab

In addition to the comment text field provided in WinDarab the program allows you to define further comment fields. These additional fields allow you to enter structured information about a particular file.

You are able to enter data into the fields you have defined during data analysis. It is also possible, however, to pre-define settings for these fields in such a way that information will be entered automatically during data transfer.

Comment fields can be used in mathematical channels as outing-specific constants.

Select the "**Tools/ Settings/ Program/ Comments**" tab if you want to define additional comment fields.
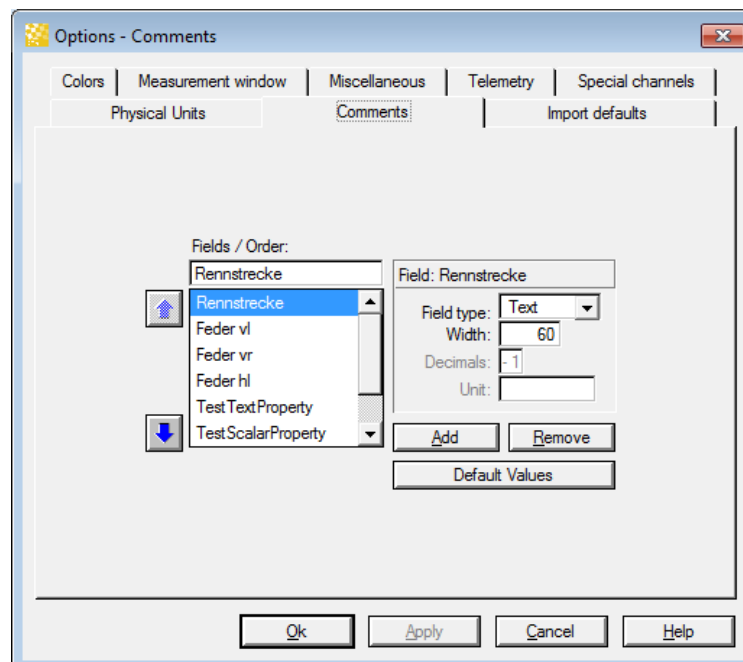
Adding a Comment Field

Follow these steps to add a comment field:

1. Enter the name of the new comment field in the "Fields/ Sequence" field.
2. In the "Field Type" field, select which type of information you want to save here.
3. Specify the width of the field in the "Width" field.
4. If you have selected "Number", you must also specify the number of decimal places and the unit of measurement.
5. Click on the "Add" button.
6. The new field will be added to the bottom of the list.

Deleting a Comment Field

Use these steps to delete a comment field:

1. Enter the name of the comment field you want to delete in the "Fields/ Sequence" field. Alternatively select the comment field from the list.
2. Click on the "Delete" button.
3. The comment field will now be deleted from the list.

> **Note:**
> Comments that have already been saved in existing Logged Data Files will not be deleted. The comment fields you have just deleted will not be included in new files you create.

Determining the Sequence of Comment Fields

You can determine the sequence in which WinDarab displays pre-defined comment fields in a Logged Data File:

Follow these steps to modify the sequence of comment fields:

1. Select the comment field which you want to move to another position in the list.
2. Click on the "Up" or "Down" arrow button to move the field up or down to the desired position.
3. Repeat this step until the fields appear in the proper sequence.

Setting Default Values

You can set the default values for the comment fields you have defined that will automatically assigned to each Logged Data File during data transfer:

1. Click on the "Default Values" button
2. A dialog box with the defined comment fields will appear.
3. Change the default values as desired.
4. Click on "OK".
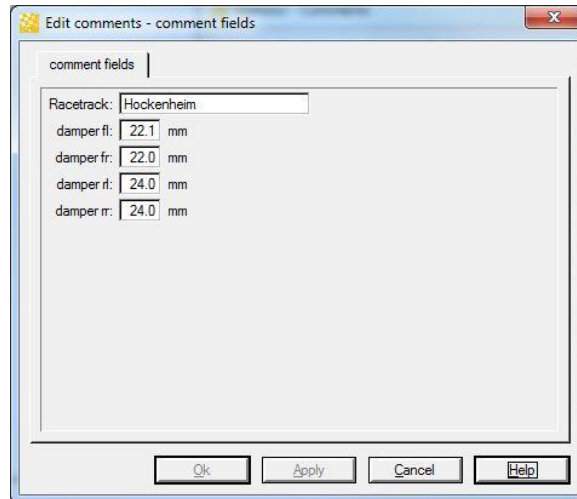
Physical Units Tab

The Physical Units tab allows you to choose/remove or add unit sets to customize your data.
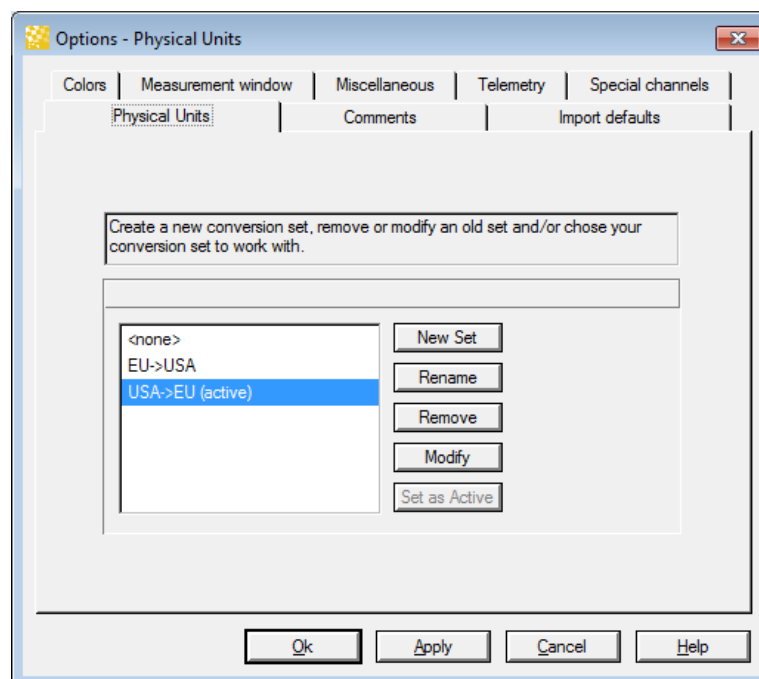
In the box on the left side, you see all of your current possible conversion sets to choose from. The **<none>** - conversion set can be selected to have no conversion set active.

With the "**Set as active**"-button, the currently selected set becomes active and all its conversions are applied to the according channels.

You can rename or remove a conversion set by pressing "**Rename**" or "**Remove**".

Removing the active conversion set automatically sets the **<none>** - conversion set as active. The **<none>** - conversion set itself can't be removed, renamed or modified.

With the "**New Set**"- and "**Modify**" – buttons, you can create a new set or modify the currently selected set. This happens in a new Dialog.

Rules-Page
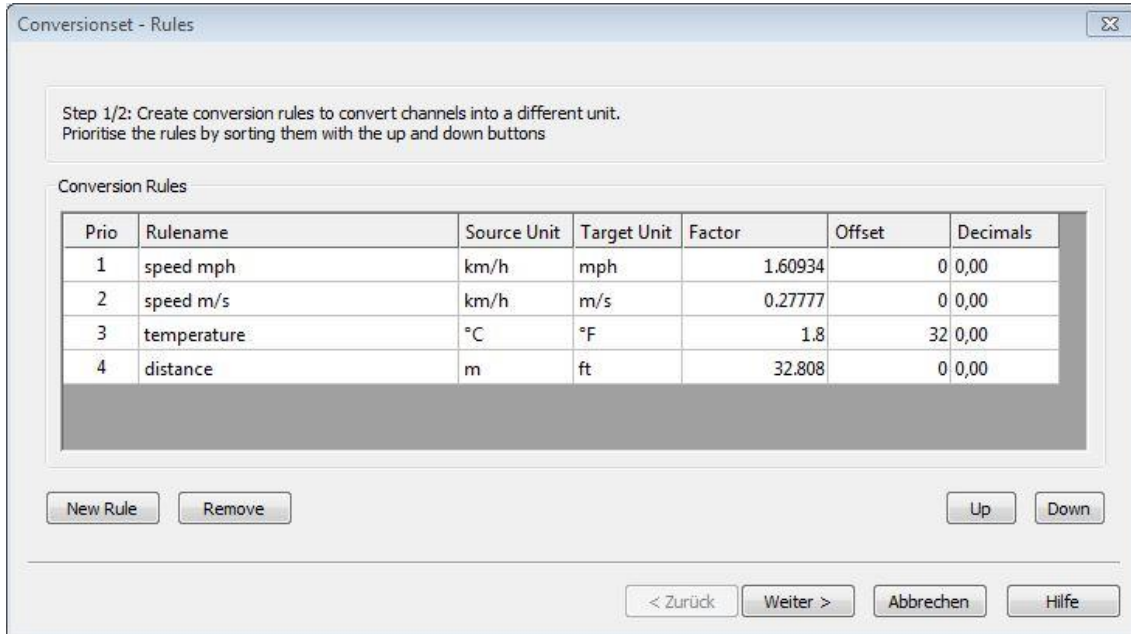
On the first page of the wizard you see all the rules that are contained by the conversion set.

Every rule has a **Priority**, a **Name**, a **source unit**, a **target unit**, a **factor**, an **offset** and a specified **number of decimals**.

The **priority** is used to define the default-converter for the channels. If there are two or more converters suitable for one channel, the one with the higher priority is applied.

The **source-** and **target unit** define what units can be converted in which destination units.

The **factor** and the **offset** define the formula for the conversion. The formula is:

Destination Value = Source value * factor + offset.

The **decimals** only specify the format in which the destination value is displayed.

You can add or remove rules by pressing the "**New Rule**" or "**Remove**" button.

Note:

The priority can be changed by pressing the "**up**" or "**down**" button. It also can be changed by *dragging* the selected row onto a different row.

Details-Page
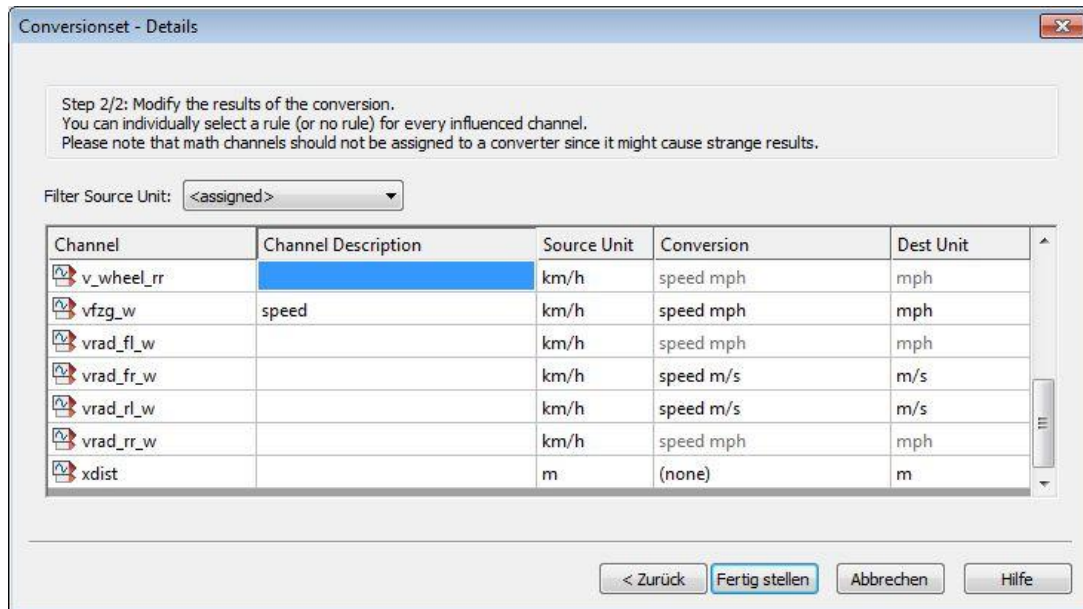
The next page is the details page. On this page you see exactly what changes will apply when you use this conversion set. The table shows every channel, its **description** and **source unit**. If the channel has a **conversion-rule** that can be applied to it the table shows the currently assigned conversion rule and the **destination unit** in which the channel is converted.

You can *filter* the channels by selecting the filter from the **filter-combobox**. Aside from every possible source unit, you can select **<none>** for no filter and **<assigned>** to only show the channels that have a possible conversion rule that can be applied.

If there is a conversion rule matching the source-unit of a channel, you can chose the conversion rule for this channel. Initially the default rule is applied to the channel. By *clicking* on the **conversion-rule name** in the conversion-column a popup menu is shown with all possible selections for this channel.

(none) is selected if no conversion rule should be applied to this channel. The channel won't be influenced by this conversion set.

(default) is selected to use the default converter. In the table, the current default conversion rule for this channel will be shown.

Select a named conversion-rule if you want to use a specific rule for this channel.

Note:

Selecting (default) or selecting the current default conversion rule has initially the same result. The difference between the two possibilities is that if you change the priority of the rules, the default conversion rule for a source unit (for example km/h) can change while the explicitly assigned rule stays the same.

Telemetry

On the Telemetry tab all options concerning the viewing of online/telemetry data can be found.
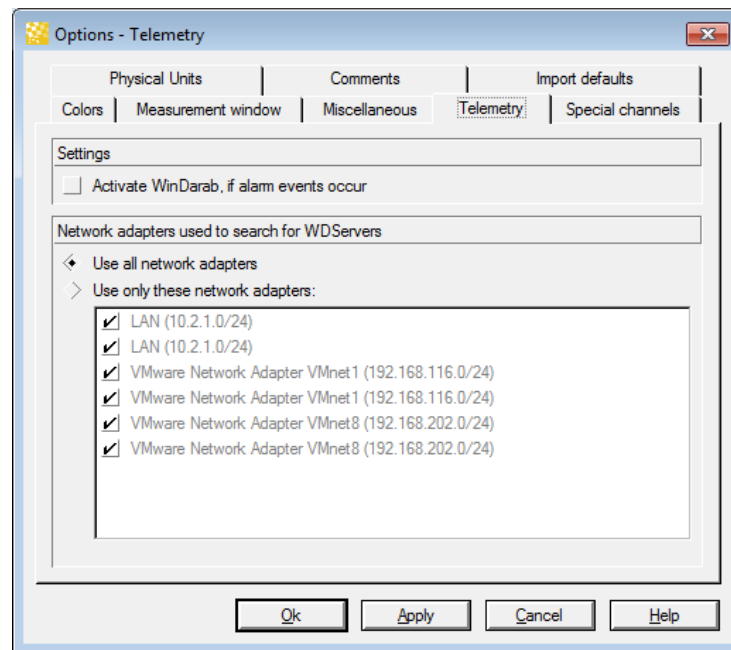
Option "Activate WinDarab …"

If this option is enabled, WinDarab will pop into the foreground, if an alert event occurs.
E.g. alert events are triggered by the gauges on an instrument panel.

Options "Network adapters used …"

In this section Network adapters can excluded from the use for telemetry/online data connections to WDServers. WinDarab will only send WDServer discovery packets on the selected network adapters.

By default WinDarab uses all connected network adapters. Only in certain cases you should exclude adapters:

- Prevent WinDarab from sending packets on network adapters to reduce the "noice" (e.g. if the network adapter is connected to the CAR LAN)

- Prevent WinDarab from sending packets on other adapters like VPN/RAS adapters – behind these adapters there shouldn't be a WDServer located.

Note:

If a new adapter is added to the system, the adapter will be added "Enabled" to the list.

Import default

The import default Tab allows you to configure different default settings for the import from your devices.
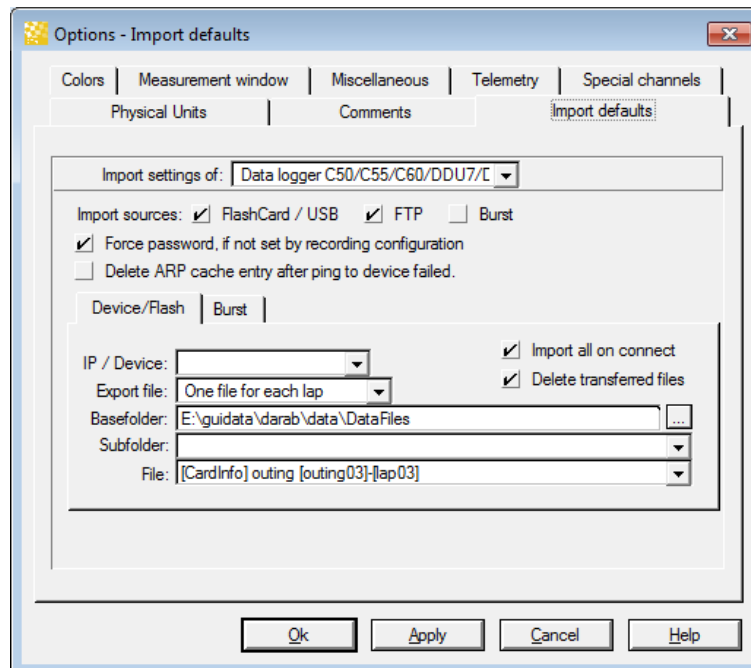
Channel Settings

You are able to make basic settings for each channel in WinDarab.
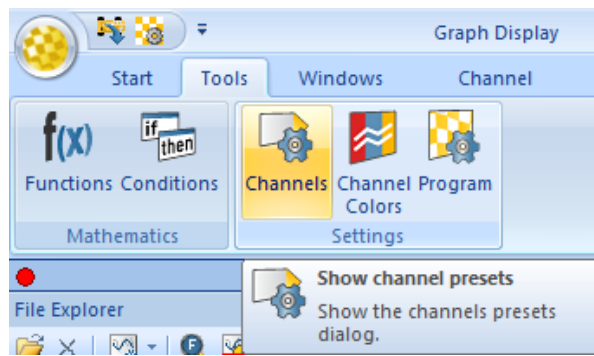
For each individual channel following settings are provided:

- Description
- Format (numerical)
- View Min
- View Max
- Signal (signal curve format)
- Separate Bits

Making Channel Settings

Select "**Tools/ Settings/ Channels**" in **Darab Ribbon** to open the "Channel Settings" dialog box.
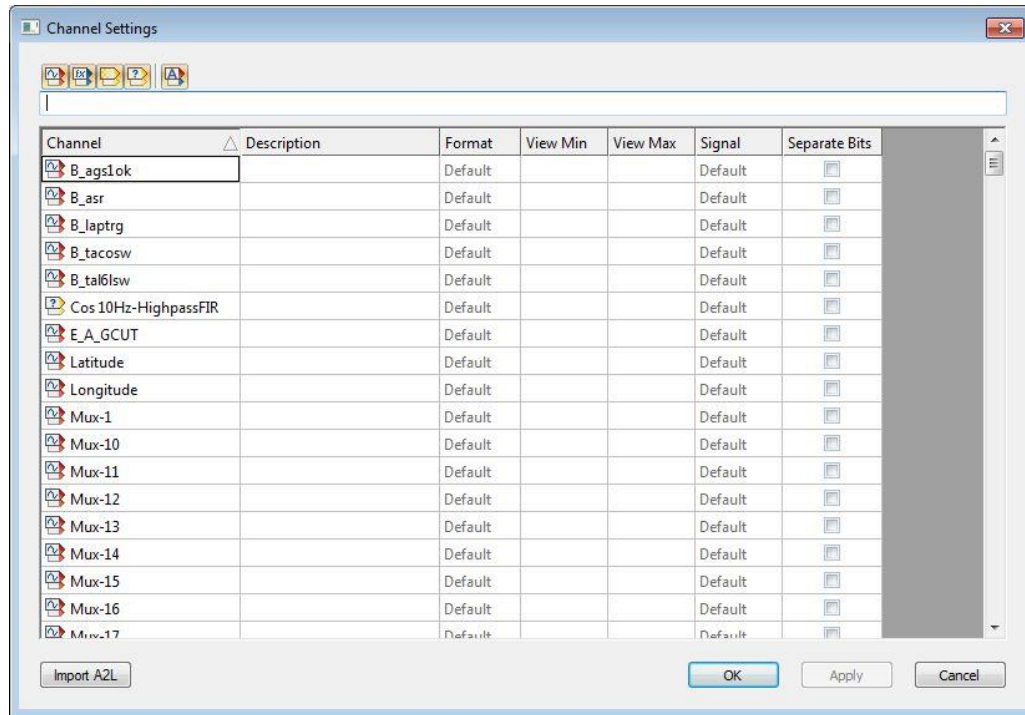
The Channels are divided into four channel list:

- Measurements channels
- Math channels
- Internal channels
- Temporary channels
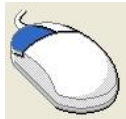
Click on **Show measurements** button to *select and display* the **Measurements channels** list
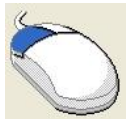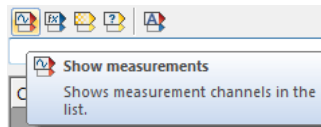
Click on **Show math channels** button to *select and display* the **Math channels** list
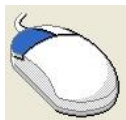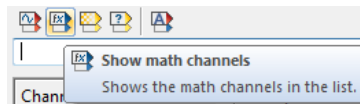
Click on **Show internal channels** button to *select and display* the **Internal channels** list
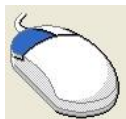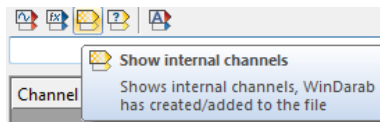
Click on **Show internal channels** button to *select and display* the **Internal channels** list

---
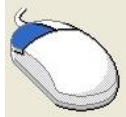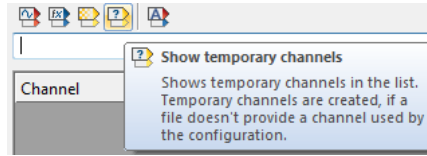
Click on **Show/Hide all channels** button to *(de-)select and display/hide* all channels
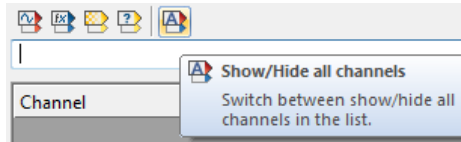
For each channel in the lists the following settings can be defined:

Description   A description for the channel can be entered in the text field.

Format   Define the display format of numbers.
The following formats are provided in the **Format** combo box of each channel:
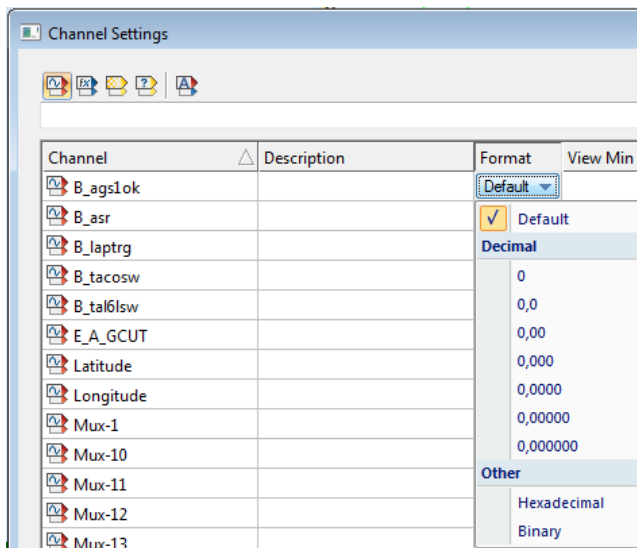
View Min   You can enter the value displayed as minimum for the channel.

View Min   You can enter the value displayed as maximum for the channel.

Signal   Determine the format for display of the signal curve.
The following formats are provided in the **Signal** combo box of each channel:
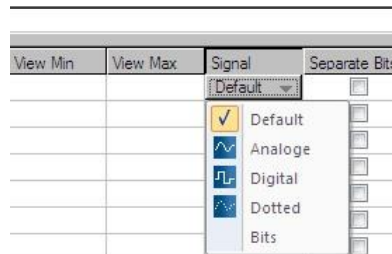
- **Analog** - values are interpolated.
- **Digital** - sampling points are represented by square wave signals
- **Dotted** - each sampling point is represented by a dot

- **Bits** - each channel bit is interpreted as a separate signal and represented by a square wave signal

Separate Bit   With this option selected, single bit channels will be created for each bit of the measured channel. If selected each single bit will be shown as a channel.

You can now select each one of these bit-channels individually in the channels pane.

Importing Descriptions

WinDarab enables you to import description for channels from a (named) object store.

Follow these steps to import a (named) object store:

1. Open the "Channel Settings" box by using the "Tools/ Settings/ Channels" menu item.
2. Click on "Import OSP/A2L".
3. Now select the (named) object store that contains the descriptions and click "OK".
4. The descriptions will now be imported.
5. Click on "OK" to close the "Channel Settings" dialog box.

Channel Colors

Select "**Tools/ Settings/ Program**" to open the Channel Colors Dialog
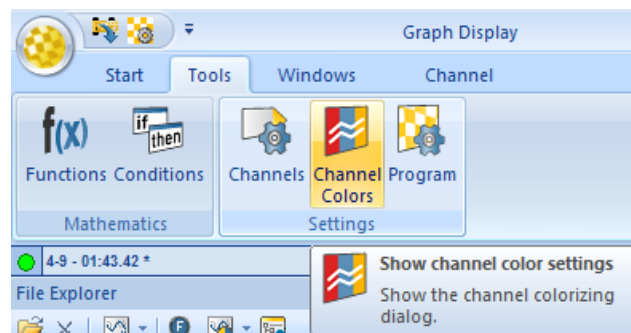
This dialog allows you to assign different colors for different overlays to specific channels. The number of overlay-columns in the dialog equals the number of overlays in the File Explorer.
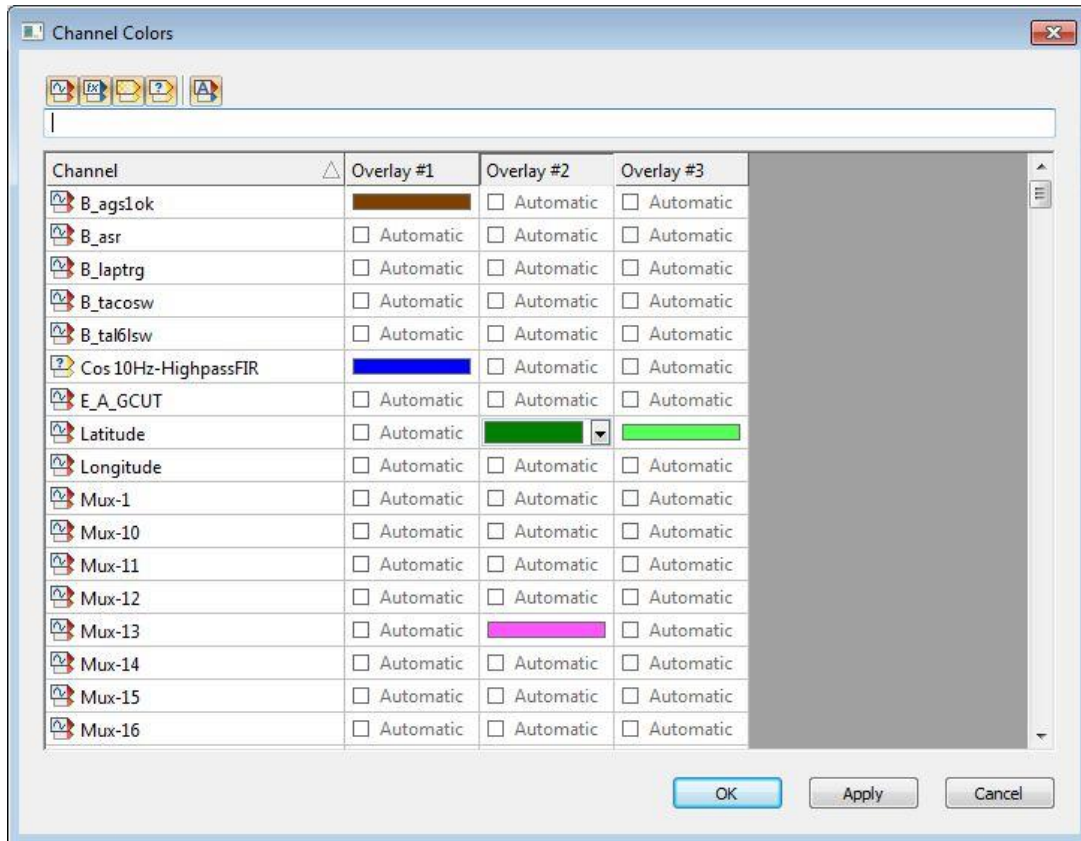
The assigned colors will automacally selected when you use this specific channel in the oscilloscope.

Miscellaneous

Extracting Logged Data

WinDarab provides a number of different functions to extract data from a logged data source file.

Extracting a File Section

Depending on the hardware you are using Logged Data Files can grow to an enormous dimension. For your work, however, you may only be interested in a certain data segment.

Therefore it is a good idea to extract excerpts of the file. You can then archive your large Logged Data File using an external data carrier such as a CD-ROM. You might even want to delete the source files altogether.
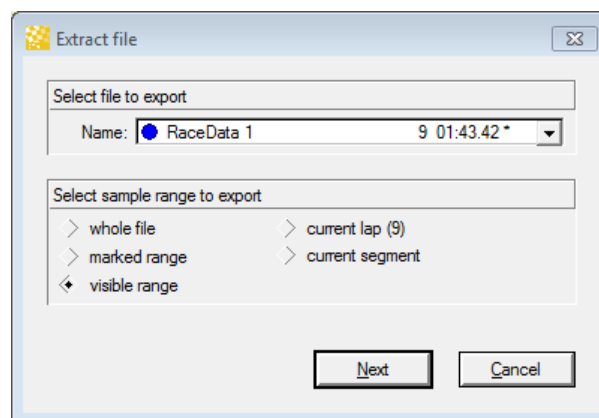
To extract a file section follow these steps:

1. Adjust the section you want to extract from the Oscilloscope.
2. Select "**WinDarab-Button / Import/Export/ Export into new measurement file**" from the menu.
3. Select the file from which you want to extract the section. Only files used in the current Oscilloscope are shown.
4. Select the area you want to extract.
5. Click on "OK".
6. The extracted section will now be stored in a temporary file.
7. Once that process is complete the dialog box "Save Logged Data File" appears.
8. Saving completes the extraction process.

Note:
The extracted file section contains all channels and information contained in the original source file.

If you want to protect your file with a password, go to [Creating a Password-protected File](#) for further information.

ASCII Extract (Logged Data)

WinDarab provides the ASCII Extract function in order to enable you to work with your data in other applications.
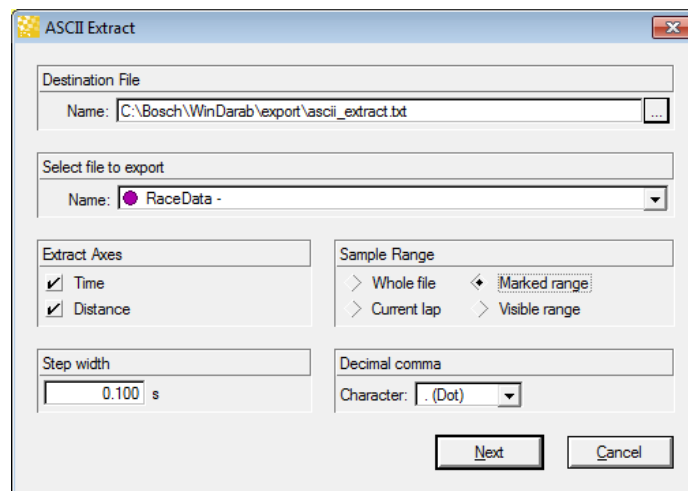
To create an ASCII Extract file follow these steps:

1. Adjust the section you want to extract from the Oscilloscope.
2. Select "**WinDarab-Button/ Import/Export/ Export into text file**" from the menu.
3. Enter the name of the ASCII file to which the data will be written. You can use the "Search" button to find a file.
4. Select the Logged Data File from which you want to extract the section. Only files used in the current Oscilloscope will be displayed.
5. Select the time and/or distance button(s) if you want WinDarab to extract distance and/or time information as well.
6. Enter the size of extraction intervals. The specified distance always refers to the x-axis in the affiliated Oscilloscope.
7. Select which area you want to extract.
8. Click on "OK".
9. The file extract will now be exported to the file you specified earlier.

Note:
When you use the ASCII Extract function to create a file, the file will only contain the channels displayed in the Oscilloscope (and distance/time information if you have selected these options).

The Appendix contains information about the format of the ASCII file that has been created.

ASCII-Extract (Analysis Window)

You can extract the results of some Analysis Windows in much the same way as in an ASCII Extract from a Logged Data File.

Use the following steps to export the results of an Analysis Window:

1. Use the Analysis Window to set up the desired data evaluation.
2. Select "**WinDarab-Button/ Import/Export/ Export into text file**" from the menu.
3. Using the "File" dialog box, select the folder to which you want to save the ASCII file.
4. Enter the file name.
5. Click on "OK".
6. The results will now be exported to the file you have specified.

Note:
The Appendix contains information about the format of the ASCII file that has been created.

Printing

You can generate print previews for and print Logged Data as well as Analysis Windows.

Note:
It could be possible that very fine lines will not be displayed in the print preview. This may be a result of a discrepancy between your screen and printer resolution.
The lines will nonetheless be visible when printed.

If you choose your colors please consider that background colors will not be printed. For example light colors against a black background are easy to see on screen, but your printed output will be difficult to read.

Print Options

Before a print job starts printing, or before the Print Preview appears on your screen, WinDarab will display a dialog box with printing options.

"Layout" Tab
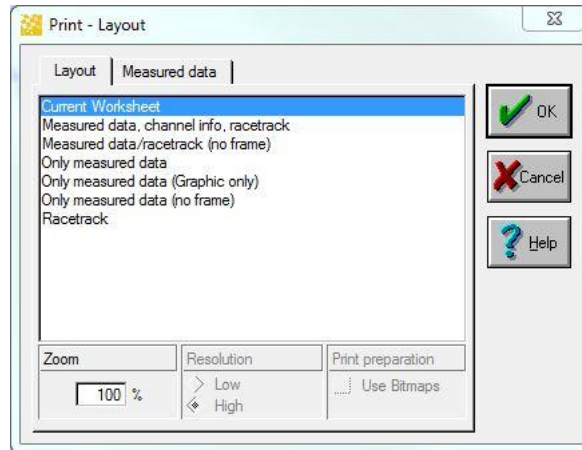
If more than one print layout is available for the window you want to print you can select the desired layout in this tab.

"Logged Data" Tab
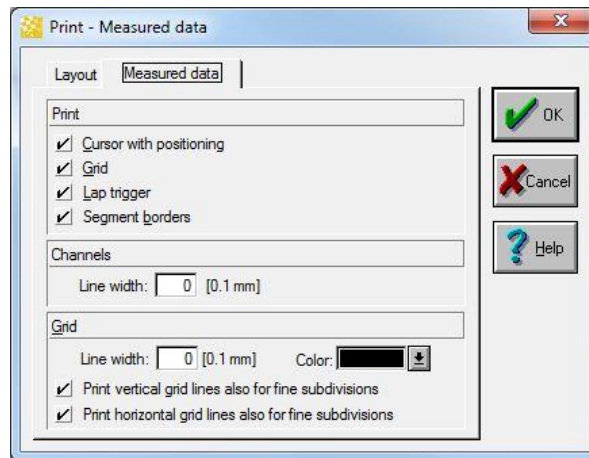
Use this tab to determine the details you want an Oscilloscope to contain for printing.

Print Preview

Use these steps to print a window:

1. Arrange the window you want to print so that the data will be displayed in the desired form.
2. Select "**WinDarab-Button/ Print/ Print Preview**" from the menu.
3. A selection of available templates for printing windows now appears. Choose the template you want.
4. The "Print Preview" window will now be displayed.
5. Use this window to

   - set a zoom factor (10 - 400%)
   - print a template
   - close the window

Print

Follow these steps to print a window directly:

Arrange the window you want to print so that the data is displayed in the desired form.

Select "**WinDarab-Button/ Print/ Print**" from the menu.

A selection of available templates may now appear. Choose the template you want.

The Windows "Print" dialog box appears next.

Click on "OK".

The template you have chosen will now be printed.

> **Note:**
> The Windows "Print" dialog box will only permit you to select your printer; all other settings have no effect.

Command line

Following command line arguments may be passed to WinDarab on start up.

-d <workdesk file>
Opens WinDarab using the given workdesk file. If no additional data files are passed the saved workdesk is completely restored – including the used files and zoom.

> If the argument is not used WinDarab loads the workdesk which was saved by WinDarab during shutdown of the application.

-f <filename> [-f <filename> …]
Use the –f argument to pass one (or more) data files which are opened instead of the files within the workdesk.

In front of each file name the argument –f has to be provided!

> **Note:**
> Don't forget to put filenames inside of two quotes ("<path/filename>") if the path or filename contains space characters.
> You may use quotes at any time to prevent misinterpreting the command line arguments.

Historical cars

Creating a Logged Data File from DOS Format

Because WinDarab cannot directly use Logged Data Files created by DOS Darab, they must be converted to the format used by WinDarab.

Follow these steps to convert a Logged Data File from DOS Darab to WinDarab:

1. Select "WinDarab-Button/ Import/Export/ Import DOS-Darab-files" in menu.
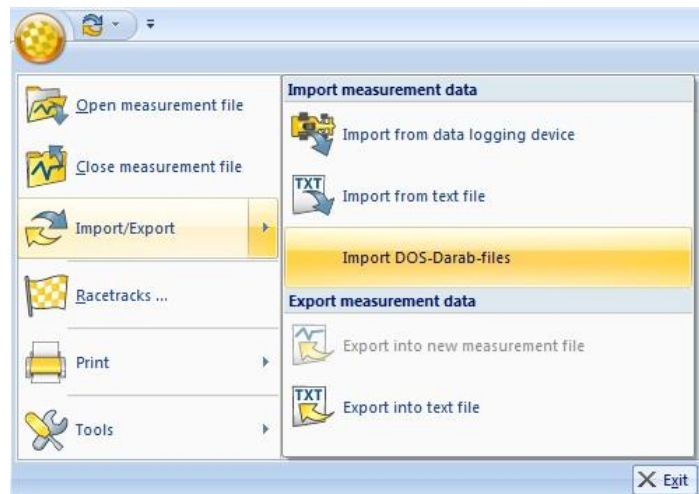
2. "Convert File" dialog box is opened.
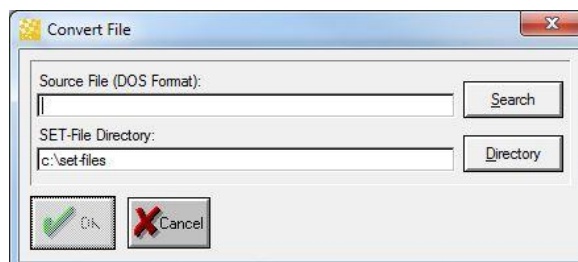
3. Select the Logged Data Files in DOS Darab. If the files do not contain any DCP information, you must also specify the location for the setup files (*.set).
4. Confirm your entries by clicking on "OK".
5. The files will now be converted. A progress bar enables you to monitor the conversion process.

6. The dialog box "Save Logged Data File" appears as soon as the conversion is complete.
7. If more than one split is contained in the source file WinDarab will now convert the next split. Step 5 to 7 are repeated for each split

Reader / CanCard Support

If you use this hardware option the reading process will follow these steps:

- Vehicle data is read with the Reader.
- CanCard is used to connect the Reader to your PC.
- WinDarab reads data from the Reader at the approximate rate of 1 Mbit/s.
- WinDarab converts the read data to WinDarab format.
- Data is available for analysis.

The hardware design limits the volume of recorded data to a maximum of approximately 4-MB per outing.

Reading Data From a Reader

When this dialog box is open WinDarab carries out continuous checks to see if a Reader is connected to the CanCard.

Waiting Status

As long as no reader is connected you have the opportunity to make data transfer and conversion settings:

Base Folder	An already existing folder that indicates the Base Folder to which the read data will be saved. The use of wildcards is not allowed in a Base Folder. If you enter information in the Sub-folder field, your file will be saved in the "Base Folder/ Sub-folder" directory.
Sub-folder	This path/folder indicates the location, within the Base Folder, to which the new file will be saved. The use of wildcards with the exception of "[n]" is permissible here. If the sub-folder does not yet exist WinDarab will automatically create one.
File	File name for the new file you want to create. The use of wildcards is permissible here.
Auto Save	The transferred data is automatically saved in the folder "Base Folder/ Sub-folder", under the name you have specified. The dialog box "Save Logged Data File" appears only if an error occurs while saving.
Auto Replace Split 1	The Logged Data File containing the first split will open automatically and replaces the Logged Data File last opened (see also Replacing a File Globally).
Auto Replace Split 2	The Logged Data File containing the second split will open automatically and replaces the Logged Data File last opened (see also Replacing a File Globally).

Do Not   If this field has been selected the dialog box remain open after data from a reader has been
Close   transferred, converted, and saved.
              WinDarab waits until a new Reader has been connected.

Reading Reader Data

1. WinDarab begins to read data from the Reader as soon as a Reader has been connected to the CanCard.

2. Once the reading process has been completed, WinDarab converts data in the first split area.

3. If you have not activated the "Auto Save" function, the "Save Logged Data File" dialog box appears after data conversion.

4. If you have selected "Auto Replace" for the current split area, a new file opens automatically, replacing the file last opened.

5. If there is more than one split area WinDarab converts data in the next split and step 3 or 4 are repeated.

Important:
Because timing is an extremely critical factor in Reader transfer protocols, you should not run any other complex programs requiring a lot of memory during a data transfer.
Data transfer may otherwise be extremely slow, produce error messages, or even fail.

WinDarab displays an error message if an error occurs during data transfer. After a few seconds the message will disappear and the program will retry reading data from the reader.

Appendix

Mathematical Expressions

The following sections list all operators and functions supported in WinDarab.

WinDarab follows conventional mathematical rules for evaluating expressions.

Constants:

e (Eulers's number)	`e = (1/0!) + (1/1!) + (1/2!) + (1/3!) + ... = 2.7182...`
HOLD	Returns the value of the latest calculation result. HOLD can be used to eliminate/replace invalid measurement values.
pi	pi = 3.1415…

Arithm operators:

* (Multiplication)
+ (Addition)
- (Subtraction)
/ (Division)

^ (<x>^<y>)	Calculates the result of <x> raised to the power of <y>
<X> BitAnd <y>	Calculate the result of the bitwise AND-combination of <x> and <y>.
<X> BitOr <y>	Calculate the result of the bitwise OR-combination of <x> and <y>.
<X> BitXor <y>	Calculate the result of the bitwise XOR-combination of <x> and <y>.

Comparisons:

< (Smaller)
<= (Smaller-Equal)
= (Equal)
<> (Unequal)
> (Greater)
>= (Greater-Equal)

Logical operators

AND (Logical And-Operator)
NOT (Logical Not-Operator)
OR (Logical Or-Operator)

General functions

If (<Condition> ; <IfTrue> ; <IfFalse>)	If the first expression is true, the result of the second expression is returned. Otherwise, the result of third expression is returned.

Math. Functions

Abs(<x>)	Returns the positive absolute value of <x>.
arccos(<x>)	Calculates the radiant angle, the cosine of which equals the parameter <x>.
ArcCosH(<x>)	Calculates the radiant angle, the hyperbolic cosine of which equals the parameter <x>.
ArcSin(<x>)	Calculates the radiant angle, the sine of which equals the parameter <x>.
ArcTan(<x>)	Calculates the radiant angle, the tangent of which equals the parameter <x>.
ArcSinH(<x>)	Calculates the radiant angle, the hyperbolic sine of which equals the parameter <x>.
ArcTanH(<x>)	Calculates the radiant angle, the hyperbolic tangent of which equals the parameter <x>.
ceil(<x>)	Calculates the smallest integer greater-equal to <x>.
Cos(<x>)	Calculates the cosine of the radiant <x>.
CosH(<x>)	Calculates the hyperbolic cosine of the radiant <x>.
Exp(<x>)	Calculates the exponential value e^<x>.
floor(<x>)	Calculates the biggest integer smaller-equal to <x>.
frac(<x>)	Returns only the decimals from <x>.
int(<x>)	Cuts all decimals from <x>.
lb(<x>)	Calculates the base-2 logarithm of <x>.
lg(<x>)	Calculates the base-10 logarithm of <x>.
ln(<x>)	Calculates the natural logarithm of <x>.
round(<x>)	Rounds <x> to the nearest integer value.
sign(<x>)	Returns the sign of <x> (-1 or 1).
Sin(<x>)	Calculates the sinus of the radiant <x>.
SinH(<x>)	Calculates the hyperbolic sine of the radiant<x>.
sqr(<x>)	Calculates the square of <x>.
sqrt(<x>)	Calculates the square root of <x>.
Tan(<x>)	Calculates the tangent of the radiant <x>.
TanH(<x>)	Calculates the hyperbolic tangent of the radiant <x>.
BitInvert(<x>)	Calculates the bitwise Inversion of <x>.
	Important: Due to limitations in precision of math results (because WinDarab is using 32-Bit IEEE-float values) there are problems, if the resulting value is too big. E.g. the result of BitInvert(128) is wrong, while the result of BitInvert(255) is correct. The precision problem occurs only, when saving the math result in memory. While evaluation the expression, there won't be any precision issues. This means that the expression can evaluate a maximum of 32-Bit values, but the final result should be smaller.

Trace functions

Ave	
average(<Channel>;>Samples>)	Calculates the average value of a channel for x(n) within the range n-<Samples> to n+<Samples>.
delta(<Channel>)	Calculates the difference between Channel(x(n)) and Channel(x(n-1))
dn	Calculates the difference Samples(x(n)) and Samples(x(n-1)).
ds	Calculates the difference between xDist(x(n)) and xDist(x(n-1)).
dt	Calculates the difference between xTime(x(n)) and xTime(x(n-1)).
LapAve(<Channel>)	Calculates the average value for a channel within the current lap.
LapMax(<Channel>)	Calculates the maximum value of a channel within the current lap.
LapMin(<Channel>)	Calculates the minimum value of a channel within the current lap.
Lookup(<TableName>;<param1>;<param2>;…)	The lookup table is called with the given parameters which specify the dimensional values.
Max(<Channel>)	Calculates the maximum value of a channel.
Min(<Channel>)	Calculates the minimum value of a channel.
sigma(<Channel>)	Calculates the sum of Channel(x(0)) to Channel(x(n)). The optional second parameter defines a reset condition that - when occurring - resets the sum and starts from zero again.
LapSigma(<Channel>)	Same as Sigma but with a lap change set as the reset condition.
ValueAtSample(<Channel>;<Sample>)	Returns the value of the channel at the given sample position.

IIR Filter Functions

WinDarab provides seven filter functions:

- bandpass
- bandpassFIR
- lowpass
- lowpassFIR
- highpass
- highpassFIR
- stopband

WinDarab supports a number of different approximation techniques and frequencies for each of these filters.

Tip:
Using the WinDarab Filter Assistant simplifies the assembly of filter functions considerably.

bandpass

Syntax:
bandpass (<Technique>;<Order>;<Frequency Range>;<Expression>)

Parameters:

<Technique>	<Order>	<Frequency Range>
Butterworth	1 / 2 / 3 / 4 / 5 / 6 / 7 / 8 / 9 / 10 / 11 / 12	50:100 / 50:200 / 100:200 / 150:200

bandpassFIR

Syntax:
bandpassFIR(<Order>, <lowerCutoff>, <upperCutoff>, <Channel>)

The channel is filtered using a FIR-bandpass-filter of the given order (>=4th order) and a cutoff frequency [Hz].

lowpass

Syntax:
lowpass (<Technique>;<Order>;<Frequency>;<Expression>)

Parameters:

<Technique>	<Order>	<Frequency>
Chebyshev-I	4 / 8 / 10 / 12	5 / 10 / 20 / 50 / 100 / 150 / 200
Chebyshev-II	4 / 8 / 10 / 12	5 / 10 / 20 / 50 / 100 / 150 / 200
Elliptic	4 / 8 / 10 / 12	5 / 10 / 20 / 50 / 100 / 150 / 200
Butterworth	1 / 2 / 3 / 4 / 5 / 6 / 7 / 8 / 9 / 10 / 11 / 12	5 / 10 / 20 / 50 / 100 / 150 / 200
FIR-Hamming	40 / 50 / 60	5 / 10 / 20 / 50 / 100
FIR-Remez	60	5 / 10 / 20 / 50 / 100

lowpassFIR

Syntax:
lowpassFIR(<Order>, <CutoffFreq>, <Channel>)

The channel is filtered using a FIR-lowpass-filter of the given order (>=4th order) and cutoff frequency [Hz].

highpass

Syntax:
highpass (<Technique>;<Order>;<Frequency>;<Expression>)

Parameters:

<Technique>	<Order>	<Frequency>
Chebyshev-I	4 / 8 / 10 / 12	5 / 10 / 20 / 50 / 100 / 150 / 200
Chebyshev-II	4 / 8 / 10 / 12	5 / 10 / 20 / 50 / 100 / 150 / 200
Elliptic	4 / 8 / 10 / 12	5 / 10 / 20 / 50 / 100 / 150 / 200

Butterworth	1 / 2 / 3 / 4 / 5 / 6 / 7 / 8 / 9 / 10 / 11 / 12	5 / 10 / 20 / 50 / 100 / 150 / 200
FIR-Hamming	40 / 50 / 60	5 / 10 / 20 / 50 / 100

highpassFIR

Syntax:
highpassFIR (<Order>, <CutoffFreq>, <Channel>)

The channel is filtered using a FIR-highpass-filter of the given order (>=4th order) and cutoff frequency [Hz].

stopband

Syntax:
stopband (<Technique>;<Order>;<Frequency Range>;<Expression>)

Parameters:

<Technique>	<Order>	<Frequency Range>*
Chebyshev-I	4 / 8 / 10 / 12	50~100 / 50~200 / 100~200 / 150~200
Chebyshev-II	4 / 8 / 10 / 12	50~100 / 50~200 / 100~200 / 150~200
Elliptic	4 / 8 / 10 / 12	50~100 / 50~200 / 100~200 / 150~200
Butterworth	1 / 2 / 3 / 4 / 5 / 6 / 7 / 8 / 9 / 10 / 11 / 12	50~100 / 50~200 / 100~200 / 150~200

The use of the diacritic mark "~" (tilde) between the upper and the lower frequency is mandatory.

Lookup Function

Syntax:

Lookup(<LookupTable>;<param1>[;<param2>…])

Parameters:

LookupTable: The base table for the lookup-value.
For more information on lookup tables see LookupTables.

param1-x: The input parameters for each dimension of the lookup table

Note:

The number of the dimensional-parameters has to match the dimension of the lookup table.

Examples

The following examples use a few channels and functions which are:

speed: The speed of the car in km/h
lapctr: The current lapnumber
HOLD: The last value calculated by this math-channel
dt: Time past since the last calculated sample of this math-channel
sigma: Sums up the expression inside for the whole file up to the current sample

Example with sigma and dt: Integration over time

There is no function for integrating a channel over time. No matter, integration over time is simply the sum of a channel multiplied with the delta-time:

In this example the speed is integrated over the file. The result is the distance driven (assuming the unit of speed is km/h, we have to divide by 3.6 to receive the distance in meters)

sigma(speed*dt) / 3.6

Example with delta and dt: Derivation of a channel

To derivate a channel by time, you have to divide the change of the channel by the change of time:

delta(channel)/dt

Example with HOLD: Filtering measurement errors

Assuming the speed of a car is between 0 and 400 km/h. If we want to clean the speed-channel from all measurement errors (values outside the valid speed-range) we have to find the error values.

If (speed <0 OR speed >400;0;speed)

Now every wrong value is set to 0. Instead of using this static value, you may want to keep the latest reasonable value. This can be accomplished by inserting the HOLD-function: HOLD returns the previous result of the math channel.

If(speed < 0 OR speed > 400; **HOLD**; speed)

Examples with sigma, lapsigma, dt, and delta: Calculating "highspeed"-duration for each lap

In this example you are shown how to use these functions to generate a math channel that calculates the duration driven with more than 200km/h for each lap.

First thing to do is to sum up the time, the car is faster than 200km/h.

sigma(if(speed > 200; dt; 0))

The if-function returns the time since the last sample when speed is greater 200 and 0 if not. Sigma sums up the results of the if-function to get the time past driving faster than 200km/h.

Note:

This current function sums up the time for the whole file.

If you want the time being calculated for each lap individual, you have to reset the sum at the each laptrigger.

Unfortunately the sigma-function works on the whole file by default, so you need to reset the function after each lap. For that matter, the sigma-function has an optional reset condition.

Sigma(if(speed > 200; dt; 0); **delta(lapctr) <> 0**)

The new part in this function does exactly that. If the lap number changed (delta(lapctr) <> 0) sigma resets the sum back to zero and starts again.

> **Note:**
>
> Instead of delta(lapctr) you can also use {laptrig} <> 0.
> Whatever your choice is, take care of the calculation period of your math channel. Especially {laptrig} is a single peak at the lap trigger and can be easily missed, if the calculation period is set to "Slowest channel". Therefore using the lap counter is better, but again: If the period is "slowest channel" and your speed is measured faster, you have to select "fastest channel"!
>
> For more information see also "Computing Tab".

There is a faster way to get the same results as well. Use LapSigma for resetting the sum at the beginning of a new lap.

LapSigma(if(speed > 200; dt; 0))

Hotkeys

This section lists all available hotkeys supported by WinDarab

Global

F1	Show help
Ctrl+F1	Minimize/maximize ribbon
F2	Create new worksheet
F3	Go to the channels pane
F5	Shows the settings dialog for the window
F6	Loads a configuration for the window
F7	Saves the configuration for the window
F9	Load desktop settings
F10	Save the desktop settings
F12	Toggle desktop layout
Ctrl+E	Open measurement file
Ctrl+S	Close a measurement file
Ctrl+I	Open the initialize flashcard dialog
Ctrl+R	Open the import dialog
Ctrl+Shift+C	Show/Hide all captions of the worksheets controls on the current worksheet
Ctrl+L	Zoom current lap and align all overlays
Ctrl+F4	Close active worksheet

Shift+F4	Close active worksheet control
Ctrl+F6	Go to next worksheet
Shift+Ctrl+F6	Go to previous worksheet
Ctrl+C	Copy
Ctrl+Insert	
Ctrl+X	Cut
Shift+Del	
Ctrl+V	Paste
Shift+Insert	

File Explorer Pane

R	Replace the active overlay
F	Go to the fastest lap of the selected overlay
Ctrl+F	Select and align the fastest laps of all overlays

Channels pane

/	Minimize tree (in tree mode only)
*	Expand tree (in tree mode only)

Oscilloscope

G	Show/Hide the grid
P	Go to the previous lap
N	Go to the next lap
F	Go to the fastest lap
Ctrl+F	Select and align the fastest laps of all the overlays
+	Zoom in
-	Zoom out
M	Zoom the marked range
L	Zoom the current lap
S	Zoom the current Segment
T	Zoom between the current marker position and a set tag
A	Show all
Ctrl+Shift+A	Select all channels
Ctrl+A	Select all channels of the current area
Insert	Show the insert menu
R	Show the replace menu
Delete	Show the delete menu
Ctrl+Delete	Clear the oscilloscope
Z	Set the zero-offset
Ctrl+Backspace	Undo the last zoom
Ctrl+Z	
F4	Enable/Disable overlay shifting
Space	Set/Clear tag
X	Toggle axis mode
Ctrl+X	Switch axis mode to custom
Ctrl+Shift+X	Switch axis mode to overlapped
Shift+X	Switch axis mode to stacked

Ctrl+D	Use the selected channel(s) as diff-channel
H	Show/Hide channel
B	Enable/Disable channel blinking
C	Show channel color selector
Y	Show channels y-range dialog
Shift+Y	Set the channels y-range to the default values
Ctrl+Y	Set the channels y-range automatically
D	Shows the channels display options menu
V	Shows the oscilloscope display menu
W	Shows the wide cursor menu
I	Show/Hide quick info of a channel
Ctrl+0	Show the zero line
Ctrl+9	Set the constant line
Ctrl+Up/Down	Shift the visible y-axis-range of the channel

Inform window

Left	Select previous overlay
Right	Select next overlay
PageUp	Select next page
PageDown	Select previous page

Racetrack

A	Show all
Ctrl+2	Switch to 2D-view
Ctrl+3	Switch to 3D-view
Ctrl+P	Toggle perspective view (3D only)
F6	Load racetrack
F7	Save racetrack
Ctrl+Z	Undo the last zoom
Ctrl+Backspace	

XY-Plot

A	Show All
I	Show Info
F11	
Ctrl+Space	Show components
Space	Show regression lines
Insert	Insert a user line
Delete	Delete a user line
Ctrl+Z	Undo the last zoom
Ctrl+Backspace	

Histogram

A	Show all
Ctrl+3	Switch to 3D view
Ctrl+B	Switch to 2D bar view
Ctrl+L	Switch to 2D line view

Ctrl+P	Toggle perspective view (3D only)
Y	Open the y-axis dialog
Ctrl+Y	Automatically adjust the y-axis settings
Ctrl+Z	Undo the last zoom
Ctrl+Backspace	

Distribution

A	Show all (3D only)
Ctrl+2	Switch to 2D-view
Ctrl+3	Switch to 3D-view
Ctrl+G	Toggle grid (3D only)
Ctrl+L	Toggle height lines (3D only)
Ctrl+P	Toggle perspective view (3D only)
Ctrl+S	Toggle solid slices (3D only)
Ctrl+V	Toggle 3D vertices (3D only)
Ctrl+Z	Undo the last zoom
Ctrl+Backspace	

Fast Fourier Transformation

A	Show all
Ctrl+2	Switch to 2D line view
Ctrl+3	Switch to 3D-view
Ctrl+B	Switch to 2D bar view
Ctrl+G	Toggle grid (3D only)
Ctrl+L	Toggle height lines (3D only)
Ctrl+P	Toggle perspective view (3D only)
Ctrl+S	Toggle solid slices (3D only)
Ctrl+V	Toggle 3D vertices (3D only)
Ctrl+Z	Undo the last zoom
Ctrl+Backspace	

Flow Chart

V	Show/Edit variables
Delete	Delete the selected item

OutingReport

F3	Show channel view
F4	Show segment view

Lap analysis

I	Show/hide information

Instrument Panel

Ctrl+D	Duplicate an item

Ctrl+F6	Add settings
Insert	Show the insert item menu
Delete	Delete an Element

ASCII Import File Format

WinDarab supports the importing of logged data series with the help of ASCII files. The following passages contain information about the structure of ASCII files as well as the (optional) Channel Definition File structure.

Special Channels:

The following channels should be defined and receive the suggested names:

- time [s]   Time
- dist [m]   Distance

Channel names "time" and "dist" will be used as "xtime" and "dist", if "xdist" or "xtime" is not available.

Miscellaneous Notes:

A channel name <Channel> may currently contain no more than 7 characters.

- An unit of measurement <Unit> can currently not be longer than 4 characters.
- The character " " (space) and "[" can not to be used in label names (channel name <Channel> or unit <Unit>) . All other characters are allowed.
- Use a decimal point when entering numeric values.
- The character "." (dot) and "," (comma) are interpreted as decimal point. For each value in a file may be used a different decimal point, but within a single value only one of both can be used.

Any numbers can use a colon (-> ":") instead of a decimal comma/point (-> "." / ",")

- Semicolon is used as field separator
- No separator is allowed for thousand grouping.
- There should be at least one space, or one tab space, between "words" and values.
- Channel values are saved as 32-Bit-Float values.
- When a value has the prefix "0x" or the suffix "h", it will be recognized as hex.

ASCII Files may contain

1. as many comment lines as you would like. Each comment line must begin with the number sign (#).
2. one line for the definition of channel columns.
   <Name1> [<Unit1>]
   <Name2> [<Unit2>] …
3. one line for channel values per record.

Example

```
#
# This is a comment
#
xTime [s]               xDist [m]   Speed [km/h]   rev [rpm]
0.0                     0.0         0.0            0.0
0.1                     2.0         100.0          1000.0
0.2                     4.0         200.0          2000.0
0.4                     10.0        160.0          2500.0
```

Channel Definition Files may contain

1. as many comment lines you would like.
   Each comment line must begin with the number sign (#).
2. one line per channel: <Name> [<Unit>] <Minimum> <Maximum>

Example

```
#
# This is a comment
#
xTime                   [s]    0.0    500.0
xDist                   [m]    0.0    10000.0
Speed                   [km/h] 0.0    300
Rev                     [rpm]  0      14000
a_thr                   [deg]  -26    26
```

ASCII Extract File Format

The format of ASCII extract files corresponds to ASCII import files.

WinDarab writes the following information (in the form of a comment) to the ASCII file:

- Date of file creation
- Name of Logged Data File
- Extracted area

Note:

- Channel columns are separated with a tab.
- Numbers are displayed with a decimal point.
- The following characters are replaced in ASCII format as follows:
  " " (space) → "_" (underline)
  "[" → "("
  "]" → ")"

Miscellaneous Settings

Some WinDarab settings can only be changed in the configuration file "setting.ini" located in WinDarabs Configuration directory.

The available options will be discussed below:

Deleting Folders and Files

Initial settings in Darab Explorer have been made to prevent the accidental deletions of folders and files. The delete function will be de-activated when you receive your software.

It is possible to delete Base Directories because deleting a Base Directory does not actually delete data, but merely a link to certain files.

Tip:
It is possible to delete folders and files at any time using Windows Explorer.

If you would like to activate (or deactivate) the default settings for the delete function yourself, please make the following entries in your configuration file:

[DarabExplorer]
AllowDeleteLocation=1
AllowDeleteFolder=1
AllowDeleteFile=1
AllowDeleteSelection=1

One „1" equals "activate"; zero „0" equals "deactivated".

Reading Data

Reading Data with FREE WinDarab version	If a user reads data from FlashCard using the Free-Version of WinDarab, the Darab file protection will be removed from the file. This means that the file can be read by any version of WinDarab.
Reading Data with non-FREE WinDarab version	Any version of WinDarab (except FREE-Version) can import any protected data from FlashCard and save the data as a Darab-Bin-File. But if the file is protected by project code which is not allowed by licensing, the file still cannot be opened.

Auto resolving of supplicated channel names

If a file contains channels with the same name WinDarab automatically appends the name of the data source to all channels except the first one.

If no data source information is available WinDarab will append a unique index. Because the first channel is not changed the second channel will get the index 2.

If a data source starts with '-', the channel will always be renamed if the data source of the other channel doesn't start with a '-'.

If a data source starts with '+', the channel won't be renamed if the data source of the other channel doesn't start with a '+'.

If the data source name of both channels start with either '+' or '-', the channel renaming will be handled as is there is no preferred data source.

If even the appending of the data source name will not result in a unique channel name WinDarab will append an additional index to the resulting channel name.
Finally all channel names will be unique.

The data source name is always appended to the channel description.

WinDarab assigns the data source "-UserDefinied" to all math channels.